Trimble

Create models
Tekla Structures 2026

March 2026

©2026 Trimble Inc. and affiliates



Contents

1 Get to know Tekla Structures basic working methods..... 19

1.1 Set up the workspace..... 19

Work with grids in the model..... 20
- Create, modify, or delete a grid..... 23
- Add or delete a single grid line..... 28
- Modify a single grid line..... 29

Model views in Tekla Structures..... 31
- Create model views..... 34
- Manage model views..... 44
- Switch between model views..... 48
- Update and refresh model views..... 49
- Model view properties..... 50
- Grid view properties..... 51

Define the work area..... 52
- Fit work area to entire model..... 52
- Fit work area to selected parts..... 52
- Fit work area using two points..... 53
- Hide the work area box..... 53
- If you place objects are outside the work area..... 53

Coordinate system..... 54
- Show or hide the work plane grid..... 56
- Shift the work plane..... 56
- Base points..... 60
- Select the work plane..... 72

Change the model background color..... 73

Change the rendering mode for model views..... 76
- Switch between the rendering options..... 77
- DirectX rendering engine..... 77
- DirectX rendering examples..... 78

1.2 Zoom and rotate the model..... 85

Zoom in and out..... 85

Rotate the model..... 85

Pan the model..... 87

1.3 Snap to positions..... 88

Snap to points by using snap switches..... 89
- Snap zone..... 90
- Snap priority..... 90
- Snap depth..... 90
- Visual cues in snapping..... 90
- Override the current snap switch settings..... 91

Snap to parallel points, lines, edges, and extension lines..... 92
- Snap to parallel points..... 92
- Snap to a line or an edge..... 94
- Snap to extension lines..... 95

Snap in orthogonal directions..... 99

Activate the Ortho tool..... 100
Snap to orthogonal points..... 100
Snap in orthogonal direction relative to previously picked points.....101
Settings for the Ortho tool..... 103
Snap to points by using exact distance or coordinates - numeric snapping.....103
Enter a distance or coordinates..... 103
Snapping example: Track along a line towards a snap point.....104
Options for coordinates in numeric snapping.....107
Change the snapping mode.....108
Snapping methods to help to set the snapping direction..... 109
Lock the modeling direction with the Shift key..... 109
Set a temporary reference point..... 110
Lock X, Y, or Z coordinate on a line..... 112
Align objects using a snap grid..... 113
Snap settings.....113
1.4 Start to create model objects..... 114
1.5 Part reference lines and part handles..... 116
Part reference lines..... 116
Part handles..... 117
1.6 Resize and reshape model objects..... 119
1.7 Select objects..... 126
Select single objects..... 126
Select multiple objects using area selection..... 127
Select all objects..... 127
Select previous objects..... 128
Select objects by identifier.....128
Select handles..... 131
Modify the selection.....132
Select assemblies, cast units, and nested objects.....133
Select assemblies and cast units.....133
Select nested objects..... 133
Select reference models, reference model objects and assemblies.....134
Select an entire reference model..... 134
Select a reference model object..... 134
Select a reference model assembly..... 135
Tips for selecting objects..... 135
Switch rollover highlight on or off.....135
Select on right-click..... 136
If you cannot select objects.....136
Interrupt object selection.....137
1.8 Copy objects in models and in drawings..... 137
Copy model objects.....139
Copy drawing objects.....141
Copy objects in a model to another object..... 142
Copy all object content in a model to another object.....143
Copy objects in a model to another plane..... 145
Additional methods for copying objects in models and drawings..... 146
Copy objects in a model or in a drawing by specifying a distance from origin..... 146
Copy objects in a model or in a drawing using drag-and-drop..... 146
Copy objects linearly in a model or in a drawing..... 148
Copy objects from another model..... 148
Copy objects using linear array tool.....149
Copy objects using radial array tool.....152

Copy objects using Array of objects (29) component 154
Move objects in models or in drawings..... 155
Move objects in a model or in a drawing..... 156
Move objects by specifying a distance from origin in a model or in a drawing.....157
Move objects using drag-and-drop in a model or in a drawing.....158
Move objects linearly in a model 160
Move objects to another plane in a model..... 161
Move objects to another object in a model..... 161
Rotate objects in models or in drawings..... 162
Rotate objects around a line in a model 162
Rotate objects around the z axis in a model 162
Rotate objects in a drawing.....164
Rotation settings..... 164
Mirror objects in models or in drawings.....165
Mirror objects in a model.....165
Mirror objects in a drawing.....165
Tips for copying and moving objects in models and in drawings.....166
1.9 Filter objects..... 168
Use existing filters..... 169
How to use a view filter..... 169
How to use a selection filter.....171
Create new filters.....172
Create a view filter.....172
Create a selection filter.....175
Create a drawing filter..... 176
Create a drawing view filter..... 178
Create a drawing selection filter.....181
Object categories and properties in filtering..... 181
Filtering techniques.....198
Select values from the model.....201
Wildcards 202
Examples of filters..... 202
Filter parts based on their name.....202
Filter main parts..... 203
Filter bolts based on their size.....204
Filter parts based on their assembly type.....205
Filter sub-assemblies..... 206
Filter objects based on their class.....206
Filter reference model objects.....207
Filter parts within component..... 208
Filter reinforcement in pour units based on pour object type..... 208
Filter all contents of a pour unit..... 209
Copy and remove filters.....210
Copy a filter to another model..... 210
Remove a filter.....210
1.10 Create model templates.....211
Create a new model template.....211
Modify an existing model template.....212
Download model templates.....212
Model template options..... 212
1.11 Tips for large models..... 214
1.12 How to model identical areas..... 216
2 Create parts, reinforcement, and construction objects..... 217

2.1 Create parts and modify part properties..... 218
Create a steel column..... 220
Modify steel column properties..... 221
Steel column properties..... 221
Create a steel beam..... 223
Modify steel beam properties..... 223
Steel beam properties..... 223
If beams are created close to each other..... 225
Create a steel polybeam..... 226
Modify steel polybeam properties..... 227
Steel beam properties..... 227
Create a curved steel beam..... 229
Modify curved beam properties..... 230
Steel beam properties..... 230
Create a twin profile..... 232
Modify twin profile properties..... 233
Twin profile properties..... 233
Create an orthogonal beam..... 235
Modify orthogonal beam properties..... 236
Orthogonal beam properties..... 236
Create a steel spiral beam..... 237
Basic concepts related to spiral beams..... 237
Create a spiral beam..... 238
Limitations..... 240
Steel spiral beam properties..... 240
Create a contour plate..... 242
Create a round contour plate..... 242
Modify contour plate properties..... 243
Contour plate properties..... 243
Create a conical or a cylindrical bent plate..... 245
Create a cylindrical bent plate..... 245
Create a conical bent plate..... 248
Modify the bend radius..... 251
Modify the shape of a bent plate..... 253
Remove curved sections..... 257
Examples..... 258
Modify bent plate properties..... 259
Bent plate properties..... 259
Create a stand-alone bent plate..... 261
Create a stand-alone bent plate..... 261
Modify the shape of a stand-alone bent plate..... 264
Modify bent plate properties..... 266
Bent plate properties..... 266
Create a steel lofted plate..... 268
Prerequisites and examples of lofted plates..... 268
Create a lofted plate..... 270
Modify the shape of a lofted plate..... 274
Split a lofted plate..... 275
Swap the end handle points to correct the geometry of a lofted plate..... 276
Unfold lofted plates..... 277
Modify lofted plate properties..... 278
Lofted plate properties..... 278
Create a concrete column..... 281
Modify concrete column properties..... 281
Concrete column properties..... 282

Create a concrete beam....284
Modify concrete beam properties....284
Concrete beam properties....284
Create a concrete polybeam....287
Modify concrete polybeam properties....288
Concrete beam properties....288
Create a concrete spiral beam....291
Basic concepts related to spiral beams....291
Create a spiral beam....291
Limitations....293
Concrete spiral beam properties....293
Create a concrete panel or wall....295
Modify concrete panel or wall properties....297
Concrete panel or wall properties....297
Create a concrete slab....299
Create a round concrete slab....300
Modify concrete slab properties....301
Concrete slab properties....301
Create a concrete lofted slab....303
Prerequisites and examples of lofted slabs....303
Create a lofted slab....305
Modify the shape of a lofted slab....309
Split a lofted slab....310
Swap the end handle points to correct the geometry of a lofted slab....310
Modify concrete lofted slab properties....311
Lofted slab properties....311
Create a pad footing....314
Modify pad footing properties....314
Pad footing properties....315
Create a strip footing....316
Modify strip footing properties....317
Strip footing properties....318
Create items....320
Create an item or a concrete item....320
Modify item or concrete item properties....322
Change the shape of an item....322
Convert a part to an item....323
Item and concrete item properties....323
Modify items....325
Start geometry editing....326
Modify the geometry of an item....327
Add an edge to an item....328
Add a vertex to an item....329
Save a modified item and shape....331
Create timber objects....331
Modify parts and part properties further....332
Select and change the profile or material of a part....332
Adjust part position....334
Bind parts to grid levels....350
Camber a part....353
Warp a part....354
Examples of user-defined attributes (UDAs) for parts....356
Create curved beams, panels, or strip footings....357
Combine parts....358
Attach parts to each other....359

Split parts... 361
2.2 Add details to parts... 363
Create bolts... 363
Create a bolt group... 364
Create a single bolt... 366
Create bolts using the Auto bolt component... 366
Create a bolt group by exploding a component... 369
Change or add bolted parts... 370
Bolt group shape... 370
Bolt properties... 371
Create studs... 375
Create bolt holes... 376
Create round holes... 376
Create oversized holes... 377
Create slotted holes... 378
Create tapped holes... 379
Create different holes with one bolt group... 380
Create welds... 381
Create a weld between parts... 381
Create a weld to a part... 382
Create a polygon weld... 383
Weld properties... 384
List of weld types... 389
Welds in components... 391
Weld preparation... 392
Set the visibility and appearance of welds... 393
Change a weld to a polygon weld... 395
Split a polygon weld... 395
Create user-defined cross sections for welds... 396
Fit part ends... 397
Create fittingsFit part ends... 397
Fit beams and columns... 400
Fit objects... 403
Cut objects with a line, a polygon, or a part... 412
Line cuts... 413
Polygon cuts... 414
Part cuts... 416
Hints and tips for efficient cutting... 418
Hide cuts in a model view... 418
Polygon cut properties... 419
Part cut properties... 419
Chamfer part corners and edges... 420
Chamfer part corners... 420
Chamfer part edges... 421
Corner chamfer properties... 422
Edge chamfer properties... 424
Add surface treatment to parts... 425
Add surface treatment to an entire part face... 426
Add surface treatment to a selected area on a part face... 426
Add surface treatment to all faces of a part... 426
Add surface treatment to cut faces... 427
Surface treatment on chamfered parts... 427
Surface treatment on parts with openings and recesses... 428
Modify surface treatment properties... 428
Surface treatment properties... 428

Define new surface treatment subtypes....430
Tiled surface treatment....431
Create an unpainted area using the No paint area component....435
Add surfaces to part faces and pour object faces....436
Add a surface to a face....437
Modify surface properties....438
Modify the adaptivity of reinforcement, surface treatment, or edge chamfers in parts....438
Define default adaptivity settings....439
Modify the adaptivity of an individual model object....439
Display detailing of a part....439
2.3 Work with assemblies....440
Assembly types and assembly hierarchy....441
Assembly types....441
How to check the assembly hierarchy in a nested assembly....444
Assembly examples....446
Use bolts and welds to create and connect assemblies and sub-assemblies....448
Use bolts to create and connect assemblies....448
Use welds to create and connect assemblies....449
Modify the assembly structure....450
Add parts to an existing assembly....450
Create nested assemblies....451
Join existing assemblies....451
Create a sub-assembly of assembly parts....452
Check and highlight objects in an assembly....452
Change the main part in an assembly....452
Change the main assembly....453
Remove objects from assemblies....453
Remove a part or a sub-assembly from an assembly....454
Explode an assembly or a sub-assembly....454
2.4 Create cast units....454
Create a cast unit....454
Check and highlight objects in a cast unit....455
Change the cast unit main part....455
Define the cast unit type of a part....455
Some best practices when you are working with cast units....456
Add objects to cast units....457
Remove objects from cast units....458
Remove objects from a cast unit....458
Explode a cast unit....458
Casting direction....458
Casting direction for concrete and non-concrete parts....458
Define the casting direction of a part....460
Show the top-in-form face....460
2.5 Batch editing of assemblies or cast units....461
Limitations and recommendations in using Batch editor....463
Edit similar assemblies or cast units with Batch editor....464
Settings in Batch editor....466
Work with property columns in Batch editor....467
2.6 Manage pours....468
Enable pour management....469
Disable pour management temporarily....470
View cast-in-place concrete structures....471
Set the appearance of cast-in-place concrete structures....471

Part view vs. pour view... 473
Define the pour phase of a part... 474
Pour objects... 475
Change the color and transparency of pour objects... 476
Modify the properties of a pour object... 477
Pour units... 477
Calculate pour units... 478
Check and inquire objects in a pour unit... 478
Add objects to a pour unit... 479
Remove objects from a pour unit... 480
Reset pour unit relations... 480
Modify the properties of a pour unit... 481
How Tekla Structures automatically adds objects to pour units... 481
Pour breaks... 482
Pour break adaptivity... 483
Set the visibility of pour breaks... 484
Create a pour break... 485
Modify a pour break... 487
Troubleshoot pours... 488
Example: Create concrete geometry and work with pours... 491
2.7 Create reinforcement... 493
Create a rebar set... 494
Basic concepts related to rebar sets... 494
Create longitudinal rebars... 496
Create crossing rebars... 498
Create rebars by face... 501
Create rebars by guidelines... 503
Create rebars by point input... 505
Rebar set properties... 506
Limitations... 507
Create a rebar set using Rebar shape placing tool... 507
Examples: Rebar sets in curved structures... 513
Create a single reinforcing bar... 520
Create a reinforcing bar group... 521
Create a reinforcing bar group using Rebar shape catalog... 523
Create a curved reinforcing bar group... 529
Create a circular reinforcing bar group... 531
Create a tapered or spiral reinforcing bar group... 533
Create a reinforcement mesh... 536
Create a rectangular reinforcement mesh... 536
Create a polygonal reinforcement mesh... 537
Create a bent reinforcement mesh... 539
Create a customized reinforcement mesh... 540
Create a reinforcement strand pattern... 541
Debond reinforcement strands... 542
Create a reinforcement splice... 543
2.8 Modify reinforcement... 545
Modify a rebar set... 545
Modify the properties of a rebar set... 546
Change the layer order of a rebar set... 546
Modify a rebar set using guidelines... 547
Change the bar plane orientation of a rebar set... 549
Modify a rebar set using leg faces and leg surfaces... 549
Modify a rebar set locally using modifiers... 557
How to cut rebar sets... 565

Distribute bars in a rebar set....566
Split a rebar set....568
Split bars in a rebar set....569
Freeze or unfreeze a rebar set....571
Modify a single reinforcing bar, bar group, or mesh....572
Distribute bars in a reinforcing bar group....576
Delete bars from a reinforcing bar group....578
Ungroup a reinforcement....579
Group reinforcement....580
Combine two reinforcing bars or reinforcing bar groups into one....581
Split a reinforcing bar group....582
Split and splice a reinforcement....582
Use handles to modify a reinforcement....584
Add hooks to reinforcing bars....585
Define the reinforcement cover thickness....588
Select the definition for a reinforcement....591
Use adaptivity to modify a reinforcement....591
Attach a reinforcement object to a concrete part....593
Check the validity of reinforcement geometry....594
Classify reinforcement to layers....595
How Tekla Structures automatically attaches rebar set bars to concrete parts....595
How to calculate the reinforcing bar length....599
How to calculate the reinforcing bar leg length....602
Reinforcement shape recognition....603
Rebar shape manager in reinforcement shape recognition....604
Hard-coded bending type identifiers in reinforcement shape recognition....617
Reinforcement in templates....641
2.9 Work with rebar assemblies....643
Create a rebar assembly....644
Create a basic rebar assembly....645
Rebar assembly properties....645
Modify a rebar assembly....646
Add objects to an existing rebar assembly....646
Create a nested rebar assembly....647
Check and highlight objects in a rebar assembly....647
Change the main object of a rebar assembly....647
Change the coordinate system of a rebar assembly....648
Remove objects from a rebar assembly....649
Remove an object from a rebar assembly....649
Explode a rebar assembly or a sub-assembly....649
2.10 Create construction objects and points....650
Create a construction line....651
Create a construction plane....652
Create a construction circle....652
Create a construction arc....653
Create a construction polycurve....655
Copy a construction object with offset....656
Modify a construction object....657
Create points....660
Create points on a line....661
Create points on a plane....662
Create points parallel to two points....662
Create points along the extension line of two points....663
Create projected points on a line....664
Create points along an arc using center and arc points....664

Create points along an arc using three arc points..... 665
Create points tangent to a circle..... 666
Create points at any position..... 666
Create bolt points..... 667
Create points at the intersection of two lines..... 667
Create points at the intersection of a plane and a line..... 667
Create points at the intersection of a part and a line..... 667
Create points at the intersection of a circle and a line..... 668
Create points at the intersection of two part axes..... 668
Import points..... 669
Point properties..... 669

3 Adjust how model objects are displayed 671

3.1 Change the rendering of parts, components, reference objects and building hierarchy..... 671

3.2 Adjust the display settings..... 676
Set the visibility and representation of model objects..... 676
Visibility and representation settings..... 678

3.3 Change the part representation to show parts with exact lines or with high accuracy..... 681
Show parts with exact lines..... 681
Show parts with high accuracy..... 681

3.4 Temporarily hide model objects or show only selected model objects.. 682
Hide parts or other objects in a model view..... 682
Show only selected parts or other objects in a model view..... 684
Temporarily show assembly and component objects in a model view..... 685

3.5 Change the color and transparency of model objects by using object representation..... 686
Use the Class property to change the color of parts and reinforcement..... 687
Define color and transparency settings for object groups..... 688
Color settings in object representation..... 689
Transparency settings in object representation..... 690
Define your own colors for object groups in object representation..... 690
Copy object representation settings to another model..... 692

3.6 Use object groups in object representation and in filters..... 693
Where object groups are used..... 693
Create an object group for object representation..... 693
Object group - Representation settings..... 694
Copy object groups to another model..... 695

3.7 If you cannot see all objects..... 695

4 Check the model..... 697

4.1 Search for model objects..... 697
Search in the entire model..... 699
Search within the selected model objects..... 699
Review the search results..... 699
Show or hide the Model search toolbar..... 700

4.2 Visualize the model with Trimble Connect Visualizer..... 700
Working in Trimble Connect Visualizer..... 701
Zoom, rotate, or pan the rendered model..... 701
Adjust the scene..... 701

Take and view snapshots....703
Create animations....704
Return to the initial view of the model....705
Enter or exit full screen mode....705
Show or hide the Trimble Connect Visualizer side pane....706
Use Trimble Connect Visualizer in VR mode....706
Modify material mappings for Trimble Connect Visualizer....707
Create and modify user-defined materials in Trimble Connect Visualizer....709
4.3 Create clip planes and clip boxes....713
Create a clip plane....713
Create a clip box....715
Create a view depth clip plane....718
4.4 Show parts, components, or assemblies in a selected view angle....719
4.5 Fly through the model....720
4.6 Inquire object properties....721
Object property report templates....723
Custom inquiry....724
Use the Custom Inquiry tool....724
Define what information is shown by Custom Inquiry tool....725
Modify the default attributes in InquiryTool.config file....727
4.7 Measure objects....728
Measure distances....728
Measure angles....729
Measure arcs....729
Measure bolt spacing....730
4.8 Detect clashes....731
Find clashes in a model....732
Check the model for clashes....732
Symbols used in clash check....734
About clash types....734
Open and save clash check sessions....737
Modify, view, and print clash check results....739
Modify clash check results....739
View clash check results....741
Print clash check results....742
4.9 Compare parts or assemblies....743
4.10 Show part and reinforcement information by using part and rebar labels....744
4.11 Check solid errors....746
Check the model for solid errors using Solid checker....746
View solid errors in a log file....747
4.12 Diagnose and repair the model....748
4.13 Find distant objects....750
5 Number the model....752
5.1 What is numbering and how to plan it....752
Numbering series....753
Plan your numbering series....754
Assign a numbering series to a part....755
Assign a numbering series to an assembly....755
Overlapping numbering series....756

Identical parts... 757
Identical reinforcement... 758
Define what affects numbering... 758
User-defined attributes in numbering... 759
5.2 Adjust the numbering settings... 760
Family numbering... 760
Assign family numbers... 761
Reinforcement sequence numbering... 762
Assign reinforcement sequence numbers... 763
Modify reinforcement sequence numbering settings... 765
5.3 Number parts and other model objects... 767
Number a series of parts or other model objects... 768
Number reinforcement... 769
Number assemblies, cast units, and rebar assemblies... 769
Number welds... 770
Save preliminary numbers... 771
5.4 Preview the numbering results... 771
5.5 Check the numbering... 777
5.6 View the numbering history... 780
5.7 Repair numbering errors... 781
5.8 Change existing numbers... 782
5.9 Clear existing numbers... 783
5.10 Renumber the model... 783
5.11 Control numbers... 784
Assign control numbers to parts... 784
Control number order... 785
Display control numbers in the model... 786
Remove control numbers... 787
Lock or unlock control numbers... 788
Example: Use control numbers to indicate the erection order... 788
5.12 Number parts by design group... 790
5.13 Numbering examples... 793
Example: Number identical beams... 793
Example: Use family numbers... 794
Example: Number selected part types... 795
Example: Number parts in selected phases... 796
5.14 Tips for numbering... 798
Numbering settings during a project... 798
Create a standard-part model... 799

6 System components and applications... 801
6.1 Work with system components... 803
Define component properties... 803
Add a component to a model... 805
Modify a component in a model... 808
View a component in a model... 809
Detailing and conceptual components... 810
6.2 Work with applications... 811
6.3 How to use the Applications & components catalog... 814

Groups in the catalog....814
Search for a component in the catalog....815
Change the view in the catalog....815
Show selected components in the catalog....816
View and modify component information in the catalog....816
Add a thumbnail image for a component in the catalog....817
Publish a component in the catalog....817
Create and modify groups in the catalog....818
Change the order of groups in the catalog....819
Hide groups and components in the catalog....819
Show the catalog message log....820
Catalog definitions....820
6.4 Automate connection creation 820
AutoConnection....820
Define AutoConnection settings and rules....821
Create a connection using AutoConnection....824
AutoDefaults....826
Define AutoDefaults settings and rules....826
Modify a connection using AutoDefaults....831
AutoConnection and AutoDefaults rules....831
Combining and iterating properties for AutoDefaults....834
AutoDefaults example: Using iteration with connection check....836
Using reaction forces and UDLs in AutoDefaults and AutoConnection....838
6.5 Advanced component settings 839
Define connection properties in the joints.def file....839
How to use the joints.def file....840
Example: How Tekla Structures uses the joints.def file....842
General defaults in the joints.def file....843
Bolt diameter and number of bolts in the joints.def file....844
Bolt and part properties in the joints.def file....846
Excel spreadsheets in connection design....852
Files used in Excel spreadsheet connection design....853
Example of an Excel spreadsheet in connection design....854
Example of visualizing the Excel connection design process....858
Showing connection status in Excel connection design....862
7 Custom components....863
7.1 Define custom components....866
Explode an existing component....866
Define a custom component in the Custom component wizard....867
Add a custom component to a model....870
Properties in the Custom component wizard....871
Typical examples of custom components....874
Custom component example: define an end plate custom component....879
Custom component example: define a nested custom component....883
Custom component example: add or move a custom part in the model....885
7.2 Edit custom components....887
7.3 Add variables to a custom component....893
Bind component objects to a plane (create distance variables)....898
Bind objects manually (create distances)....898
Bind objects automatically (create automatic distances)....902
Test a binding....904
Check a binding....905

Delete a binding..... 906
Example: Bind an end plate to a plane.....906
Bind component objects using magnetic construction planes or lines.....908
Bind handles using a magnetic construction plane.....908
Bind handles using a magnetic construction line..... 910
Add a distance between component objects (create reference distance variables)...911
Use parametric variables to set object properties..... 913
Example: Create a parametric variable for weld size.....914
Copy properties and property references from another object.....916
Create a variable formula.....917
Example: Create a variable formula to set the weld size based on flange thickness 918
Functions in variable formulas..... 920
Arithmetic operators.....920
Logical and comparison operators..... 920
Reference functions..... 921
ASCII file as a reference function.....922
Mathematical functions.....923
Statistical functions..... 924
Data type conversion functions.....925
String operations.....926
Trigonometric functions..... 928
Market size function..... 928
Framing condition functions.....929
How to avoid cyclic dependencies in variable formulas.....931
7.4 Tips for creating custom components..... 932
7.5 Examples of parametric variables and variable formulas in custom components..... 934
Variable properties in custom components 936
Example of a variable formula: Create new component objects.....942
Example of a variable formula: Change the end plate material..... 944
Example of a variable formula: Create a nested custom connection with stiffeners. 945
Example of a variable formula: Replace a sub-component in a nested component.. 949
Example of a variable formula: Modify a sub-component by using a component attribute file in a nested component..... 951
Example of a variable formula: Define the bolt size and bolt standard..... 953
Example of a variable formula: Calculate the bolt group distance from the beam flange..... 956
Example of a variable formula: Calculate the number of bolt rows..... 958
Example of a variable formula: Define the stiffener position using construction planes.....960
Example of a variable formula: Link parametric variables to user-defined attributes..... 964
Example of a variable formula: Calculate the number of handrail posts using a template attribute..... 966
Example of a variable formula: Create a custom stiffener detail with variables.....970
Create a basic custom stiffener detail..... 970
Create bindings to control the stiffener shape.....972
Create bindings to control the stiffener position..... 975
Create variables to control the stiffener thickness..... 976
Create variables to control the creation of stiffener plates..... 978
Example of a variable formula: Link an Excel spreadsheet to a custom component. 980
Examples of a variable formula: Rebar set modifiers in custom components.....982
Example: Define the class and size of rebar set bars using a property modifier...982

Example: Create and modify rebar hooks using an end detail modifier....984
7.6 Import and export custom components....987
Export a custom component....987
Import a custom component....988
7.7 Frequently asked questions about custom components....989
General....989
Creating distance variables (binding)....990
Editing variables....992
Custom component browser....994
Editing an input file....995
7.8 Customize the custom component dialogs....995
Modify a custom component dialog....995
Custom component input files....997
Lock or unlock the custom component input file....998
Custom component dialog editor settings....998
Customize the custom component dialogs by using a text editor....999
Add new tabs....1000
Add text boxes....1000
Add images....1001
Change the order of boxes....1002
Change the location of boxes....1002
Example: Add a group of checkboxes in the custom component dialog....1003
Default properties of custom component dialogs....1007
Default properties of custom connections, details, and seams....1007
Default properties of custom parts....1009
Example: Customize the dialog of a custom stiffener detail....1011
Example: Add a list with images in a custom component dialog....1012
Example: Arrange text boxes and labels in a custom component dialog....1017
Example: Dim unavailable options in a custom component dialog....1019
8 Predefined parametric profiles available in Tekla Structures....1023
8.1 I profiles....1023
8.2 I beams (steel)....1024
8.3 L profiles....1024
8.4 Z profiles....1025
8.5 U profiles....1026
8.6 C profiles....1026
8.7 T profiles....1027
8.8 Welded box profiles....1027
8.9 Welded beam profiles....1027
8.10 Box profiles....1030
8.11 WQ profiles....1031
8.12 Rectangular sections....1031
8.13 Circular sections....1032
8.14 Rectangular hollow sections....1032
8.15 Circular hollow sections....1033
8.16 Cold rolled profiles....1033

8.17 Folded plates....1036
8.18 Hat profiles....1043
8.19 I beams (concrete)....1044
8.20 Ledger beams (concrete)....1044
8.21 T profiles (concrete)....1045
8.22 Irregular beams (concrete)....1047
8.23 Panels....1050
8.24 Variable cross sections....1053
8.25 Others....1055

9 Modeling settings....1057
9.1 Numbering settings....1057
General numbering settings....1057
Weld numbering settings....1059
Control number settings....1060
9.2 Reinforcement settings....1061
Reinforcing bar and bar group properties....1061
Reinforcement mesh properties....1064
Custom reinforcement mesh properties....1066
Rebar set properties....1068
Secondary guideline properties....1073
Leg face properties....1075
Leg surface properties....1076
Property modifier properties....1078
End detail modifier properties....1081
Splitter properties....1085
Reinforcement strand properties....1088

10 Disclaimer....1091

1 Get to know Tekla Structures basic working methods

Before starting to create models and drawings, get to know some basic working methods that you will need to work efficiently both in the Tekla Structures models and in the drawings.

We recommend you to familiarize yourself how to

- work with grids (page 20) and views
- define the work area (page 52) and coordinates (page 54) that affect your work
- zoom and rotate (page 85) the model
- create (page 217), select (page 125) and move (page 155) objects
- position objects by using snapping (page 88)
- filter objects (page 168) both in the modeling mode and in the drawing mode.

See also

Create parts and modify part properties (page 217)

1.1 Set up the workspace

Before starting to model, check that your Tekla Structures workspace is set up correctly.

1. Define the units and decimals you will use.
2. Modify the grid to suit your needs. (page 20) Create a modular grid if needed.
3. Create some views (page 31) to examine the model from different angles and elevations.

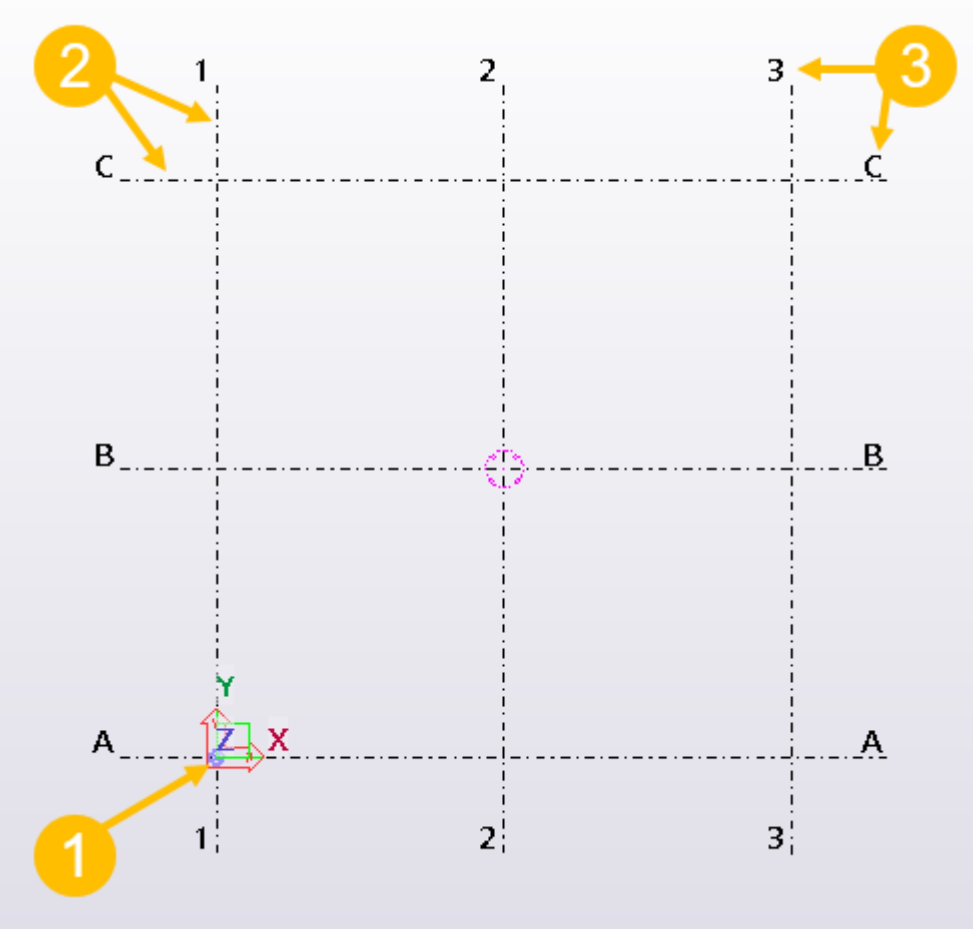
4. Resize the work area to suit your project. (page 52)
5. Get familiar with the coordinate system (page 54). If you are modeling sloped structures, shift the work plane accordingly. (page 56)

Work with grids in the model

A *grid* represents a three-dimensional complex of horizontal and vertical planes. The grid is shown on the view plane using dash-and-dot lines. You can create both rectangular and radial grids. Use grids as an aid in locating objects in a model. You can make rectangular grids and *grid lines* act magnetically so that the objects on the grid lines of a rectangular grid follow if you move the grid line.

- Create, modify, or delete a grid (page 23)
- Add or delete a single grid line (page 27)
- Modify a single grid line (page 29)

Grid terminology



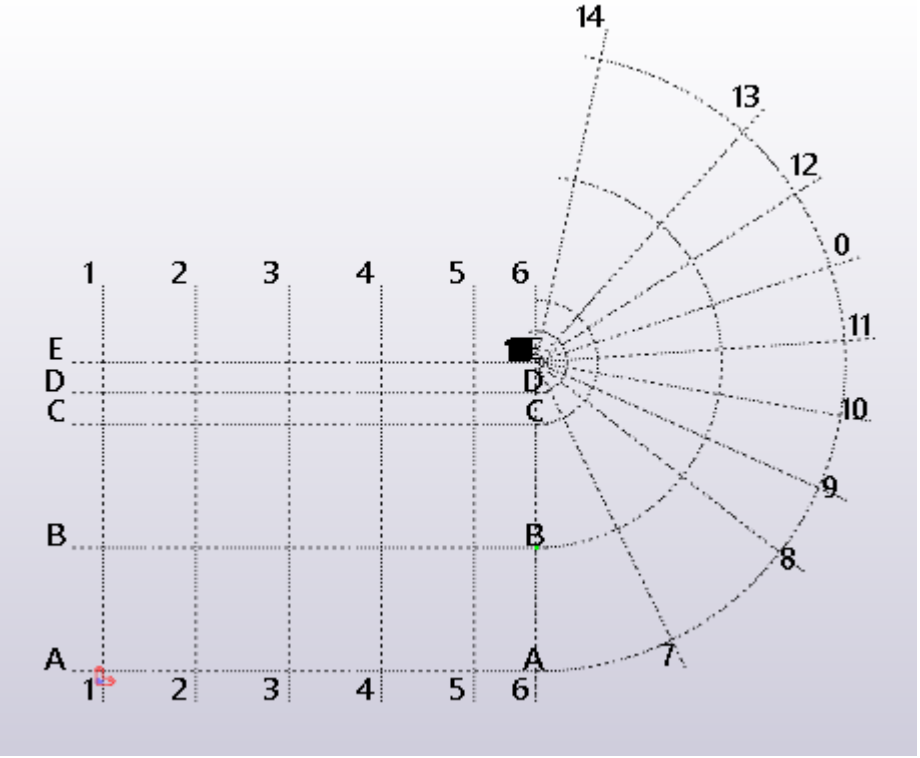
(1) Grid origin is the point where the zero points of each coordinate axis intersect

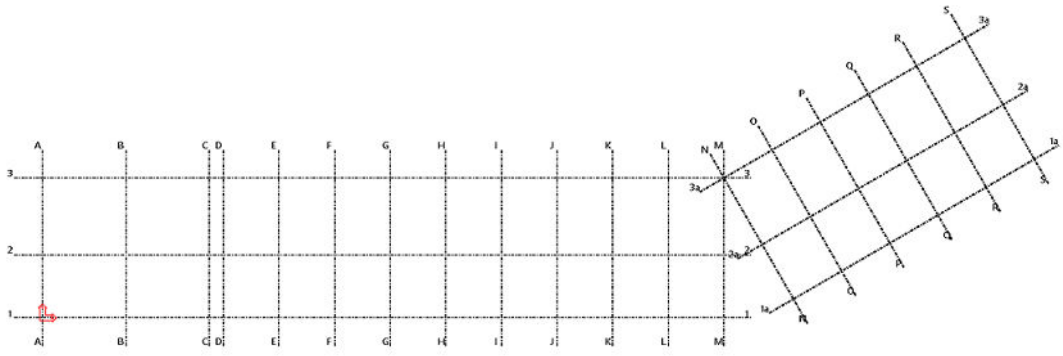
(2) Grid line extensions define how far the grid lines extend in each direction

(3) Grid labels are the names of the grid lines shown in views

Several grids in one model

You can have more than one grid in a model. You can create a large-scale grid for the entire structure, and smaller grids for some detailed sections. Create as many grids as needed, so that you can easily place objects in your model.

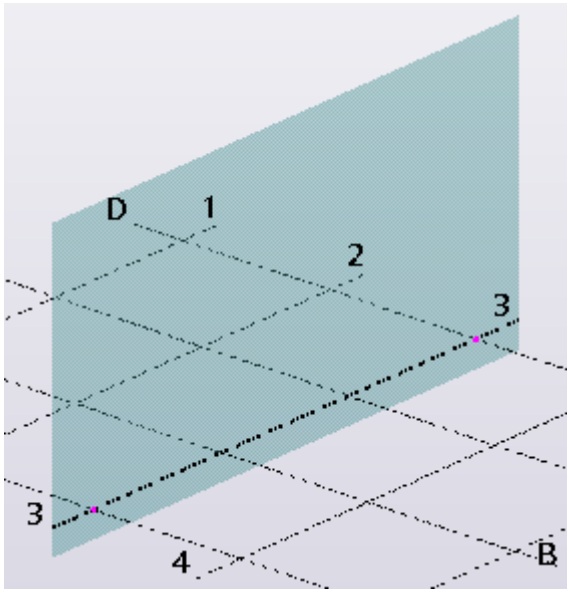


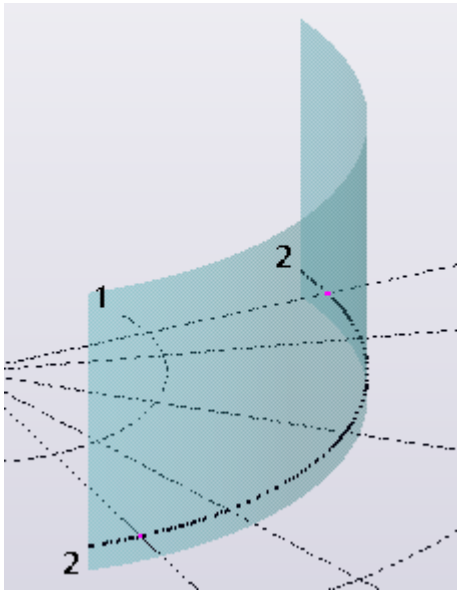


Single grid lines

You can create single grid lines and attach them to an existing grid.

Single grid lines have handles (page 117). If the  **Select grid line** selection switch is active and you select a grid line, the handles appear in magenta. If you move the handles to make a skewed grid, you can do this only on the local XY plane (page 31) of the grid.





If you cannot see grid in the model

Sometimes grid is not visible in the model. Possible reasons can be:

- **View depth**

  View depth has no effect on grid visibility. The grid is only shown in the view if any of the grids' planes intersect with the view plane. To move the view

plane to make the grid visible, click the view, right-click and select **Move** --> **Linear**.

- **Grid visibility**

  Grid can be hidden. You can control the grid visibility in the **Display** settings. Double-click the view to open the **View Properties** dialog. Click the **Display...** button to open the **Display** dialog, and make sure that the **Grids** checkbox is selected.

- **Work area**

  Grid can be outside the work area and hence it is not visible in the current view. To resize the work area to include grids, click the view, right-click and select **Fit work area to entire model**.

- **Grid does not exist**

  Grid may have been deleted, or it has been not created yet.

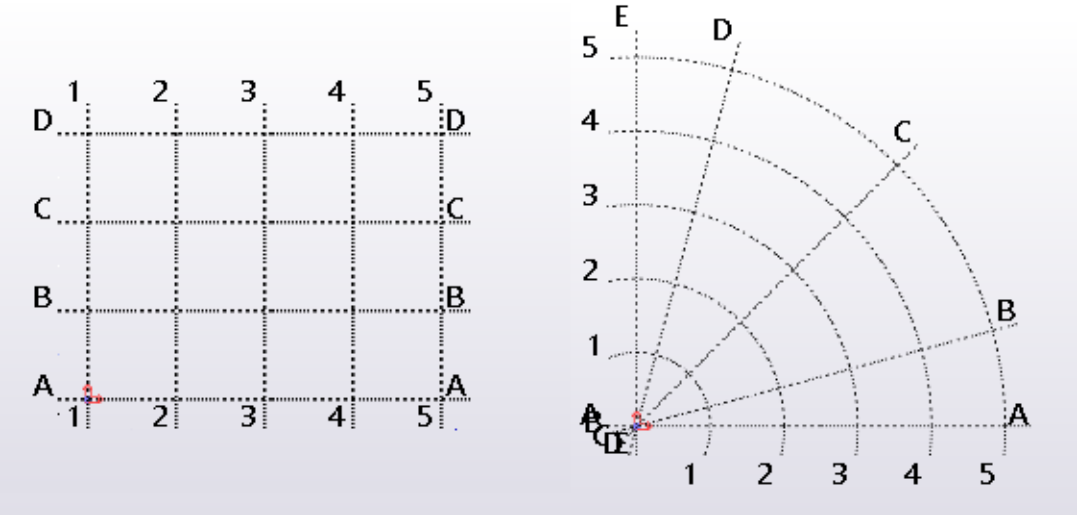
For more information, see Why is the grid not visible in the 3D view?

Create, modify, or delete a grid

When you create a new model, Tekla Structures automatically creates a rectangular grid and a view according to the saved standard properties. If needed, you can later modify the grid properties, or create new rectangular and radial grids. You can modify an existing grid, or delete it. Use the grid properties in the property pane to view and modify the grids.

Create a rectangular grid or a radial grid

You can create both rectangular and radial grids and modify their properties in the property pane.



1. Select whether to create a rectangular or a radial grid.

- To create a rectangular grid: on the **Edit** tab, click **Grid** --> **Create rectangular grid**.
- To create a radial grid: on the **Edit** tab, click **Grid** --> **Create radial grid**.

Tekla Structures shows a preview of the grid. You can modify the grid properties in the property pane before inserting the grid. The grid preview changes according to the changes in the property pane.

2. Pick a point in the model to indicate the origin of the grid, or click the middle mouse button to accept the grid property values in the property pane.
   - If you pick an origin point to the grid, the grid is created using the properties in the property pane, and the origin is the location you picked.
   - If you click the middle mouse button, the grid is created using the properties in the property pane, and to the origin defined in the property pane.

     The coordinates of the origin are shown in the property pane in the **Origin** section as the **X**, **Y**, and **Z** values.

NOTE When working with very large grids, always having the grid labels visible might slow down Tekla Structures. To hide the grid labels when you zoom in, use the advanced option XS_ADJUST_GRID_LABELS.

Create a radial grid (alternative method)

This is an alternative method to create a radial grid. You can create a radial grid using a component called **Radial Grid**. Note that the curved grid lines created by the **Radial Grid** component are not truly curved but straight.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Start typing `radial grid` in the search box.
3. Double-click **Radial Grid** to open the properties dialog.
4. Modify the grid properties.

   In the coordinate properties
   - **X** defines the location of the curved grid lines and the distance between the grid lines.

     The first value is the radius of the innermost curved grid line.
   - **Y (degrees)** defines the location of the straight grid lines and the distance between the grid lines in degrees.

     The first value defines how the grid is rotated. The grid is rotated counter-clockwise from the x axis in the current work plane.
5. Click **OK**.

6. Pick a point to indicate the origin of the grid.

   The grid is created automatically.

Modify a grid

Double-click an existing grid to modify it.

1. Ensure that the  **Select grid** selection switch is active.

   The **Select grid** switch is on the **Selecting** toolbar. The toolbar is by default located at the bottom of the screen.
2. Double-click a grid line.

   Depending on the type of the grid, the **Rectangular grid** or the **Radial grid** properties open in the property pane.
3. Change the grid properties.
4. Click **Modify** to apply the changes.

Delete a grid

When you delete an entire grid, ensure that you do not have any other objects selected. Otherwise Tekla Structures only deletes the objects, not the grid.

1. Ensure that **only** the  **Select grid** selection switch is active.

   The **Select grid** switch is on the **Selecting** toolbar. The toolbar is by default located at the bottom of the screen.
2. Select the grid.
3. Press **Delete**.
4. Confirm that you want to delete the grid.

Grid properties

Use the **Rectangular grid** or the **Radial grid** properties in the property pane to view and modify the properties of a grid. To open the properties, double-click the grid. The file name extension of a rectangular grid property file is `*.grd`. The file name extension of a radial grid property file is `*.rgrd`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
Coordinates	In the **Rectangular grid** properties: Coordinates of the grid in the x, y, and z directions. **X**: grid lines vertical to the work plane **Y**: grid lines horizontal to the work plane

Setting	Description

	Z: elevations in the structure
	You can enter a maximum of 1024 characters. Use a zero at the start to represent a grid on the 0,0 coordinate and spaces as separators for coordinates.
	The x and y coordinates are relative, which means that the entries for **X** and **Y** are always relative to the previous entry. The **Z** coordinates are absolute, which means that entries for z are absolute distances from the work plane origin.
	You can either define the coordinates individually, or you can define several grid lines with equal spacing. Both of the following coordinate entries create three grid lines with the spacing of 4000:
	`0 4000 4000`
	`0 2*4000`

	In the **Radial grid** properties:
	Radial and angular values of the grid lines. Elevations is the value of the grid in z direction.
	Radial: curved grid lines. If you enter only one value for radius, it must be >0.
	Angular: straight grid lines. You can enter negative angle values, if needed.
	Elevations: elevations in the structure
	Use a zero at the start to represent a grid on the 0,0 coordinate and spaces as separators for coordinates.
	The **Radial** and **Angular** coordinates are relative, which means that the entries are always relative to the previous entry. The **Elevations** are absolute, which means that entries are absolute distances from the work plane origin.
	You can either define the coordinates individually, or you can define several grid lines with equal spacing. Both of the following coordinate entries create three grid lines with the spacing of 4000:
	`0 4000 4000`
	`0 2*4000`

Labels	In the **Rectangular grid** properties: Names of the grid lines shown in views.

Setting	Description
	Names in the **X** box are associated with the grid lines parallel to the y axis and vice versa. The **Z** box is for the names of levels parallel to the work plane. If you wish, you can leave the label boxes empty.
	In the **Radial grid** properties: Names of the grid lines shown in views. Names in the **Radial** box are associated with the curved grid lines. Names in the **Angular** box are associated with the straight grid lines. The **Elevations** box is for the names of levels parallel to the work plane. If you wish, you can leave the label boxes empty.
Line extensions	In the **Rectangular grid** properties: Define how far the grid lines extend in the directions **Left/Below** and **Right/Above**.
	In the **Radial grid** properties: Define how far the grid lines extend in the directions **Start** and **End**.
Origin	Coordinates of the grid origin in the x, y, and z directions. These values offset the grid from the work plane origin, not from the global model origin.
Magnetic grid plane	In the **Rectangular grid** properties: Select whether objects are bound to grid lines (page 29). If you bind the objects to grid lines, the objects follow if you move the grid line.
Grid color	Use the color picker to pick the grid color.
Label font size and color	Define the label font size. Use the color picker to pick the label color.
Name	Enter a name for the grid. The name can be used, for example, when you bind parts to grid levels.
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the grid. These can be used for customizing drawing grid labels.

Add or delete a single grid line

You can add new grid lines either between existing grid lines or between two freely chosen points that you define in the model.

Add a grid line between existing grid lines

You can add new grid lines between existing grid lines.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid** selection switch is active.

   The **Direct modification** and **Select grid** switches are on the **Selecting** toolbar. The toolbar is by default located at the bottom of the screen.
3. Select an existing grid to attach the grid line to.
4. Click the  symbol between two existing grid lines or outside the grid.

   Tekla Structures creates the grid line and gives it a label using the labels of the adjacent grid lines. For example, a new grid line between the grid lines 1 and 2 receives the label 12*.

Add a grid line between two points

You can add new grid lines between two picked points.

1. On the **Edit** tab, click **Grid** --> **Add grid line**.
2. Select an existing grid to attach the grid line to.
3. Pick the start point of the grid line.
4. Pick the end point of the grid line.

   Tekla Structures creates the grid line.

Delete a grid line

1. Ensure that the  **Direct modification** switch is active.

   The **Direct modification** switch is on the **Selecting** toolbar. The toolbar is by default located at the bottom of the screen.
2. Select the grid.
3. Select the grid line you want to delete.
4. Press **Delete**.

   Or, you can right-click and select **Delete** from the pop-up menu.

Delete a grid line (alternative method)

When you delete a grid line, ensure that you do not have any other objects selected. Otherwise Tekla Structures only deletes the objects, not the grid line.

1. Ensure that the  **Select grid line** selection switch is active.

   The **Select grid line** switch is on the **Selecting** toolbar. The toolbar is by default located at the bottom of the screen.
2. Select the grid line you want to delete.
3. Right-click and select **Delete** from the pop-up menu.
4. Confirm that you want to delete the grid line.

Modify a single grid line

You can modify single grid line properties. You can also move grid lines or change the grid line labels.

Modify grid line properties

You can edit the properties of a single grid line in the property pane.

1. Ensure that the  **Select grid line** selection switch is active.

   The **Select grid line** switch is on the **Selecting** toolbar, which is by default located at the bottom of the screen.
2. If the property pane is not open, double-click the grid line to open the **Grid line** properties.
3. Change the properties as needed.
4. Click **Modify** to apply the changes.

Move a grid line

Use direct modification to move single grid lines.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid** selection switch is active.

   The **Direct modification** and **Select grid** switches are on the **Selecting** toolbar, which is by default located at the bottom of the screen.
3. Select the grid.
4. Select the grid line you want to move.
5. Drag the grid line to a new location.

   You can also use the keyboard to enter a numeric location.

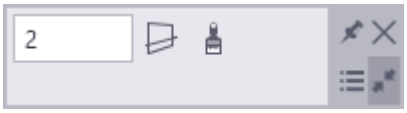
   To start with the negative sign (-), use the numeric keypad. To enter an absolute coordinate, first enter $, then the value. Press **Enter** to confirm.

Change a grid line label

Use the contextual toolbar to change the label of a single grid line.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid line** selection switch is active.

   The **Direct modification** and **Select grid** switches are on the **Selecting** toolbar, which is by default located at the bottom of the screen.
3. Select a grid line.
4. On the contextual toolbar, enter a new label.



Stretch, shrink, or incline a grid line

Use direct modification to stretch, shrink, or incline single grid lines in rectangular grids.

Note that this works only for the single grid lines that you have added in rectangular grids by using the **Add grid line** command.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid** selection switch is active.

   The **Direct modification** and **Select grid** switches are on the **Selecting** toolbar, which is by default located at the bottom of the screen.
3. Select the rectangular grid.
4. Select the grid line.
5. Drag a grid line handle  to a new location.

Turn grid line stretching off

If you move the outermost grid lines in rectangular grids using the line handles, Tekla Structures stretches or shrinks the perpendicular, crossing grid lines accordingly by default. You can switch this off temporarily.

1. Ensure that the  **Direct modification** switch is active.

2. Ensure that the  **Select grid** selection switch is active.

   The **Direct modification** and **Select grid** switches are on the **Selecting** toolbar, which is by default located at the bottom of the screen.

3. Select the grid line.

4. On the contextual toolbar, click the  **Turn grid line stretching off** button.

Grid line properties

Use the **Grid line** properties in the property pane to view and modify the properties of a single grid line. To open the properties, double-click a single grid line. The file name extension of a grid line property file is `.grdp`.

The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.

Setting	Description
Label	Name of the grid line.
Depth in view plane	Height of the grid plane perpendicular to the view plane.
Line extension Left/ Below	Define how far the grid lines extend in the directions **Left/Below** and **Right/Above**.
Line extension Right/ Above	
Magnetic	Select whether objects are bind to straight grid lines. If you bind the objects to grid lines, the objects follow if you move the grid line.
Visible in drawing	Select whether grid lines are made visible in drawings.
Grid line automatic dimensioning	Select whether single grid lines are used in grid dimensioning.
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the grid line.

Model views in Tekla Structures

A *model view* is a representation of a model from a specific location. Each view is represented in its own window within Tekla Structures. Selecting a part in a model view highlights the part in all open views.

View plane

Each model view has a view plane on which the grids are visible. The view plane is an imaginary flat surface.

View depth (page 50) is the thickness of the displayed slice of the model, and it is measured starting from the view plane in both positive and negative directions. Only objects positioned within the view depth are visible in the model.

Points located on the view plane are represented as blue crosses. Points that are located outside the view plane are represented as red crosses.

You can move the view plane like any other object. When you move it, Tekla Structures only uses the vector that is perpendicular to the view plane. To move the view plane, click the model view, right-click and select **Move** --> **Linear**.

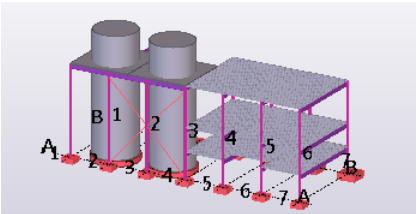
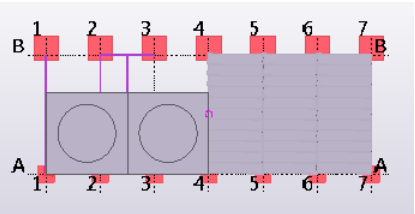
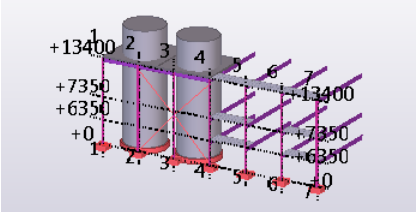
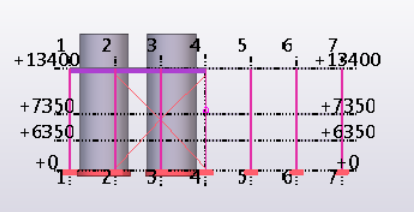
Basic views

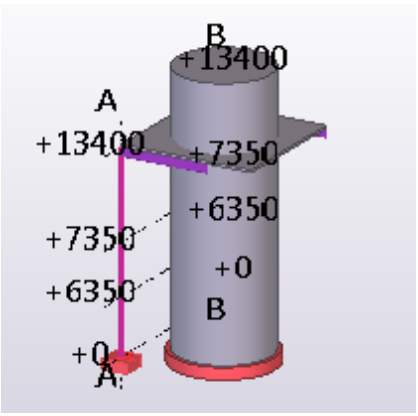
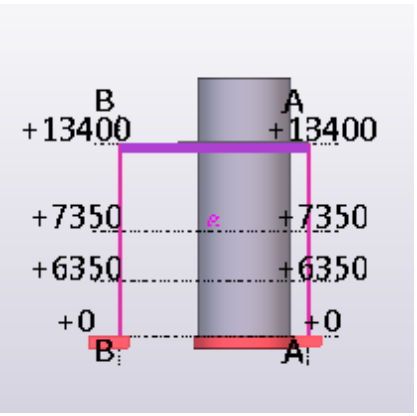
Basic views are those parallel to the global basic planes (xy, xz, and zy). In basic views, two axes always define the view plane and the axes appear in the plane name. The third axis is perpendicular to the view plane. It does not appear in the plane name. In the basic plane view, the model is shown from the direction of the third axis.

When you create basic views, you must define the view plane's distance (the view plane coordinate) from the global origin in the direction of the third axis. For more information, see Create model views (page 34).

Tekla Structures automatically assigns names for the basic views based on the plane and the coordinate.

Examples of basic views:

Plane	3D view	Plane view
XY plan view, z-axis being the perpendicular axis		
XZ elevation view, y-axis being the perpendicular axis		

Plane	3D view	Plane view
ZY elevation view, x-axis being the perpendicular axis		

Other views

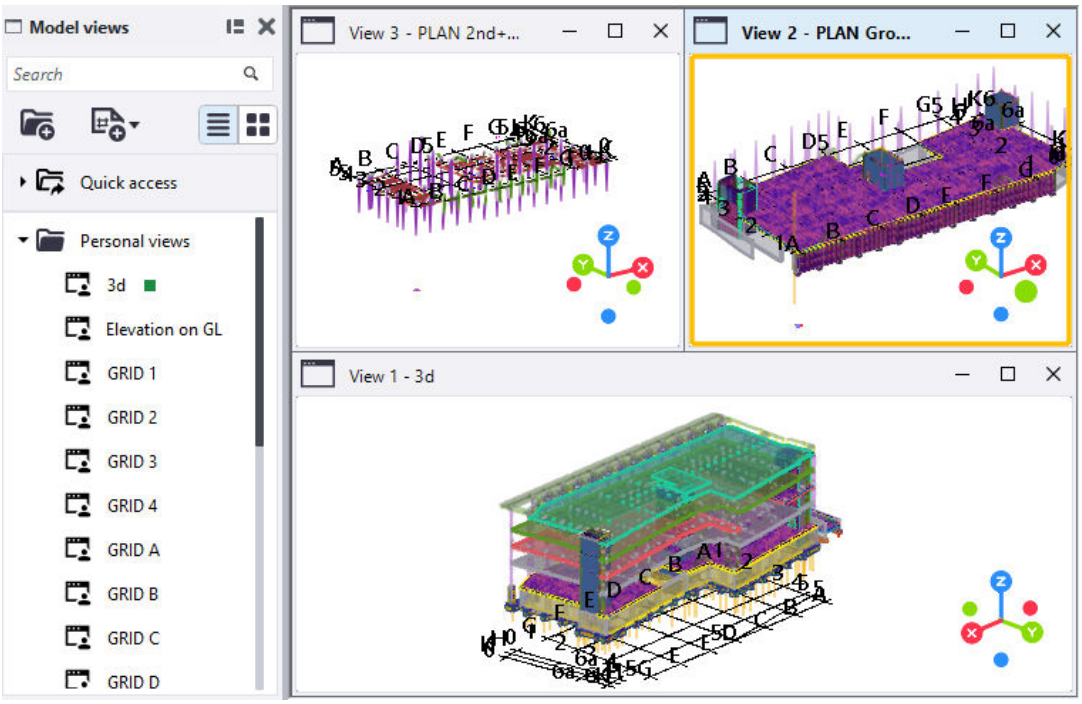
For other view types, you either define the view plane and coordinate by picking points, or the points are defined automatically, depending on the creation method. For more information, see Create model views (page 34).

Position and arrange model views and drawings in the Tekla Structures window

Model views and drawings can be positioned and arranged around the Tekla Structures window using different docking zones.

To dock a model view or a drawing, drag the model view or drawing by its title bar towards to the Tekla Structures window edge in the direction where you want to dock it. As you drag and move closer to the window edge, blue highlighting shows where you can dock the model view or the drawing. There are nine predefined docking zones around the Tekla Structures window. Note that you can only dock the model views or drawings to the predefined docking zones; the model views or drawings do not dynamically identify empty areas. The docked positions for model views and drawings are saved and restored the next time you open the model.

Model views and drawings automatically resize and reposition themselves to fit the screen and avoid overlapping with each other or any open side pane windows. When you stretch one view, the others are automatically resized and adjusted. If you undock a model view or drawing, it reverts to its original size and position. The current active view is highlighted to ensure that you are working in the correct view.



Should I model in a 3D or plane view?

Model views can be 3D or plane views. 3D, plane, and also elevation views provide different types of information, which is useful for different tasks.

One common technique is to open several model views:

- A 3D view to see a real-life version of the model
- A plane view, where you can add and connect parts
- An elevation view to check the level

If you are working with several screens, maximize your work area by putting the different views on different screens.

You can easily switch between the 3D and plane view by using the keyboard shortcut **Ctrl+P**. For more information, see Switch between model views (page 48).

Create model views

You can create model views in several ways in Tekla Structures.

For example, you can create views

- of the entire model (3D view) (page 35)
- of selected parts (page 40)
- of selected components (page 41)
- along grid lines (axis views) (page 36)

- to the surface of an object (page 41)

Each view has properties which define its appearance. You can change the appearance of a view after you have created it. To view and modify the properties of each model view (page 50), double-click anywhere on the model background to open the **View properties** dialog.

If you create views along grid lines, you can view and modify the properties (page 51) using the **Creation of views along grid lines** dialog.

If you need to reopen views for later use, you can find them in the **Model views** side pane. Tekla Structures only saves the named views when you exit the model.

Create a basic view of the model

You can create a basic view along two coordinate axes. Use this view for the overall viewing of the model.

1. On the **View** tab, click  **New view** --> **Basic view**.
2. Select a view plane from the **Plane** list.

   The view plane is defined by two axes.
3. In the **Coordinate** box, enter the view level.

   This value defines the distance of the view plane from the global origin, perpendicular to the view plane.
4. Click **Create**.

If you cannot see the model after creating a basic view, check that there are no distant objects (page 750) in the model that are very far. Distant objects can make the work area so big that the initial model is in some corner and not visible.

To find the distant objects, on the **File** menu, click **Diagnose & repair** --> **Find distant objects**. Select an object in the list, right-click and select a command. You can, for example, inquire or delete the object.

Repeat this several times until there are no distant objects found in the model.

Create a view using two points

You can create a view using two points you pick: the origin and a point in the direction of the x axis.

1. On the **View** tab, click  **New view** --> **Using two points**.
2. Pick a point to indicate the origin of the view plane.
3. Pick a second point to indicate the direction of the x axis.

The y axis is perpendicular to the view plane on which you picked the second point.

Create a view using three points

You can create a view using three points you pick: the origin, a point in the direction of the x axis, and the third point in the direction of the y axis.

1. On the **View** tab, click  **New view** --> **Using three points**.
2. Pick a point to indicate the origin of the view plane.
3. Pick a second point to indicate the direction of the x axis.
4. Pick a third point to indicate the direction of the y axis.

Create a view of the work plane

You can create a view of the work plane using the current view properties.

- On the **View** tab, click  **New view** --> **On work plane**.

Create grid views

You can create views along the grid lines you select.

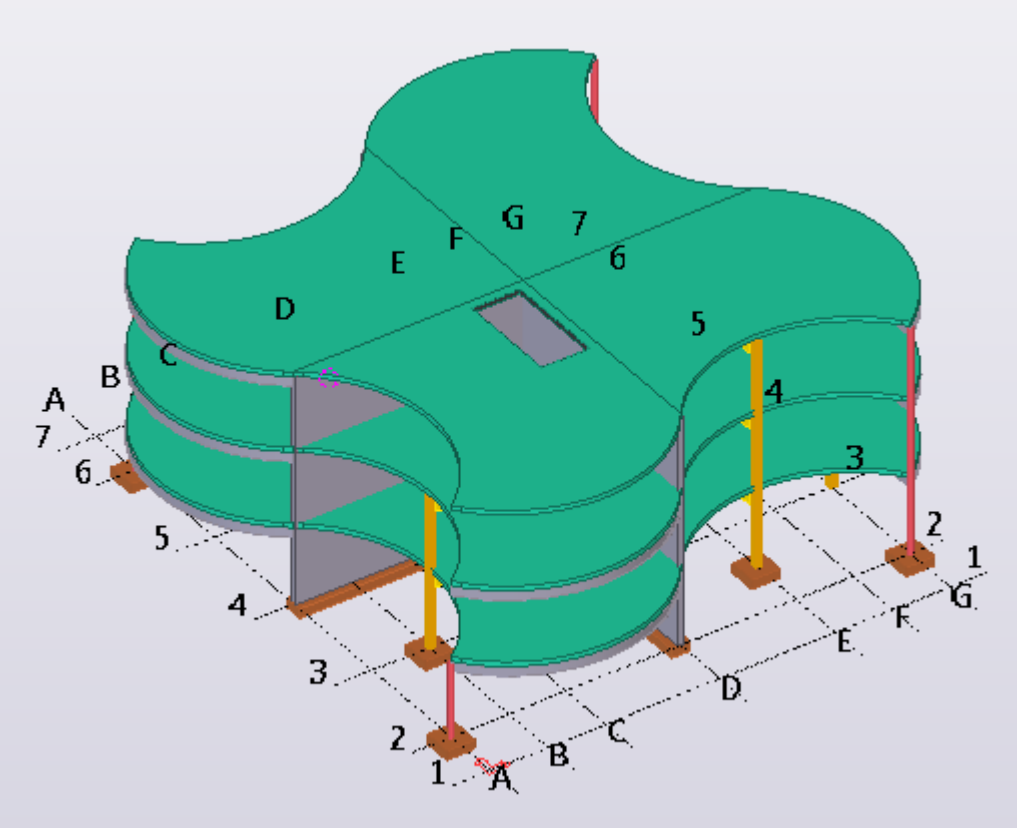
Before you start, create a view that contains a grid, and check the grid properties. Note that with radial grid you can create the grid views only on the straight grid lines, not on the arched grid lines.

If the grid properties are incorrect in some way, Tekla Structures may cut the views at the wrong elevations or they may be named incorrectly. If you change the grid labels or the elevation or grids later on, the views will not be automatically renamed.

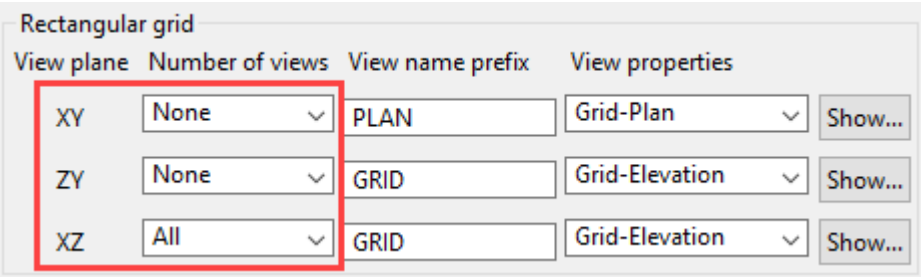
1. Select the grid.
2. On the **View** tab, click  **New view** --> **Along grid lines**.
3. Modify the grid view properties if needed.
   a. In the **Number of views** list, select how many views you want to create.
   b. In the **View name prefix** box, enter a prefix.
   c. In the **View properties** list, define which view properties (applied or saved) you want to use.
4. Click **Create**.

Example

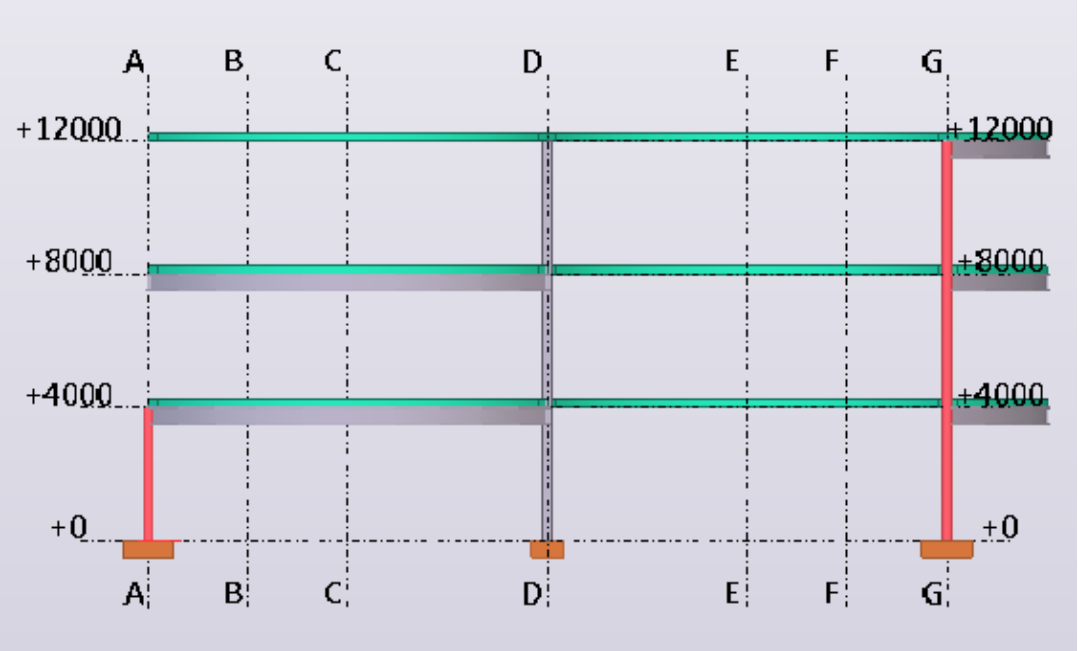
This example shows how to create vertical views of the grid lines 1–7 on the following model:



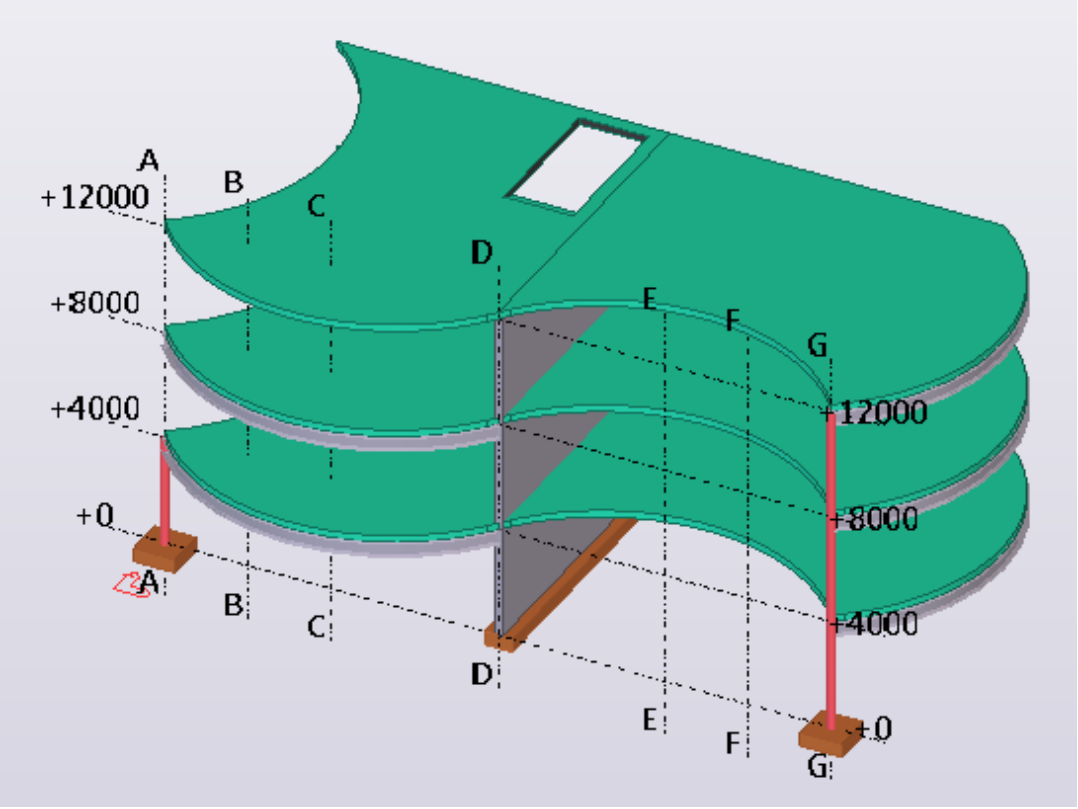
In the **Creation of views along grid lines** dialog, select **All** for the view plane XZ and **None** for the view planes XY and ZY. Use the default settings for the view name prefix and the view properties.



The grid view is displayed as a plane view in a new window:



Rotate the view to see it in 3D:



Create views on selected building storeys

You can create model views of the building storeys that you select in the **Building hierarchy** side pane or in the model.

1. On the **View** tab, click  **New view** --> **On selected building storeys**.
2. Select building storeys in the **Building hierarchy** side pane or in the model.
3. In the **Create views on selected building storeys** dialog, define the view name, and select the view properties.
4. Select whether the view is created at the building storey offset.
5. Click **Create**.

Create a view on a plane

You can create a view on the work plane or almost any plane of an existing part.

1. On the **View** tab, click  **New view** --> **On plane**.

   When you hover the mouse cursor over model objects, Tekla Structures highlights the available planes in light blue.
2. Select the desired plane.

TIP Alternatively, to create a view on the front, top, back, or bottom plane of a part, you can also use the following commands available on the **View** tab:

- **On part front plane**
- **On part top plane**
- **On part back plane**
- **On part bottom plane**

To use these commands, first select a command and then the part.

Create a 3D view of a part

When you need to see a specific part clearly, create a 3D view of the part. The part is placed in the center of the view.

1. On the **View** tab, click  **New view** --> **3D view of part**.
2. Select the part.

   Tekla Structures creates the view using the properties defined in the `part_basic_view` property file. The view plane y axis is the global z axis

of the model. The x axis is the projection of the part's local x axis onto the global xy plane.

Tekla Structures fits the work area to include the selected part.

If you want Tekla Structures to rotate the view once whenever you create a new 3D view, select the Basic view auto rotation setting in the **File** menu.

Create default part views

You can create four views of a part: front, top, end, and 3D view. Tekla Structures creates these views all at once with the same command. The front, top, and end views are plane views by default.

1. On the **View** tab, click  **New view** --> **Default part views**.
2. Select the part.

   Tekla Structures creates the four default views all at once, using the properties defined in `part_front_view`, `part_top_view`, `part_end_view`, and `part_persp_view` property files.

Create an undeformed part view

You can create a view that shows a deformed part in undeformed form. This works only for beams and columns.

1. On the **View** tab, click  **New view** --> **Undeformed part view**.
2. Select the part.

   For example, select a warped beam. Tekla Structures displays the beam in a separate view in undeformed form.

Create a 3D view of a component

When you need to see a specific component clearly, create a 3D view of the component. The component is placed in the center of the view.

1. On the **View** tab, click  **New view** --> **3D view of component**.
2. Select the component.

   Tekla Structures creates the view, using the properties defined in the `component_basic_view` property file. The view plane y axis is the global z axis of the model. The x axis is the projection of the first secondary

part local x axis onto the global xy plane. Work area depth is 1 m in all directions.

The component you select automatically defines the work area.

If you want Tekla Structures to rotate the view once whenever you create a new 3D view, select the Basic view auto rotation setting in the **File** menu.

Create default component views

You can create four views of a component: front, top, end, and 3D view. Tekla Structures creates these views all at once with the same command. The front, top, and end views are plane views by default.

1. On the **View** tab, click  **New view** --> **Default views of component**.
2. Select the component.

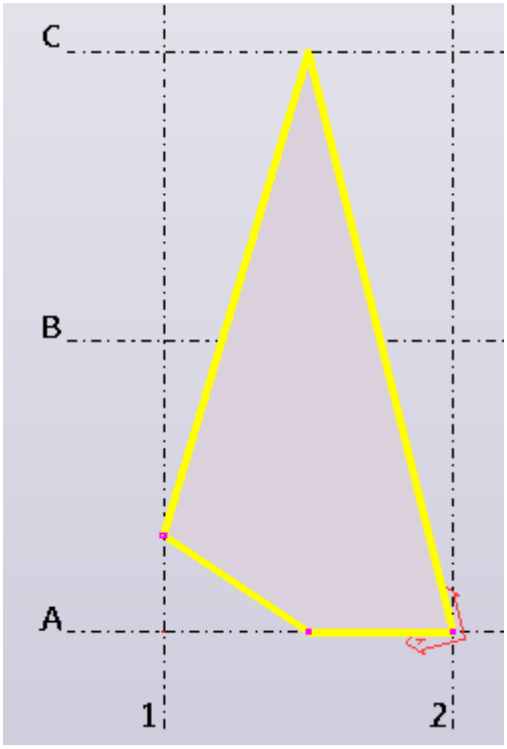
   Tekla Structures creates the four default views all at once, using the properties defined in the `component_front_view`, `component_top_view`, `component_end_view`, and `component_persp_view` property files.

Create a surface view

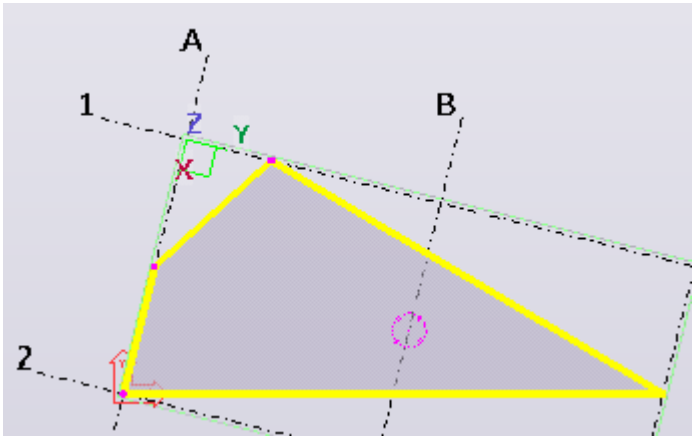
Use the **Create surface view** macro to create an automatically aligned surface view. This can be useful when modeling bolt groups, stiffener plates, and hole penetrations on complex geometry.

To be able to select the surface of a part, ensure that you are using a model view that shows part faces. On the **View** tab, click **Rendering**, and select either the **Parts grayscale** or the **Parts rendered** option.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **Create surface view** to start the macro.
4. Select the surface of the part.



Tekla Structures creates a new temporary view and moves the work plane typically along the longest edge of the part face. You can model in the surface view and see your modeling work being done in your original 3D view at the same time.



5. Press **Esc** to stop the macro.
6. To return the work plane back to the origin:
   a. Repeat steps 1–2 to open the **Applications** list.
   b. Double-click the **Work plane global** macro.

      The work plane returns back to the origin and is aligned with the global x,y, and z planes of the model.

Create a surface view along selected edge

Use the **Create surface view wEdge** macro to create a surface view and align the work plane along the edge you select. This can be useful when modeling bolt groups, stiffener plates, and hole penetrations on complex geometry.

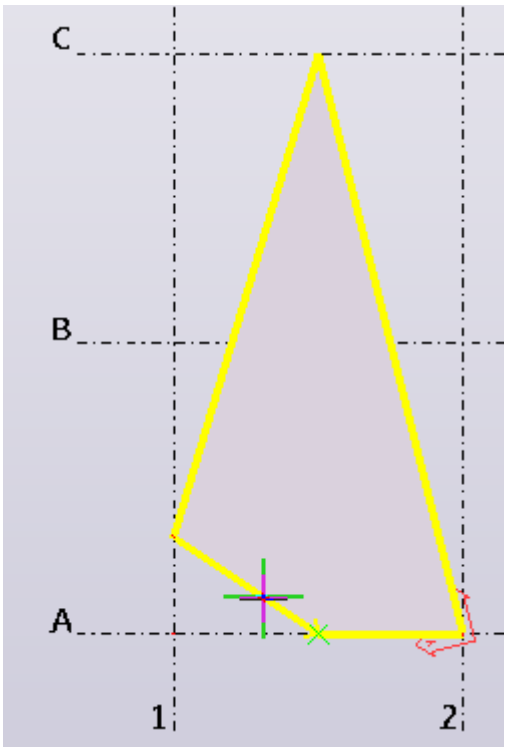
To be able to select the surface of a part, ensure that you are using a model view that shows part faces. On the **View** tab, click **Rendering**, and select either the **Parts grayscale** or the **Parts rendered** option.

1. Ensure that the **Snap to geometry lines / points** snap switch (page 89) is active.

   This allows you to pick along an edge to define the direction.

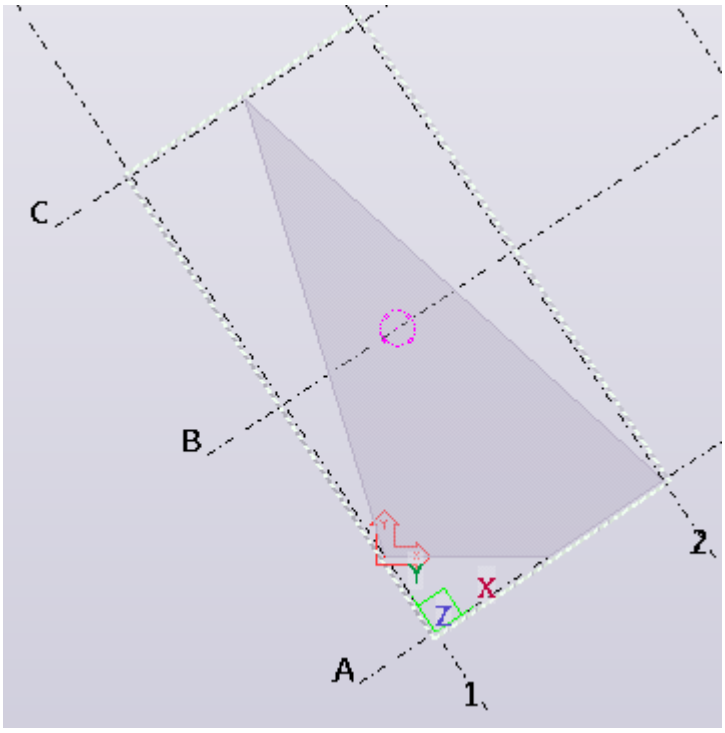
2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
3. Click the arrow next to **Applications** to open the applications list.
4. Double-click **Create surface view wEdge** to start the macro.
5. Select the surface of the part.

   When you hover the mouse pointer over the part edges, a yellow arrow symbol is displayed to indicate the possible edges you can align the view to. The head of the arrow represents the positive direction of the x axis. The view will be rotated in this direction to form the flat horizontal edge of the view. The origin of the view and work plane will be at the start of the arrow snap line.



6. Pick the desired edge.

   Tekla Structures creates a new temporary view, and the selected edge forms the x axis of the view. You can model in the surface view and see your modeling work being done in your original 3D view at the same time.



7. Press **Esc** to stop the macro.
8. To return the work plane back to the origin:
   a. Repeat steps 2–3 to open the **Applications** list.
   b. Double-click the **Work plane global** macro.

      The work plane returns back to the origin and is aligned with the global x,y, and z planes of the model.

Manage model views

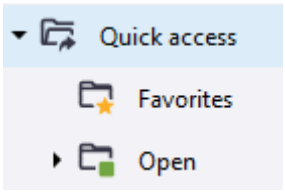
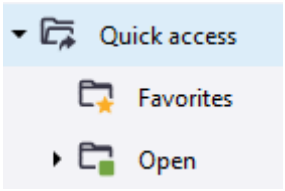
Use the **Model views** side pane window to manage your personal model views and model views that have been shared in Tekla Model Sharing.

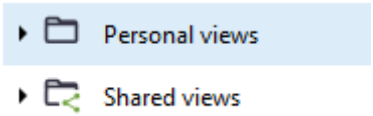
Organize and manage a large number of model views in a controlled manner using the **Model views** side pane. Additionally, you can create general arrangement drawings of the selected model views.

Click the  button in the side pane to open the **Model views** side pane window. The **Model views** side pane opens on the left side of the screen and lists all the views of that model. You can organize the model views as you like.

You can have up to nine views open at the same time. To arrange the model views to suitable layout in the Tekla Structures window, dock the views around the window. Drag the model view by its title bar towards to the Tekla Structures window edge in the direction where you want to dock it.

To	Do this
Open or close the selected view Open or close all views in a folder	• Right-click a view and select **Open view** or **Close view**. • Right-click a folder or select **Open all views in folder** or **Close all views in folder**. You can have up to nine views open at the same time. To open multiple views, use the **Shift** or **Ctrl** keys when you select views.
Rename a view Delete a view	• Right-click a view and select **Rename**. • Right-click a view and select **Delete**.
Use the indicators to quickly check the status of the view	•  the view is the currently active view. •  the view is currently open. •  the view is a temporary view. Rename the view to make it a permanent view. Temporary views have a default name in parentheses. If you need to reopen views later on, give each view a unique name. Do not use parentheses when naming a view, or the view will not be saved for later use. When you exit the model, Tekla Structures only saves the named views. Temporary views disappear when you close them. •  the view is shared in Tekla Model Sharing.
Bring a selected view to the front on top of the views	Right-click a view and select **Bring to front**. Alternatively, double-click a view in the side pane to bring it to the front as the topmost of all visible views.

To	Do this
Switch between the list view and the thumbnail view	Click  to switch between the list view and the thumbnail view. The thumbnail view shows a preview of each view. Thumbnail is visible only if you have opened the view. If a view has never been opened, it does not have a thumbnail.
Open the **Quick access** folder	The **Quick access** folder displays all open and temporary views, and those views that you have marked as favorites. 
Add a view to favorites	Right-click a view and select **Add to Favorites**. The view is then added to the **Favorites** folder under **Quick access**. 
Use the **Search** box	Use the **Search** box to find specific views.
Create a new folder	Right-click and select **New folder** to create your own folders where you can organize your views. You can also have subfolders. Alternatively, create a new folder by clicking .
Move a view to another folder	Drag the selected view to a suitable folder. Alternatively, right-click a view and select **Cut**. Go to another folder, right-click, and select **Paste**.
Copy a view to another folder	Right-click a view and select **Copy**. Go to another folder, right-click, and select **Paste**.
Open view properties	Right-click a view and select **Properties**. The **View properties** dialog opens.
Create drawings	• If you have selected one view and want to create a general arrangement drawing of the selected view, select **Create GA drawing**.

To	Do this
	• If you have selected several views and want to create general arrangement drawings for each selected view, select **Create GA drawing for each selected view**. • If you have selected several views and want to create a general arrangement drawing that includes all the views, select **Create GA drawing containing selected views**. Alternatively, create an empty drawing, or define the drawing properties by clicking . To open the created drawing, click the **Drawing created. Click here to open the drawing.** button.
If the model has been shared using Tekla Model Sharing	Personal views and shared views are organized in separate folders.  Views in the **Personal views** folder are not shared, whereas the views in the **Shared views** are shared with other Tekla Model Sharing users. • To share a view with Tekla Model Sharing, right-click a view and select **Share**. If you select multiple views, you can share all of them at once. Alternatively, drag the selected views to the **Shared views** folder. • To remove a view from being shared, right-click a shared view and select **Remove from sharing**. Alternatively, drag the selected views to the **Personal views** folder. Note that the **Share** option is available only if the model has been shared using Tekla Model Sharing. Shared views are moved to the **Shared views** folder, marked as shared , and shared the next time when the model is written out to the sharing service. The folder structure of the **Shared views** folder is shared as well.

Switch between model views

You can easily switch between all open views while modeling in Tekla Structures. You can also switch between the 3D and plane view to examine the current view from different perspectives, switch between plane views, or switch between view angles to show the model in different angles.

You can find all the created model views in the **Model views** side pane. Tekla Structures only saves the named views when you exit the model.

Switch between open model views

To switch between open model views, do one of the following:

- In the **Model views** side pane, double-click a view to bring it to the front as the topmost of all visible views, or right-click a view and select **Bring to front**.
- Use the keyboard shortcut **Ctrl+Tab** to cycle through the open model views.
- Click **Window** in the top right corner of the Tekla Structures ribbon and select a model view from the list.

  The views are listed in alphabetical order.
- Right-click a model view, then select **Next window** from the pop-up menu.

  The next open view becomes active.

Switch between 3D and plane view

Use the **Switch to 3D or plane** command to examine the current model view from different perspectives.

On the **View** tab, click  **Switch to 3D or plane**.

Alternatively, use the keyboard shortcut **Ctrl+P**.

Rotate model view from 3D to a selected plane view

Use the navigation control axes in the local coordinate system symbol to rotate the current model view from 3D to a selected plane view.



Click the selected navigation control axis to rotate the model view around the current rotation point to the selected 2D plane. To check the model using a different view, click another control axis in the symbol.

To return to the 3D view, rotate the model using your mouse.

Switch between plane views

Use the **View angle** command to rotate the model view or the selected object to a plane view.

1. Select a model view or an object, and on the contextual toolbar click **View angle**.
2. Select a top, back, right, bottom, front, or left view.



3. To return to the original 3D view, click the empty button in the middle of the view angle options.

Update and refresh model views

Use the **Update window** and **Redraw** commands in Tekla Structures to refresh a single model view or all model views at once.

- **Update**: Removes temporary graphics (such as measured distances) but does not redraw the view. Faster than redrawing.
- **Redraw**: Redraws the view completely and shows all previously hidden objects.

To	Do this
Update the current model view	Right-click the view and select **Update window**.
Update all of the model views	On the **View** tab, click **Redraw** --> **Erase temporary graphics**. 
Redraw the current model view	Right-click the model view and select **Redraw view**.
Redraw all of the model views	On the **View** tab, click .

See also

Create model views (page 34)

Manage model views (page 44)

Switch between model views (page 48)

Model view properties

Use the **View properties** dialog to view and modify the model view properties. To open the **View properties** dialog, double-click anywhere on the model background.

Option	Description
Name	The name of the view.
Angle	Whether the view angle is **Plane** or **3D**.
Projection	The projection type of views. **Orthogonal**: All objects are of equal size (no perspective). When you zoom, text and point size remains the same. In addition, the zoom remains on object faces. **Perspective**: Distant objects appear smaller than close ones, as do text and points. You can zoom, rotate, or fly through the model.
Rotation	How the view is rotated around the z and x axes. Rotation is view-specific. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.
Share	This option is available only if the model has been shared with Tekla Model Sharing. In Tekla Model Sharing, views are not shared by default. Views are shared if they have a name, and the **Share** option is set to **Shared**.
Color and transparency in all views	The color and transparency setting that is used in all views (according to the status of the objects in the model).
Representation...	Opens the **Object Representation** dialog for defining color and transparency settings.
View depth	The thickness of the displayed slice of model. You can define the depth separately upwards and downwards from the view plane. Only objects positioned within the view depth are visible in the model. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.

Option	Description
Display...	Opens the **Display** dialog for defining which objects are displayed (page 676) in the view and how.
Visible object group	Which object group is displayed in the view.
Object group...	Opens the **Object group - View filter** dialog for creating and modifying object groups.

See also

Create model views (page 34)

Manage model views (page 44)

Grid view properties

Use the **Creation of Views Along Grid Lines** dialog to view and modify the properties of grid views.

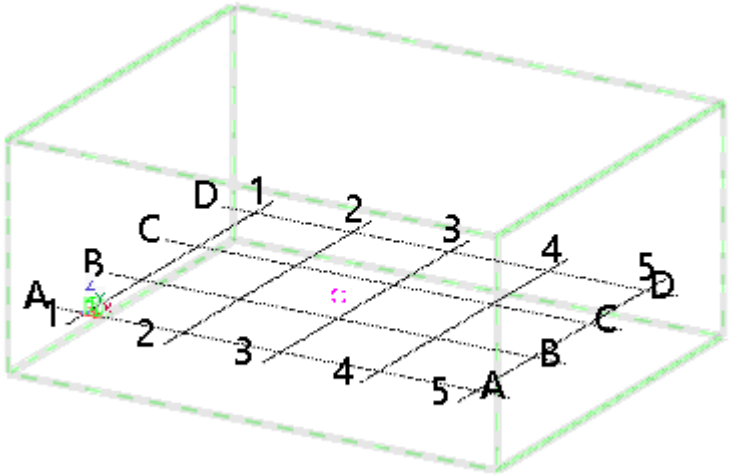
Option	Description
View plane	The plane of the view defined by two axes similarly to the default view.
Number of views	Defines which grid lines the views will be created of. **None** does not create any views. **One (First)** only creates the view closest to the grid origin. **One (Last)** only creates the view furthest from the grid origin. **All** creates all views in grid planes in the relevant direction.
View name prefix	The prefix to be used with the grid label in the view name. This name overrides the name in the view properties. View names consist of a prefix and a grid label, e.g. PLAN +3000. If the **View name prefix** box is left empty, no prefix is used. Tekla Structures adds a dash and a running number to the view name if view names are otherwise identical.
View properties	Defines which view properties (applied or saved) will be used. Each view plane has its own view properties. You can load the properties from the current view properties with the option **<applied values>** or from saved view properties. The **Show** button displays the current view properties.

See also

Create model views (page 34)

Define the work area

Tekla Structures indicates the work area of a view using green, dashed lines. Objects outside the work area exist, but they are not visible. You can shrink and expand the work area to suit particular situations, for example to concentrate on a particular area of the model. You can temporarily hide the work area box.



Fit work area to entire model

You can resize the work area to include all model objects, either in all views or in selected views only.

1. On the **View** tab, click **Work area** and select one of the following:
   - **To entire model in all views**

     Fits the work area to include all model objects in all visible views.
   - **To entire model in selected views**

     Fits the work area to include all model objects in the selected views.

Fit work area to selected parts

You can resize the work area to include only selected parts, either in all views or in selected views only.

1. Select the objects you want to include.

2. On the **View** tab, click **Work area** and select one of the following:
   - **To selected parts in all views**

     Fits the work area to include the selected model objects in all views.
   - **To selected parts in selected views**

     Fits the work area to include the selected model objects in the selected views.

Fit work area using two points

You can resize the work area based on two corner points you pick on the view plane. The depth of the work area is the same as the view depth.

1. On the **View** tab, click **Work area** and select **Using two points**.
2. Pick the first point.
3. Pick the second point.

Hide the work area box

You can temporarily hide the work area box in a view. This can be useful, for example, when creating screenshots for presentations.

1. Hold down the **Ctrl** and **Shift** keys simultaneously.
2. On the **View** tab, click **Redraw** --> **Redraw all views**.



3. To make the box visible again, click **Redraw** --> **Redraw all views** again.

TIP Alternatively, use the advanced option XS_HIDE_WORKAREA.

If you place objects are outside the work area

If you place new objects outside the work area, or copy or move objects outside the work area, Tekla Structures displays a warning message **Objects outside the work area**. You can expand the work area to include the new objects.

If you do not wish to see thew warning message again, you have the option to hide future warnings of the same type. You can also have Tekla Structures show these warnings again.

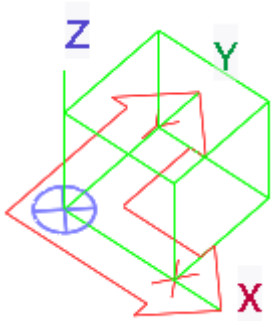
- To hide future warnings of the same type, select the **Do not show this message again** checkbox.
- To re-display the warnings, hold down the **Shift** key while running a command that should normally induce a warning message. For example, if you deliberately copy or move objects outside the work area while holding down the **Shift** key, the associated warning message reappears.

Coordinate system

Tekla Structures uses two coordinate systems: the global and the local coordinate system. The local coordinate system is also known as the work plane.

Global coordinate system

Global coordinate system is the main reference for positiong objects in Tekla Structures. It is represented by a green cube symbol at the global point of origin (x=0, y=0, z=0). The global coordinate system is static and it cannot be changed.

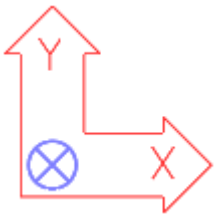


Do not place the model far away from the origin. If you create model objects far away from the origin, snapping to ponts in the model views may become inaccurate. The further away from the origin you model, the less precise all computations become.

If you need to use another coordinate system for inserting reference models or exporting IFC models, you can use base points. When you use base points, you can keep the coordinates small and locate the model wherever needed. For instuctions, see Base points (page 60).

Local coordinate system (Work plane)

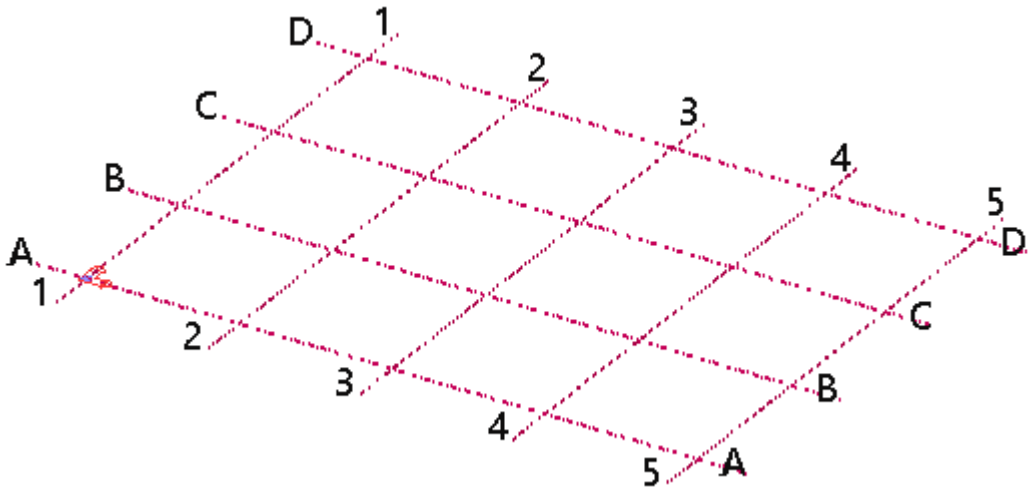
Local coordinate system, also known as the work plane, can be adjusted according to modeling needs. It is indicated by a red work plane arrow symbol which shows the xy plane. The z direction follows the right-hand rule.



Most of the commands that are dependent on the coordinate system use the work plane coordinates. For example creating points, part positioning, and copying always comply with the work plane coordinate system. The work plane is model-specific, so it is the same in all views.

You can set the work plane to any position by picking points or by selecting a plane. This can make it easier to place parts accurately when, for example, modeling sloped parts. For instructions on setting the work plane, see Shift the work plane (page 56).

The work plane also has its own red grid, which can be used for positioning parts. You can show or hide this grid as needed. For instructions, see Show or hide the work plane grid (page 56).



Coordinate symbol

The local coordinate system symbol, which is located in the lower right corner of the model view, follows the work plane and indicates the direction of the model.



If you want to rotate the model view from 3D to a 2D plane, click the navigation control axes in the local coordinate system symbol. The model view

rotates around the current rotation point to the selected 2D plane. To check the model using a different view, click another control axis in the symbol. To return to the 3D view, rotate the model using mouse.

Work plane handler toolbar

If you have defined base points or saved work planes, you can use the **Work plane handler** toolbar to select which work plane you currently have in use in the model. For instructions, see Select the work plane (page 72).

See also

Adjust part position (page 334)

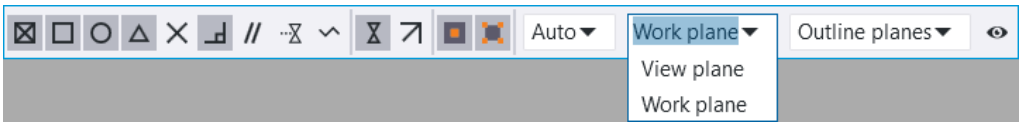
Snap to positions (page 88)

Show or hide the work plane grid

Work plane represents the local coordinate system of the Tekla Structures model. The work plane also has its own dark red grid, which can be used for positioning parts.

You can show or hide the work plane grid as needed. The work plane grid is hidden by default. Use the options on the **Snapping** toolbar to show or hide the work plane grid.

1. To show the work plane grid, on the **Snapping** toolbar, select **Work plane** from the second list.



2. To hide the work plane grid, select **View plane** from the same list.
3. If you want to change the work plane grid color, use the XS_GRID_COLOR_FOR_WORK_PLANE advanced option.

See also

Coordinate system (page 54)

Shift the work plane (page 56)

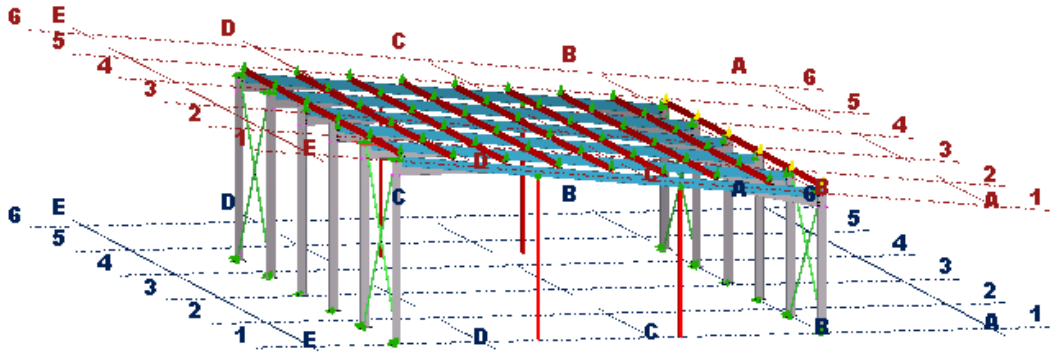
Select the work plane (page 72)

Shift the work plane

The current work plane defines the local coordinate system of a Tekla Structures. You can set the work plane to any position by picking points or by selecting a plane. The work plane is model-specific, so it is the same in all views.

When modeling sloped parts, shifting the work plane helps you to place parts accurately.

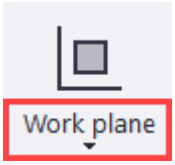
For example, you can shift the work plane to the slope of the roof to make it easier to model horizontal bracing and purlins in a sloped roof.



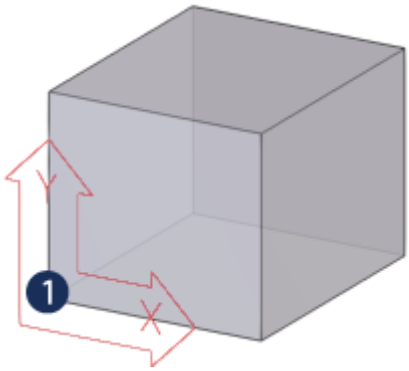
Set work plane to any part plane

Use the **Work plane tool** command to set the work plane to any part plane or grid intersection. Note that you can select only native Tekla Structures object planes. You cannot select reference model object planes.

1. On the **View** tab, click **Work plane** --> **Work plane tool**.



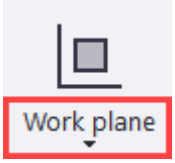
2. Move your mouse over the model to preview the work plane direction.
3. Click the left mouse button to set the work plane.



Set work plane parallel to xyz plane

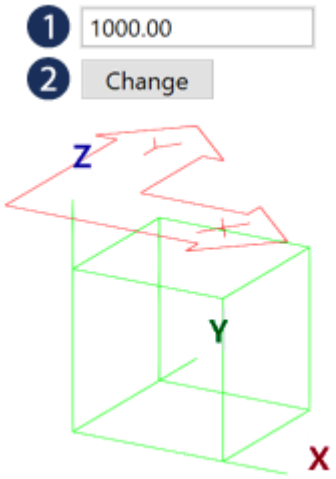
You can set the work plane parallel to the xy, xz, or zy plane.

1. On the **View** tab, click **Work plane** and select **Parallel to XY(Z) plane**.



2. In the **Plane** list box, select the plane parallel to the work plane.
3. Enter the depth coordinate.

   The depth coordinate defines the distance of the work plane from the global origin along a line perpendicular to the plane parallel to the third axis.
4. Click **Change**.



Set work plane using one point

You can set the work plane using one picked point. The work plane stays parallel to the current work plane, but moves to a new position. The x and y directions are unchanged.

1. On the **View** tab, click **Work plane** and select **Using one point**.

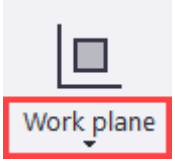


2. Pick the new position of the work plane.

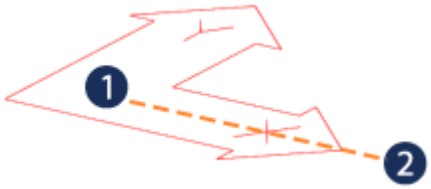
Set work plane using two points

You can set the work plane using two picked points. The first point you pick is the origin, the second defines the x direction of the work plane. The y direction remains the same as the previous work plane.

1. On the **View** tab, click **Work plane** and select **Using two points**.



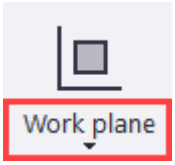
2. Pick the origin of the work plane.
3. Pick a point in the work plane, in the positive x direction.



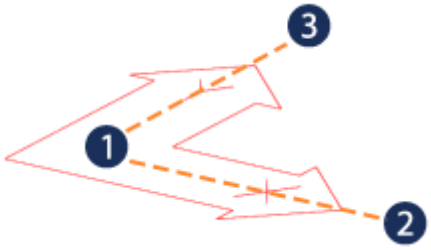
Set work plane using three points

You can set the work plane using three picked points. The first point you pick is the origin, the second defines the x direction, and the third defines the y direction of the work plane. Tekla Structures fixes the z direction according to the right-hand rule.

1. On the **View** tab, click **Work plane** and select **Using three points**.



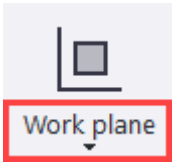
2. Pick the origin for the work plane.
3. Pick a point in the positive x direction.
4. Pick a point in the positive y direction.



Set work plane parallel to view plane

You can set the work plane to be the same as the view plane of a selected view.

1. On the **View** tab, click **Work plane** and select **Parallel to view plane**.

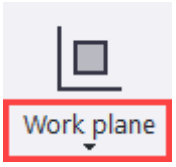


2. Select the view.



Restore the default work plane
Remember to change back to the default work plane when you have finished modeling sloped structures.

1. On the **View** tab, click **Work plane** --> **Parallel to XY(Z) plane**.



2. In the **Plane** list, select **XY**.
3. In the **Depth coordinate** box, enter **0**.
4. Click **Change**.

Base points

The base points (control points) allow you to use a coordinate system based on the civil origin or other coordinate system for interoperability and collaboration. For example, you can use base points when inserting reference models, exporting IFC models, in drawings, in **Layout manager**, and in reports and templates.

Civil origin is the datum point or fundamental benchmark point of the national land survey network.

When you use base points, you can keep the coordinates small and locate the model wherever needed. You can create as many base points as you need, and select one of those to be the project base point.

Note the following:

- Reference models should not have any additional lines to the origin.
- Reference models should not include objects that are very far from each other because otherwise using the model may get difficult.

- Tekla Structures native objects, including reference models, should not be inserted very far from the Tekla Structures model origin.

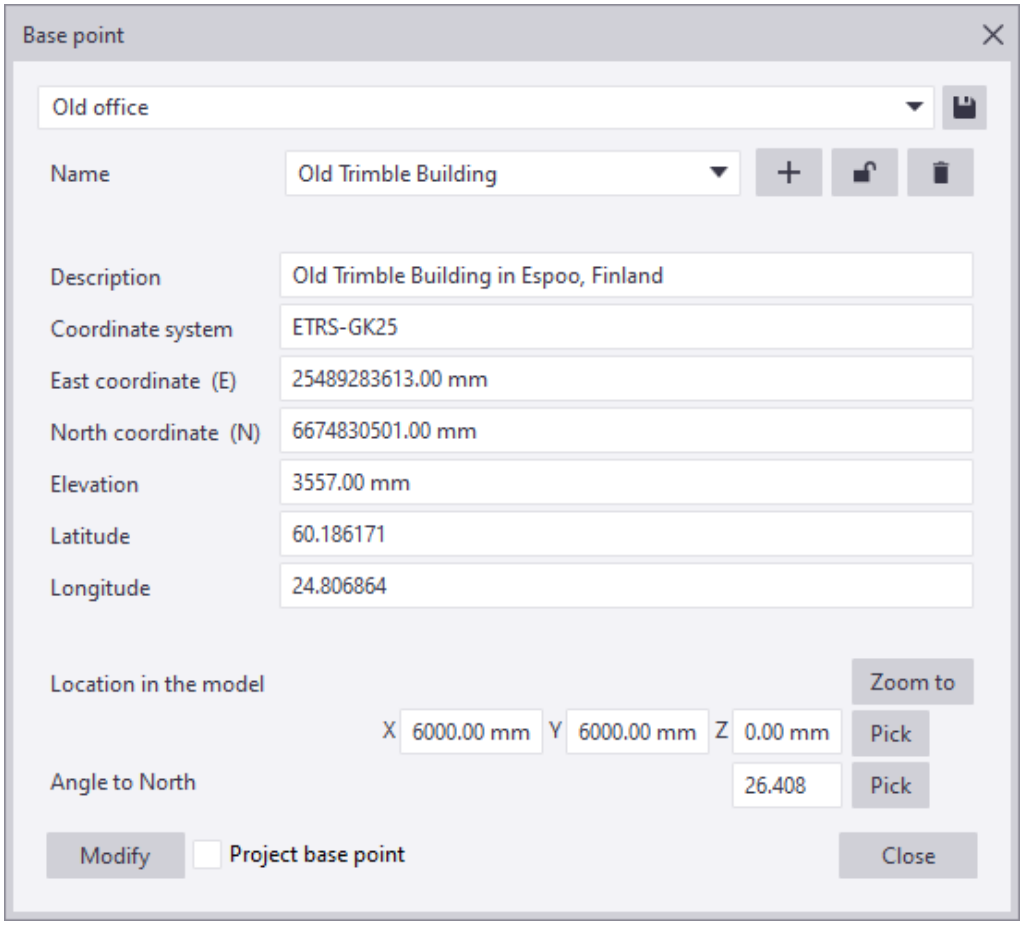
Define a base point

You can define base points in **Project properties**. In case you need to import or export a reference model, you need to know the coordinates of the reference model that you are importing, or the coordinates that you want to use in the resulting IFC export file.

TIP You can inquire the point coordinates in Tekla Structures: On the ribbon, click the down arrow next to , and then select **Point coordinates**. For more information, see Inquire object properties (page 721).

---

1. Open a Tekla Structures.
2. Click **File** --> **Project properties** --> **Base points** to open the **Base point** dialog.
3. Fill in the needed information:



You can load existing base point settings. To do this, copy your settings file `<settings name>.basePoint.json` from the `\attributes` folder under the model where you have the base point settings you want to use to the `\attributes` folder under the current model folder. In the **Base point** dialog, select the base point settings from the list.

Note that base point settings also work with project and firm folders: Copy the base point settings file to the `\attributes` folder under the firm and project folders.

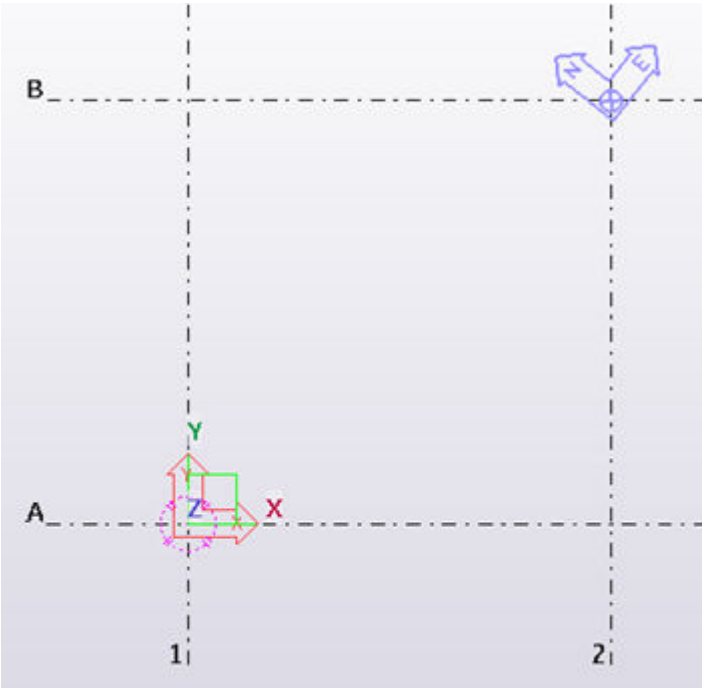
Name, **Description**	Enter a name and a description for the base point. Giving a name is obligatory.
Coordinate System	Enter the name of the coordinate system you are using.
East coordinate (E)	Enter the **East coordinate (E)** that represents the X coordinate related to the civil origin.
North coordinate (N)	Enter the **North coordinate (N)** that represents the Y coordinate related to the civil origin.
Elevation	Enter the **Elevation** that represents the Z coordinate related to the civil origin. The Z value of the project base point coordinates sums with the **Elevation** value defined separately.
Latitude,**Longitude**	Enter the **Latitude** and **Longitude** of the base point to be used in the IFC export. **Latitude** and **Longitude** is additional information, which some software can use. In the IFC file, this is written to `IFCSITE` information. If the total number of digits in **Longitude** is more than 15, the value is rounded up to the nearest if it is > 99.999999999999999. To convert **Latitude** and **Longitude** information between decimal format and degree/minute/second (DMS) format, see Convert Latitude/Longitude to decimal.
Location in the model	Pick or enter a location for the base point in the Tekla Structures model. The distance is measured from the model origin. The base point location in the model can have a maximum value of +/- 10 km from the model origin. Modeling is meant to be done near the model origin, and the offset is to be given with **East coordinate** and **North coordinate** values.

Angle to North	Pick or enter the **Angle to North**, which is the angle between Y and North directions. The maximum number of decimals for the angle is 13.
Project base point	If you want to set a coordinate system as the project base point, select a base point from the list at the top and then select the **Project base point** checkbox.

4. To save the base point settings, type a unique name for the settings and click **Save**. To use the same base point settings in another model, copy your settings file `<settings name>.basePoint.json` from the `\attributes` folder under the current model folder to the `\attributes` folder under the model where you wish to use the same base point settings. The settings also work with project and firm folders: Copy the base point settings file to the `\attributes` folder under the firm and project folders.
5. To avoid unwanted changes by others working in the same model, lock the base point by clicking the **Lock/Unlock** button next to the base point name. The button changes to . To unlock the base point, click the button again.
6. Click **Modify** to save the base point.

   A blue symbol is added in the model.

   If you later on make changes to the base point, the base point location in the model changes according to the location or rotation changes you make in the **Base point** dialog when you press **Enter** or click another input field, and there is no need to click **Modify**. Note that if the base point is locked, you cannot modify it.



Now you can insert a reference model, or export an IFC model using the defined base point, for example.

Set a coordinate system as the project base point

One base point can be set to be as the project base point. Model origin is the default project base point value if the model does not contain any base points, or if none of the existing base points has been set to be the project base point. You can check and change the current project base point through **File** --> **Project properties** --> **Location by**.

Do not change the project base point during a project.

1. Click **File** --> **Project properties**.

   You can see the current base point in the **Location by** box.

2. To select the project base point, click the **Base points** button, select a base point from the **Location by** list, and then select the **Project base point** checkbox.

3. Click **Modify**.

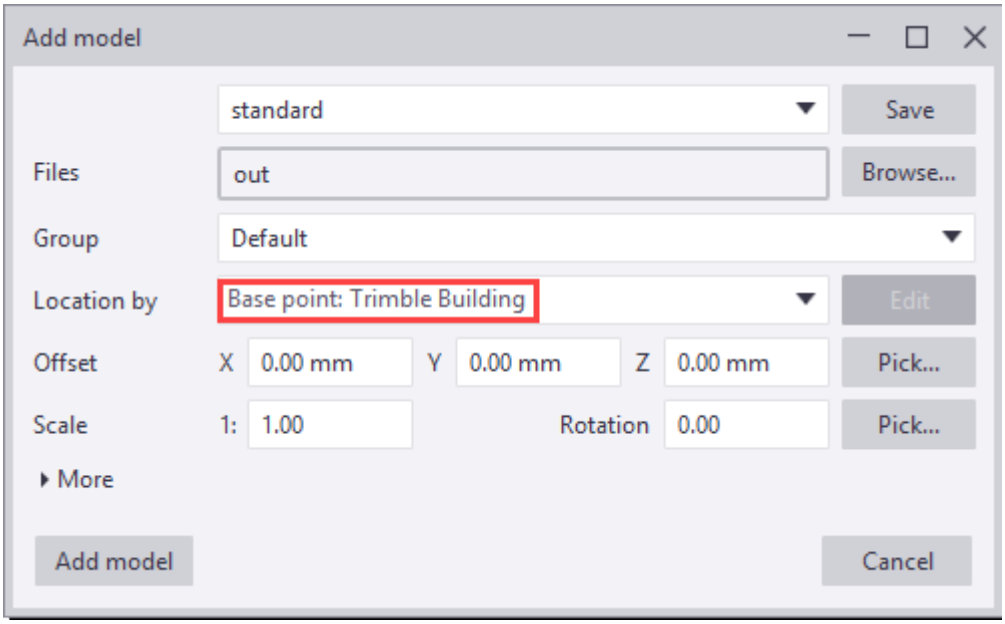
Insert a reference model using a base point

Before you can insert a reference at the base points, you need to create a base point in your model. To create the base point, you need to know the coordinates of the reference model that you are importing.

1. Open the **Reference models** list by clicking the **Reference models** button in the side pane .

2. In the **Reference models** list, click the **Add model** button.
3. In the **Add model** dialog, if you have any previously created reference model properties files, load the desired file by selecting the file from the properties file list at the top.
4. Browse for the reference model by clicking **Browse...**.
5. In **Group**, select a group for the reference model, or enter a name for a new group.

   If you do not enter a name for the group, the reference model is inserted in the **Default** group.
6. In **Location by**, select the base point you want to use.



7. Click the **Add model** button. Tekla Structures inserts the reference model relative to the selected base point by using the coordinate system values, elevation and angle in the base point definition in the model **Project properties**. For example, both IFC2x3 and IFC4 reference model inserting support base points.

Export an IFC model using a base point

Before you can export an IFC file using a base point, you need to create a base point in your model.

1. Click **File** --> **Export** --> **IFC** to open the **Export to IFC** dialog.
2. In **Location by**, select a base point that you have created.
3. Define other necessary IFC export information.

Important in IFC4 and IFC4.3 export: When using a base point, select the method to use in **Base point export** to ensure that the collaboration works with other parties in the project. Select one of the following:

IfcMapConversion: Converts the model's local coordinate system into the global coordinate system. This option is needed, when the IFC4 format that complies with the IFC4 schema is required.

IfcSite coordinate system: Converts coordinates in the IFC model for each object separately. This option can be used to produce an IFC4 file viewable in most IFC viewers, including Trimble Connect. The same conversion method is also used in the IFC2x3 export.

4. Click **Export**. The base point option exports the IFC model relative to the base point using the coordinate system values, elevation, latitude, longitude and angle in the base point definition in the model **Project properties**.

Base points in drawings

It is possible to use base-point-defined coordinate system values in drawings. If you change the project base point Z or the elevation value, the level value will change accordingly when a drawing is opened.

- Base point data can be used on the drawing and drawing view level to set the coordinate system. The base point can be used instead of the datum offset.
- When the base point is set, level attributes and template attributes in marks provide values in the specific base-point-defined coordinate system. Attributes with `_BASEPOINT` at the end give coordinates relative to the base point that has been selected in the view, and attributes with `_PROJECT` at the end give coordinates relative to the base point selected as the project base point.
- When the base point is set at the drawing level, the `_BASEPOINT` template attributes can be used in drawing templates.

In the drawing view properties, you can set **Location by** to model origin, project base point, or any base point defined coordinate system. **Location by** uses the project base point as the default value.

If you have not set **Location by** to any base point, the `_BASEPOINT` attributes give the same result as the basic attributes. For example, `COG_Z` would give same result as `COG_Z_BASEPOINT`. However, `COG_Z_PROJECT` always gives the value according to the project base point.

The datum level in the view properties only affects the attributes `TOP_LEVEL` and `TOP_LEVEL_UNFORMATTED` when **Location by** is set to **Model origin** or to the project base point that is in the model origin.

To change the **Location by** value:

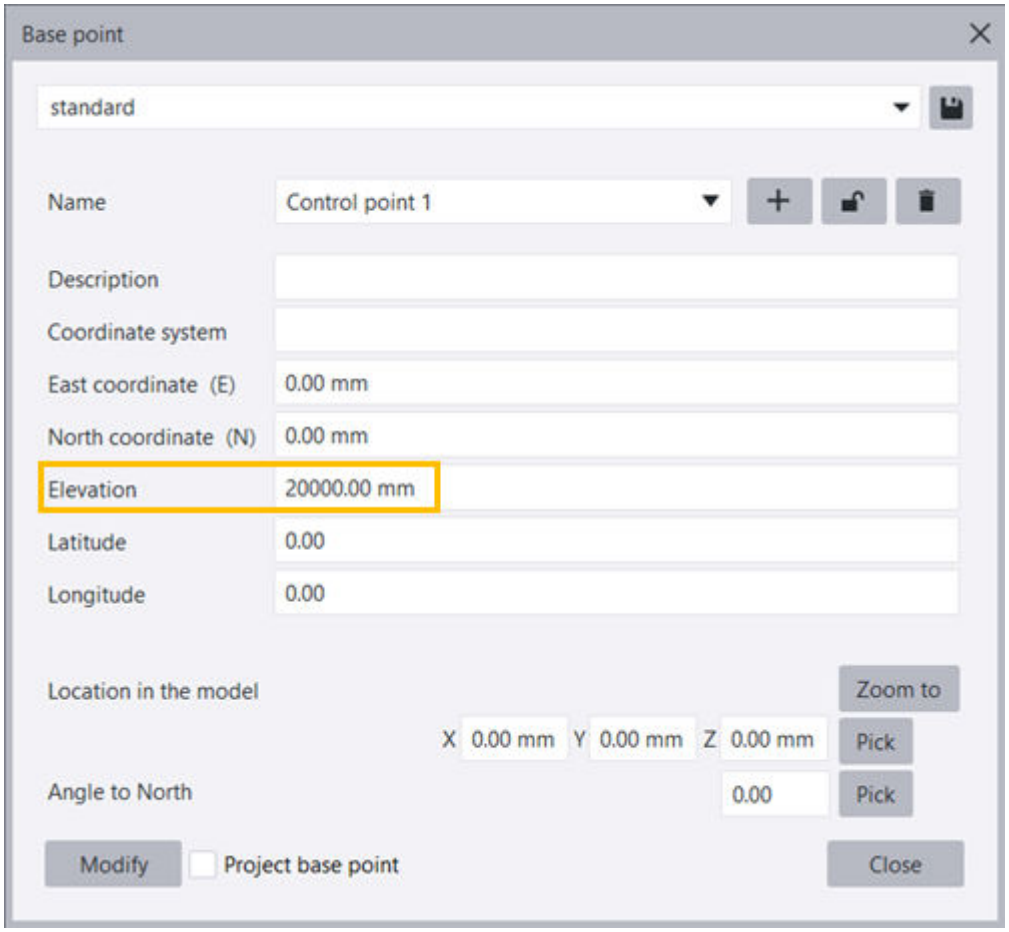
1. In an open drawing, double-click the drawing view frame to open the **View properties** dialog.

2. On the **Attributes 2** tab, set **Location by** to a new base point or to the model origin.
3. Click **Modify**.

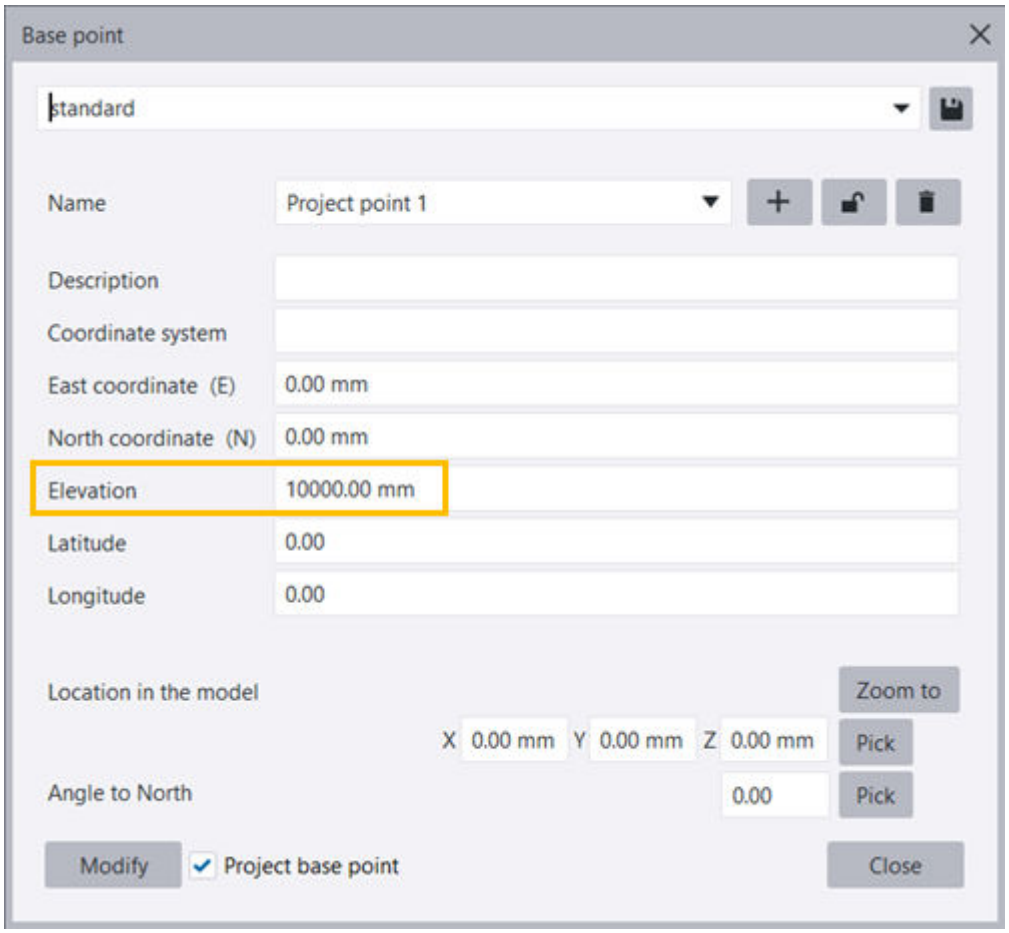
Example of base point usage in a drawing

In the following example, do the following:

1. Create a slab with a thickness of 200 mm with the slab's top at the level 0 in the model.
2. Create a new base point "Control point 1" with elevation 20000 mm.

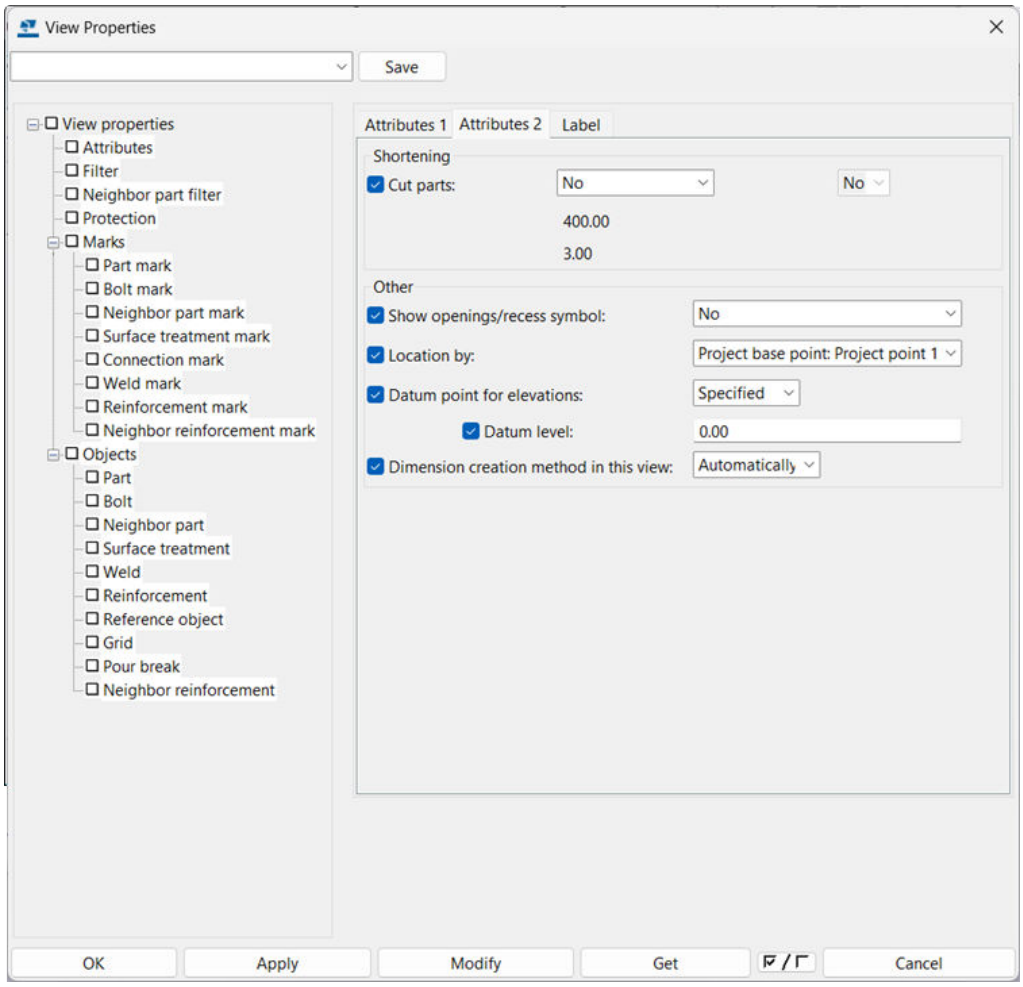


3. Create a new base point "Project point 1" with elevation 10000 mm, and set is as project base point by selecting the **Project base point** checkbox.



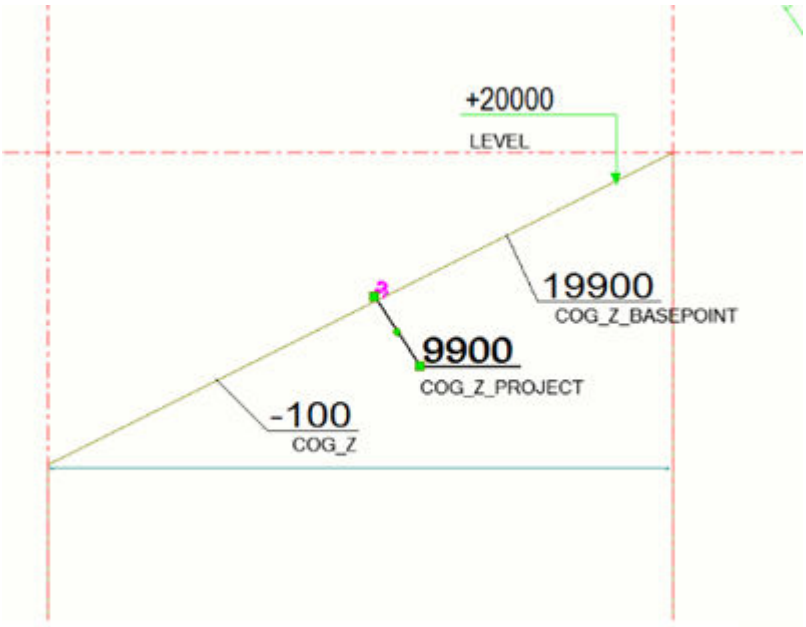
4. Create a GA drawing in plan view.
5. Open the GA drawing, double-click the view frame to open the **View properties** dialog.

6. On the **Attributes 2** tab, set **Location by** to the new base point Project base point: Project point 1 and click **Modify**.



7. Add a level mark using the following template attributes:
   - `COG_Z`
   - `COG_Z_PROJECT`
   - `COG_Z_BASEPOINT`
8. Reopen the drawing.

   Note that changing the value does not update the template attribute automatically but after reopening the drawing.



Base points in Layout manager
You can use base points in **Layout manager** when defining the location of layout points.

- You can use base points as location coordinates when exporting and importing layout points.
- When you add, modify or delete base points, you need to reopen or refresh **Layout manager** to make the changed base point data available in **Layout manager**.

Base point in reports and templates
You can inquire project base point and current base point value in reports and templates.

The following table lists the template attributes where you can use `_PROJECT` and `_BASEPOINT` at the end, for example, `ASSEMBLY_BOTTOM_LEVEL_PROJECT` or `ASSEMBLY_BOTTOM_LEVEL_BASEPOINT`. Attributes with `_BASEPOINT` at the end give coordinates relative to the base point selected in the view, and attributes with `_PROJECT` at the end give coordinates relative to the base point selected as the project base point. `_BASEPOINT` uses the current base point in the same way as the work plane uses the current work plane: If there is no current base point defined, `_BASEPOINT` provides values relative to model origin (Global).

Content type	Attributes
`ASSEMBLY`, `CAST_UNIT` and `PART`	ASSEMBLY_BOTTOM_LEVEL ASSEMBLY_BOTTOM_LEVEL_UNFORMATTED

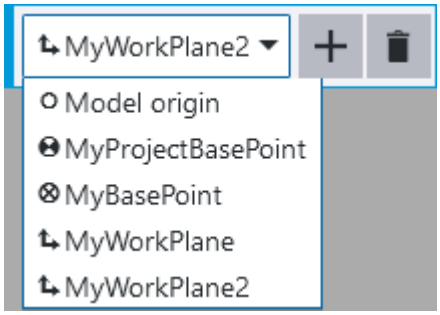
Content type	Attributes
	ASSEMBLY_TOP_LEVEL ASSEMBLY_TOP_LEVEL_UNFORMATTED BOTTOM_LEVEL BOTTOM_LEVEL_UNFORMATTED BOUNDING_BOX_MIN_X BOUNDING_BOX_MIN_Y BOUNDING_BOX_MIN_Z BOUNDING_BOX_MAX_X BOUNDING_BOX_MAX_Y BOUNDING_BOX_MAX_Z BOUNDING_BOX_X BOUNDING_BOX_Y BOUNDING_BOX_Z COG_X COG_Y COG_Z START_X START_Y START_Z END_X END_Y END_Z TOP_LEVEL TOP_LEVEL_UNFORMATTED LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION ASSEMBLY.LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION
REFERENCE MODEL, REFERENCE OBJECT and REFERENCE_ ASSEMBLY	BOUNDING_BOX_MIN_X BOUNDING_BOX_MIN_Y BOUNDING_BOX_MIN_Z BOUNDING_BOX_MAX_X BOUNDING_BOX_MAX_Y

Content type	Attributes
	BOUNDING_BOX_MAX_Z LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION
POUR OBJECT	BOTTOM_LEVEL BOTTOM_LEVEL_UNFORMATTED TOP_LEVEL TOP_LEVEL_UNFORMATTED LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION
CONNECTION	ORIGIN_X ORIGIN_Y ORIGIN_Z
HIERARCHIC OBJECT	LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION

Select the work plane

If you have defined base points or saved work planes in your Tekla Structures model, you can use the **Work plane handler** toolbar to select which work plane you currently have in use in the model.

By default, the **Work plane handler** toolbar is located at the bottom of the screen.



You can select among the following work planes:

- Model origin (if project base point is set somewhere else)
- Project base point
- All base points you have defined

  For instuctions on defining base points, see Base points (page 60).
- All work planes you have set and saved

  For instructions on setting work planes, see Shift the work plane (page 56).

The symbol for model origin is different if the project base point is set to model origin, or if the project base point is set somewhere else.

When a model is reopened, the last used work plane or base point is used.

Add a work plane to the toolbar

1. To add a work plane to the **Work plane handler** toolbar:
   a. Set a work plane in the model.

      For instructions, see Shift the work plane (page 56).

   b. On the **Work plane handler** toolbar, enter a name for the work plane in the **Select work plane** box.

   c. Click the **Save work plane**  button to add the new work plane to the list.

   d. Repeat as many times as needed.
2. To rename your work plane, double-click the work plane and enter a new name.
3. To switch between different work planes or base points in the model, select a suitable work plane or base point from the list.
4. To remove a work plane from the list, click the **Remove work plane**  button.

By default, the **Work plane handler** toolbar is located at the bottom of the screen. If you are unable to find the toolbar, click **File** --> **Settings**, and in the **Toolbars** list ensure that the **Work plane handler toolbar** is selected.

See also

Coordinate system (page 54)

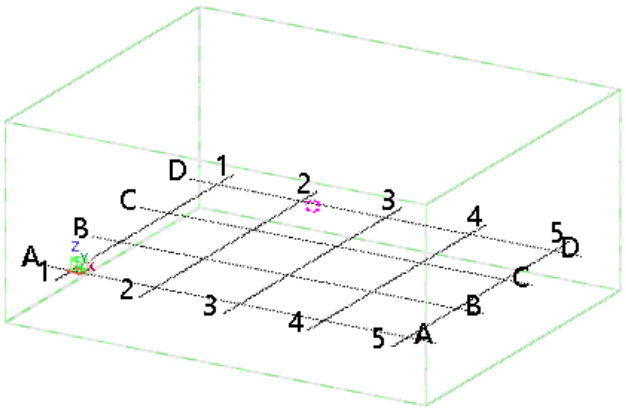
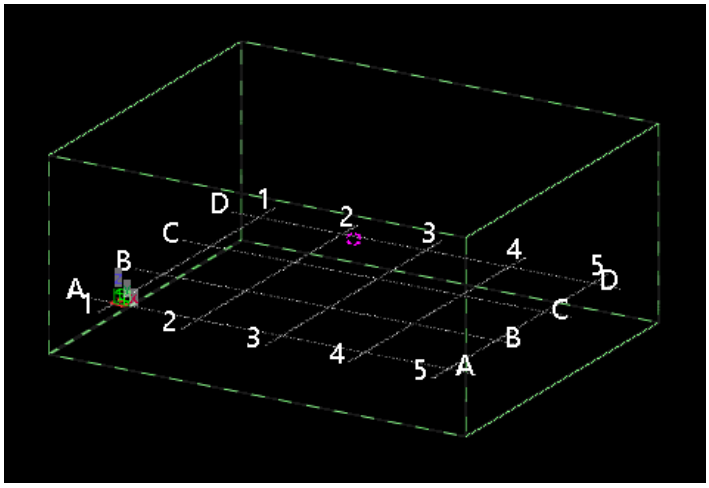
Shift the work plane (page 56)

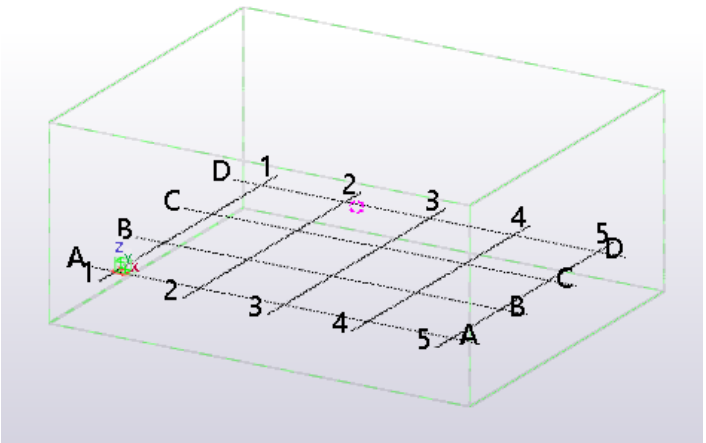
Base points (page 60)

Change the model background color

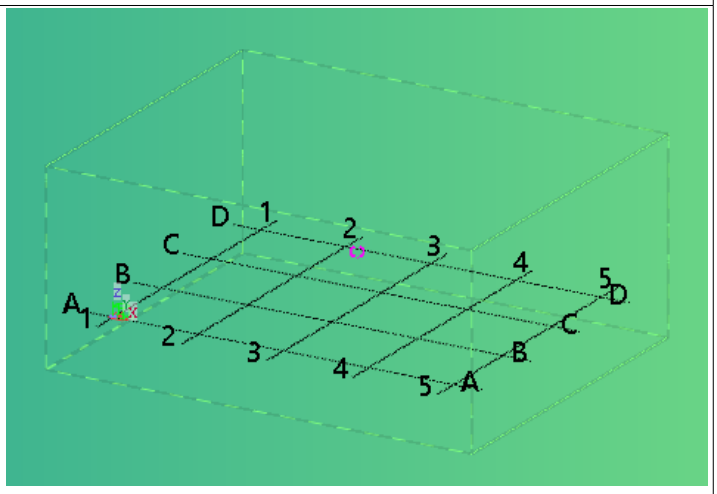
You can easily change the model background color.

1. On the **View** tab, click **Color theme**.
2. Select one of the available background color options.

Option	Example
For white background in all model views: **Light**	 The grid color is adjusted according to the background color. Grid line labels are set to black.
For black background in all model views: **Dark**	 The grid color is adjusted according to the background color. Grid line labels are set to white.

Option	Example
For the default gradient background in all model views: **Tekla default**	 The grid color is adjusted according to the background color. Grid line labels are set to black.

For your own background color in all open model views: **Advanced**	Use the `XS_BACKGROUND_COLOR1`, `XS_BACKGROUND_COLOR2`, `XS_BACKGROUND_COLOR3`, and `XS_BACKGROUND_COLOR4` advanced options to set the background color.
	Use RGB values on a scale from 0.0 to 1.0. You can control the color of each corner of the background separately. To use a single-colored background, set the same color code for all four corners of the background.
	For example, for green background color, set the advanced options to
	`0.1 0.6 0.6`
	`0.5 0.9 0.5`
	`0.1 0.6 0.6`
	`0.5 0.9 0.5`.

Option	Example
	 Click **OK** to save the changes. Close and reopen the view to see the changes.

3. If you want to change the drawing background color to black, use the advanced option XS_BLACK_DRAWING_BACKGROUND.

Change the rendering mode for model views

Tekla Structures model view rendering uses either the DirectX technology or the legacy OpenGL technology.

By default, Tekla Structures uses the DirectX rendering. The DirectX rendering adds a subtle shading effect to Tekla Structures objects, making the 3D visualizations clearer and more defined.

The rendering setting is model view specific, which means that you can use different rendering options in different model views. If you switch between the rendering options, you need to reopen the model view to activate the rendering option.

Switch between the rendering options

If you want to switch between the DirectX and the legacy OpenGL rendering options, use the **Use legacy rendering** switch.

1. In the upper-left corner of the screen, click **File**.



2. Go to **Settings**.
3. Under **Switches**, go to the **Use legacy rendering** option.
4.
   - When the checkbox is cleared, Tekla Structures uses the default DirectX rendering.
   - When the checkbox is selected, Tekla Structures uses the legacy OpenGL rendering.

   You can use different rendering options in different model views. If you switch between the rendering options, you need to reopen the model view to activate the selected rendering option.

DirectX rendering engine

The DirectX rendering mode is better optimized for modern graphics cards, and the graphics performance is better on the recommended NVIDIA GeForce GTX graphics cards than with graphics cards that have a lower end or no graphics processing unit (GPU). For more information on the graphics cards, see Tekla Structures hardware recommendations.

If you want to measure the performance of your DirectX 3D rendering engine, you can use the Tekla Structures graphics hardware test tool from Tekla Warehouse. The tool tests how fast your computer handles graphical information typically used in Tekla Structures, for example, processor speed, load time, average draw time, and per frame details. For more information, see the support article that shows graphs of the collected performance data on different hardware setups.

NOTE If you use Tekla Structures via remote connections, the DirectX rendering may not work as expected: parts that you have created may not show in the model, or the model is working slowly. If you experience such problems, switch the DirectX rendering off.

Use the following advanced options to fine-tune the DirectX rendering:

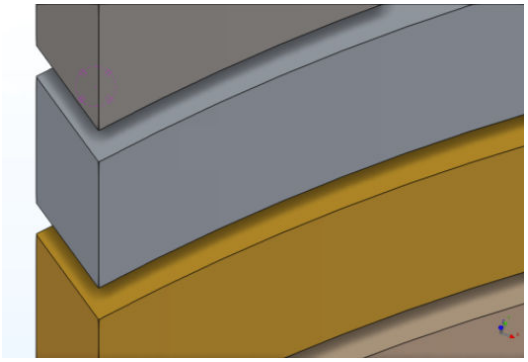
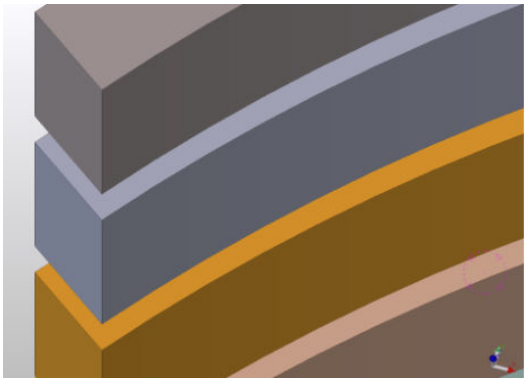
- XS_USE_ANTI_ALIASING_IN_DX

- XS_SHOW_SHADOW_FOR_ORTHO_IN_DX
- XS_SHOW_SHADOW_FOR_PERSPECTIVE_IN_DX
- XS_SHOW_STATISTICS_IN_DX

DirectX rendering examples

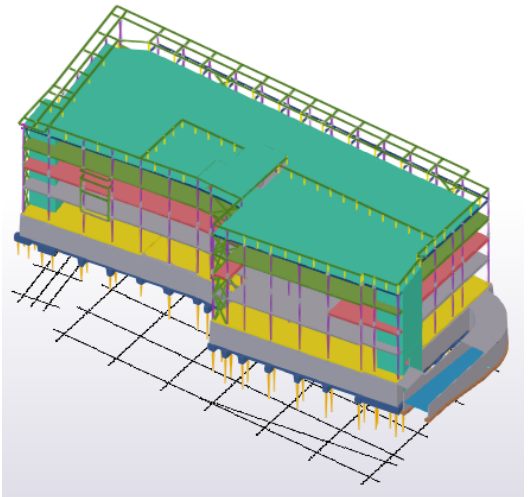
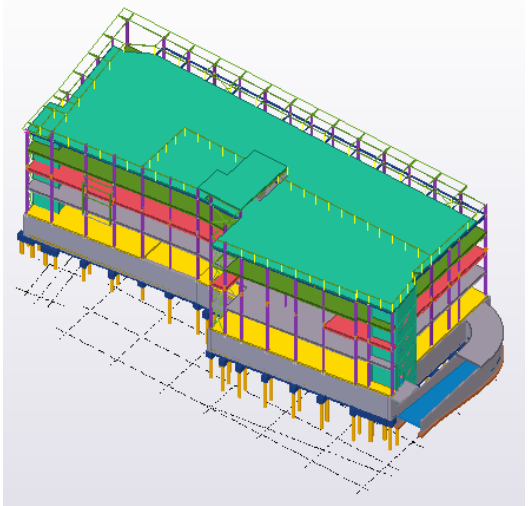
Visualization of distance

In DirectX rendered model views, the visualization of the distances uses subtle shadows and ambient occlusion. This gives a better understanding of the structure and distances.

DirectX	OpenGL
	

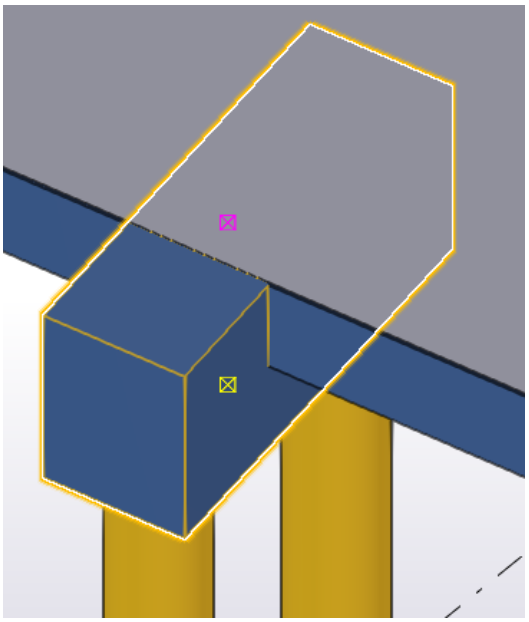
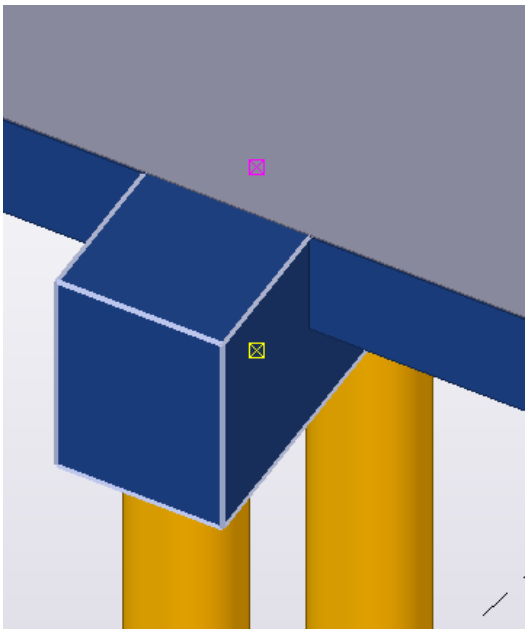
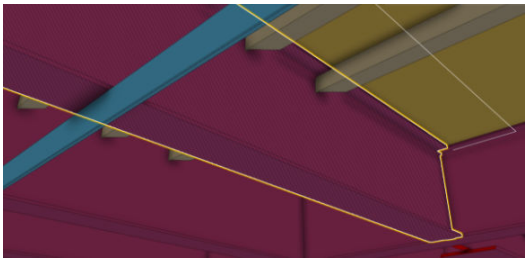
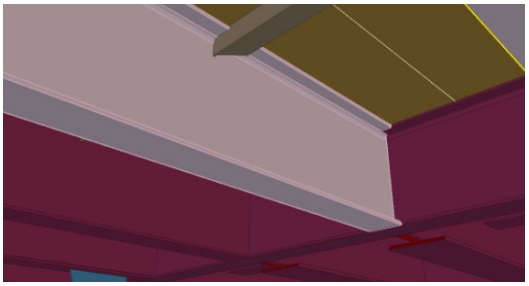
Depth accuracy

In DirectX rendered model views, the accuracy in depth buffer is improved so that when the model is zoomed, parts are not shown through other parts' faces as often as previously.

DirectX	OpenGL
	

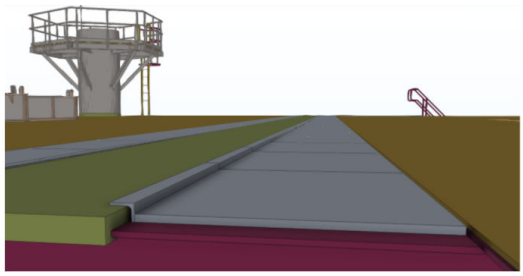
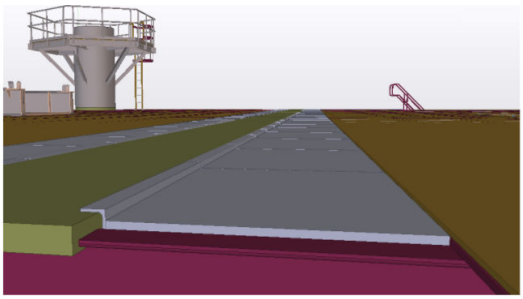
Dynamic states

In DirectX rendered model views, in dynamic states, such as in selection and in preselection highlight, the selection is clearer and the highlight is less intrusive.

DirectX	OpenGL
	
	

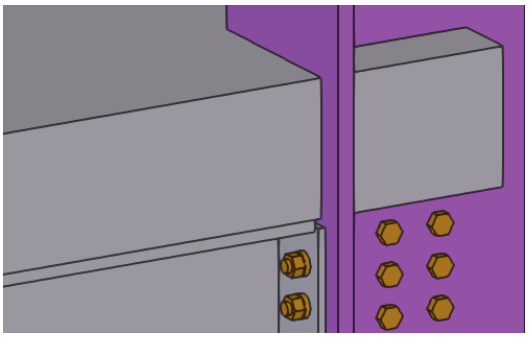
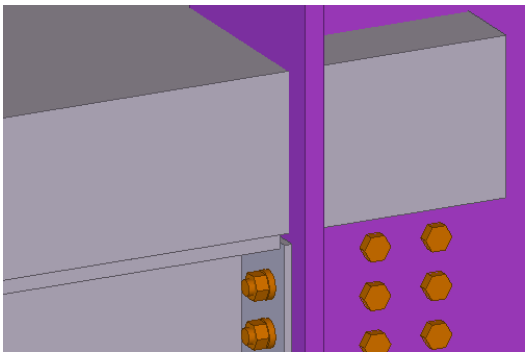
Antialiasing for higher fidelity

In DirectX rendered model views, the image quality is by default better than in OpenGL rendered model view, with less flickering.

DirectX	OpenGL
	

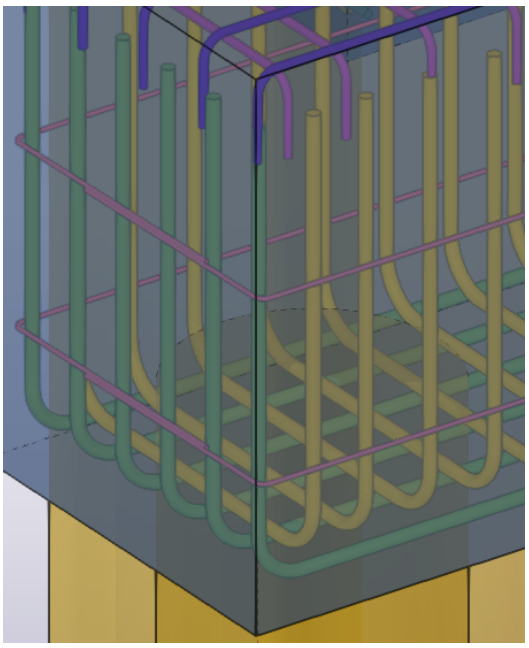
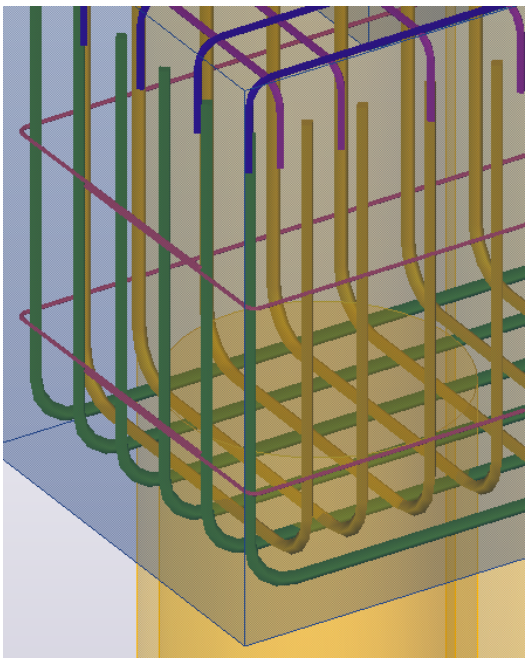
High fidelity edge line

In DirectX rendered model views, there are no flickering zigzag lines but continuous, smooth edges.

DirectX	OpenGL
	

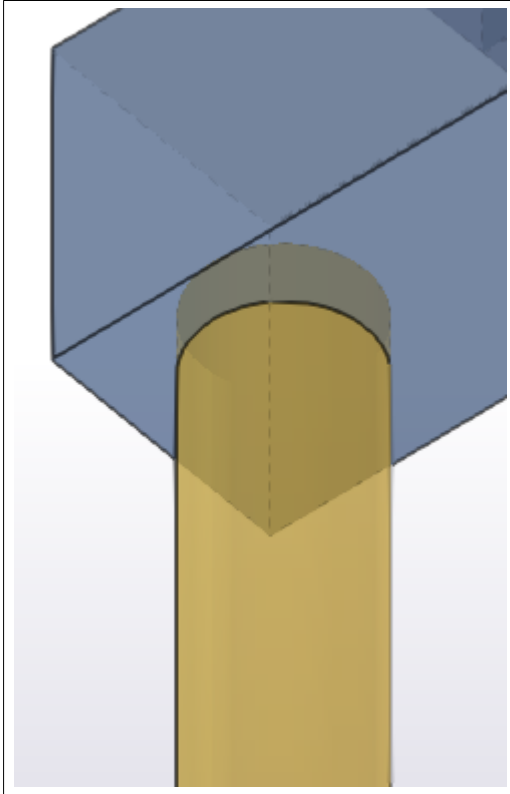
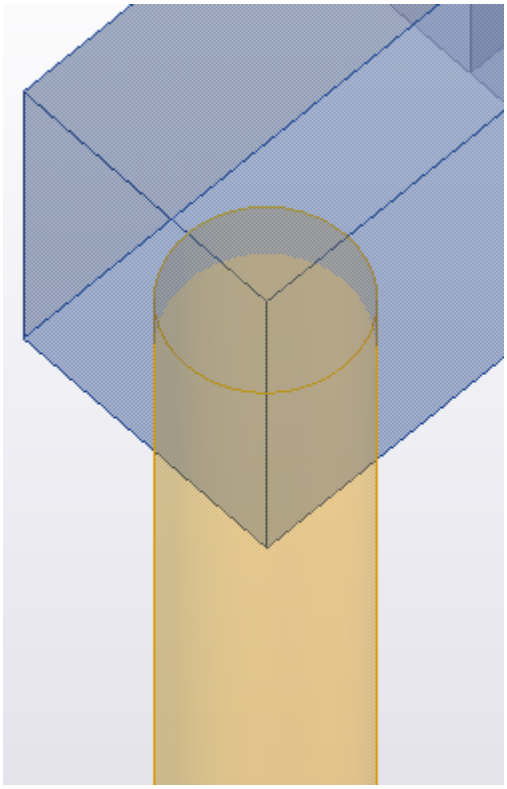
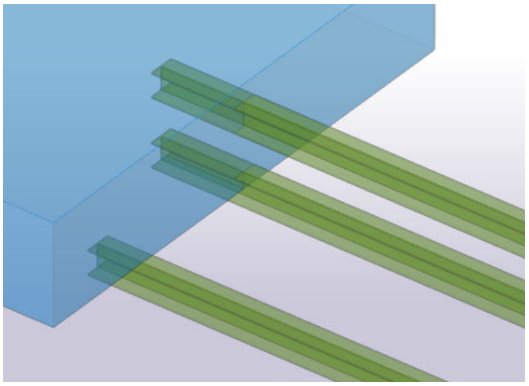
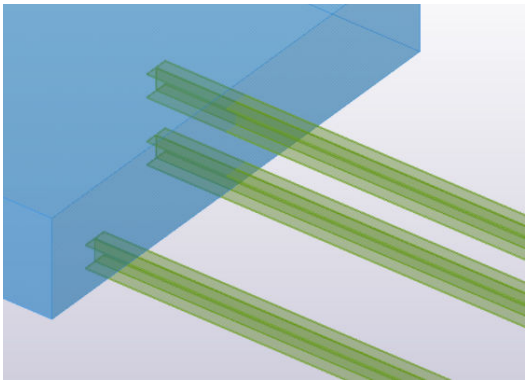
Accurate reinforcing bars

In DirectX rendered model views, reinforcing bars have edge lines. When you zoom in, the reinforcing bars are shown as round.

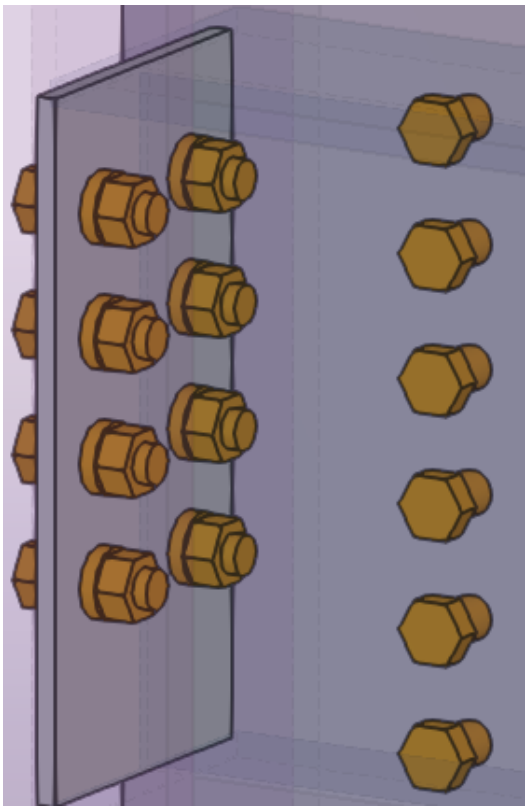
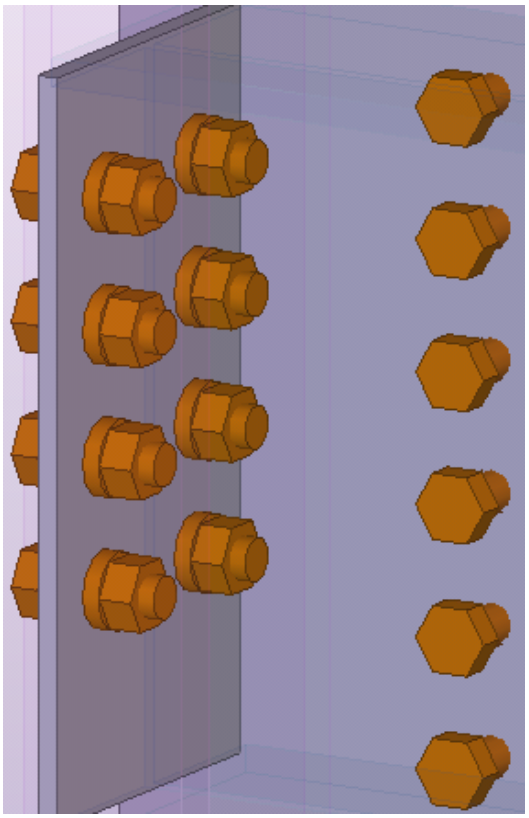
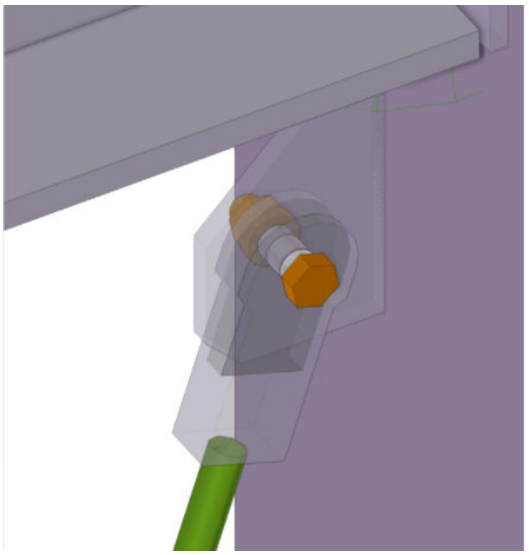
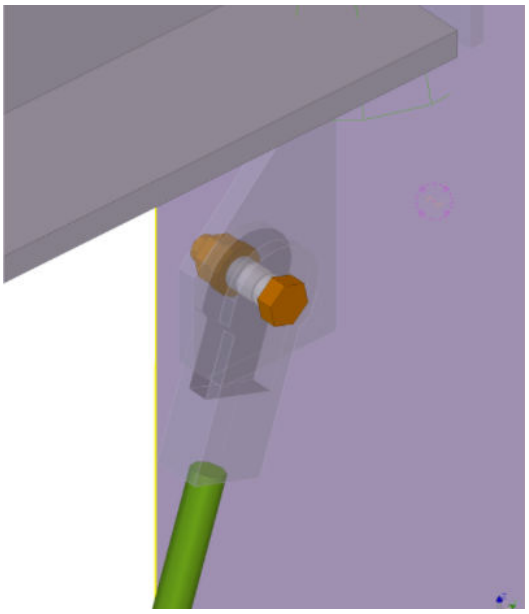
DirectX	OpenGL
	

Automatic edge lines for intersecting material in transparent view

In DirectX rendered model views, you can see where there are intersecting materials in the model.

DirectX	OpenGL
	
	

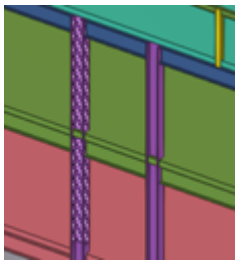
Accuracy and clarity on details

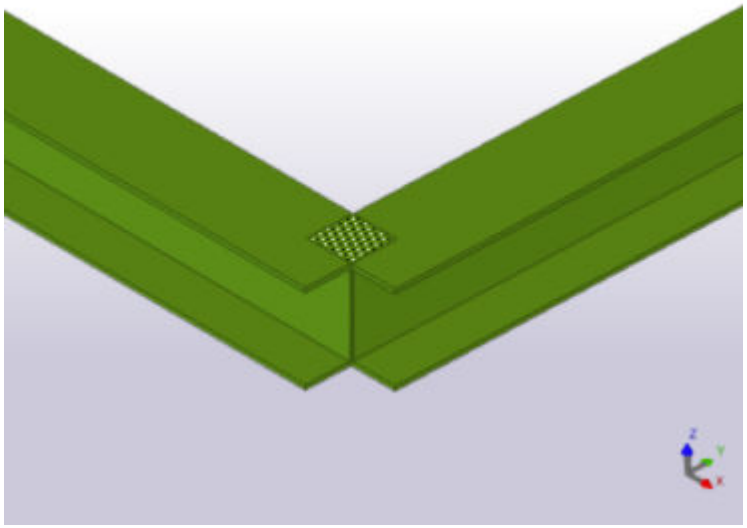
DirectX	OpenGL
	
	

Automatic hatching for overlapping surfaces on same plane

In DirectX rendered model views, duplicate objects or overlapping parts are visualized with a hatch in non-transparent views (**Ctrl+4** for parts and **Shift+4** for components).

Switch the hatching on in **File menu** --> **Settings** --> **Switches** --> **Hatching of overlapping surfaces**.

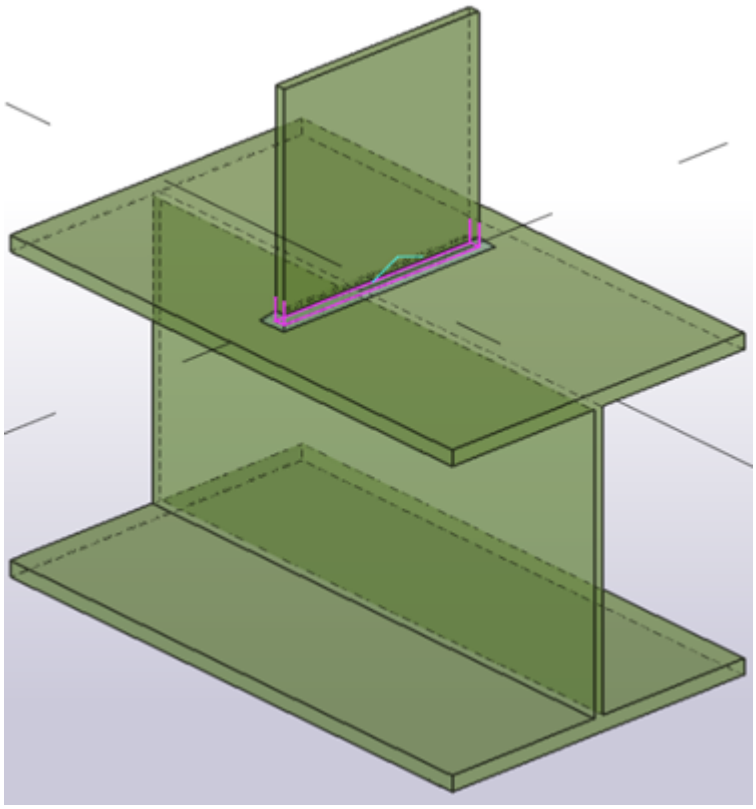


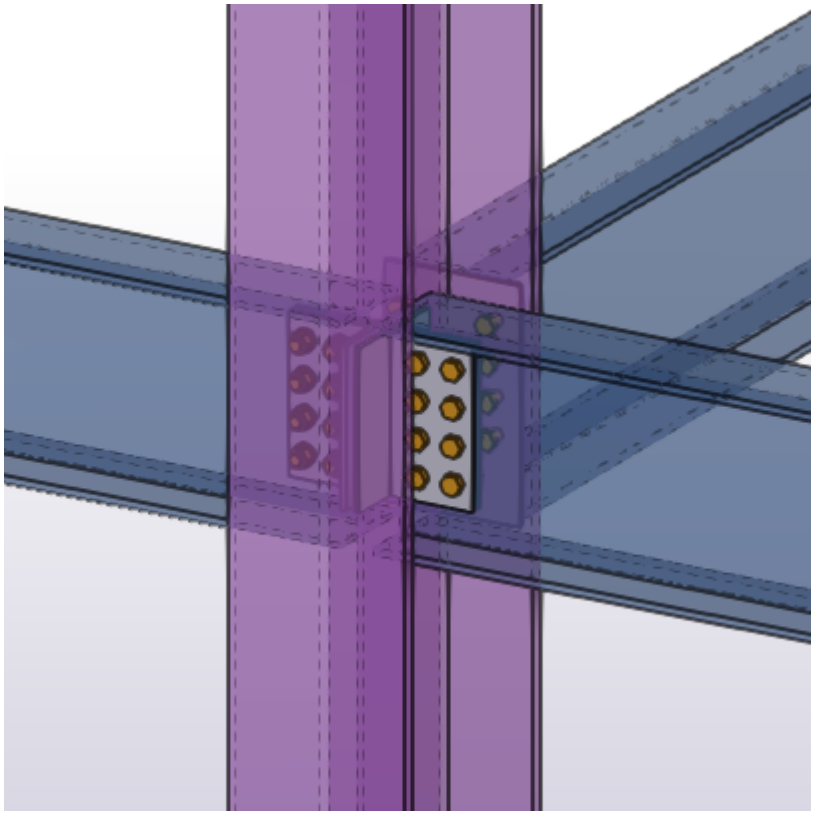


Dashed hidden lines

In DirectX rendered model views, the part edge lines that are hidden behind another part are shown as dashed lines in transparent views (**Ctrl+1, 2, 3, or 5** for parts and **Shift+1, 2, 3, or 5** for components).

Switch the dashed lines on in **File menu** --> **Settings** --> **Switches** --> **Dashed line for hidden line**.





1.2 Zoom and rotate the model

The commands on the **View** tab allow you to focus on a particular area, or pull out for a wider view. You can use a mouse, command, keyboard shortcut, or a combination of these.

Zoom in and out

You can use a variety of tools to zoom in and out in the model. By default, the mouse pointer position determines the center point of zooming.

To	Do this
Zoom in	Scroll forward with the mouse wheel. Alternatively, press **Page Up**.
Zoom out	Scroll backward with the mouse wheel. Alternatively, press **Page Down**.
Zoom to selected objects	1. Select the objects. 2. On the **View** tab, click **Zoom** --> **Zoom selected**. Alternatively, press **Shift+Space**.
Zoom with menu commands	On the **View** tab, click **Zoom** and select one of the zoom commands.
Keep the center point of zooming in the middle of the view	On the **File** menu, click **Settings** and select **Centered zooms**. When the option is **on**, the center point of zooming is kept in the middle of the view, regardless of the mouse pointer position.
Define the zoom ratio	Use these advanced options: XS_ZOOM_STEP_RATIO XS_ZOOM_STEP_RATIO_IN_MOUSEWHEEL_MODE XS_ZOOM_STEP_RATIO_IN_SCROLL_MODE

Rotate the model

You can use either the middle or left mouse button, or keyboard, to rotate the model in a view.

To	Do this
Rotate using the **middle** mouse button	1. On the **View** tab, click **Navigate** --> **Set view point**. You can also press **V**. 2. To set the view point, pick a position in the view. The following symbol appears in the model:  3. Hold down the **Ctrl** key, and click and drag the model with the middle mouse button. Tekla Structures rotates the model around the view point you defined in step 2.
Rotate using the **left** mouse button	1. On the **View** tab, click **Navigate** --> **Rotate with mouse**. You can also press **Ctrl+R**. 2. To set the view point, pick a position in the view. The following symbol appears in the model:  3. Click and drag the model with the left mouse button. Tekla Structures rotates the model around the view point you defined in step 2.
Rotate using keyboard	Use the keyboard shortcuts **Ctrl+arrow keys** and **Shift+arrow keys**. **Ctrl+arrow keys** rotates the model in 15 degree increments.

To	Do this
	Shift+arrow keys rotates the model in 5 degree increments. If you cannot rotate the model using the keyboard shortcuts, check that the **Disable view rotation** command is not active. The command is available only in **Quick Launch**, or by pressing F8.
Rotate once	**Shift+R**
Rotate continuously	**Shift+T**

Pan the model

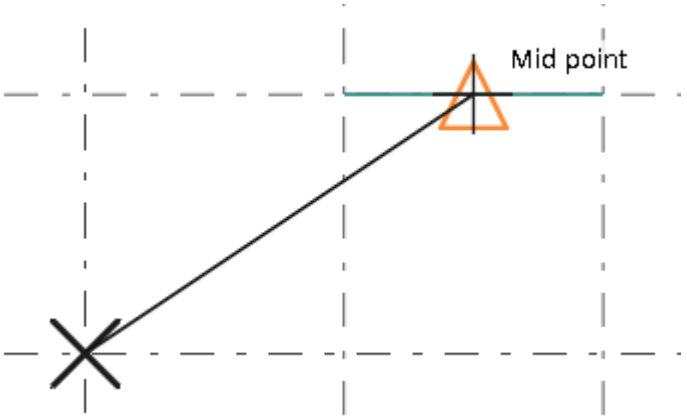
To move the model in a view, you can use either the middle or left mouse button to pan the model.

To	Do this
Move the model using the **middle** mouse button	1. On the **File** menu, click **Settings** and check that the **Middle button pan** checkbox is selected. Alternatively, • you can press **Shift+M** to switch the middle button pan on/off • go to **Quick Launch**, enter **Toggle middle button pan** and select the command from the list. 2. Hold down the middle mouse button and drag the model. When middle mouse button panning is active, the text **Pan** is shown at the bottom of the Tekla Structures window. You can also move the model using the arrow keys.
Move the model using the **left** mouse button	1. To activate dynamic panning, go to the **View** tab and click **Navigate** --> **Pan**. Alternatively, • you can press **P**

To	Do this
	• go to **Quick Launch**, enter **Pan** and select the command from the list. The mouse pointer changes to a hand:  2. Hold down the left mouse button and drag the model. 3. To stop panning, press **Esc**. You can also move the model using the arrow keys.

1.3 Snap to positions

Most commands ask you to pick points to place objects in the model or drawing. This is called *snapping*. When you are creating a new object, Tekla Structures displays snap symbols and snap tooltips for the available snap points and a light gray line between the snap point and the last point picked.



Use the snap switches (page 89) on the Snapping toolbar to control which positions you can snap to.

For example, you can snap to

- different points, such as end points and midpoints
- centers
- intersections
- lines and edges
- object faces
- dimensions and mark lines, drawing layout items and drawing frames

If you want to use exact distances or coordinates when snapping to positions, use numeric snapping (page 103).

With the combination of different snapping tools, you can, for example, snap to the closest orthogonal point (page 99) on the plane, both in the model and in the drawings. Additionally, you can follow a line and pick a point at a specified distance along the line, or create a temporary reference point to use as a local origin, both in the model and in the drawings.

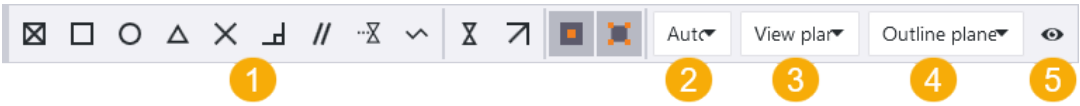
Tekla Structures displays snap dimensions in the model, which means you can easily create objects of a desired length. Use the advanced option XS_DISPLAY_DIMENSIONS_WHEN_CREATING_OBJECTS to switch the snap dimensions on or off.

TIP Use the snapping keyboard shortcuts to speed up your work.

Snap to points by using snap switches

Use the snap switches to control which positions you can pick in the model or drawing. By using snap switches, you can position objects precisely without having to know the coordinates. You can use snap switches whenever Tekla Structures prompts you to pick a point.

By default, the **Snapping** toolbar is located at the bottom of the screen. Click the snap switches on the **Snapping** toolbar to switch them on or off. If there is more than one point available to snap to, press the **Tab** key to cycle forward through the snap points, and **Shift+Tab** to cycle backwards through them. Click the left mouse button to select the appropriate point.



(1) Use the snap switches to control which positions you can pick when placing objects. Snap switches define exact locations on objects, for example, end points, midpoints, and intersections.

(2) Use the first list to define the snap depth.

(3) Use the second list to switch between the view plane and work plane (page 56).

(4) Use the third list to set the plane types. The plane type defines what planes you can select in the model.

(5) You can hide selected switches from the toolbar.

Alternatively, you can control the snap switches with **Quick Launch**. Start typing the name of the snap switch, for example, `snap`, and click the name of the snap switch in the search results list to activate the switch.

Snap zone

Each object has a snap zone. It defines how close you need to pick to hit a position. When you pick within the snap zone of an object, Tekla Structures automatically snaps to the closest pickable point on that object.

You can set the snap zone using the advanced option XS_PIXEL_TOLERANCE.

Snap priority

If you pick and hit several positions simultaneously, Tekla Structures automatically snaps to the point with the highest snap priority. To control which positions you can pick, use snap switches. Snap switches define the snap priority of positions.

Snap depth

The first list on the **Snapping** toolbar defines the depth of each position you pick. You have the following options:

- **Plane**: You can snap to positions either on the view plane (page 31) or the work plane (page 54), depending on what you have selected in the second list on the **Snapping** toolbar.
- **Auto**: In perspective views, this option works like the **3D** option. In non-perspective views, it works like the **Plane** option.
- **3D**: You can snap to positions in the entire 3D space.

Visual cues in snapping

Tekla Structures indicates where you can snap in the model and which snap switches can be used to snap to certain positions.

When you start a command that requires picking points and you move the mouse pointer over objects, the mouse pointer locks to a snap point and Tekla Structures displays in the model

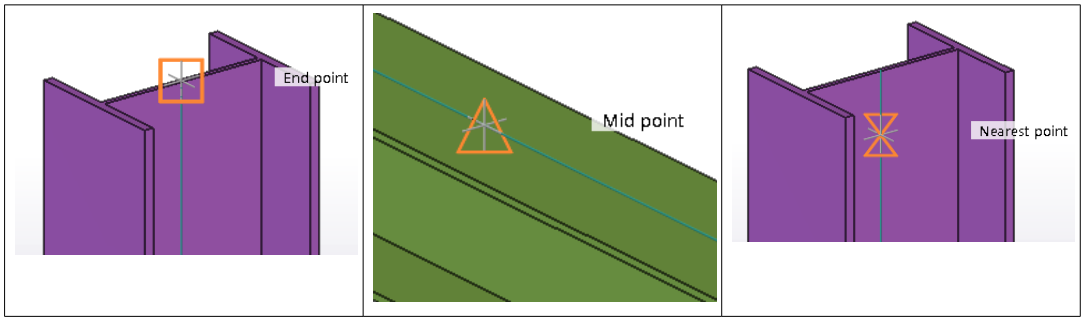
- a snap symbol

  The snap symbol changes according to the possible snap point. Tekla Structures automatically highlights the points where you can snap to.

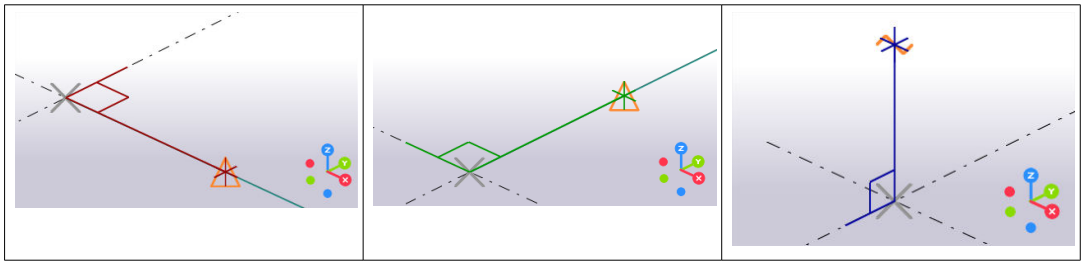
  Use the snap settings (page 113) to show or hide the snap symbols.
- a snap tooltip which shows the name of the possible snap point

  To show or hide the snap tooltips, click **File** --> **Settings** and select the **Snap tooltips** check box.
- a turquoise reference line or geometry line of the object. The turquoise line shows the line or the edge to which the snap point belongs to.

For example:



Additionally, Tekla Structures indicates in which direction the picked points are located. When the Ortho (page 99) tool is active, Tekla Structures displays a rubber band line between the last point picked and the snap point. The color of the cursor and the rubber band line follow the color of the work plane axis: red for x-axis, green for y-axis and blue for z-axis. For any other direction the color of the rubber band line and the cursor is black.



As a general rule of thumb, you can only snap to something that is visible.

For example, if you use the rendering options **Parts rendered** or **Components rendered** (**Ctrl/Shift +4**), the object surfaces are displayed and the objects are not transparent. This means that you cannot snap to the object's geometry lines or reference lines which cannot be seen through the object.

Override the current snap switch settings

You can temporarily override the current snap switch settings, and activate only the selected snap switch. The selected snap switch overrides the other snap settings for the next point you pick.

1. Run a command that asks you to pick a point.

   For example, start creating a beam.

2. To override the current snap switches, do one of the following:
   - Right-click to show a list of snap options, and then select one of the options.
   - Click **File** --> **Settings**, and in the list of toolbars select **Snap override toolbar**.

     A new toolbar appears. Click a button to activate the selected snap switch.



- Use Quick Launch to activate a snap override switch. Type `override` in the **Quick Launch** box and select the needed override switch in the search results list.

Snap to parallel points, lines, edges, and extension lines

You can snap to lines when you model objects that should be lined up with or parallel to an existing object or with a grid line. You can also snap to the extension lines of part reference lines, or to the extension lines of nearby objects' reference lines.

Snap to parallel points

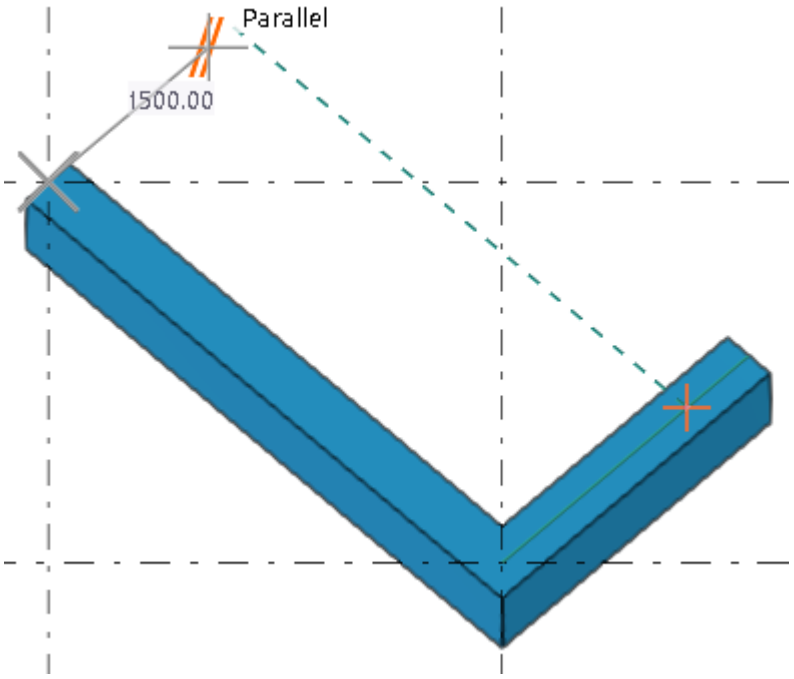
With the **Snap to parallel points** snap switch you can snap to points that are parallel to grid lines, reference lines, and the edge lines of existing objects or reference objects. Use the **Snap to parallel points** switch, for example, when you need to model new parts or structures that are parallel to an existing structure or parallel to a grid line, such as strip footings or walls, or if you need to cut polygonal shapes that are parallel to edge lines of irregular shapes.

1. Ensure that **Snap to parallel points**  snap switch is active.
2. Run a command that requires you to pick two or more points.

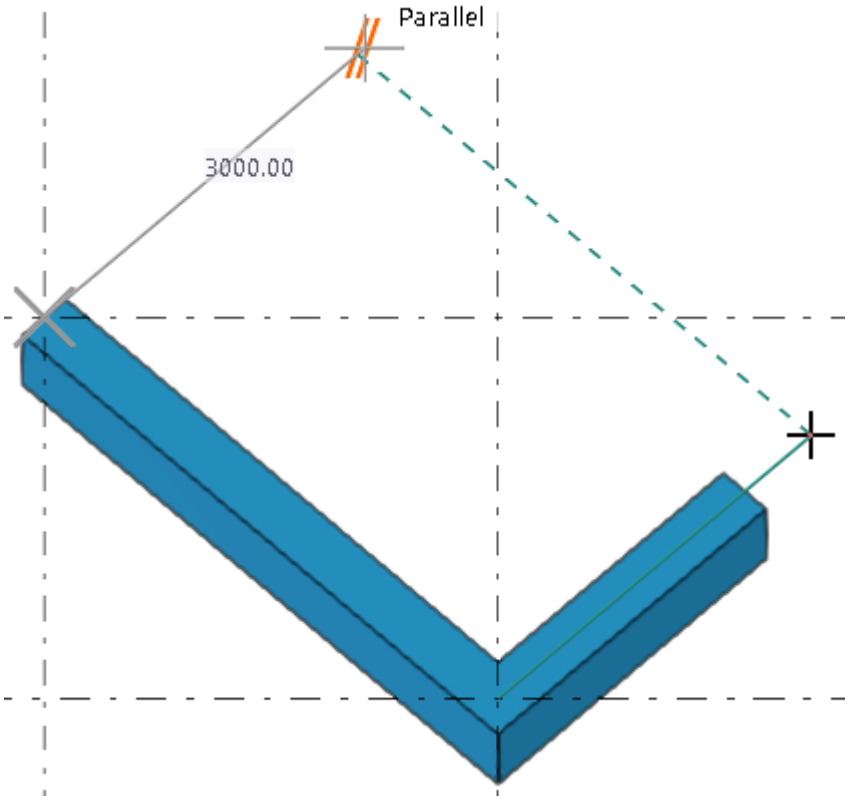
   For example, a beam, a wall, or a strip footing.
3. Pick the start point in the model.

   Alternatively, you can pick a temporary reference point by holding down the **Ctrl** key and then pick a point that is used as a reference point.
4. Place the mouse pointer on top of a line or an edge that you want to use as basis for the parallel snap, and using the mouse pointer follow the line or the edge to the desired direction. Keep the mouse pointer close to the line or the edge.

Tekla Structures shows the parallel snap symbol and a green, dashed line that indicates the dependency between the picked point and the parallel point.



If you move the mouse pointer further away in the direction of the line or the edge, the parallel snap follows the extension of the line or the edge.



5. Pick the rest of the points, or enter a distance by using the keyboard and press **Enter**.

   Alternatively, if you had previously picked a temporary reference point, pick the start point.

Snap to a line or an edge

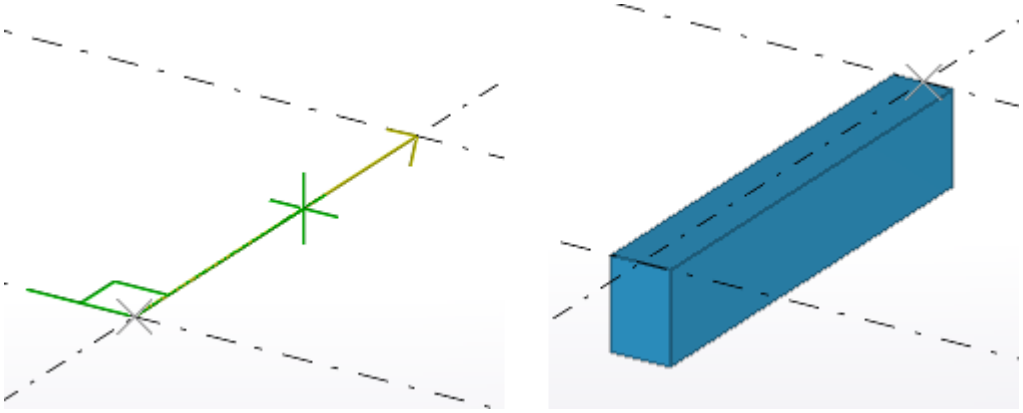
Use the **Snap to line** snap switch when you need to snap to another line in the model. You can snap to grid lines, reference lines, and the edges of existing objects.

Use the **Snap to line** snap switch when you need to create, for example, several beams one after another on a grid line. With the **Snap to line** snap switch you do not need separately pick the start point and the end point of the beam.

1. Ensure that the **Snap to line**  snap switch is active.
2. Run a command that requires you to pick two or more points.

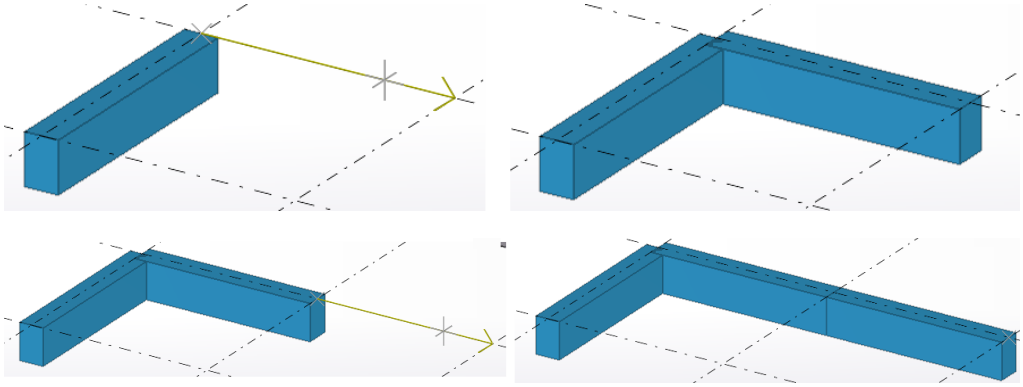
   For example, start creating a beam. When you move the mouse pointer over a grid line or a nearby object, Tekla Structures automatically picks

both ends of the line. The yellow arrow symbol indicates the direction of the points.



3. To switch direction, move the mouse pointer closer to the opposite end of the line.
4. Click the left mouse button to confirm the snap position.

   Tekla Structures creates the object. For example:



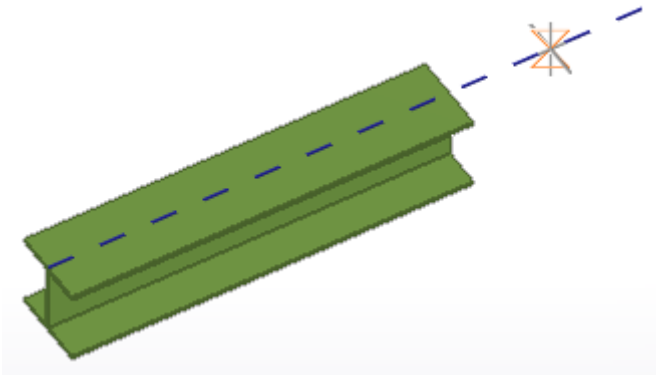
NOTE If you use the **Snap to line** snap switch with a command that requires only one point to be picked, for example when creating a column, only the start point of the line is used to position the part.

Snap to extension lines

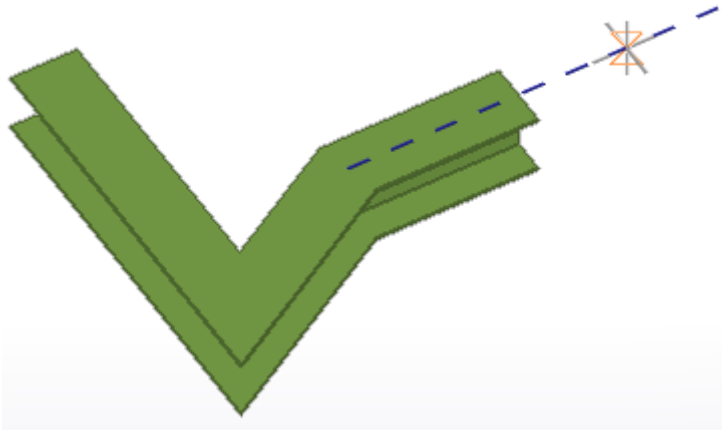
Use the **Snap to extension lines** snap switch when you need to snap to the extensions of the part reference lines, which are the lines between part handles, or to the extensions of nearby objects' reference lines. The extension line is shown as a blue, dashed line. You can use extension lines as an aid when you want to align objects with one another, or for creating objects parallel to an existing object.

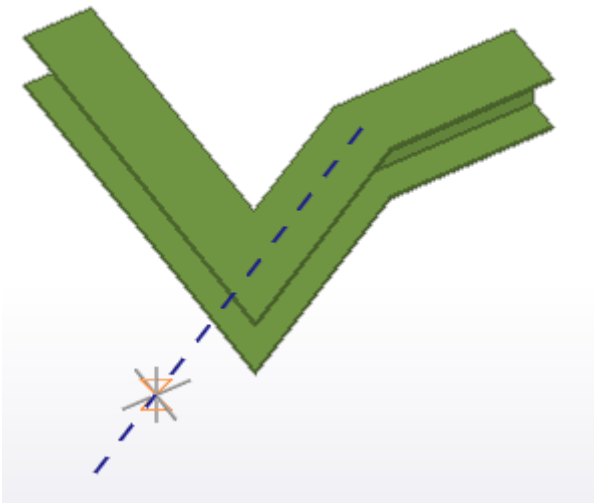
The **Snap to extension lines** snap switch works with beams, polybeams, plates, and slabs.

- With beams, the extension line is the line that passes through both of the part handles.

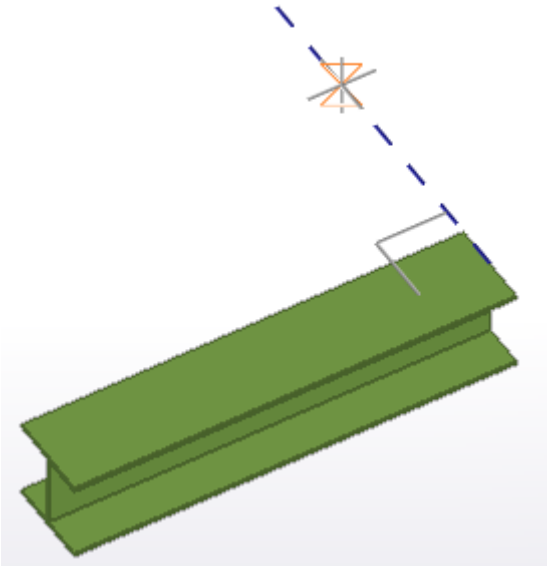


- With polybeams, plates, and slabs, the extension lines are the lines that pass through to the consecutive part handles.





- With beams and polybeams, you can snap to the line that passes through the handle at the end of the beam, and which is perpendicular to the direction of the beam.

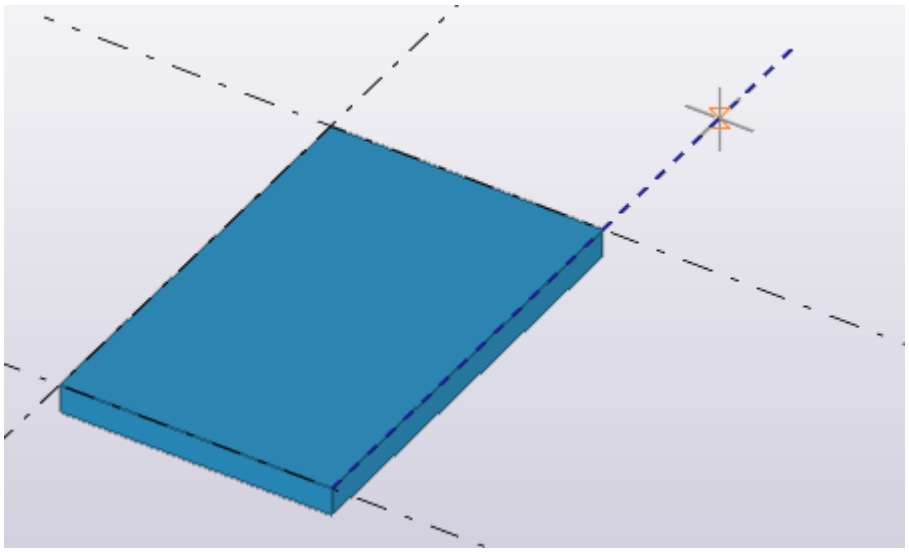


NOTE The **Snap to reference lines and points** and **Snap to geometry lines and points** snap switches do not affect the **Snap to extension lines** snap switch.

---

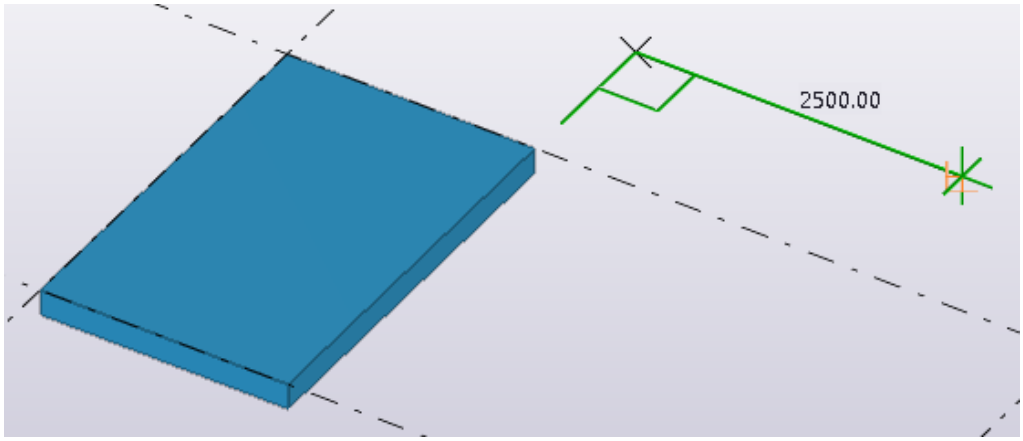
1. Ensure that the correct snap switches are active:
   - Switch on **Snap to extension lines** .
   - Switch on either **Snap to intersection points** or **Snap to nearest points (points on line)** if you are snapping to the intersection of an extension line and a grid line.
   - Switch off **Snap to end points** if you are working in 3D.
2. Run a command that requires you to pick points.

   For example, start creating a beam, a plate, or slab.
3. Move the mouse pointer over an existing object to see the extension lines.

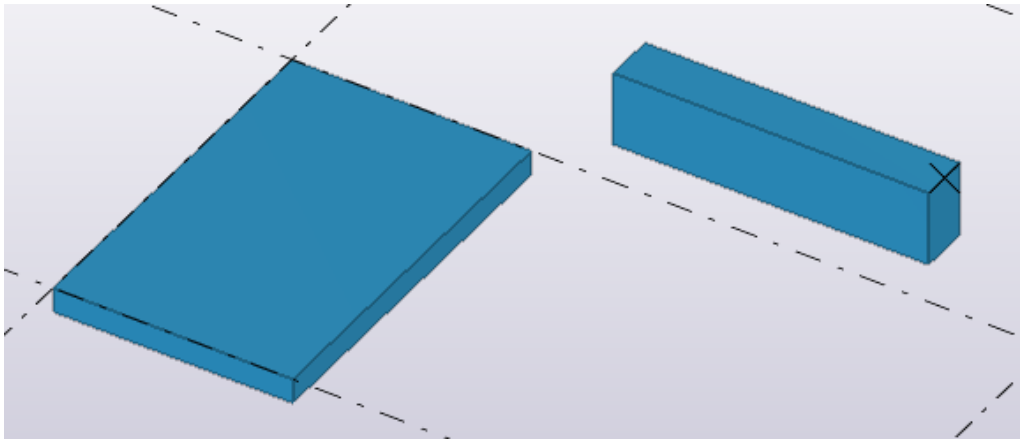


When a line is found, you can move the pointer further away while keeping the snap.

4. Pick the rest of the points.



Tekla Structures creates the object:



Tips on how use extension lines as a snapping aid

- When you snap to the extension lines of nearby objects, the extension line snaps to the direction of the nearby object and the extension line indicates the direction that is followed in snapping. Snapping to the extension lines of nearby objects can be useful, for example, when you want to align objects with one another.



- If you need to create objects parallel to existing objects, for example parallel beams, use the extension line of a nearby object as a snapping aid.

  For example, start the **Create steel beam** command, and pick the start point. Move the mouse on top of the object whose extension line you are using to create the new parallel beam. Click on the extension line to indicate the end point of the new parallel part.

Snap in orthogonal directions

You can snap to orthogonal points in models and in drawings using the **Ortho** tool. If you create objects that require you to pick multiple points, you can snap in orthogonal directions relative to the two previously picked points.

NOTE The **Ortho** tool has the lowest priority among snap points.

Even if you have activated the **Ortho** tool, but Tekla Structures detects any other possible snap point than an orthogonal point, Tekla Structures uses the

found snap point instead of the orthogonal snap point. If there are no other possible snap points found, then Tekla Structures uses the orthogonal snap point.

Activate the Ortho tool

Before you can snap in orthogonal directions, ensure that the **Ortho** tool is active. The letter **O** in the status bar at the bottom of the Tekla Structures main window indicates that **Ortho** is active.

If **Ortho** is not active

- press **O** to activate it
- alternatively, click **File** --> **Settings** and select the **Ortho** checkbox.

Snap to orthogonal points

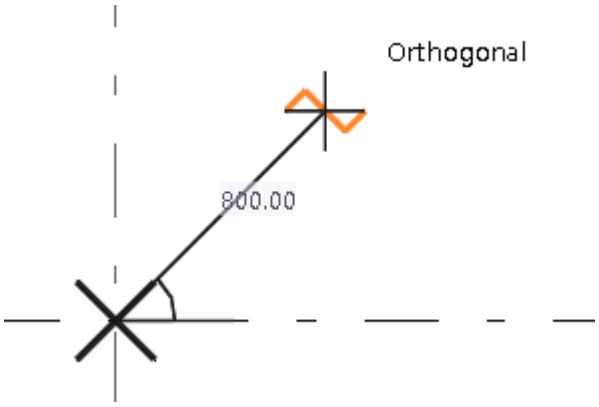
Use the **Ortho** tool to snap to the closest orthogonal point on the plane (0, 45, 90, 135, 180 degrees, and so on). The mouse pointer automatically snaps to positions at even distances in the given direction. This can be useful, for example, if you need to place marks in a consistent manner in exact locations in a drawing.

1. Ensure that the **Ortho** tool is active.
   - Press **O** to activate **Ortho** if the tool is not active.
   - Alternatively, on the **File** menu, click **Settings** and select the **Ortho** checkbox.
2. Run a command that requires you to pick points.

   For example, start creating a beam. Tekla Structures displays an angle symbol to indicate the direction of snapping.

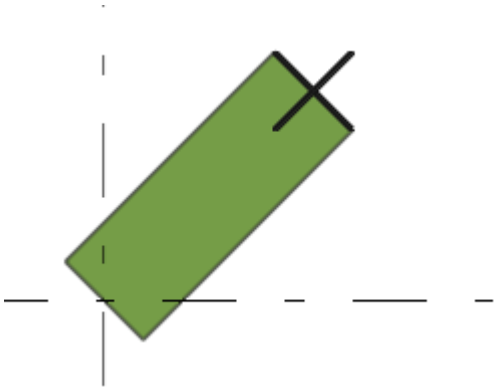
   The snapping precision depends on the current zoom level.

   The angle interval (page 113) depends on the settings in the **Model snap settings** dialog.



3. Click the left mouse button to confirm the snap position.

   Tekla Structures creates the object. For example:



Snap in orthogonal direction relative to previously picked points

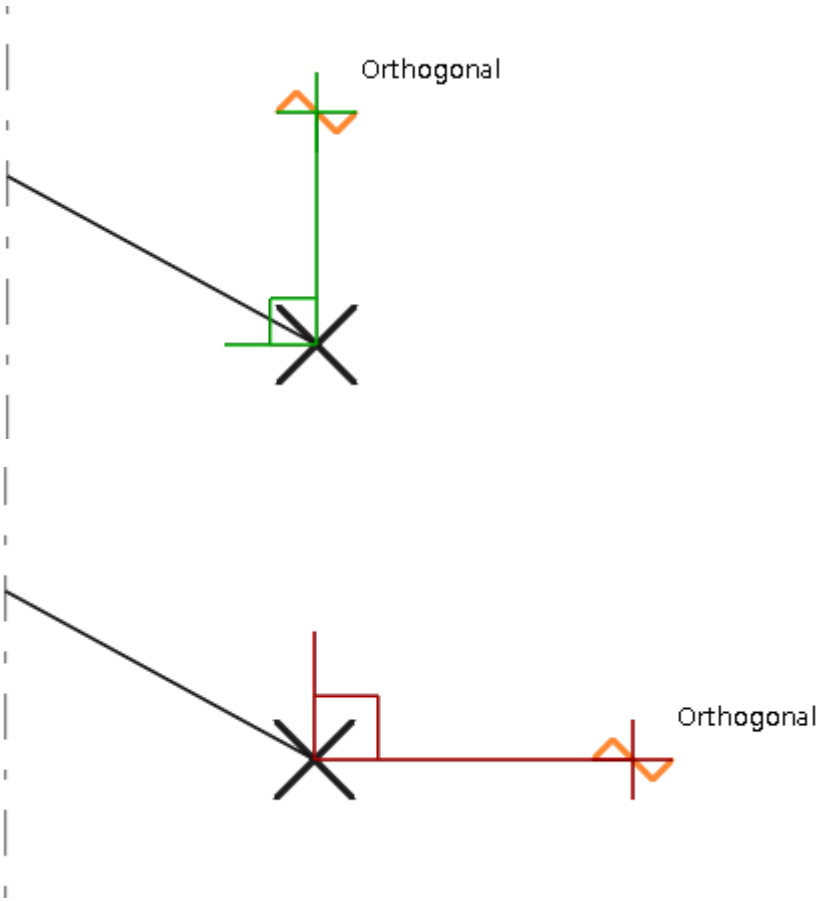
When you create objects that require you to pick more than two points, for example when you create a polybeam or a contour plate, you can snap in orthogonal directions relative to the two previously picked points. This can be useful, for example, if you need to create a rectangular slab that is on the view plane but not along the x and y axes.

1. Ensure that the **Ortho** tool is active.
   - On the **File** menu, click **Settings** and select the **Ortho** checkbox.
   - Alternatively, you can press **O**.
2. Run a command that requires you to pick multiple points.

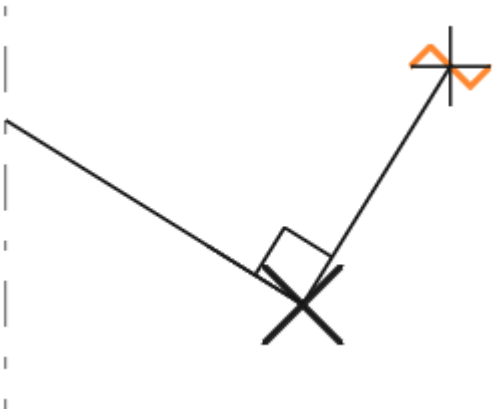
   For example, start creating a polybeam or a rectangular slab.
3. Pick the first two points.

   Tekla Structures displays an angle symbol to indicate the direction of snapping.
4. Move the mouse pointer in the model to see the angle symbol.

When the snap is orthogonal to a work plane axis, the color of the angle symbol follows the color of the work plane axis: red for x-axis, green for y-axis and blue for z-axis.



When the snap is orthogonal to the previous points, the color of the angle symbol is black.



5. Pick the rest of the points.

Tekla Structures creates the object. For example:



Settings for the Ortho tool

Use the settings (page 113) in the **Snap settings** dialog to set the angle interval for **Ortho**. Use the **Angle interval** or the **Custom angles** option.

The default angle interval value is 90 degrees.

Snap to points by using exact distance or coordinates - numeric snapping

You can enter exact distances and coordinates when snapping to a position. This is called *numeric snapping*.

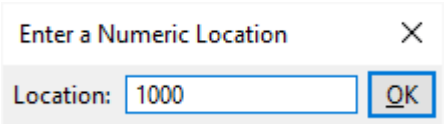
Enter a distance or coordinates

Use the **Enter a Numeric Location** dialog to specify the distance or coordinates to a position you want to snap to.

1. Run a command that requires you to pick points.

   For example, start creating a beam.
2. Pick the first point.
3. Move the mouse pointer to indicate the direction of snapping.
4. Enter a distance or coordinates by using the keyboard.

   For example, type `1000` as the distance from the last point picked. When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog automatically.

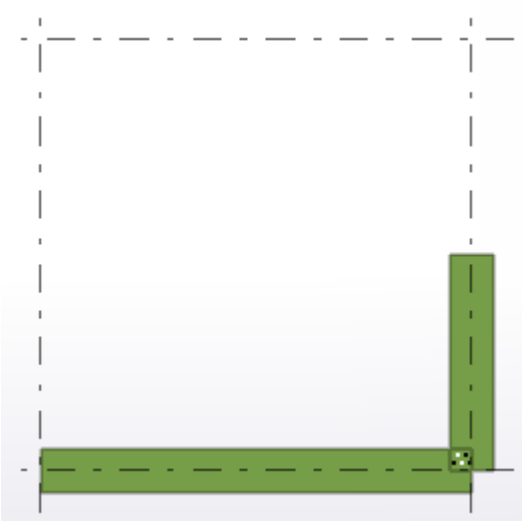


5. After entering the distance or coordinates, click **OK** or press **Enter** to snap to the position.

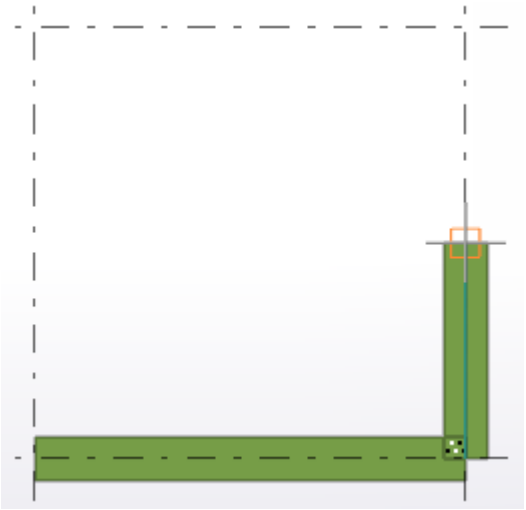
Snapping example: Track along a line towards a snap point

Tracking means that you follow a line and pick a point at a specified distance along the line. You usually use tracking in combination with numeric coordinates and other snapping tools, such as snap switches and orthogonal snapping. This example shows how to pick a point at a specified distance along a line. Use the **Enter a Numeric Location** dialog to specify the distance from the last point picked.

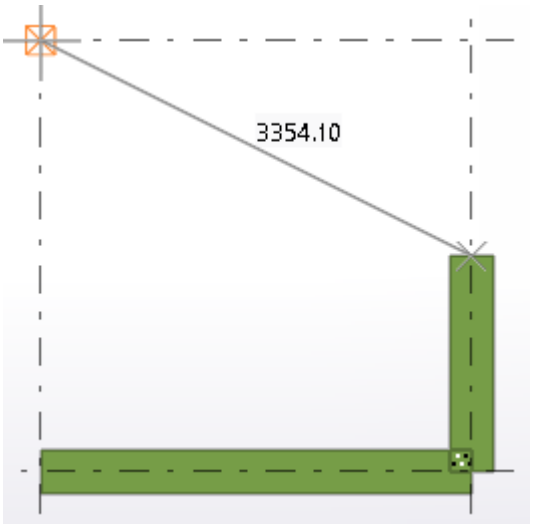
1. Create two beams and place them as shown below:



2. Activate the beam command, to create one more beam.
3. Pick the first point.

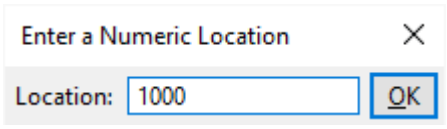


4. Move the mouse pointer over the grid line intersection so that it locks onto the snap point, but **do not** click the mouse button.



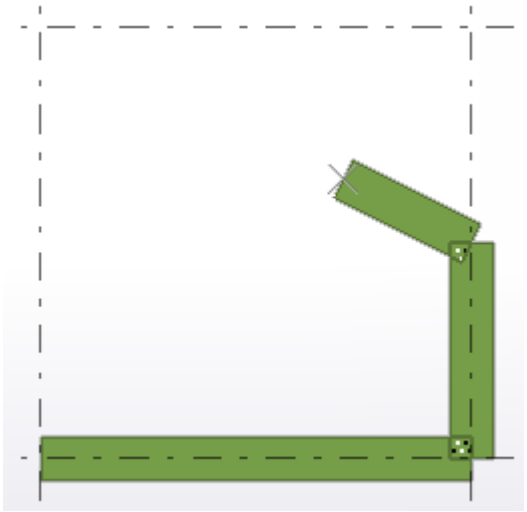
5. Type `1000`.

   When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog.



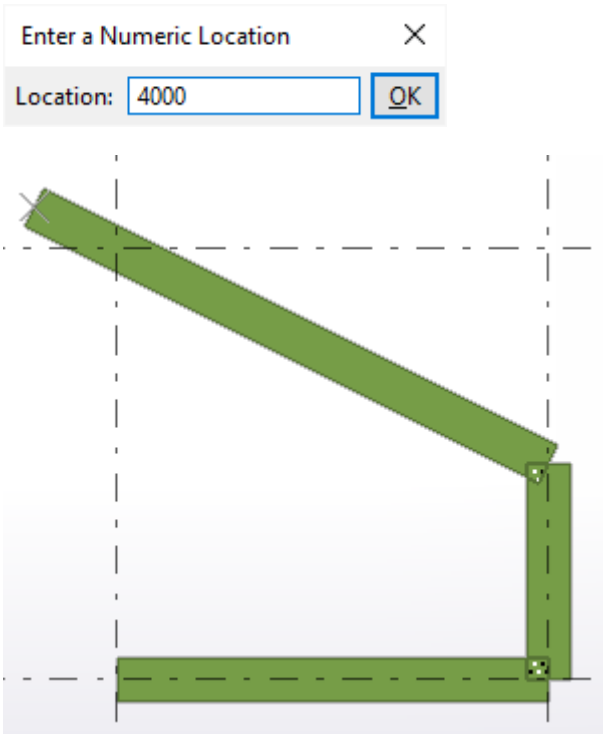
6. Click **OK** to confirm the distance.

   Tekla Structures creates a beam, which is 1000 units long and positioned between the points you defined:

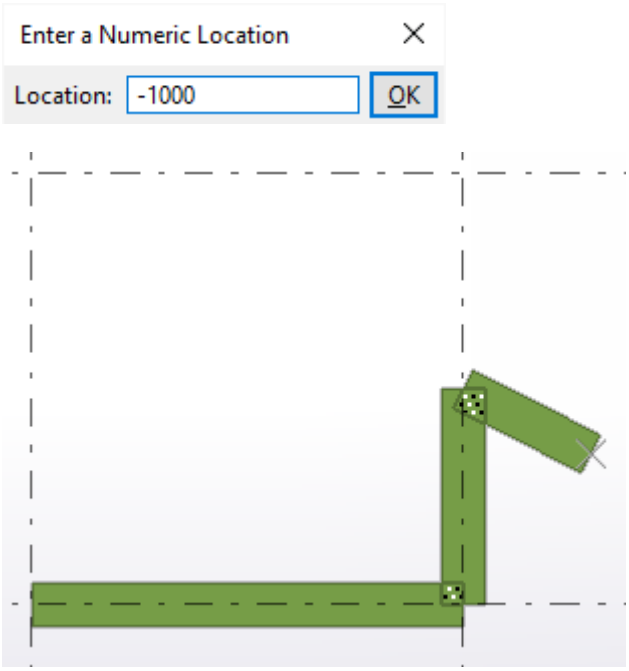


You can also:

- Track beyond the snap point, for example 4000 units from the first point:



- Track in the opposite direction by entering a negative value, for example `-1000`:



For an example of how to use numeric snapping in drawings, see Place a sketch object at a specified distance.

Options for coordinates in numeric snapping

The table below explains the types of information you can enter in the **Enter a Numeric Location** dialog.

Note that Tekla Structures has three *snapping modes*: relative, absolute, and global. You can temporarily override the default snapping mode by using a special character in front of the coordinates in the **Enter a Numeric Location** dialog.

You can enter	Description	Special character
One coordinate	A distance to an indicated direction.	
Two coordinates Three coordinates	If you omit the last coordinate (z) or angle, Tekla Structures assumes that the value is 0. In drawings, Tekla Structures ignores the third coordinate.	
Cartesian coordinates	The x, y, and z coordinates of a position separated by commas. For example, `100,-50,-200`.	, (comma)
Polar coordinates	A distance, an angle on the xy plane, and an angle from the xy plane separated by angle brackets. For example, `1000<90<45`. Angles increase in the counterclockwise direction.	<
Relative coordinates	The coordinates relative to the last position picked. For example, `@1000,500` or `@500<30`.	@
Absolute coordinates	The coordinates based on the origin of the work plane. For example, `$0,0,1000`.	$
Global coordinates	The coordinates relative to the global origin and the global x and y directions. For example, `6000,12000,0`. This is useful, for example, when you have set the work plane to a part plane and want to snap to a position defined in the global coordinate system without changing the work plane to global.	!

You can enter	Description	Special character
Coordinate axis prefixes	When using direct modification, with relative and absolute coordinates, you can also use axis prefixes to allow snapping in the prefixed directions only. For example, `@z500` or `$y6000,z-500`. Axis prefixes cannot be used with global coordinates. If any of the entered coordinate values has an axis prefix, the other values need to have prefixes, too. The axis prefixes are not case-sensitive, and the prefixed values can be entered in any order.	x y z

Change the snapping mode

Tekla Structures has three snapping modes: relative, absolute, and global. Use the advanced option `XS_KEYIN_DEFAULT_MODE` to indicate the default snapping mode.

1. On the **File** menu, click **Settings** --> **Advanced options** and go to the **Modeling properties** category.
2. Set the advanced option XS_KEYIN_DEFAULT_MODE to `RELATIVE`, `ABSOLUTE`, or `GLOBAL`.
   - In the relative snapping mode, the coordinates you enter in the **Enter a Numeric Location** dialog as such without any prefix are relative to the last position picked.
   - In the absolute snapping mode, the coordinates are based on the origin of the work plane.
   - In the global snapping mode, the coordinates are based on the global origin and the global x and y directions.
3. Click **OK** to save the changes.
4. If you want to temporarily override the default snapping mode, enter a special character in front of the coordinates when entering a numeric location.

   By default, the special characters are:

   - @ for relative coordinates
   - $ for absolute coordinates

- ! for global coordinates

Alternatively, you can start numeric snapping and bring up the special characters by typing R, A, or G. R is for relative coordinates, A is for absolute coordinates, and G is for global coordinates.

NOTE If you want to change the special character for any of the three snapping modes, use the advanced options XS_KEYIN_RELATIVE_PREFIX, XS_KEYIN_ABSOLUTE_PREFIX, and XS_KEYIN_GLOBAL_PREFIX.

Snapping methods to help to set the snapping direction

In addition to the snap switches and the numeric snapping, you can use some additional methods that help you to pick the correct points when snapping.

Lock the modeling direction with the Shift key

You can temporarily lock the modeling direction with the **Shift** key when hovering over a snapping point on a line or on a point. When you release the **Shift** key, the direction locking is released.

1. Run a command that requires you to pick points.

   For example, start creating a beam.

2. To lock the direction for the first point to be picked, ensure that the **Snap to nearest points (points on line)**  snap switch is active.

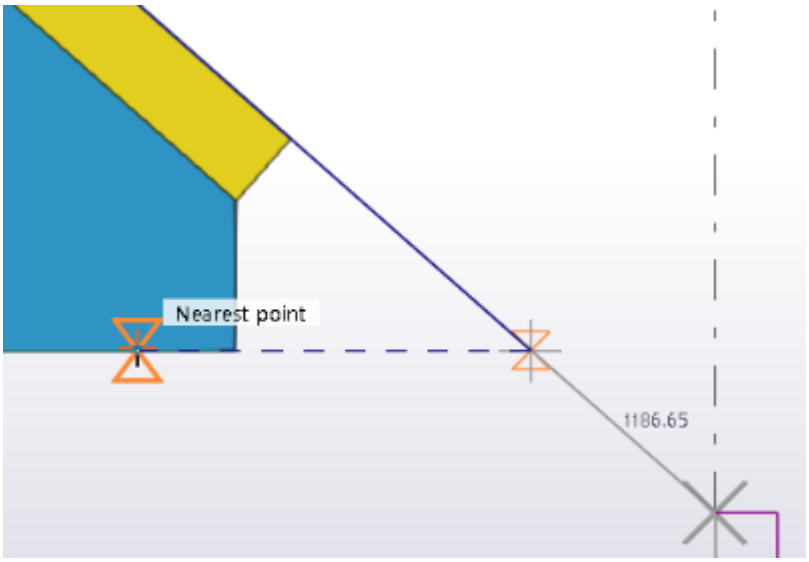
3. Hover over a snap position on a line, such as part edge line, part reference line, or grid line.

4. Hold down the **Shift** key to lock the modeling direction.

   As long as you hold down **Shift**, the direction is locked, and any point that can be snapped is projected to the locked line.

   You can snap either to a line or a point:

   - if you snap to a line (for example, using the **Snap to nearest points (points on line)** snap switch), the direction locking extends the line to the intersection of the first locked line.
   - if you snap to a point (for example, using **Snap to end points** or **Snap to mid points** snap switch), the point is projected perpendicular to the locked line.



A perpendicular symbol is displayed when the snapped point is projected perpendicularly to the locked line.

5. Pick the first point.
6. To lock the direction for the second point to be picked, ensure that a snap switch that enables snapping either to lines (such as **Snap to nearest points (points on line)**) or to points (such as **Snap to end points** or **Snap to mid points**) is active.
7. When the snap symbol is visible, hold down the **Shift** key to lock the direction.
8. Pick the second point by snapping either to a point or a line, similarly as when picking the first point.
9. Continue to pick more points, if needed.

Note that the direction locking works on only in modeling mode, not in drawings. Direction locking does not work with numeric snapping.

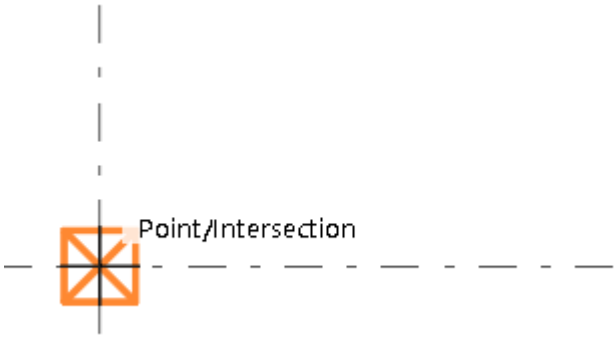
Set a temporary reference point

You can set a temporary reference point to be used as a local origin when snapping in models and drawings. Typically the **Ortho** tool and the **Snap to perpendicular points** snap switch use the reference point information.

The reference point information is automatically set to the last picked point, and shown as a black cross. When you interrupt a command, the reference point information, meaning the last picked point, is cleared. If you need to use the reference point, set the temporary reference point manually.

1. Run a command that requires you to pick points.

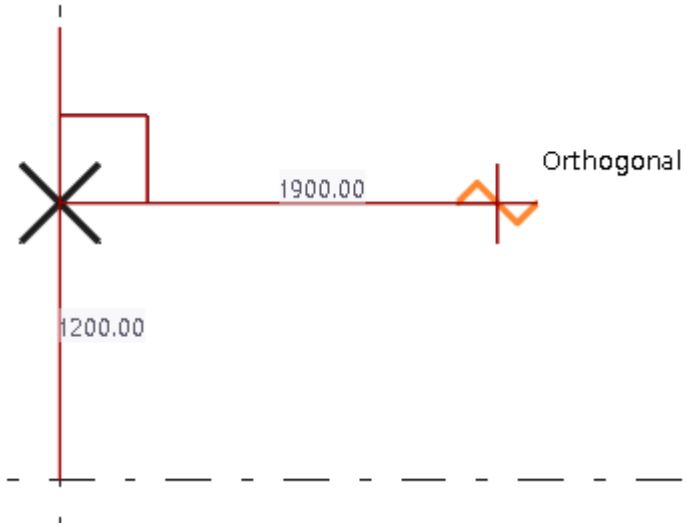
   For example, start creating a beam.
2. Pick the start point.



3. Hold down the **Ctrl** key and pick a point.

   Alternatively, right-click and select **Define temporary snap reference point** and pick a point.

   A black cross indicates that this point is now a temporary reference point. You can continue snapping from the temporary reference point.

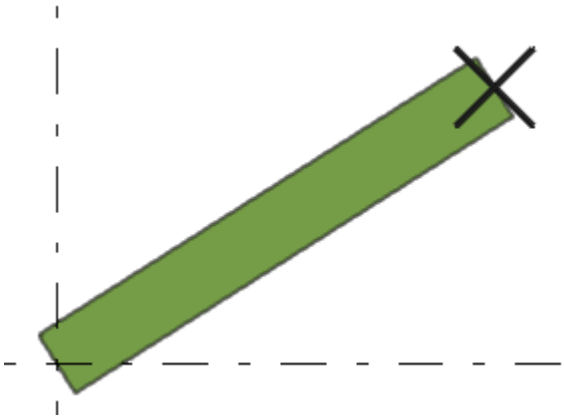


4. Repeat step 3 to create as many reference points as needed.

   If you use the **Define temporary snap reference point** command, you need to activate it for every reference point you pick.

5. Release the **Ctrl** key and pick the end point.

Tekla Structures creates the object between the start point and the end point. For example:



Lock X, Y, or Z coordinate on a line

You can lock the x, y, and z coordinates on a line. The coordinate locks work both in modeling mode and in drawing mode.

Locking the coordinates is useful when you need to determine a point to pick and the needed point does not exist on the line. When a coordinate is locked, you can only snap to points in that direction.

In drawing mode the locking is useful, for example, if you need to drag a view along x or y axis only.

1. Run a command that requires you to pick points.

   For example, start creating a beam.

2. Lock a coordinate:

   - To lock the x coordinate, press **X**.
   - To lock the y coordinate, press **Y**.
   - To lock the z coordinate, press **Z**.

   Now you can only snap to points in the chosen direction.

   Tekla Structures indicates the locked coordinate with the letters **X**, **Y**, or **Z** in the status bar at the bottom of the Tekla Structures main window.

3. To unlock the coordinate, press the same letter (**X**, **Y**, or **Z**) again.

Alternatively, you can control the coordinate locks with **Quick Launch**. Start typing `toggle coordinate lock`, and click the name of the coordinate lock in the search results list to activate the switch.

Align objects using a snap grid

A snap grid makes it easier to align objects in a model, because it allows you to snap to points only at set intervals (page 113). Use a snap grid when you pick points using the  **Snap to any position** snap switch.

1. On the **File** menu, click **Settings** --> **Snap settings**.
2. Define the grid spacing intervals in the **Spacing** boxes.

   For example, if the spacing of the x coordinate is 500, you can snap to points at intervals of 500 units in the x direction.
3. If needed, define offsets for the snap grid origin in the **Origin** boxes.
4. To activate the snap grid, select the **Active (when free snap is on)** checkbox.
5. Click **OK**.

   Now when you pick points using the  **Snap to any position** snap switch, you can only snap to points at set intervals. The snap grid itself is invisible in the model.

Snap settings

Use the **Model snap settings** dialog in **File menu** --> **Settings** --> **Snap settings** to view and modify the snap settings in the model. The **Drawing snap settings** dialog has the same options for drawings. These settings are user specific.

Option	Description
Symbol	Show or hide the snap symbols. Select the checkbox to show the snap symbols, and clear the checkbox to hide them.
Active (when free snap is on)	Select the checkbox to activate the snap grid.
Spacing	Define grid spacing intervals for the snap grid origin. For example, if the spacing of the x coordinate is 500, you are can snap to positions at intervals of 500 units in the x direction.
Origin	Define offsets for the snap grid origin.

Option	Description
Angle interval	Set the angle interval for the **Ortho** tool. This setting is used when you snap to orthogonal points (page 100). For example, if you set the interval to **10**, the **Ortho** tool will snap to angles at intervals of 10 degrees in the model or drawing.
Custom angles	Define custom angles for the **Ortho** tool. This setting is used when you snap to orthogonal points (page 100). Separate the values with empty spaces. For example, if you enter `12.5 60`, the **Ortho** tool will snap to the angles 12.5 and 60 in the model or drawing.

See also

Snap to points by using snap switches (page 89)

Snapping methods to help to set the snapping direction (page 109)

1.4 Start to create model objects

When you model in Tekla Structures, you create and work with different types of model objects. In most cases, a model object represents a building object that will exist in the real building or structure, or that will be closely related to it. A model object can also be a modeling aid that represents information that is only relevant when you are creating the model. Model objects are either created in the model or imported to it.

You can create different types of model objects, such as parts and items, bolts, reinforcement, and cuts, by using the commands on the ribbon.

Some ribbon commands have a keyboard shortcut that speed up your modeling work. You can customize the keyboard shortcuts and assign your own shortcuts for the most used commands.

Alternatively, you can use Quick Launch or the property pane to start many of the commands that create model objects.

After you have created the model objects, you can view and modify the model object properties by using the property pane.

Examples of model objects

Examples of model objects include:

- Parts (page 217) and items (page 320)
- Bolts (page 363) and welds (page 381)
- Reinforcement (page 493) and embeds
- Surface treatment (page 425) and surfaces (page 436)
- Cuts (page 412), fittings (page 397), bolt holes (page 376), and chamfers (page 420)
- Pour breaks (page 482)
- Loads

Model objects may also be created by components (page 803).

To resize and reshape model objects, use the direct modification (page 119) handles.

The following modeling aids can be used in the model:

- Grids (page 23) and grid lines (page 27)
- Construction objects (page 650) and points (page 660)
- Reference models

You can combine model objects to bigger entities by creating assemblies (page 440), cast units (page 454), and pour units (page 477).

You can manage model objects by using object groups (page 692), Organizer, and other planning tools.

Create or delete a model object

1. Run a command that creates a model object, such as a part.

   - On the ribbon: click a command. For example, click  to create a steel beam.
   - By using **Quick Launch**: enter a search term. For example, type `steel beam` to find the **Create steel beam** command.
   - In the property pane: ensure that you have nothing selected in the model. Click the **Object type list** button  and select from the list the object you want to create.

2. Pick points to place the model object in the model.

   Tekla Structures creates the model object using the current properties of the object type.

3. Follow the status bar messages to get instructions on how to proceed.
4. To create more model objects with the same properties, pick more points.

   The command runs until you end it or start another command.

5. If you want to delete a model object, select the object and press the **Delete** button.

1.5 Part reference lines and part handles

When you create a part in Tekla Structures, you position the part by picking points. These points are the part reference points. *Part reference line* is line between two reference points, and the reference line has *part handles* at the line ends. Part handles can be used to move a model object or to modify the model object's shape or size.

Part reference lines

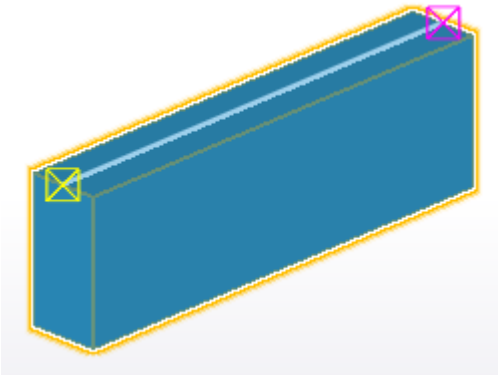
When you pick two points to position a part, the picked points form the part reference line, and part handles appear at the line ends. The position of a part is always relative to its reference points.

If you change the **Position** values of the part in the property pane, the part profile moves around the reference line; location of the reference line in the model does not move.

By default, the part reference line is invisible in the model. It may be useful to have it visible when snapping to the middle points of parts, for example.

1. To make the part reference line visible in the model, double the view to open the **View properties** dialog.
2. Click **Display...** to open the **Display** dialog.
3. On the **Advanced** tab, select the **Part reference line** checkbox.
4. Click **Modify**.

   The part reference line is displayed.



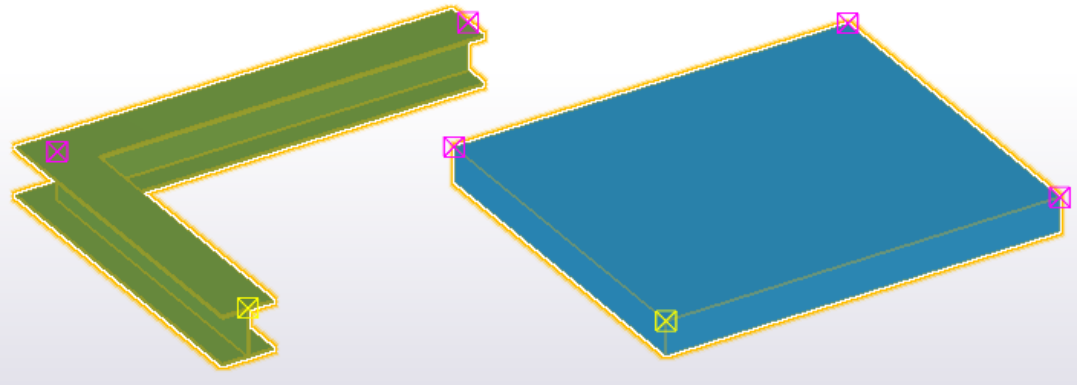
Note that when you create horizontal parts, such as beams, always pick points in the same direction, for example, from left to right, and from bottom to top (in positive x, y directions). This ensures that Tekla Structures places and

dimensions the parts in the same way in drawings, and that part marks automatically appear at the same part end.

To ensure that beam rotation is correct in drawings, set part **Rotation** to **Top** in the part properties.

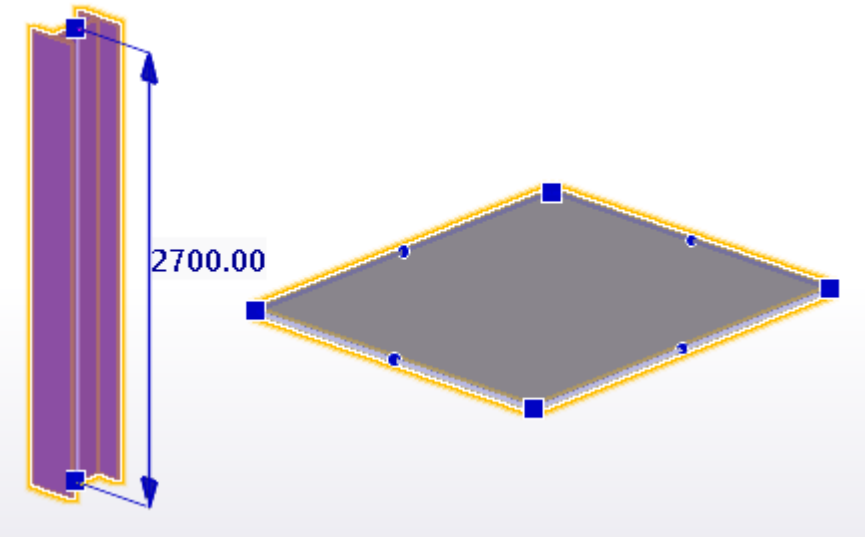
Part handles

Tekla Structures indicates the direction of a part with handles. When you select a part, Tekla Structures shows the handles. The handle at the part start point is yellow, the rest are magenta.



For information on how to select only the handles of a part, see Select objects (page 125).

If **Direct modification** is on, Tekla Structures displays the direct modification handles for the reference points, corners, segments, and segment midpoints of the selected part. These handles are blue.



Part handles and direct modification handles are not visible at the same time. If you do not want to use the direct modification handles, ensure that direct modification (page 119) is switched off.

You can use the part handles to modify the length of a part, if you do not want to use direct modification.

1. To start modifying the length of a part using part handles, ensure that the **Direct modification** switch  is **not** active.
2. Select the part.

   Tekla Structures highlights the handles of the part.
3. Click one of the handles to select it.
4. Move the handle like any other object in Tekla Structures.

   For example, right-click and select **Move**.

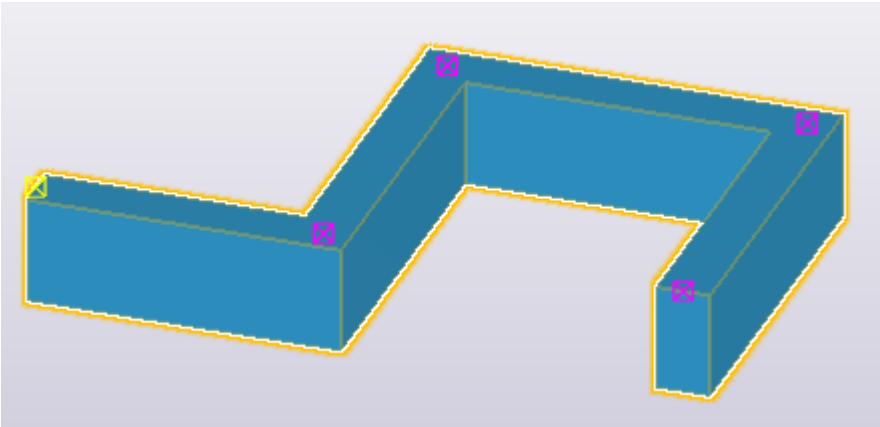
   If **Drag & drop** is active (page 158), just drag the handle to a new location.

Swap part handles

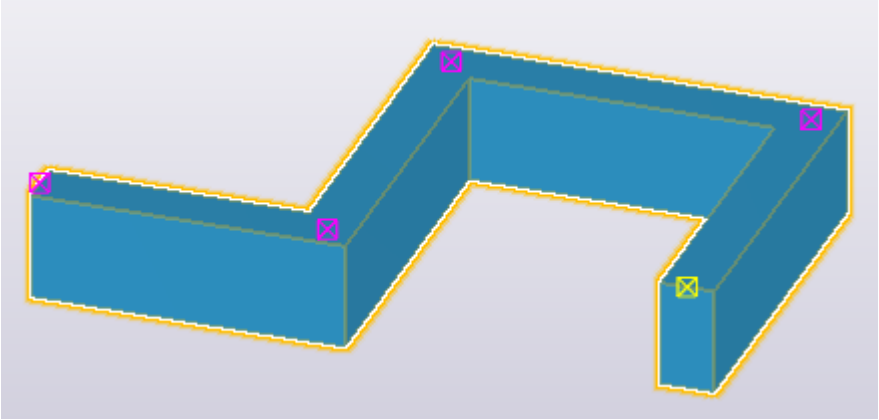
You can change the modeling direction of a part by using the **Swap handles** macro. This changes the yellow start handle to magenta, and the other way around.

1. Select the part whose modeling direction you want to change.

   Tekla Structures shows the part handles.

   
2. Go to **Quick Launch**, start typing `swap handles`, and select the **Macro.Swap handles** command from the list that appears.

   Tekla Structures changes the modeling direction of the part, and swaps the start and end handles.



1.6 Resize and reshape model objects

You can resize, reshape, and move model objects by using *direct modification handles*. When you select an object, Tekla Structures displays the direct modification handles and dimensions that are specific for that model object.

Direct modification can be used with the following object types:

- Parts
- Construction objects
- Grids and grid lines
- Line cuts and polygon cuts
- Reinforcement
- Rebar set guidelines, modifiers, and leg faces
- Pour breaks
- Custom parts
- Loads

1. Ensure that **Direct modification** is switched on.

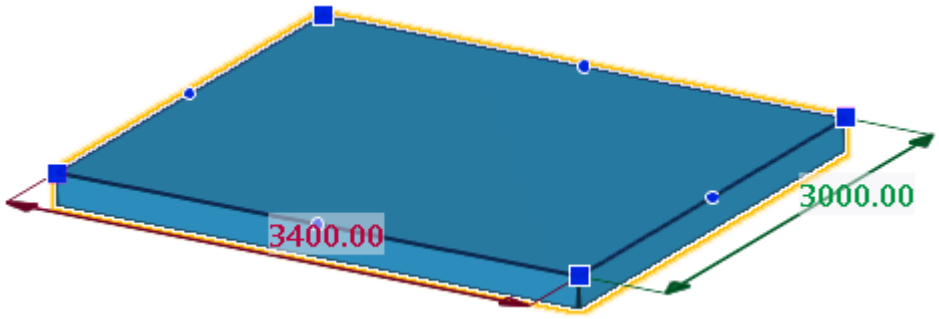
   To switch direct modification on or off, click  or press **D**.

   The **Direct modification** switch is on the **Selecting** toolbar, which is by default located at the bottom of the screen.

   Note that if **Direct modification** is switched on, Tekla Structures displays the direct modification handles, and not the yellow and magenta part handles. Direct modification handles and part handles are not visible at the same time. If you do not want to use the direct modification handles, ensure that direct modification is switched off.

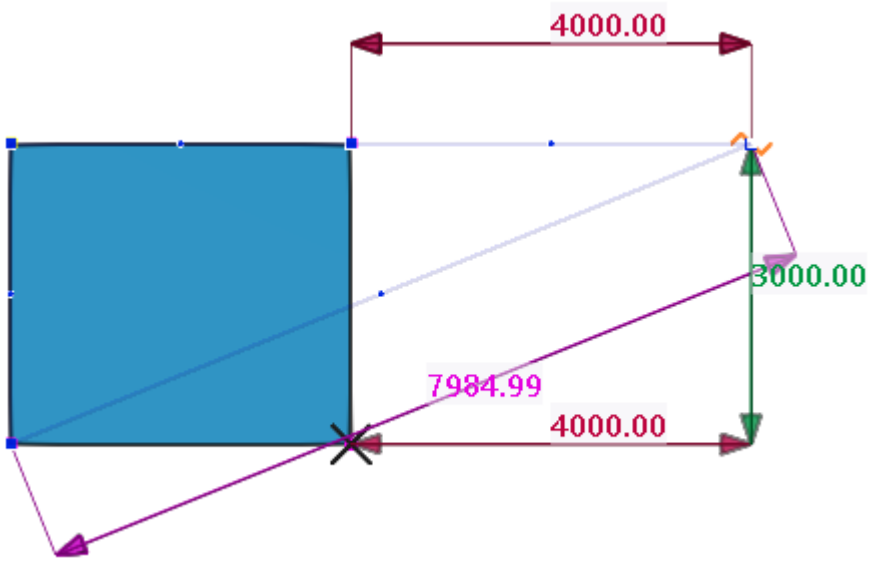
2. Click the object to select it.

Tekla Structures displays the handles that you can use to modify the object.

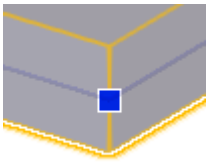


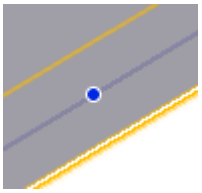
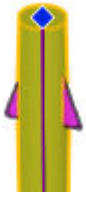
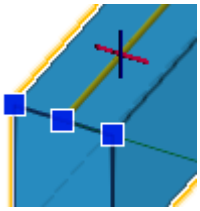
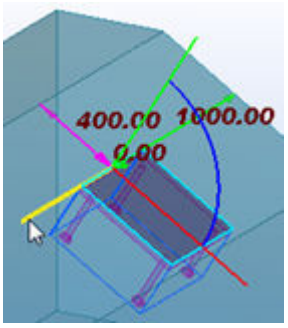
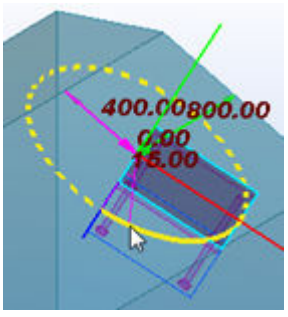
The relevant dimensions are shown when you move the mouse pointer slowly over the object's edges. The dimension colors follow the colors of the work plane coordinate axes: red in the X direction, green in the Y direction, and blue in the Z direction. Diagonal dimensions are magenta.

3. To reshape the object, drag any of the handles.



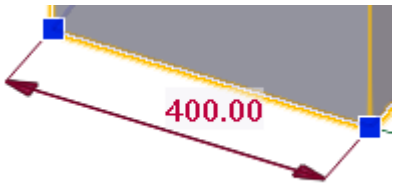
Here are some examples of direct modification handles:

Handle	Description
	Reference point handle

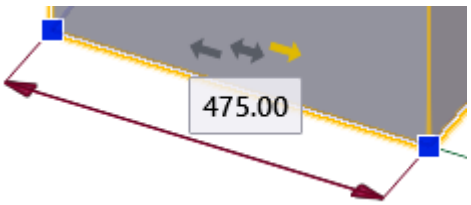
Handle	Description
	Midpoint handle
	End point handle (for reinforcing bars only)
	Plane handle
	Line handle
	Axis handle (for items (page 320) and custom parts only)
	Rotation handle (for items and custom parts only)

TIP You can use the snap switches (page 89) when dragging a handle. To temporarily disable the snap switches, hold down the **Alt** key when dragging a handle.

4. To give an exact value for a dimension, modify the dimension value.
   a. Click a dimension to select it.



   b. Type a new value.



   The yellow arrow defines the direction in which the object is extended or shortened. You can change the direction by clicking the arrows.

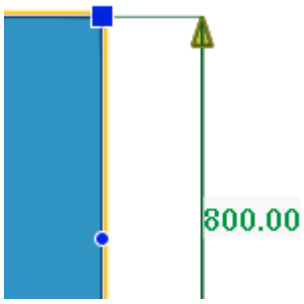
   c. Press **Enter** to confirm the new value.



5. To change a dimension only from one end, move the dimension arrowheads.

   You can either drag an arrowhead to a new location, or you can type an exact distance or coordinates.

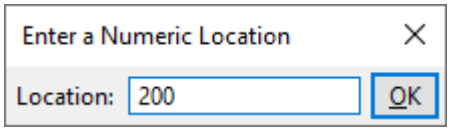
   a. Select the dimension arrowhead you want to move. For example:



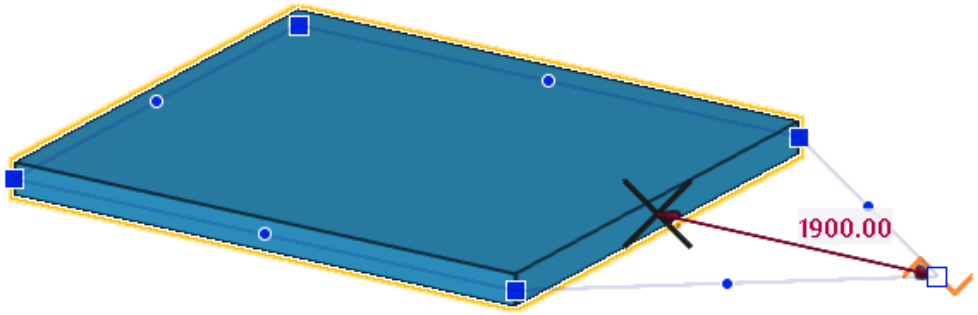
To change a dimension at both ends, select both arrowheads.

b. Type the distance or coordinates.

When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog. Click **OK** to confirm the dimension.

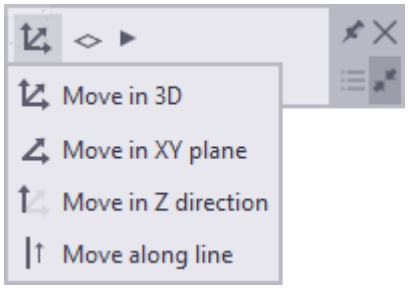
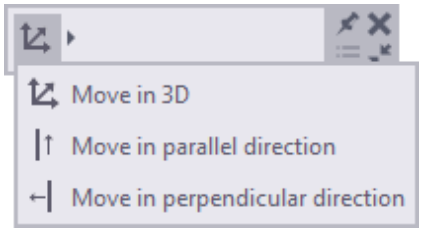
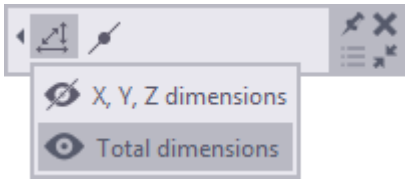


6. To add a new object corner, drag a midpoint handle . For example:



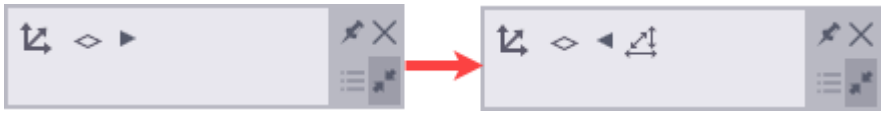
7. To display more modification options, select a handle.

A contextual toolbar appears with more options. The availability of the options depends on the object and handle you have selected.

Click this button	To do this	Location
	Move a handle to any location in the 3D space.	
	Move a handle in the XY plane only.	
	Move a handle in the Z direction only.	
	Move a handle along the reference line only.	
	Move a handle in the parallel direction only.	
	Move a handle in the perpendicular direction only.	
	Move a handle parallel to a certain plane only. Select the plane and drag the handle to a new location. This option can be useful when working with a sloped roof, for example.	
	Control the visibility of direct modification dimensions. Click the eye symbol to show or hide dimensions. • **X, Y, Z dimensions**: All orthogonal dimensions in	

Click this button	To do this	Location
	the work plane directions X, Y, and Z are displayed. • **Total dimensions**: Only the total length is displayed.	
	Show or hide midpoint handles.	
	Add a new point at the end of an object. Only available for objects that pass through multiple points, such as polybeams, panels, strip footings, and rebar set modifiers.	

NOTE Some of these options are located in an expandable section on the contextual toolbar. Click the small triangle symbol on the contextual toolbar to show or hide the options:



8. To delete a handle, select it and press **Delete**.

See also

Modify a construction object (page 657)

Modify a single grid line (page 29)

Modify a single reinforcing bar, bar group, or mesh (page 572)

Modify a rebar set (page 545)

Modify a pour break (page 487)

Custom component example: add or move a custom part in the model (page 885)

1.7 Select objects

Many Tekla Structures commands require to select objects. You can make single selections and area selections. Tekla Structures highlights the selected objects. The number of selected objects and handles is displayed in the bottom right corner of the status bar. For example:

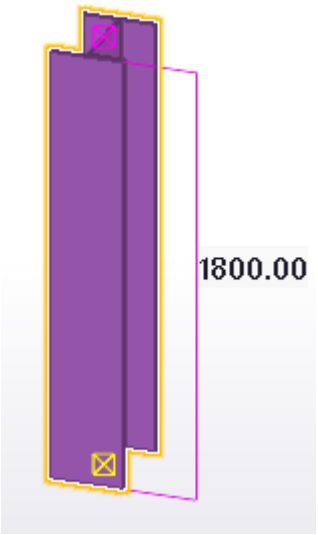
22 objects and 3 handles selected

Use the different commands and methods to select the objects. To control which type of objects can be selected, use the selecting toolbar and the selection switches.

Select single objects

1. Make sure that the correct selection switches are active.
2. Click an object to select it.

   Tekla Structures displays dimensions and dimension lines for column, beam, reinforcing bar group, and for rebar set. If you want to hide the dimensions, use the advanced options XS_DISPLAY_DIMENSIONS_WHEN_SELECTING_OBJECTS and XS_DISPLAY_DIMENSIONS_WHEN_SELECTING_REBARS.

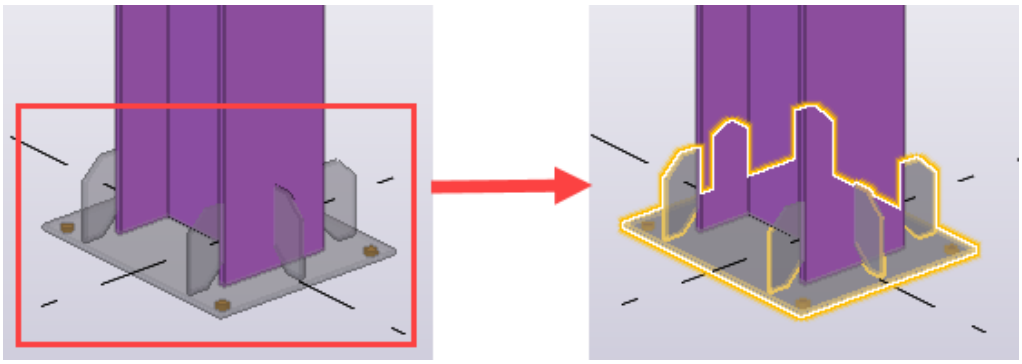


NOTE When direct modification (page 119)  is switched on, the object dimensions and dimension lines are always hidden. Only the direct modification dimensions are then shown. This makes it easier to know which dimensions can be edited.

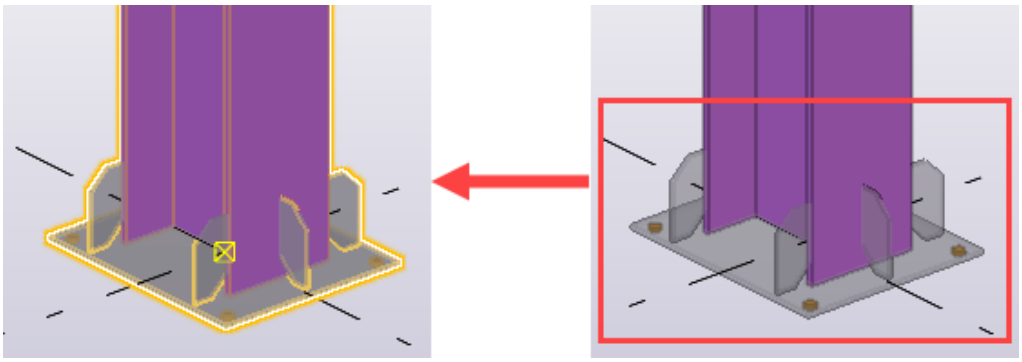
Select multiple objects using area selection

You can select multiple objects using area selection. By default, the dragging direction affects the selection of objects.

1. Make sure that the correct selection switches are active.
2. To select all objects that are completely within a rectangular area, hold down the left mouse button and drag the mouse from **left to right**.



3. To select all objects that are at least partly within a rectangular area, hold down the left mouse button and drag the mouse from **right to left**.

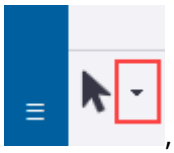


4. If you want to change how area selection works, click **File menu** --> **Settings** and select or clear the **Crossing selection** checkbox.

   By default, the option is switched off. When the option is **off**, the dragging direction affects the selection of objects. When the option is **on**, all objects that fall at least partially inside the rectangular area are selected, regardless of the dragging direction.

Select all objects

To select all objects at once, do one of the following:

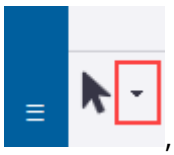
- On the ribbon, click the small down arrow next to the arrow button , and then click **Select all objects**.
- Press **Ctrl+A**.

Note that

- the **Select all objects** command respects the selection filter so that only the objects that are matching the current selection filter are selected.
- the **Select all objects** command selects also those objects that are hidden by a view filter, work area, or that have been hidden by using the **Hide** command, if they match the selection filter.

Select previous objects

Sometimes you need to select again the same objects that you selected previously but have then deselected. To select previously selected objects, do one of the following:

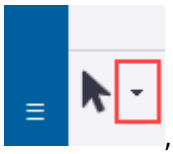
- On the ribbon, click the small down arrow next to the arrow button , and then click **Select previous objects**.
- Press **Alt+P**.

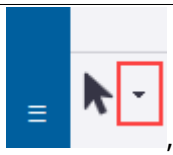
Select objects by identifier

If you know the GUID (globally unique identifier), or the ID of an object, or the IFC GUID of a reference object, you can use the **Select by identifier** command to locate the objects in a model or in a drawing.

You can often find information about the object GUID or ID, for example, in reports and log files. By using the **Select by identifier** command you can quickly find the objects in a model or in a drawing, instead of defining a view filter or a selection filter with the specific GUID or ID. You can use IFC GUIDs to find IFC reference objects. This is useful if you need to track updates and changes in IFC reference models.

Additionally, you can use the **Select by identifier** command to inquire the GUIDs of selected objects, instead of using the traditional inquiry (page 721).

To	Do this
Find objects based on the object GUID, ID, or IFC GUID identifier	1. Do one of the following: • In the modeling mode: on the ribbon, click the small down arrow next to the arrow button , and then click **Select by identifier**. • In the drawing mode: in **Quick Launch**, type **Select by identifier**. The **Select by identifier** dialog opens. 2. Copy the object identifier, for example from a log file, to the dialog. You can enter multiple identifiers in the dialog. Either enter each identifier on its own row, or separate them with semicolon `;`. 3. To define the search, select the needed check boxes. • **Reference objects**: Tekla Structures selects IFC objects based on their GUID or IFC GUID. • **Keep selection**: Tekla Structures keeps the currently selected object and appends it with new selection. • **Zoom to selected**: Tekla Structures selects the object and zooms to it. 4. Click **Select**. Tekla Structures selects the objects based on the GUID in the model or in the drawing. If there are identifiers that are not found in the model or in the drawing, they are listed in the status bar as `identifier?`.
Find a model object in a drawing	You can select an object in a model, get its identifier, and then find it in a drawing based on the identifier. 1. In the modeling mode: on the ribbon, click the small down arrow next to the arrow

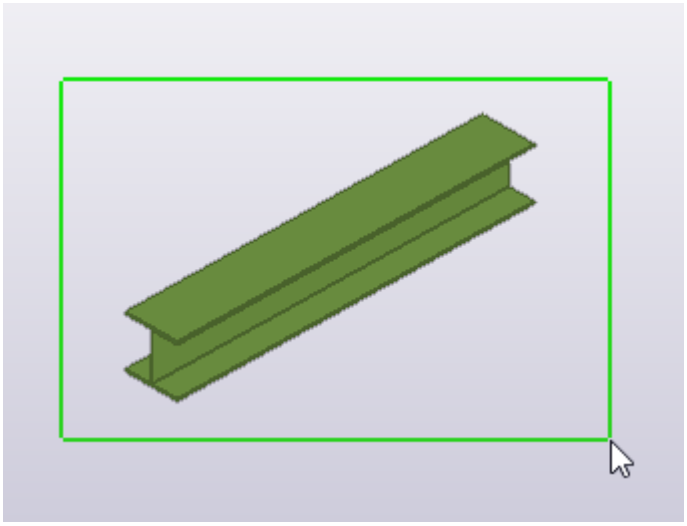
To	Do this
	button , and then click **Select by identifier**. The **Select by identifier** dialog opens. 2. Select an object or objects in the model. 3. Click **Get**. The **Select by identifier** dialog lists the identifiers of the selected objects. If you want to get IFC GUIDs, make sure that the **Reference objects** checkbox is selected. 4. Keep the dialog open. 5. Open a drawing. 6. In the drawing mode, click **Select** to find the objects in the drawing. You can then continue working with the found objects.
Find a drawing object in a model	You can select an object in a drawing, get its identifier, and then find it in a model based on the identifier. 1. In the drawing mode: in **Quick Launch**, type **Select by identifier**. The **Select by identifier** dialog opens. 2. Select an object or objects in the drawing. 3. Click **Get**. The **Select by identifier** dialog lists the identifiers of the selected objects. 4. Keep the dialog open. 5. Close the drawing. 6. In the modeling mode, click **Select** to find the objects in the model. You can then continue working with the found objects.

Select handles

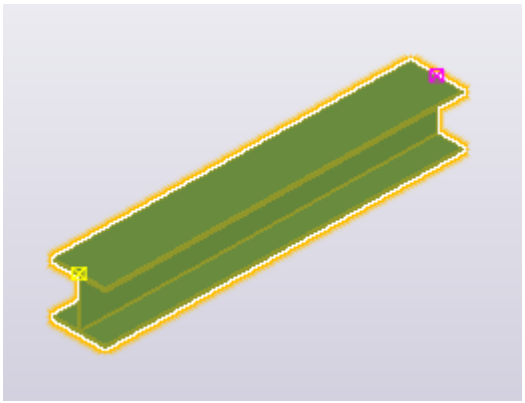
Sometimes you need to select only the handles of a part, for example when moving the part.

Before you start, make sure that **Crossing selection** is switched off, and that the **Direct modification** switch  is not active.

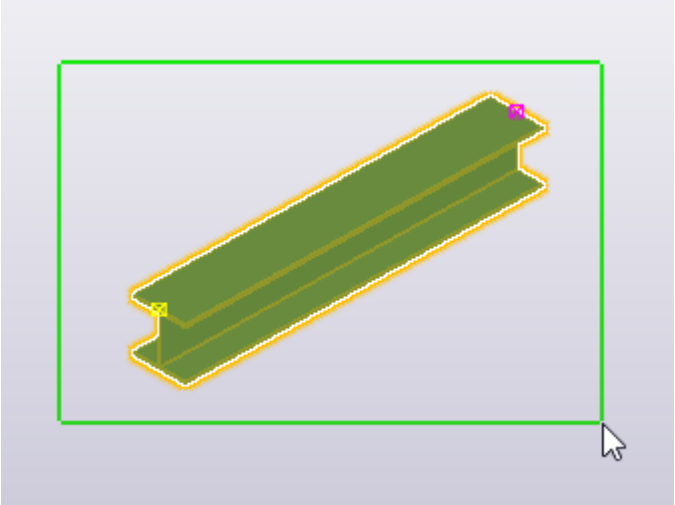
1. On the **File** menu, click **Settings** and make sure that **Crossing selection** is switched off. If the **Crossing selection** is not switched off, selecting the handles with the **Alt** key does not work.
2. Make sure that the correct selection switches are active.
3. Hold down the left mouse button and drag the mouse from left to right to include the entire part.



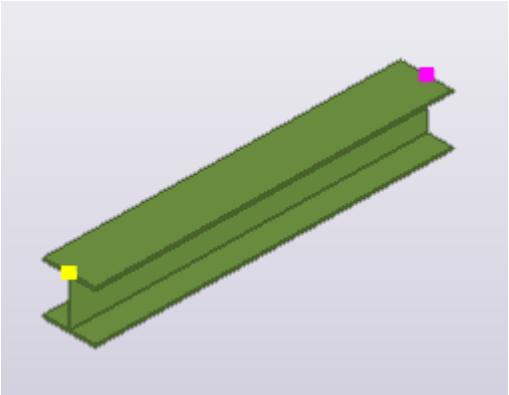
The part becomes selected:



4. Hold down the **Alt** key and drag the mouse from left to right again.



Now only the part handles are selected:



NOTE When direct modification (page 119)  is switched on, Tekla Structures also displays the direct modification handles for reference points, corners, segments, and segment midpoints of the selected part. These handles are blue.

Modify the selection

You can add objects to the current selection, or remove objects from the selection.

1. To add objects to the current selection, press the **Shift** key and select more objects.
2. To switch the selection of an object on or off, press the **Ctrl** key during the selection.

   Tekla Structures deselects the objects that were already selected and selects those that were previously not selected.

3. To clear the selection of all objects and handles, click somewhere else.

   For example, click on the empty background of the current view.

Select assemblies, cast units, and nested objects

You can select either assemblies or cast units, or single objects in nested assemblies or nested components.

Select assemblies and cast units

Use the **Select assemblies** selection switch to select assemblies and cast units (page 454).

1. Ensure that the  **Select assemblies** selection switch is active.
2. Select a part.

   Tekla Structures selects the entire cast unit or assembly that contains the selected part.

Alternatively, to select an entire assembly or cast unit, hold down the **Alt** key and select one of the parts in the assembly or cast unit. Tekla Structures selects the entire cast unit or assembly that contains the selected part.

The number of objects in the assembly or in the cast unit is shown at the bottom right corner of the status bar.

Select nested objects

You can select nested assemblies and components. The active selection switch defines on which level you start and toward which direction you move in the component or assembly hierarchy. The status bar shows the steps you take in the hierarchy.

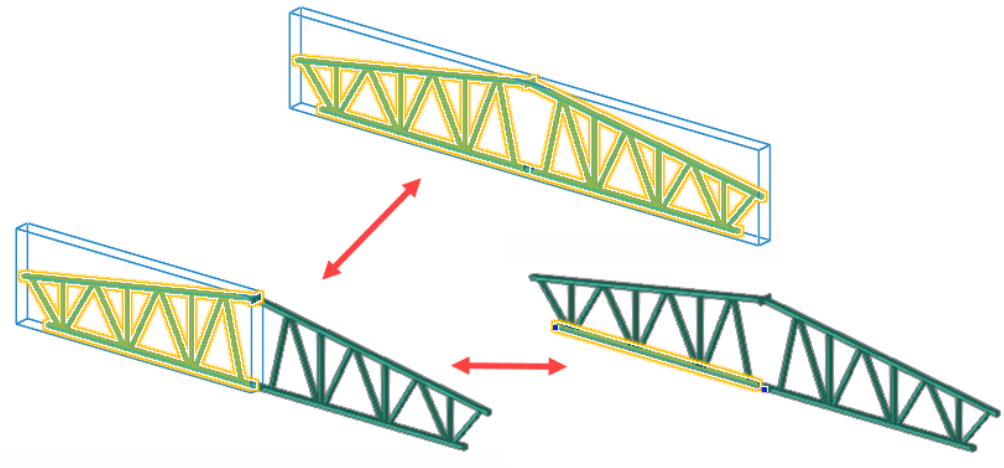
1. Ensure that the correct selection switch is active.

   - : to start from the assemblies on the highest level, move to their sub-assemblies, and finally select single parts, bolts, and so on
   - : to start from single objects and move to higher levels of nested assemblies
   - : to start from the components on the highest level, move to their sub-components, and finally select single parts, bolts, and so on

- : to start from single objects and move to higher levels of nested components

2. Place the mouse pointer on any part in the assembly or in the component.
3. Hold down the **Shift** key.
4. Scroll with the mouse wheel.

   The blue highlighting box indicates the assembly or the component that you can select.



Select reference models, reference model objects and assemblies

You can select either entire reference models, or single objects and assemblies that are part of a reference model. The use of selection switches differs in each case.

Select an entire reference model

1. Activate the **Select reference models** selection switch.
2. Activate the **Select components** selection switch.
3. Select the reference model.

Select a reference model object

1. Activate the **Select reference models** selection switch.

2. Activate the  **Select objects in components** selection switch.
3. Select the desired object in the reference model.

Select a reference model assembly

1. Activate the  **Select reference models** selection switch.
2. Activate the  **Select assemblies** selection switch.
3. Select the desired assembly in the reference model.

Tips for selecting objects

Here are listed some tips that can help when you are selecting objects.

Switch rollover highlight on or off

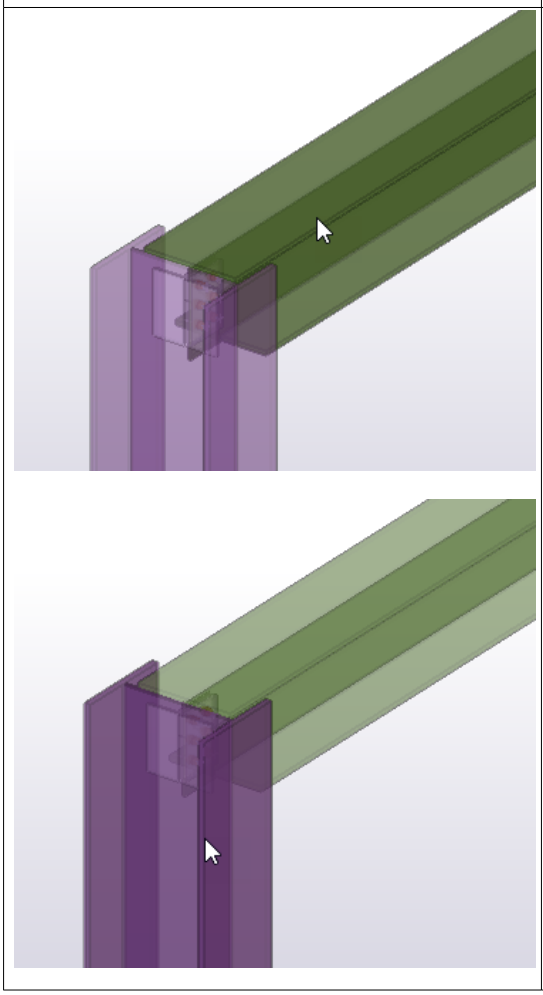
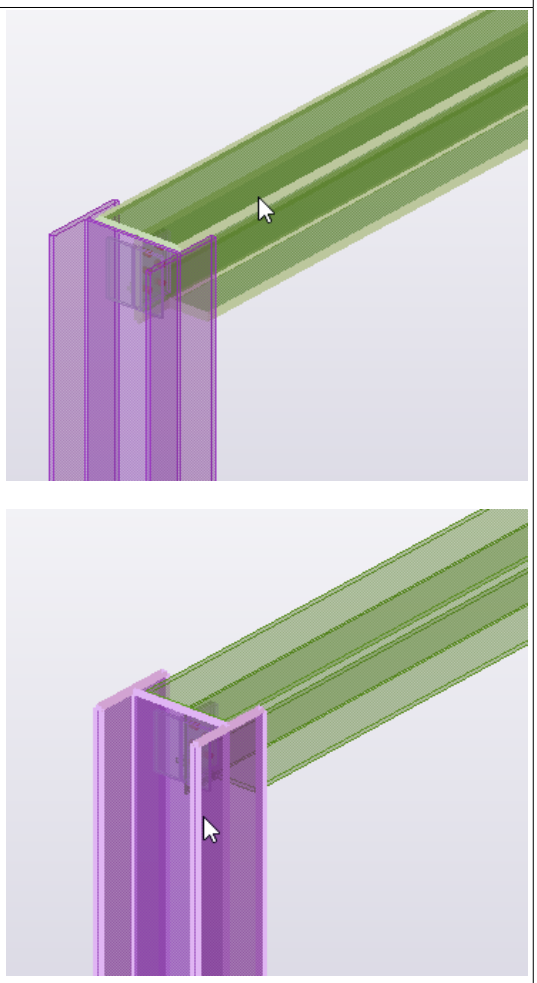
By default, Tekla Structures highlights the objects that you can select. You can switch the highlighting on or off.

To switch the rollover highlight on or off, on the **File** menu, click **Settings** and select or clear the **Rollover highlight** checkbox. Alternatively, you can press **H**.

Depending on the rendering engine (page 76) you are using, OpenGL or DirectX, Tekla Structures highlights the objects differently when rollover highlight is on.

In the example below, the rendering of parts (page 671) is set to **Parts shaded wireframe**.

DirectX	OpenGL
Tekla Structures highlights the objects by showing them with darker object color. For example:	Tekla Structures highlights the objects by showing them with light edge line color. For example:

DirectX	OpenGL
	

Select on right-click

You can change the settings so that you can select objects also with the right mouse button.

1. On the **File** menu, click **Settings** and select the following checkboxes:
   - **Select on right-click**
   - **Rollover highlight**
2. Right-click an object to select it. Tekla Structures highlights the object and shows the related shortcut menu.

If you cannot select objects

If you cannot select desired objects in the model, check the selection switches and the filter settings.

- Check that you have switched on all the needed selection switches.

- If you still cannot select the objects, check the selection filter settings. You can select a different filter or modify the current filter.

Interrupt object selection

You can have Tekla Structures interrupt the object selection process if the selection takes over a defined period of time. For example, if you are working on a large model and you accidentally select all or part of the model, you can interrupt the selection if it takes over 5000 milliseconds (5 seconds) to complete.

1. Define the time after which Tekla Structures asks if you want to interrupt object selection.
   a. On the **File** menu, click **Settings** --> **Advanced options** and go to the **Modeling properties** category.
   b. Modify the advanced option XS_OBJECT_SELECTION_CONFIRMATION.

      The default value is 5000 milliseconds.
   c. Click **OK**.
2. Select (page 125) all or part of the model.
3. When Tekla Structures asks if you want to interrupt object selection, click **Cancel**.

1.8 Copy objects in models and in drawings

In Tekla Structures, each time you copy model objects, such as parts, assemblies, or reinforcement, or drawing objects, such as independent annotation objects and sketch objects, they are automatically copied to the clipboard. This allows you to easily paste the copied objects into the current model or any other model, or into the current drawing or any other drawing in the same or another model.

Copying model objects using the clipboard

Model objects are copied to the clipboard when the following commands are used on the **Edit** tab:

- **Copy**
- **Copy** --> **To another object**
- **Copy** --> **To another plane**
- **Copy** --> **All content to another object**

On the **Edit** tab, use the **Copy** --> **Copy continuously** switch and the **Edit** --> **Paste** command in conjunction with the above copying commands to copy and paste model objects.

Copying drawing objects using the clipboard

Drawing objects are copied to the clipboard when you use the **Copy** command on the **Drawing** tab.

On the **Drawing** tab, use the **Copy** --> **Copy continuously** switch and the **Drawing** --> **Paste** command in conjunction with the **Copy** command to copy and paste drawing objects.

Controlling clipboard copying

- When the **Copy continuously** checkbox is selected, the selected objects are copied to the clipboard, and the copy command remains active. You can insert the copied objects repeatedly into the model, or into the drawing, or use the **Paste** command to insert the copied objects from the clipboard.
- When the **Copy continuously** checkbox is cleared, the selected objects are copied to the clipboard and the copy command is interrupted. Use the **Paste** command to insert the copied objects into the same model or into another model, or into the current drawing or any other drawing in the same or another model.

The last copied model objects always remain on the clipboard, and you can later use the **Paste** command to insert the objects from the clipboard.

Keyboard shortcuts

Use the keyboard shortcuts **Ctrl+C** and **Ctrl+V** for quick copying and pasting from the clipboard.

Notes and limitations

- The model and drawing objects that you want to copy using the clipboard to another model have to be saved in the source model before starting the **Copy** command.
- If the source model objects or drawing objects that have been copied to the clipboard are modified in the model or in the drawing, the changes will also be updated in the clipboard. Therefore, when you start the **Paste** command, you will not get the object you copied originally, but the latest version of the object.
- Copying to and pasting from the clipboard only work with specific drawing object types. The supported objects are:
  - 2D library details
  - lines
  - polylines
  - rectangles
  - polygons
  - circles

    - arcs
    - insulations
    - clouds
    - symbols
    - texts
    - embedded text files
    - embedded DWG and DXF files
    - embedded images
    - revision marks
    - section marks
    - detail marks
    - weld marks
- You cannot use clipboard copying for the following drawing objects:
    - associative annotations: associative notes, part marks, bolt marks, rebar marks, and so on
    - level marks
    - drawing views
    - templates
    - drawing layouts
- Copying of symbols and embedded images, DWGs/DXFs, and text files in drawings: Currently, Tekla Structures attempts to copy a file into a destination folder only if it is originally located in the model folder. If the file is located in any other folder, including shared environment or firm/project folders, it will not be copied. Objects located in shared folders are intended to be shared, not copied.
- When copying sketch objects between drawing views and drawings, consider the XS_SCALE_COPIED_OR_MOVED_OBJECTS_IN_DRAWINGS advanced option. This advanced option automatically adjusts the dimensions of sketch objects to match the scale of the target view.

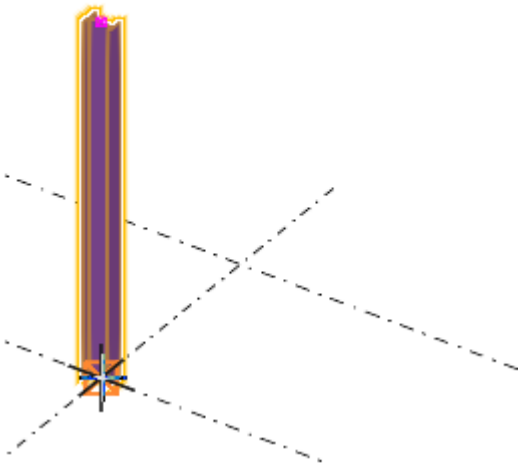
Copy model objects

Copy the selected model objects to another location in the same or another model by picking the origin and one or more destination points.

Before you start copying, use the **Edit** --> **Copy** --> **Copy continuously** checkbox to enable or disable the continuous copying of model objects.

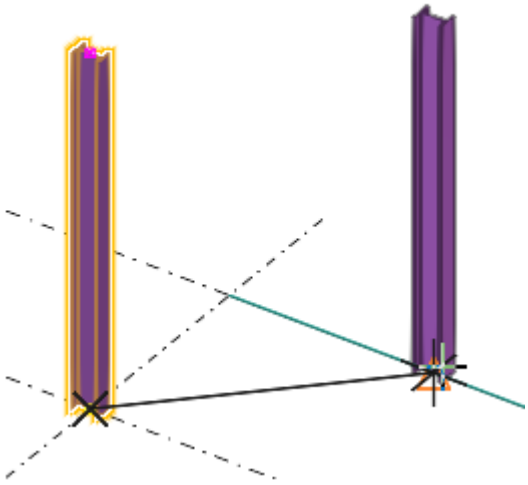
Use the keyboard shortcuts **Ctrl+C** and **Ctrl+V** for quick copying and pasting.

1. Select the objects you want to copy.
2. On the **Edit** tab, click **Copy**.
3. Pick the origin for copying.



The selected objects are copied to the clipboard.

4. If **Copy continuously** is enabled:
   a. Pick one or more destination points.



Tekla Structures displays a rubber band line between the first picked point and the cursor position. This is a preview of the position where the objects will be copied. Move the cursor to see how the preview changes.

Note that Tekla Structures always displays the preview in the position where the objects will be copied, not in the position where the cursor is when you pick the destination point.

If you want to limit the number of objects shown in the preview, use the advanced option `XS_PREVIEW_LIMIT`. The default value is 1000. When the value is 0, the preview is off.

The objects are copied immediately. The **Copy** command remains active.

b. If you want to undo the latest copy operation, click the **Undo** button in the top left corner of the Tekla Structures main window.

The **Copy** command remains active.

c. To stop copying, press **Esc**.

d. The last copied objects remain on the clipboard. If you want to insert them later into the model, click **Paste** and pick one or more destination points.

5. If **Copy continuously** is disabled:

a. Navigate to your destination location in the same model or in another model.

b. Click **Paste**.

c. Pick one or more destination points.

The objects are pasted from the clipboard.

d. The last copied objects remain on the clipboard. If you want to insert them later into the model, click **Paste** and pick one or more destination points.

Copy drawing objects

Copy the selected drawing objects to another location in the current drawing or in any other drawing in the same or another model by picking the origin and one or more destination points.

Before you start copying, use the **Drawing** --> **Copy** --> **Copy continuously** checkbox to enable or disable the continuous copying of drawing objects.

Use the keyboard shortcuts **Ctrl+C** and **Ctrl+V** for quick copying and pasting.

1. In an open drawing, select the drawing objects you want to copy.
2. On the **Drawing** tab, click **Copy**.
3. Pick the origin for copying.

The selected objects are copied to the clipboard.

4. If **Copy continuously** is enabled:

a. Pick one or more destination points.

   The objects are copied immediately. The **Copy** command remains active.

b. If you want to undo the latest copy operation, click the **Undo** button in the top left corner of the Tekla Structures main window.

   The **Copy** command remains active.

c. To stop copying, press **Esc**.

d. The last copied objects remain on the clipboard. If you want to insert them later in the drawing, click **Paste** and pick one or more destination points.

5. If **Copy continuously** is disabled:

   a. Navigate to your destination location in the same drawing or in another drawing in the same or another model.

   b. Click **Paste**.

   c. Pick one or more destination points.

      The objects are pasted from the clipboard.

   d. The last copied objects remain on the clipboard. If you want to insert them later in a drawing, click **Paste** and pick one or more destination points.

Copy objects in a model to another object

In the model, you can copy reinforcement, surface treatments, and single-part welds between similar objects and make them adaptive to the object they are copied to. This is useful, for example, when you detail previously modeled parts. The objects that you can copy between can have different dimensions, length, and rotation. If you copy objects from an assembly or cast unit, Tekla Structures copies the assembly structure as well, if possible. For example, sub-assemblies are copied as sub-assemblies if a parent object is found. Additionally, you can copy custom components from one model to another.

TIP Instead of copying objects from an assembly or cast unit to other identical assemblies or cast units, you can use the **Batch editor** tool. **Batch editor** detects matching objects in the target assemblies or cast units, and edits the matching objects by modifying their geometry and properties.

If you copy reinforcement or surface treatments and want them to adapt to the part they are copied to, note the limitations:

- The reinforcement or surface treatment handles must be in part corners.

- The parts between which you copy must have the same number of cross-section corners.
- Circular parts must have the same cross-section dimensions.

Before you start copying, use the **Edit** --> **Copy** --> **Copy continuously** checkbox to enable or disable the continuous copying of model objects.

1. Select the objects you want to copy.
2. On the **Edit** tab, click **Copy** --> **To another object**.
3. Select the object to copy from (source object).

   The selected objects are copied to the clipboard.
4. If **Copy continuously** is enabled:
   a. Select the objects to copy to (destination objects).

      The objects are copied immediately. The **To another object** command remains active.
   b. To stop copying, press **Esc**.
   c. The last copied objects remain on the clipboard. If you want to insert them later into the model, click **Paste** and pick one or more destination points.
5. If **Copy continuously** is disabled:
   a. Navigate to your destination location in the same model or in another model.
   b. Click **Paste**.
   c. Select the objects to copy to (destination objects).

      The objects are pasted from the clipboard.
   d. The last copied objects remain on the clipboard. If you want to insert them later into the model, click **Paste** and pick one or more destination objects.

Copy all object content in a model to another object

In the model, you can copy objects from an assembly or cast unit to other similar assemblies or cast units without individually selecting each object to copy. This is useful, for example, when you have detailed an assembly and want to copy all details to another similar assembly.

TIP Instead of copying objects from an assembly or cast unit to other identical assemblies or cast units, you can use the **Batch editor** tool. **Batch editor** detects matching objects in the target assemblies or cast

units and edits the matching objects by modifying their geometry and properties.

Before you start copying, use the **Edit** --> **Copy** --> **Copy continuously** checkbox to enable or disable the continuous copying of model objects.

1. Ensure that the  **Select assemblies** selection switch is active.
2. Select the assembly or cast unit to copy from (source object).
3. On the **Edit** tab, click **Copy** --> **All content to another object**.

   The selected objects are copied to the clipboard.
4. If **Copy continuously** is enabled:
   a. Select the assemblies or cast units to copy to (destination objects).

      The objects are copied immediately. The **All content to another object** command remains active.
   b. To stop copying, press **Esc**.
   c. The last copied objects remain on the clipboard. If you want to insert them later into the model, click **Paste** and pick one or more destination points.
5. If **Copy continuously** is disabled:
   a. Navigate to your destination location in the same model or in another model.
   b. Click **Paste**.
   c. Select the assemblies or cast units to copy to (destination objects).

      The objects are pasted from the clipboard.
   d. The last copied objects remain on the clipboard. If you want to insert them later into the model, click **Paste** and pick one or more destination objects.

As a result, Tekla Structures copies the following objects:

- Secondary parts
- Reinforcement, bolts, and welds
- Cuts, fittings, and edge chamfers
- Sub-assemblies
- Components

NOTE Tekla Structures does not copy pour breaks or secondary parts created by a component that has also created the assembly main part. If some of the objects to be copied already exist in the assembly or cast unit to copy to, Tekla Structures may create duplicate

objects. Tekla Structures warns you about duplicate secondary parts, reinforcement, and sub-assemblies, but not about duplicate bolts, welds, cuts, or components.

Copy objects in a model to another plane

In the model, you can copy objects from the first plane you specify to the second, third, and so on, plane you specify. The position of the copied objects relative to the second, third, and so on, plane remains the same as the position of the original objects relative to the first plane.

Before you start copying, use the **Edit** --> **Copy** --> **Copy continuously** checkbox to enable or disable the continuous copying of model objects.

1. Select the objects you want to copy.
2. On the **Edit** tab, click **Copy** --> **To another plane**.

   The selected objects are copied to the clipboard.
3. If **Copy continuously** is enabled:
   a. Pick the point of origin of the first plane.
   b. Pick a point on the first plane in the positive x direction.
   c. Pick a point on the first plane in the positive y direction.

      The objects are copied immediately. The **To another plane** command remains active.
   d. Repeat the above steps for all destination planes.
   e. To stop copying, press **Esc**.
   f. The last copied objects remain on the clipboard. If you want to insert them later into the model, click **Paste** and pick one or more destination points.
4. If **Copy continuously** is disabled:
   a. Navigate to your destination location in the same model or in another model.
   b. Click **Paste**.
   c. Pick the point of origin of the first plane.
   d. Pick a point on the first plane in the positive x direction.
   e. Pick a point on the first plane in the positive y direction.

      The objects are pasted from the clipboard.
   f. Repeat the above steps for all destination planes.

g. The last copied objects remain on the clipboard. If you want to insert them later into the model, click **Paste** and pick one or more destination objects.

Additional methods for copying objects in models and drawings

You can copy objects in Tekla Structures in a number of different ways. When you copy an object, Tekla Structures copies all objects connected to it, including the components.

If you do not wish to use copying commands that copy the selected objects to clipboard (page 137), there are additional copying methods available.

Copy objects in a model or in a drawing by specifying a distance from origin

You can copy objects to a new position in the model or drawing by specifying a distance from the origin. Use the **Enter a Numeric Location** dialog to specify the distance.

1. Select the objects you want to copy.
2. Run the **Copy** command:
   - In the model, on the **Edit** tab, click **Copy**.
   - In the drawing, on the **Drawing** tab, click **Copy**.
3. Pick the origin for copying.
4. Move the cursor in the direction you want to copy the objects, but do not pick the point.
5. Type the distance.

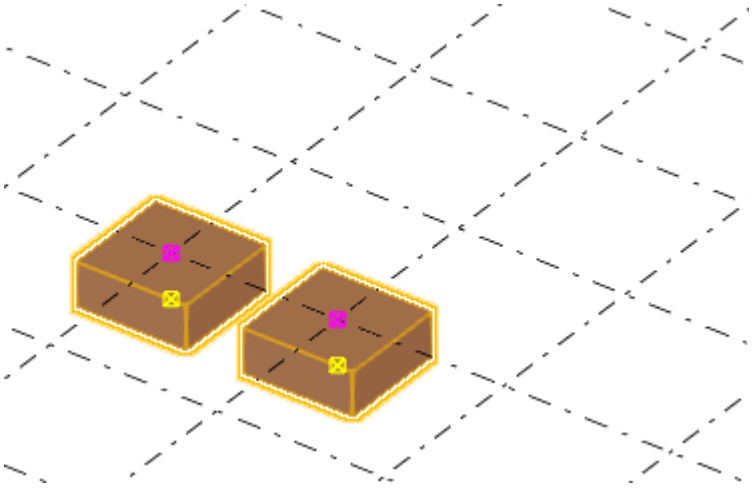
   When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog automatically.
6. Click **OK**.

Copy objects in a model or in a drawing using drag-and-drop

You can copy objects by using drag-and-drop.

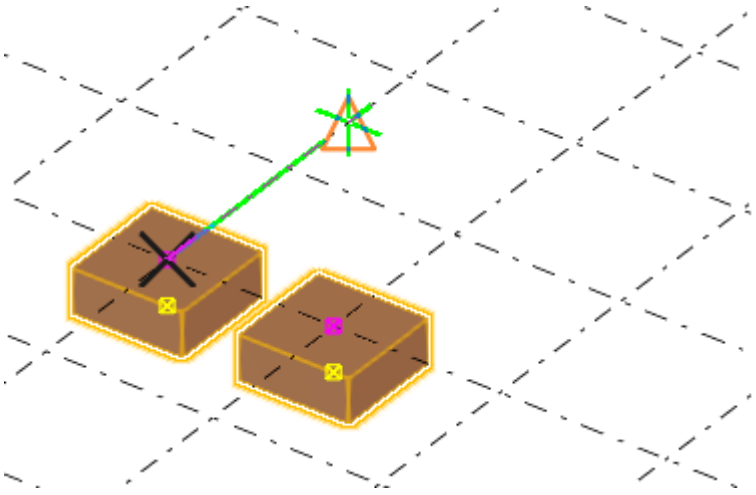
1. Activate drag-and-drop.
   - In the model, on the **File** menu, click **Settings** and select the **Drag & drop** checkbox.
   - In the drawing, on the **File** menu, click **Settings** and select the **Drawing drag & drop** checkbox.
2. Select the objects you want to copy.

In drawings, you can copy annotations, sketch objects, and grid labels by dragging. To copy grid labels, first select the grid label and then either activate the  **Select grid line** selection switch or select the grid label handle.

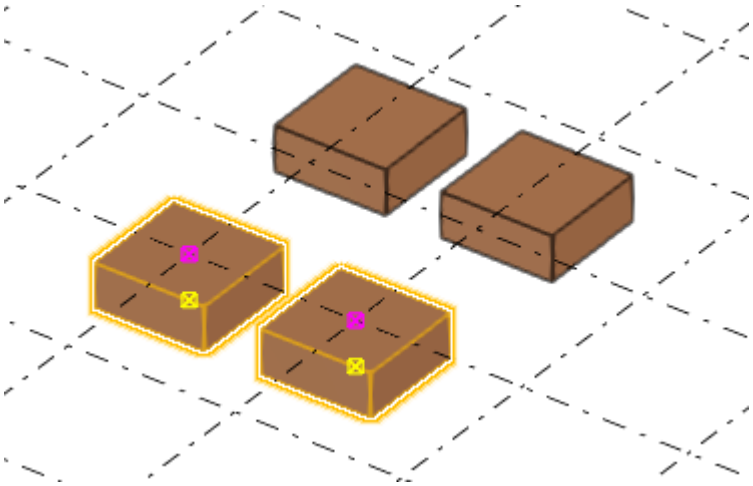


3. Hold down the **Ctrl** key and drag the objects to a new location.

   In a model, the point you start dragging from (center, corner, or middle point) affects the object's alignment in the new location.



   Tekla Structures copies the objects:



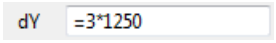
Copy objects linearly in a model or in a drawing

In the model or drawing, you can create multiple copies of an object in the same linear direction.

1. Select the objects you want to copy.
2. Run the **Linear** command:
   - In the model, on the **Edit** tab, click **Copy** --> **Linear**.
   - In the drawing, on the **Drawing** tab, click **Copy** --> **Linear**.

   The **Copy - Linear** dialog opens.
3. Pick two points, or enter the coordinates in the **dX**, **dY**, and **dZ** boxes.

   You can also use a formula to calculate the x, y, and z displacements. For example:

   In drawings, **dZ** is not defined.
4. Enter the number of copies.
5. Click **Copy**.
6. To stop copying, press **Esc**.

TIP If the dialog is open but the command is no longer active, click the **Pick...** button to re-activate the command.

Copy objects from another model

An alternative method for copying model objects from another model is to use phase numbers. Note that Tekla Structures copies secondary parts from

the model only if they belong to the same phase as their main part. This also applies to component objects.

Note that you can use the **Copy**, **To another object**,**To another plane**, or **All content to another object** commands to copy model objects from model to another. For more information, see Copy model objects and drawing objects using the clipboard (page 137).

1. On the **Edit** tab, click **Copy** --> **From another model**.

   The **Copy from Model** dialog opens.

2. In the **Model directories** list, select the model to copy from.

   This is the source model. Note that the target model must have been created using the same or newer version of Tekla Structures as the source model. You cannot copy from a newer version to an older version.

3. In the **Phase numbers** box, enter the numbers of the phases from which to copy objects, separated by spaces.

   For example, `2 7`.

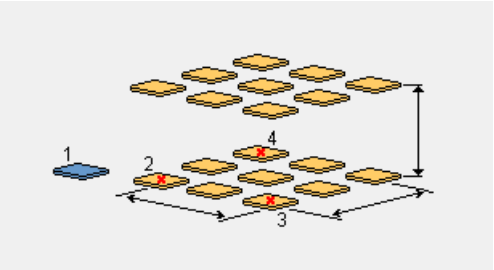
4. Click **Copy**.

5. Close the dialog.

NOTE If you copy pour breaks from another model, the copied pour breaks automatically adapt to the target model. Always check that the copied pour breaks have adapted correctly.

Copy objects using linear array tool

Use **Linear array tool** to copy selected objects linearly along multiple directions at defined intervals or spacing. Tekla Structures does not check for duplicates when you copy objects using this method.



How to use Linear array tool

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for **Linear array tool**, and then double-click to open it.
3. Select the **Copy method**. The options are:

- **Selected objects only**

  This is the default. Only the selected objects are copied.

- **All associated objects**

  Selected objects and all objects associated with them are copied. For example, cuts and fittings applied to a part.

- **Advanced**

  This option is similar to **All associated objects**, but works better with modifications. For example, when you have stairs that have posts welded to the steps, and you modify the distance between steps.

4. Select the **Copy origin**. The options are:
   - **Object to be copied**

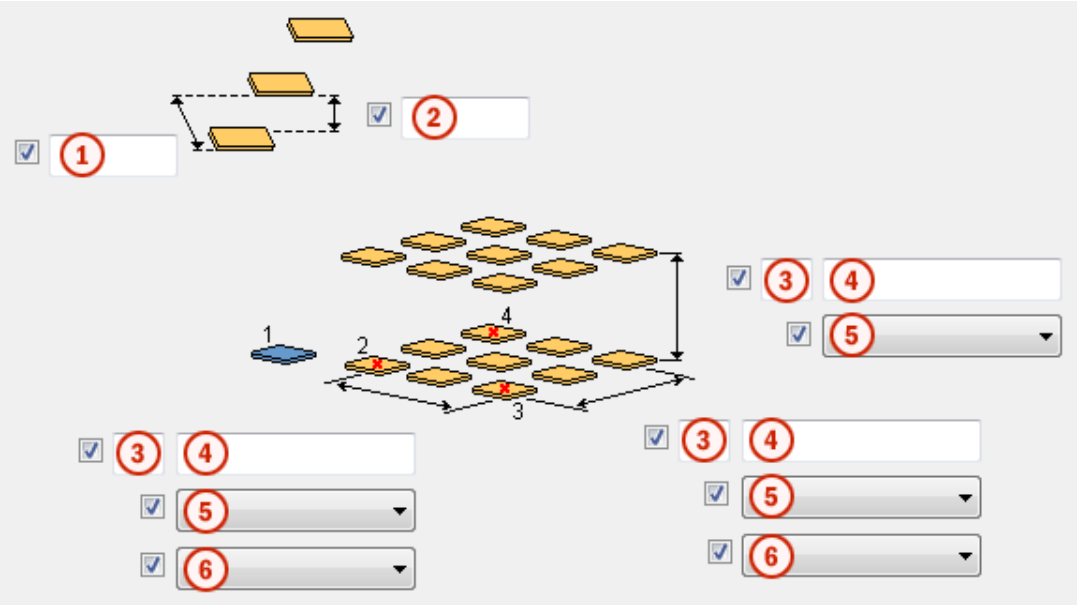
     This is the default. Copies are relative to the input objects.

   - **Origin point**

     Copies are relative to the input origin point.
5. Define the settings.
6. Select the objects to copy.
7. Click **OK** to close the dialog.
8. Click the middle mouse button.
9. Pick origin point.
10. Pick axis direction X.
11. Pick axis direction Y.

    The selected objects are copied.

How to define the settings

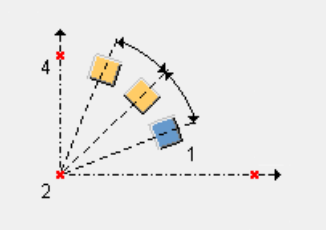


1	Offset along the Y axis. The default value is 0 mm.
2	Offset along the Z axis. The default value is 0 mm.
3	Number of copies. The default value is 0. If you leave this empty, the number of copies is taken from the **Space between copies** field.
4	Space between copies. The default value is 0 mm. Use the space character to separate values. Enter a value for each space between copies. This option is not available if you select **Equal** as the spacing method.
5	Copy direction. The options are: • **Normal** (default) Spacing values are calculated from the origin in positive direction along the axis. • **Reverse** Spacing values are calculated from the origin in negative direction along the axis. • **Centered** Copies are centered on the origin. • **Mirror** Spacing values are calculated from the origin in both positive and negative direction. Mirrored copying doubles the number of copies.

6	Spacing method. The options are: • **Equal** (default) Copies are equally spaced based on the length of the X or Y axis. • **Specified** Copies are spaced according to the number and spacing values given.

Copy objects using radial array tool

Use **Radial array tool** to copy selected objects radially along multiple directions at defined intervals or spacing. Tekla Structures does not check for duplicates when you copy objects using this method.



How to use Radial array tool

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for **Radial array tool**, and then double-click to open it.
3. Select the **Copy method**. The options are:
   - **Selected objects only**

     This is the default. Only the selected objects are copied.
   - **All associated objects**

     Selected objects and all objects associated with them are copied. For example, cuts, welds, and bolts.
   - **Advanced**

     This option is similar to **All associated objects**, but works better with modifications. For example, when you have stairs that have posts welded to the steps, and you modify the distance between steps.
4. Select the **Rotate copies** option.

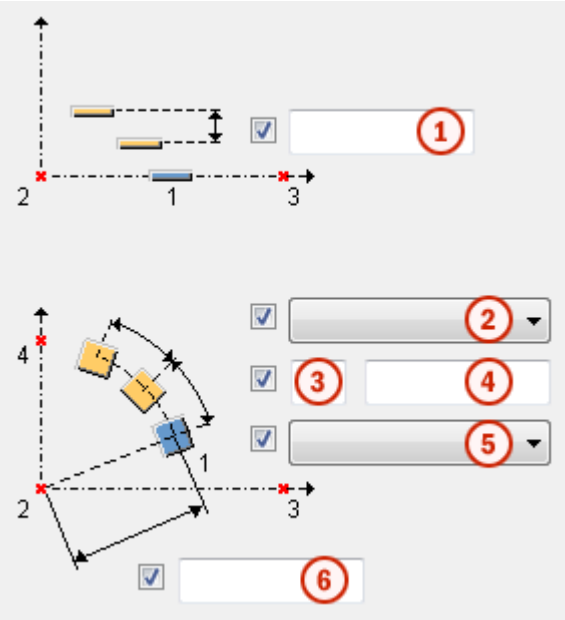
   The default is **Yes**.
5. Define the rotation axis.

   The default is **X**.
6. Define the settings.

7. Select the objects to copy.
8. Click **OK** to close the dialog.
9. Click the middle mouse button.
10. Pick origin point.
11. Pick axis direction X.
12. Pick axis direction Y.

    The selected objects are copied.

How to define the settings



1	Distance between copies. The default value is 0.
2	Rotation. The options are: • **Angle** (default) The copies are rotated by angle. • **Distance** The copies are rotated by distance.
3	Number of angles or distances. The default value is 0. If you leave this empty, the number of copies is taken from the **Space between copies** field.
4	Space between copies. Use the space character to separate values. Enter a value for each space between copies.

5	Copy direction. The options are: • **Normal** (default) Spacing values are calculated from the origin in positive direction along the axis. • **Reverse** Spacing values are calculated from the origin in negative direction along the axis. • **Centered** Copies are centered on the origin. • **Mirror** Spacing values are calculated from the origin in both positive and negative direction. Mirrored copying doubles the number of copies.
6	Radial distance. The radial distance should be equivalent to the distance you picked when applying the component. If the radial distance is smaller or greater than the picked distance, the spacing between the copied objects is not the same as given in the **Space between copies** box (4). Tekla Structures calculates the rotation angle according to the dialog values (spacing and radial distance), and the rotation angle overrides the spacing given in the dialog.

Copy objects using Array of objects (29) component

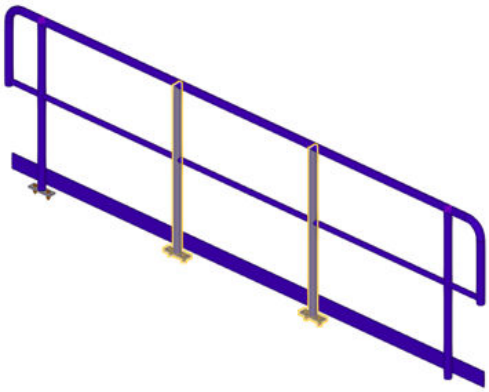
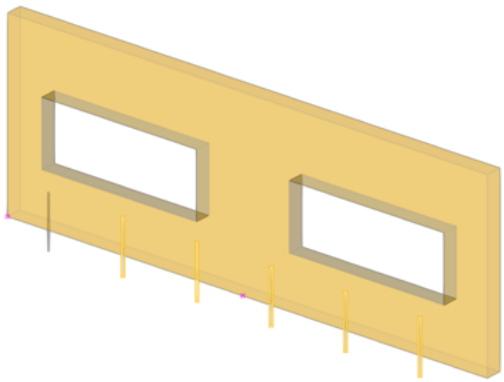
Use the **Array of objects (29)** component to copy model objects along a line. If you modify the original object, Tekla Structures also changes the copied objects.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for the **Array of objects (29)** component, and then double-click to open it.
3. Define the settings:
   - **Number of copies**: Enter the number of copies you want to create.
   - **Spacing values**: Define the spacing of the objects.
   - **Copy to the opposite direction**: Select **Yes** if you want to copy in the direction opposite to the points you pick.
   - **Start point for copying**: Choose either the object to be copied or the first input point.

- **Copy at equal distances (Ignore spacing values)**: Select **Yes** if you want to create the objects at equal distances. **Spacing value** will be ignored.

4. Click **OK** to save the settings.
5. Select the objects to copy.
6. Click the middle mouse button to finish selecting.
7. Pick a point to indicate the start of the line along which to arrange copied objects.
8. Pick a point to indicate the end of the line.

Examples

Example	Description
	An array of steel objects.
	An array of concrete objects.

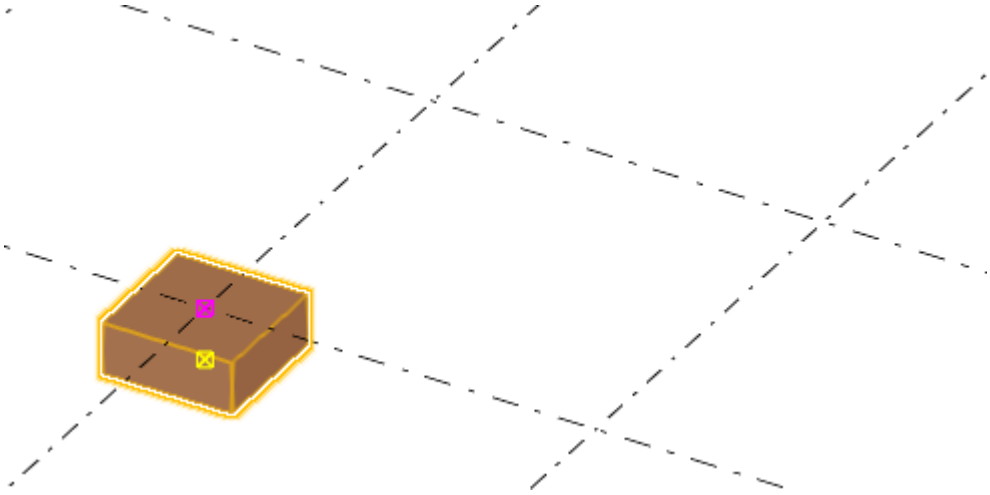
Move objects in models or in drawings

You can move objects in Tekla Structures in a number of different ways, especially in models. When you move an object, Tekla Structures also copies all objects connected to it, including the components.

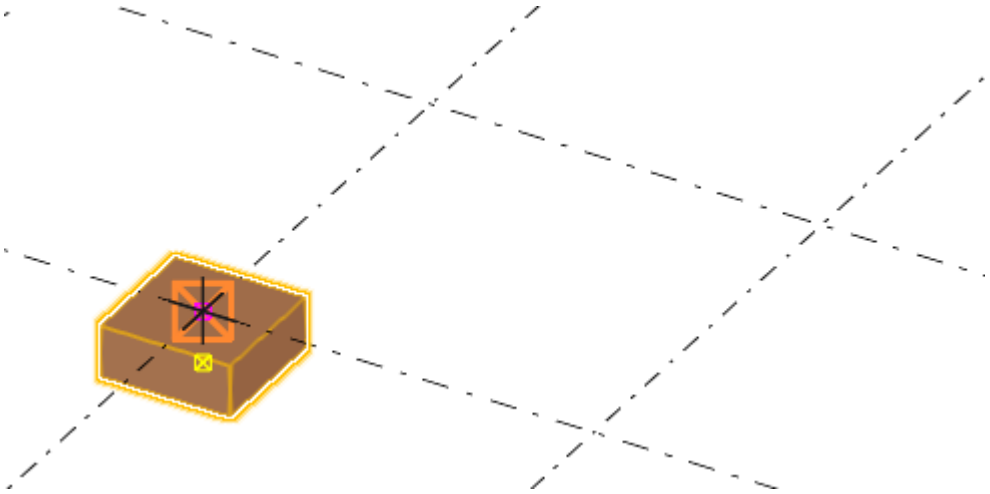
Move objects in a model or in a drawing

The basic way to move objects in a model or drawing is by defining the origin and one or more destination points.

1. Select the objects you want to move.



2. Run the **Move** command:
   - In the model, on the **Edit** tab, click **Move**.
   - In the drawing, on the **Drawing** tab, click **Move**.
3. Pick the origin for moving.

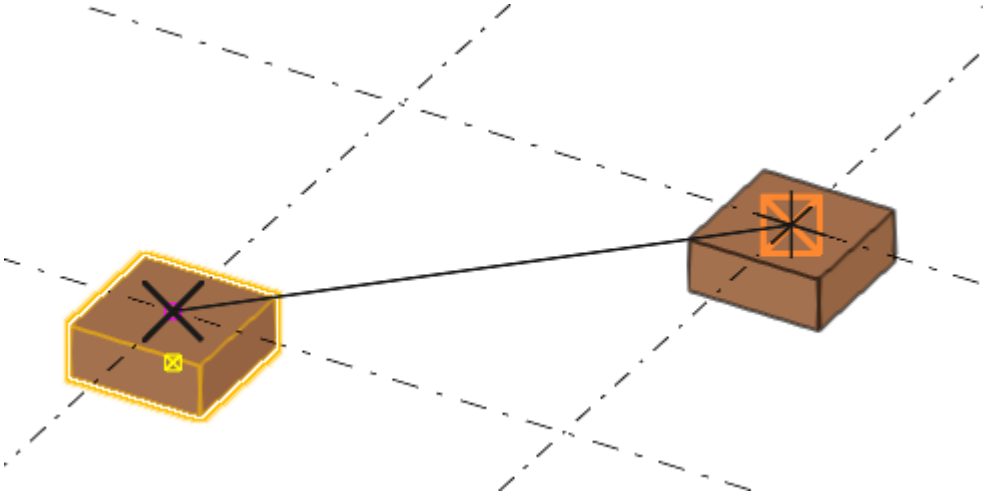


In the model, Tekla Structures displays a rubber band line between the first picked point and the cursor position. This is a preview of the position where the objects will be moved. Move the cursor to see how the preview changes.

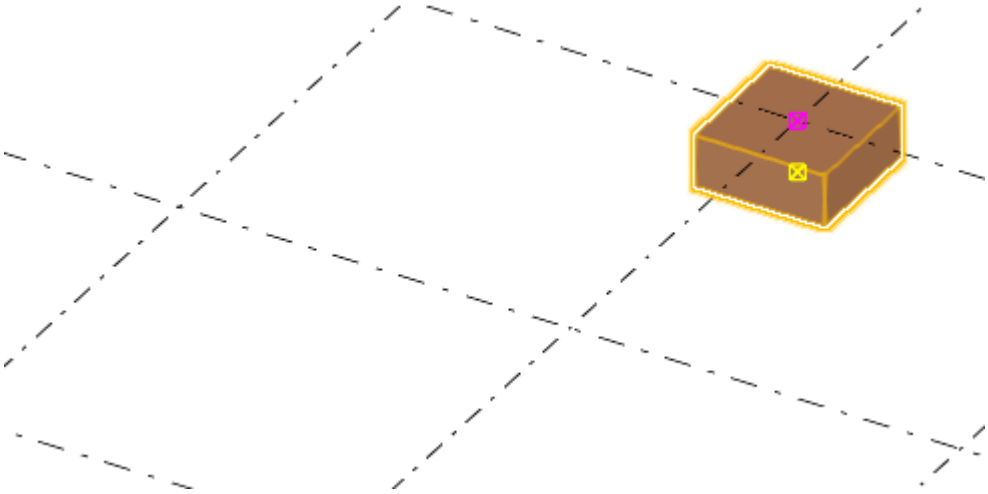
Note that Tekla Structures always displays the preview in the position where the objects will be moved, not in the position where the cursor is when you pick the destination point.

If you want to limit the number of objects shown in the preview, use the advanced option `XS_PREVIEW_LIMIT`. The default value is 1000. When the value is 0, the preview is off.

4. Pick the destination point.



The objects are moved immediately. The **Move** command does not remain active.



Move objects by specifying a distance from origin in a model or in a drawing

You can move objects to a new position in the model or drawing by specifying a distance from the origin. Use the **Enter a Numeric Location** dialog to specify the distance.

1. Select the objects you want to move.

2. Run the **Move** command:
   - In the model, on the **Edit** tab, click **Move**.
   - In the drawing, on the **Drawing** tab, click **Move**.
3. Pick the origin for moving.
4. Move the cursor in the direction you want to move the objects, but do not pick the point.
5. Type the distance.

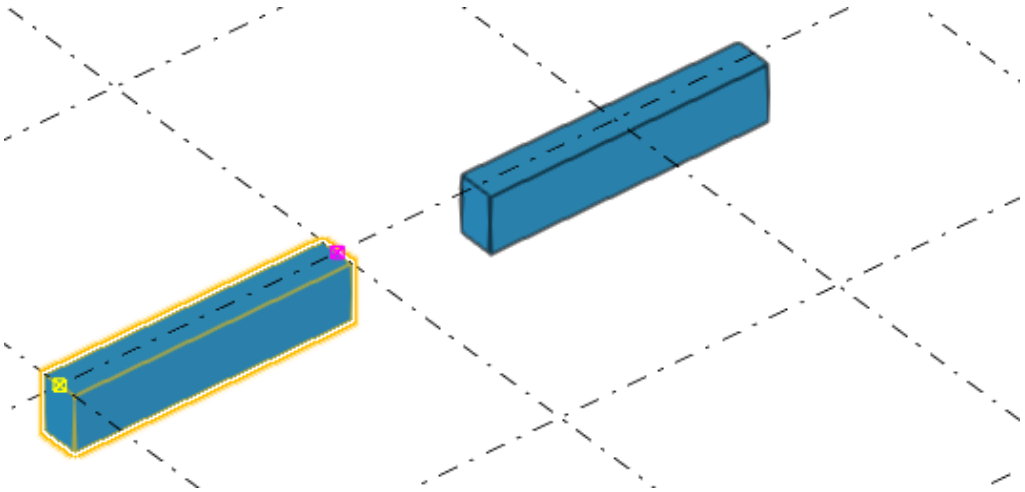
   When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog automatically.
6. Click **OK**.

Move objects using drag-and-drop in a model or in a drawing

You can move objects in a model or in a drawing by dragging them to a new location.

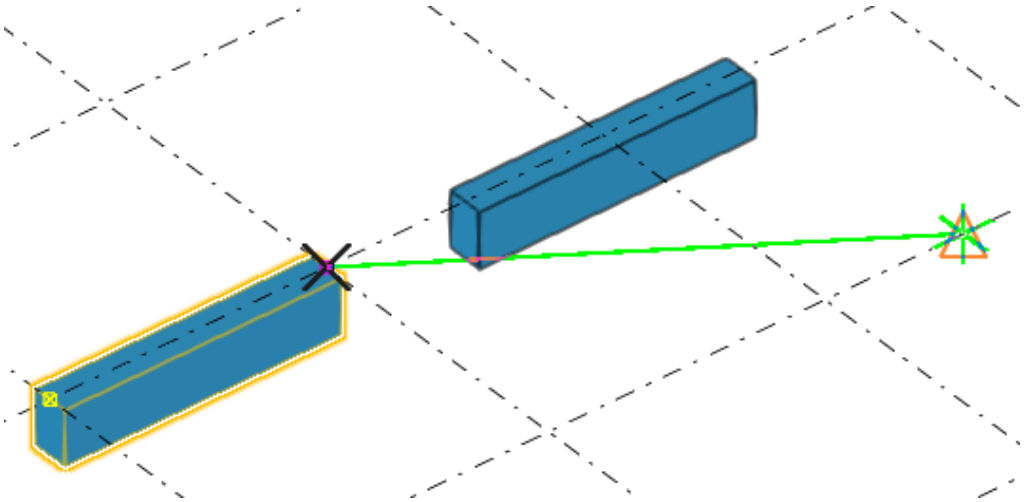
1. Activate drag-and-drop.
   - In the model, on the **File** menu, click **Settings** and select the **Drag & drop** checkbox.
   - In the drawing, on the **File** menu, click **Settings** and select the **Drawing drag & drop** checkbox.
2. Select the objects you want to move.

   In drawings, you can drag annotations, sketch objects, and grid labels. To move grid labels, first select the grid label and then either activate the  **Select grid line** selection switch or select the grid label handle.

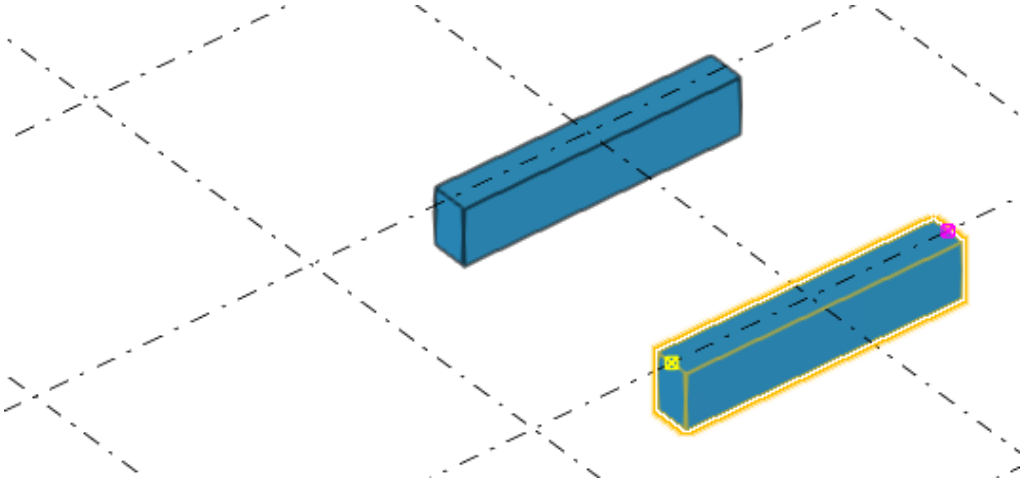


3. Drag the objects to a new location.

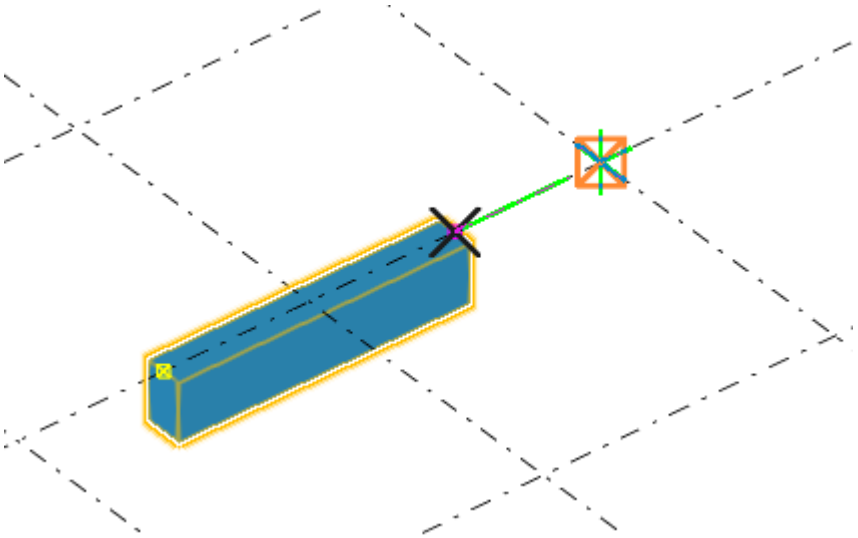
   In a model, the point you start dragging from (center, corner, or middle point) affects the object's alignment in the new location.



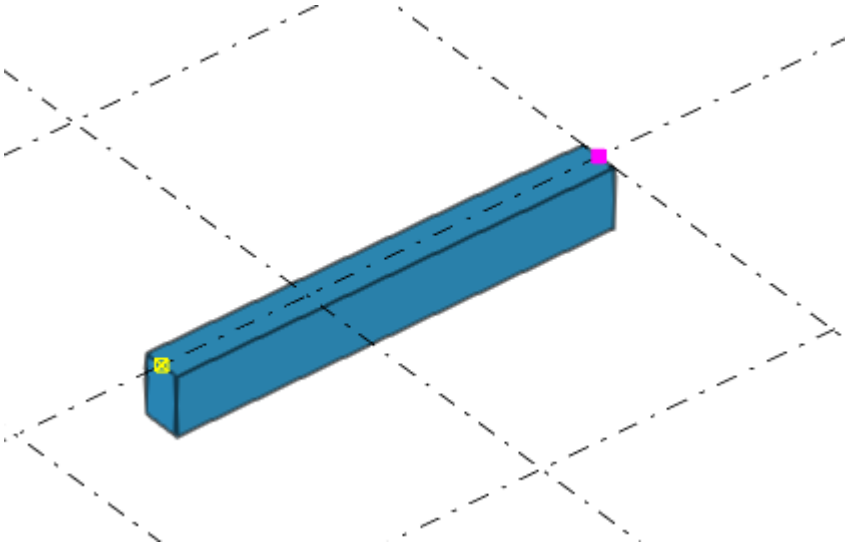
   The objects are moved immediately.



4. To move an end point using drag-and-drop:
   a. Select the handle.
   b. Hold down the left mouse button, and drag the handle to a new location.



The end point is moved accordingly:



Note that for some objects you may need to switch on **Smart select** to drag from handles without selecting them first. To switch it on, click **File** --> **Settings** and select the **Smart select** checkbox.

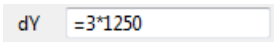
Move objects linearly in a model

In a model, you can move objects linearly to a new position.

1. Select the objects you want to move.
2. On the **Edit** tab, click **Move** --> **Linear**.

   The **Move - Linear** dialog opens.
3. Pick two points in the model, or enter the coordinates in the **dX**, **dY**, and **dZ** boxes.

You can also use a formula to calculate the x, y, and z displacements. For example:



4. Click **Move**.

TIP If the dialog is open but the command is not active anymore, click the **Pick...** button to re-activate the command.

Move objects to another plane in a model

In a model, you can move objects from the first plane you specify to the second, third, and so on, plane you specify. The moved objects remain in the same position in the second plane, third, and so on, plane as the original objects in the first plane.

1. Select the objects you want to move.
2. On the **Edit** tab, click **Move** --> **To another plane**.
3. Pick the point of origin of the first plane.
4. Pick a point on the first plane in the positive x direction.
5. Pick a point on the first plane in the positive y direction.
6. Repeat steps 3–5 for the destination planes.

Move objects to another object in a model

In a model, you can move objects from an object to other, similar objects. This is useful, for example, when you detail previously modeled parts. The objects that you move between can have different dimensions, length, and rotation.

If you move reinforcement or surface treatments and want them to adapt to the part they are moved to, note the limitations:

- The reinforcement or surface treatment handles must be in part corners.
- The parts between which you move must have the same number of cross-section corners.
- Circular parts must have the same cross-section dimensions.

1. Select the objects you want to move.
2. On the **Edit** tab, click **Move** --> **To another object**.
3. Select the object to move from (source object).
4. Select the objects to move to (target object).

Rotate objects in models or in drawings

In Tekla Structures, you can copy or move an object in a model by rotating it around any line you choose. In a drawing, you can copy or move an object by rotating it around a given line on the work plane.

NOTE Positive rotation is clockwise when viewed from the start point of the rotation axis.

Rotate objects around a line in a model

Use the **line** option in the **Rotate** dialog when you want to copy and rotate, or move and rotate objects around any given line in the model.

1. Select the objects you want to copy or move.
2. Activate the rotation command.
   - To copy and rotate, go to the **Edit** tab and click **Copy** --> **Rotate**.

     The **Copy - Rotate** dialog opens.
   - To move and rotate, go to the **Edit** tab and click **Move** --> **Rotate**.

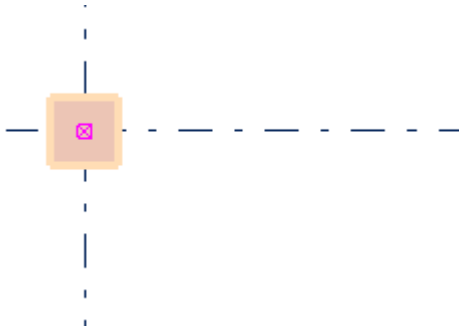
     The **Move - Rotate** dialog opens.
3. In the **Around** list, select **line**.
4. Pick the start point of the rotation axis, or enter its coordinates.
5. Pick the end point of the rotation axis, or enter its coordinates.
6. If you are copying, enter the number of copies.
7. If needed, enter the **dZ** value, which is the difference in position between the original and copied object in the z direction.
8. Enter the rotation angle.
9. Click **Copy** or **Move**.

   The objects are rotated accordingly.

Rotate objects around the z axis in a model

Use the **Z** option in the **Rotate** dialog when you want to copy and rotate, or move and rotate objects around the z axis in the model.

1. Select the objects you want to copy or move. For example:

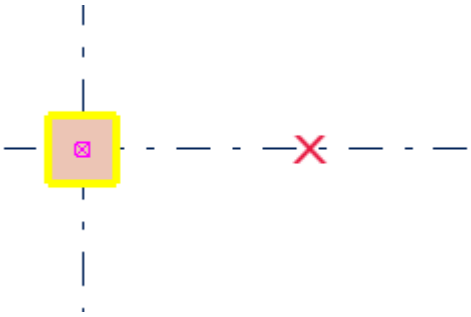


2. Activate the rotation command.
   - To copy and rotate, go to the **Edit** tab and click **Copy** --> **Rotate**.

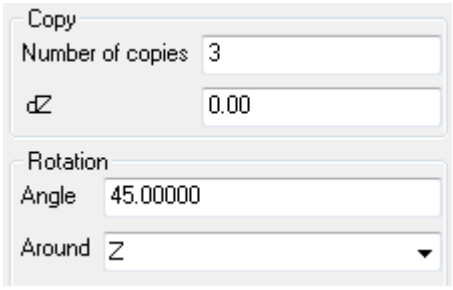
     The **Copy - Rotate** dialog opens.
   - To move and rotate, go to the **Edit** tab and click **Move** --> **Rotate**.

     The **Move - Rotate** dialog opens.
3. Select **Z** in the **Around** list.
4. Pick a point to define the rotation axis, or enter its coordinates.

   In the example below, the red cross indicates the picked point.

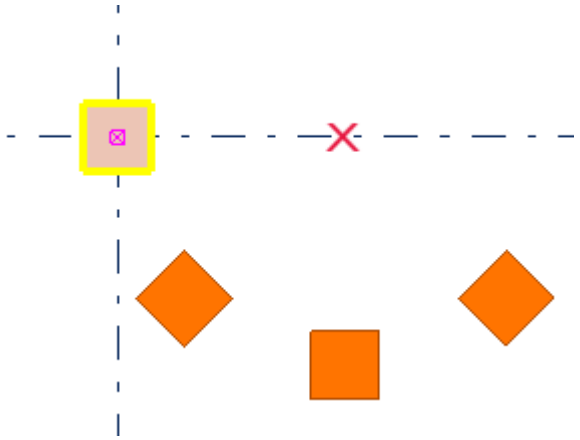


5. If you are copying, enter the number of copies.
6. If needed, enter the **dZ** value, which is the difference in position between the original and copied object in the z direction.
7. Enter the rotation angle. For example:



8. Click **Copy** or **Move**.

The objects are rotated accordingly.



Rotate objects in a drawing

Use this option when you want to rotate drawing objects on the work plane.

1. Select the objects you want to copy or move.
2. Activate the rotation command.
   - To copy and rotate, go to the **Drawing** tab and click **Copy** --> **Rotate**.

     The **Copy - Rotate** dialog opens.
   - To move and rotate, go to the **Drawing** tab and click **Move** --> **Rotate**.

     The **Move - Rotate** dialog opens.
3. Pick a point, or enter its coordinates.
4. If you are copying, enter the number of copies.
5. Enter the rotation angle.
6. Click **Copy** or **Move**.

Rotation settings

Use the **Copy - Rotate** and **Move - Rotate** dialogs to view and modify the settings that are used when you rotate objects in Tekla Structures. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.

Option	Description
X0	The x and y coordinates of the start point of the rotation axis.
Y0	
Origin angle	The angle of the rotation axis when rotating around a line on the work plane.
Number of copies	Define the number of copies created.

Option	Description
dZ	The difference in position between the original and copied object in the z direction.
Rotation angle	The rotation angle between the original and new position.
Around	Define whether the rotation axis is a **line** on the work plane or in the **z** direction.

Mirror objects in models or in drawings

In Tekla Structures, when you copy or move objects, you can mirror them through a plane that is perpendicular to the work plane and passes through a line you specify.

Note that Tekla Structures cannot create mirrored copies of object properties. For example, the **Copy** --> **Mirror** command does not fully mirror objects if they include components with asymmetrically positioned parts, or reinforcement objects with asymmetric properties such as spacing.

Mirror objects in a model

Use this method to copy and mirror, or move and mirror objects in a model.

1. Select the objects you want to copy or move.
2. Activate the mirroring command.
   - To copy and mirror, go to the **Edit** tab and click **Copy** --> **Mirror**.

     The **Copy - Mirror** dialog opens.
   - To move and mirror, go to the **Edit** tab and click **Move** --> **Mirror**.

     The **Move - Mirror** dialog opens.
3. Pick the start point of the mirroring plane, or enter its coordinates and angle.
4. Pick the end point of the mirroring plane, or enter its coordinates and angle.

   The first picked point (X0, Y0) is the origin, and the second picked point calculates the angle around the origin.
5. Click **Copy** or **Move**.

Mirror objects in a drawing

Use this method to copy and mirror, or move and mirror objects in a drawing.

1. Select the objects you want to copy or move.

2. Activate the desired mirroring command.
   - To copy and mirror, go to the **Drawing** tab and click **Copy** --> **Mirror**.

     The **Copy - Mirror** dialog opens.
   - To move and mirror, go to the **Drawing** tab and click **Move** --> **Mirror**.

     The **Move - Mirror** dialog opens.
3. Pick the start point of the mirroring plane, or enter its coordinates in the dialog.
4. Pick the end point of the mirroring plane, or enter its coordinates in the dialog.
5. Enter the angle in the dialog.
6. Click **Copy** or **Move**.

Tips for copying and moving objects in models and in drawings

The basic functionality of copying and moving objects in Tekla Structures is the same in models and drawings.

- **Copy carefully**

  Always copy the objects carefully, and ensure that you are copying the intended objects.
- **Control over copying**

  At first, when you familiarize yourself with snapping, we recommend using the **Copy - Linear** method to have more control over the copying action.
- **Compare copied objects**

  Any comparison between the copied objects should be done before running numbering. The numbering itself works as the final check.
- **Select a suitable command for copying**
  - To ensure that the object is copied in the intended plane, use the **Copy** --> **Linear** command. The **Copy - Linear** dialog can be used as an explicit check to confirm that copying distance is in the intended direction and round values.
  - To copy objects, such as reinforcement, between similar objects, use the **Copy** --> **To another object** command. Always ensure that the object you copy from and the object you copy to are of similar type and have similar shape. For example, a polygonal slab and a rectangular column have different types of part handles, and their front faces are of different shape and location.
  - To copy objects around a specified line on the work plane, use the **Copy** --> **Rotate** command. When you use this command, copy carefully and

always check the outcome. If the results are not as expected, copy in smaller chunks, for example, one component at a time.

- To copy objects from one model to another, use the **Copy** command. Alternatively, you can use the **Copy** --> **From another model** command. When using the **From another model** command, copying is based on the phase numbers in the original model. Successful copying requires that you have set the objects correctly, without any additional objects, in a specific phase in the source model. Otherwise all the objects included in the phase are copied.

  Note that if you copy objects from another model, only the model objects are copied. The drawings are not copied.

- The source object for the copying determines the object orientation.

  When the objects are copied with the **Copy** command, the object orientation of the target object remains the same as the orientation of the source object.

  When the objects are copied with the **Copy** --> **To another object** command, the object orientation is defined in relation to the source object's internal coordinate system, and this orientation will be translated into the target object's internal coordinate system.

  Custom components have their own logic for object orientation. It may be more straightforward to add the component in the model than to use copying, especially if the target object's main geometry differs considerably from the source object's geometry.

- **Duplicate objects**

  After copying and moving, check that the outcome is as expected and there are no accidentally created duplicates in the model.

  Two objects are considered duplicates if they have the same properties and location. Tekla Structures checks for duplicate, overlapping objects when you copy and move objects or create new objects in the same location as an existing object. If duplicates are found, you can choose whether to keep or delete them. If you choose to keep the duplicates, they are hard to detect later on.

  Use the advanced option XS_DUPLICATE_CHECK_LIMIT_FOR_COPY_AND_MOVE to define the maximum number of objects that can be counted as duplicates while copying or moving objects.

  NOTE Tekla Structures does not check for duplicates when you copy objects using a modeling tool, such as the **Array of objects (29)** component.

- **Assemblies and cast units**

  If you copy or move objects from an assembly or cast unit, Tekla Structures copies the assembly structure if possible. For example, sub-assemblies are copied as sub-assemblies if a parent object is found.

  When selecting the correct content to be copied, first use model selection filters (page 169), and secondly assembly, part, or component selection switches.

  To easily select all objects within an assembly or a cast unit according to the selection filter, hold down the **Alt** key and click any object in the assembly or cast unit.

- **Drawing objects**

  You can copy and move drawing objects between drawing views that have different scales. To define how the copied or moved objects are scaled, use the advanced option XS_SCALE_COPIED_OR_MOVED_OBJECTS_IN_DRAWINGS.

- **Reinforcement and surface treatment**

  If you copy or move reinforcement or surface treatments (page 425), and want them to adapt to the part they are copied or moved to:

  - The reinforcement handle or surface treatment handles must be in part corners.
  - The parts between which you copy or move must have the same number of cross-section corners.
  - Circular parts must have the same cross-section dimensions.

- **Copy and move efficiently**

  You can keep the **Move** and **Copy** dialogs open if you are going to use them often.

  After you have run the **Copy - Linear**, **Copy - Mirror**, **Copy - Rotate**, or the **Move - Linear**, **Move - Mirror** or **Move - Rotate** command, interrupt the command and leave the dialog open. When you want to continue copying or moving, click the dialog to activate it and continue to copy or move objects.

1.9 Filter objects

Use filters to restrict what can be selected or what is visible in a view. You can create filters of your own, or you can use any of the standard filters available in Tekla Structures.

Here are some examples of what filters can be used for:

- **To select a large number of objects**

  Use selection filters when you need to change a specific object property that is common for many objects. The rest of the objects will not be affected, even if you try to include them in the selection.

- **To check the model**

  Use view filters to ensure that beams are called beams, columns are called columns, and so on. You can highlight several groups of objects, one at a time, to check that all required objects are included in a given group.

- **To hide objects**

  Use view filters to temporarily hide the columns in a view so that it easier to select all the beams, for example.

- **To find objects**

  You can create a selection filter to find all locations where ½" reinforcing bars are in the model, for example. Once the filter is active, you can make an area selection that includes the entire model. All specified reinforcing bars become selected, but the other objects will not be affected.

See also

Use existing filters (page 169)

Create new filters (page 172)

Object categories and properties in filtering (page 181)

Filtering techniques (page 198)

Select values from the model (page 201)

Wildcards (page 202)

Examples of filters (page 202)

Copy and remove filters (page 210)

Use existing filters

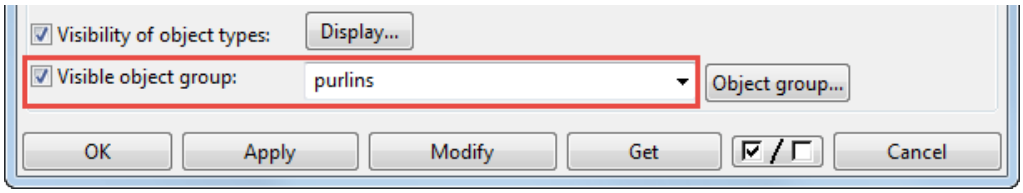
Before creating new custom filters, check out the existing view and selection filters available in Tekla Structures.

How to use a view filter

Use view filters to define which objects are displayed in a model view.

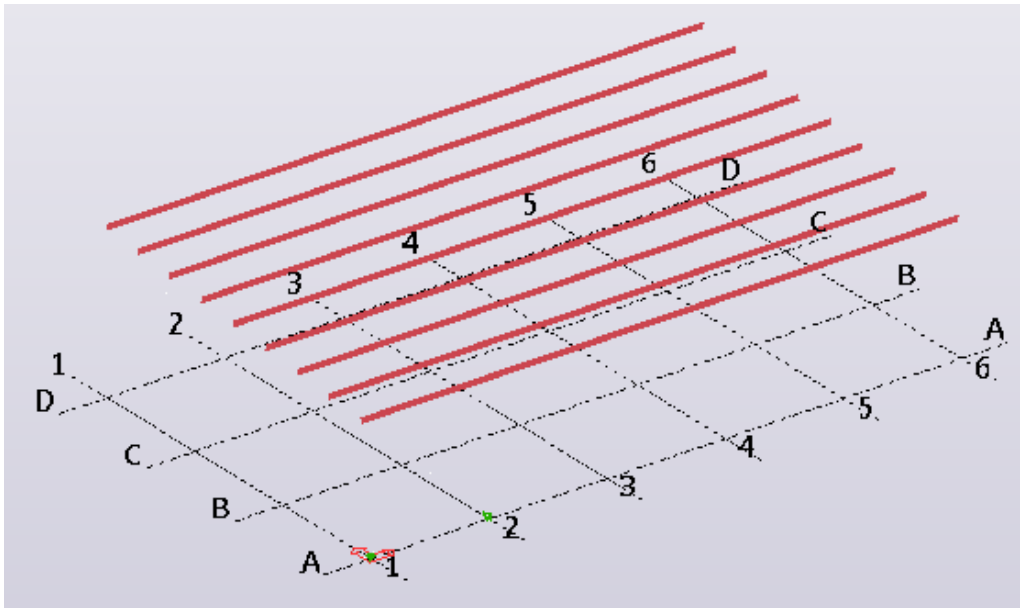
1. Double-click the view to open the **View properties** dialog.
2. Select a filter from the **Visible object group** list.

For example, select **purlins**.



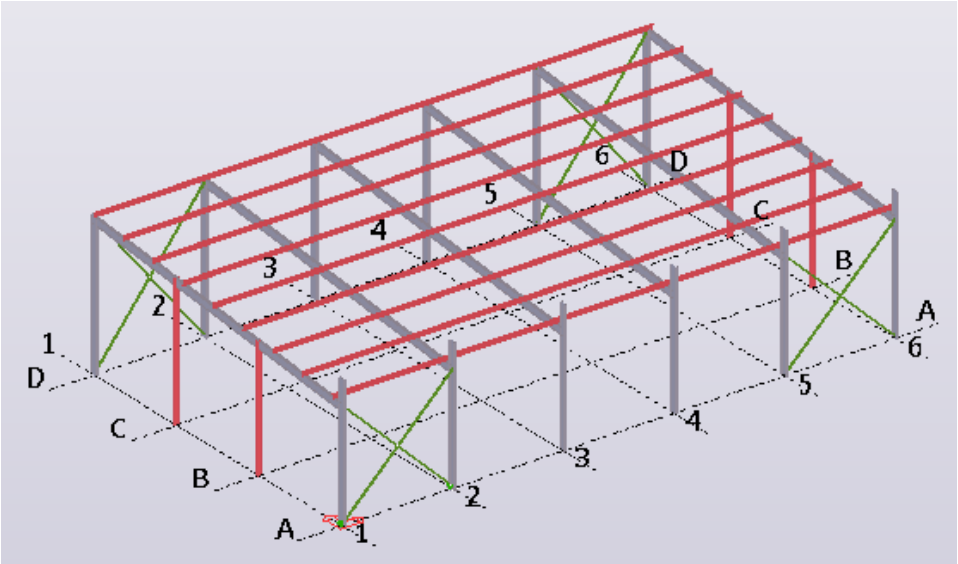
3. Click **Modify**.

   Now only the objects defined by the filter are visible. For example, the purlins:



4. To stop using the filter:
   a. Double-click the view to open the **View properties** dialog.
   b. In the **Visible object group** list, select the **standard** filter.
   c. Click **Modify**.

All objects are visible again:



NOTE If you cannot see all desired objects (page 695), note that the work area, view depth, view setup, and object representation settings also affect the visibility of objects.

How to use a selection filter

Use selection filters to define which objects can be selected in the model. Note that an object must be visible in the model to be selectable.

1. On the **Selecting** toolbar, select a filter from the 

list.

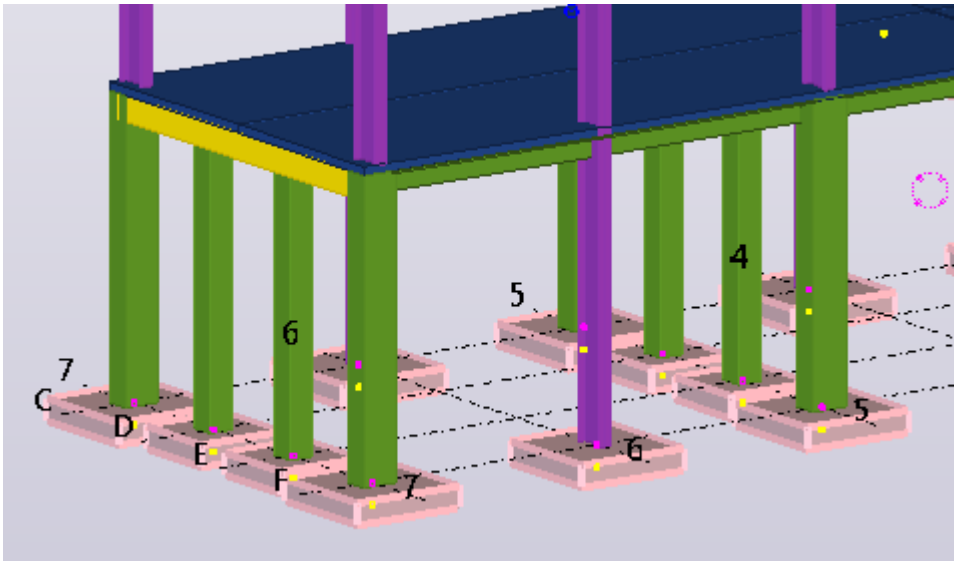
   The list is by default located at the bottom of the Tekla Structures main window.

   For example, select the **Name - Footing** filter.

2. Select the desired objects in the model.

   You can select multiple objects or even the entire model at once. Now that the filter is active, only objects defined by the filter become selected. For

example, if the **Name - Footing** filter is active, only footings are selectable and the rest of the objects stay intact:



3. If you cannot select all objects defined by the selection filter, check your view filter settings and ensure that you have switched on all the needed selection switches.
4. To stop using the filter, go to the **Selecting** toolbar and select the **standard** filter.

   All objects are selectable again.

Create new filters

You can create custom filters to define which objects are visible and selectable in the model and drawings. Add new filter rules, one on each row, to define which objects should be included or excluded.

Create a view filter

You can create your own custom filters to define which objects are visible in a model.

1. Double-click the view to open the **View properties** dialog.
2. Click **Object group**.

   The **Object group - View filter** dialog opens, showing the currently active filter.
3. Click **New filter** to create a new filter from scratch.
4. Click **Add row** to add a new filter rule.
5. In the **Category** list, select an object category.

You have the following options:

- Part
- Component
- Bolt
- Weld
- Reinforcing bar
- Surface
- Assembly
- Construction object
- Load
- Template
- Reference assembly
- Reference object
- Location breakdown structure
- Pour object
- Pour break
- Pour unit
- Task
- Object

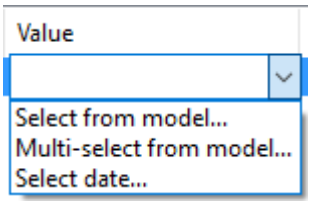
6. In the **Property** list, select a suitable object property (page 181).

   The options vary depending on the object category you chose in step 5.

7. In the **Condition** list, select a suitable condition (page 198).

8. In the **Value** list, type a value.

   Alternatively, to use the current value of an existing object, click **Select from model...** and select (page 201) the desired object from the model. To use the values of multiple objects, click **Multi-select from model**, select the objects from the model, and then click the middle mouse button. For date values, also the **Select date...** option is available.

   The values can be complete strings, such as the profile name `UC310*97`. You can also use incomplete strings together with wildcards (page 202). For example, the value `UC*` will match with all parts whose profile name

begins with the characters `UC*`. Empty values are matched to empty object properties.

If you use multiple values, separate the strings with blank spaces (for example, `12 5`). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, `"custom panel"`), or use a question mark (for example, `custom?panel`) to replace the space.

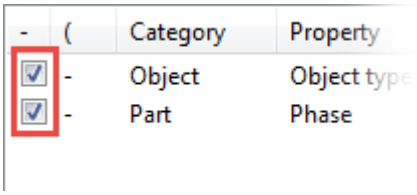
9. Repeat steps 4–8 to create as many filter rules as needed.

   You can apply several filter rules at the same time.

10. Use the **And/Or** options and brackets (page 198) to define how multiple rows work together.

11. Select the checkboxes next to all filter rules that you want to enable.

    If the checkbox is selected, the filter rule is enabled and effective. For example:



    By default, each new rule is disabled.

12. Define the filter type.

    a. Click >> to display more settings.

    b. Select or clear the checkboxes to define where the filter will be used.

       For example, the same filter may be used both as a view filter and as a selection filter. You can also define that the filter is used in **Organizer** or when defining the IFC property sets to be exported.

13. Enter a unique name in the box next to the **Save as** button.

NOTE
- Filters are case sensitive.
- Do not use spaces in filter names.
- We recommend that you use _ (underscores) in your naming convention.
- To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.

---

14. Click **Save as** to save the filter.

15. To apply the filter to the current view, click **Modify**.

Create a selection filter

You can create your own custom filters to help you select objects in a model.

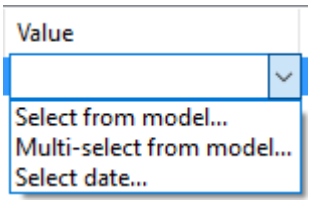
1. On the **Selecting** toolbar, click  to open the **Object group - Selection filter** dialog.

2. Click **New filter** to create a new filter.
3. Click **Add row** to add a new filter rule.
4. In the **Category** list, select an object category.
5. In the **Property** list, select a suitable object property (page 181).

   The options vary depending on the object category that you selected.
6. In the **Condition** list, select the condition (page 198) that defines how the filter criteria relate to one another..
7. In the **Value** list, select or enter a value.

   If you use multiple values, separate the strings with blank spaces (for example, `12 5`). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, `"custom panel"`), or use a question mark (for example, `custom?panel`) to replace the space.

   - To use the current value of an existing object, click **Select from model...** and select (page 201) the desired object from the model.
   - To use the values of multiple objects, click **Multi-select from model**, select the objects from the model, and then click the middle mouse button.
   - For date values, click **Select date...**.

   The values can be complete strings, such as the profile name `UC310*97`. You can also use incomplete strings with wildcards (page 202). For example, the value `UC*` matches all parts whose profile name begins with the characters `UC*`. Empty values match empty object properties.
8. Use the **And/Or** options and brackets (page 198) to define how multiple rows work together.
9. Define the filter type.

   a. Click  to show more settings.

b. Select or clear the checkboxes to define where the filter is used.

   For example, the same filter can be used both as a view filter and as a selection filter. You can also define that the filter is used in **Organizer** or when defining the IFC property sets to be exported.

10. Enter a unique name in the box next to the **Save as** button.

NOTE
- Filters are case sensitive.
- Do not use spaces in filter names.
- We recommend that you use _ (underscores) in your naming convention.
- To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.

11. Click **Save as** to save the filter.

Create a drawing filter

For general arrangement drawings, you can create drawing filters that affect the whole drawing, not just a specific view. Drawing filters select objects in the whole drawing.

You can use drawing filters together with saved object property files when you create and apply object level settings in the whole drawing. For example, you might create a filter that selects all beams, then save an object property file that defines that the part color is blue, and then create and apply an object level settings file that changes all beams to blue in the whole drawing.

1. In a GA drawing, on the **Drawing** tab, click **Properties** --> **Drawing**.
2. Click **Filter**.
3. Click **New filter** to create a new filter from scratch.
4. Click **Add row** to add a new filter rule.
5. In the **Category** list, select an object category.

   You have the following options:

   - Part
   - Component
   - Bolt
   - Weld
   - Reinforcing bar
   - Surface
   - Assembly
   - Construction object

- Template
- Reference assembly
- Reference object
- Location breakdown structure
- Pour object
- Pour break
- Pour unit
- Task
- Object

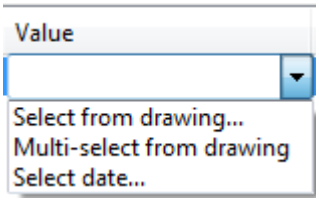
6. In the **Property** list, select a suitable object property (page 181).

   The options vary depending on the object category you chose in step 5.

7. In the **Condition** list, select a suitable condition (page 198).

8. In the **Value** list, type a value.

   Alternatively, to use the current value of an existing object, click **Select from drawing** and select the desired object from the drawing. To use the values of multiple objects, click **Multi-select from drawing**, select the objects from the drawing, and then click the middle mouse button. For date values, also the **Select date...** option is available.

   The values can be complete strings, such as the profile name `UC310*97`. You can also use incomplete strings together with wildcards (page 202). For example, the value `UC*` will match with all parts whose profile name begins with the characters `UC*`. Empty values are matched to empty object properties.

   If you use multiple values, separate the strings with blank spaces (for example, `12 5`). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, `"custom panel"`), or use a question mark (for example, `custom?panel`) to replace the space.

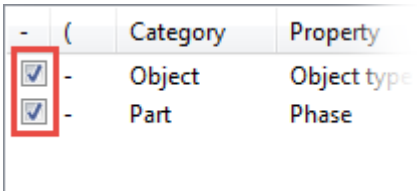
9. Repeat steps 4–8 to create as many filter rules as needed.

   You can apply several filter rules at the same time.

10. Use the **And/Or** options and brackets (page 198) to define how multiple rows work together.

11. Select the checkboxes next to all filter rules that you want to enable.

If the checkbox is selected, the filter rule is enabled and effective. For example:



By default, each new rule is disabled.

12. Define the filter type.

    a. Click >> to display more settings.

    b. Select or clear the checkboxes to define where the filter will be used.

       For example, the same drawing filter may be used both as a model view filter and as a model selection filter, and also as an Organizer filter.

13. Enter a unique name in the box next to the **Save as** button.

    NOTE
    - Filters are case sensitive.
    - Do not use spaces in filter names.
    - We recommend that you use _ (underscores) in your naming convention.
    - To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.

14. Click **Save as** to save the filter.

15. When you are done, click **Cancel** to close the filter properties dialog.

Create a drawing view filter

You can create your own custom view filters to help you select a specific group of view objects in a drawing view.

You can use drawing view filters for changing the appearance of a certain object group, or for selecting which objects are hidden in a drawing view.

You can also use drawing view filters together with saved object property files when you create and apply object level settings in the selected view. For example, you might create a view filter that selects all columns in a view, then save an object property file that defines that the part color is red, and then create and apply an object level settings file that changes all columns to red in the selected view.

1. Open a drawing.
2. Double-click a drawing view frame.

3. Click **Filter**.
4. Click **New filter** to create a new filter from scratch.
5. Click **Add row** to add a new filter rule.
6. In the **Category** list, select an object category.

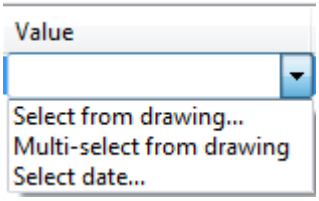
   You have the following options:

   - Part
   - Component
   - Bolt
   - Weld
   - Reinforcing bar
   - Surface
   - Assembly
   - Construction object
   - Template
   - Reference assembly
   - Reference object
   - Location breakdown structure
   - Pour object
   - Pour break
   - Pour unit
   - Task
   - Object
7. In the **Property** list, select a suitable object property (page 181).

   The options vary depending on the object category you chose in step 5.
8. In the **Condition** list, select a suitable condition (page 198).
9. In the **Value** list, type a value.

   Alternatively, to use the current value of an existing object, click **Select from drawing** and select the desired object from the drawing. To use the values of multiple objects, click **Multi-select from drawing**, select the

objects from the drawing, and then click the middle mouse button. For date values, also the **Select date...** option is available.



The values can be complete strings, such as the profile name `UC310*97`. You can also use incomplete strings together with wildcards (page 202). For example, the value `UC*` will match with all parts whose profile name begins with the characters `UC*`. Empty values are matched to empty object properties.

If you use multiple values, separate the strings with blank spaces (for example, `12 5`). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, `"custom panel"`), or use a question mark (for example, `custom?panel`) to replace the space.

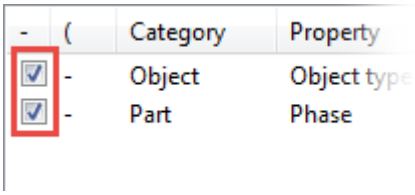
10. Repeat steps 4–8 to create as many filter rules as needed.

    You can apply several filter rules at the same time.

11. Use the **And/Or** options and brackets (page 198) to define how multiple rows work together.

12. Select the checkboxes next to all filter rules that you want to enable.

    If the checkbox is selected, the filter rule is enabled and effective. For example:



    By default, each new rule is disabled.

13. Define the filter type.

    a. Click >> to display more settings.

    b. Select or clear the checkboxes to define where the filter will be used.

       For example, the same drawing view filter may be used both as a model view filter and as a model selection filter, and also as an Organizer filter.

14. Enter a unique name in the box next to the **Save as** button.

NOTE • Filters are case sensitive.

- Do not use spaces in filter names.
- We recommend that you use _ (underscores) in your naming convention.
- To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.

15. When you are done, click **Cancel** to close the filter properties dialog.

Create a drawing selection filter

You can create your own custom filters to help you select objects in a drawing.

You can use selection filters in drawings if you want to hide certain parts from drawings or drawing views, or change the part color or representation for certain parts.

Also, if you have some different looking part marks for different types of parts, you can select the specific parts using a selection filter and then only modify part marks for those parts.

1. In an open drawing, on the **Selecting** toolbar, click  (**Ctrl+G**).

   The **Selection filter** dialog opens.
2. Follow the instructions above on how to create a drawing or drawing view filter.

   The same instructions apply to drawing selection filters.
3. Click **Apply** or **OK** to select the parts according to the filter.

Object categories and properties in filtering

You can select from a wide variety of object properties when creating new filters. In the tables below, the properties are listed according to their object category. In addition to these, almost all categories contain user-defined attributes and template attributes that can also be used in filtering.

Category: Object

Use the **Object** category to filter objects based on their object level properties.

Property	Description
GUID	To filter objects based on their globally unique identifier (GUID). For example, "Object GUID begins with `ID7554C9EB-C8B4`."

Property	Description
Phase	To filter objects based on their phase number. For example, "Object phase does not equal 3."
Object type	To filter objects based on their type. Select an object type from the **Value** list, or use the **Select from model...** or **Multi-select from model** option. NOTE: We recommend that you include one filter rule for the **Object type** property in each filter you create. This ensures that only objects of this type are filtered in. If you leave the object type out, the filtering outcome will be different, and objects that do not match the category in further filter rules may be filtered in. The following object types can be selected from the list: • Assembly • Bolt group • Connection • Part • Pour break • Pour object • Pour unit • Reference object • Reinforcing bar • Surface • Surface treatment • Weld The following object types are shown as numeric values only: • 1 = point • 9 = fitting • 11 = polygon cut • 12 = line cut • 24 = construction line

Property	Description
	• 30 = construction plane • 38 = added material • 42 = construction circle • 43 = construction arc • 48 = reference model • 76 = analysis part • 79 = edge chamfer • 103 = construction polycurve
Is component	To filter objects based on whether or not they are components. The options are **Yes** and **No**. For example, "Object is component equals `Yes`."

Some object types are not visible directly, but only when the objects that make them up are visible. For example, assemblies are visible when parts are visible, and pour units are visible when pour objects are visible. Therefore, using the object type **Assembly** or **Pour unit** alone in a view filter in the model or in drawings does not show anything. However, selection filters are able to select object types like assemblies and pour units.

Category: Part

Use the **Part** category to filter parts (page 217) based on their common properties.

Property	Description
Name	To filter objects based on their name. For example, "Part name equals `SLAB`."
Profile	To filter objects based on their profile. For example, "Part profile does not equal `L20*2`."
Material	To filter objects based on their material grade. For example, "Part material equals `C25/30`."
Finish	To filter objects based on how the part surface has been treated. For example, "Part finish equals `"FP - Fire proofing"`."
Prefix	To filter objects based on their numbering prefix. For example, "Part prefix equals `P`."

Property	Description
Start number	To filter objects based on their start number. For example, "Part start number greater than `100`."
Numbering series	To filter objects based on their numbering series information. For example, "Part numbering series equals `TP/1`." Note that position number separator can be a dot (.), comma (,), slash (/), or hyphen (-), depending on what you have defined in **File menu** --> **Settings** --> **Options** --> **Numbering**.
Position number	To filter objects based on their position number. For example, "Part position number does not equal `P/5`." Note that position number separator can be a dot (.), comma (,), slash (/), or hyphen (-), depending on what you have defined in **File menu** --> **Settings** --> **Options** --> **Numbering**.
Class	To filter objects based on their class number. For example, "Part class equals `210`."
Phase	To filter objects based on their phase number. For example, "Part phase equals `1 2`."
Lot	To filter objects based on their lot number. For example, "Part lot greater than `1`."
Primary part	To filter objects based on whether they are main or secondary parts in an assembly or cast unit. **1** = primary part, **0** = secondary part. For example, "Part primary part equals `1`."
Bind to levels	To filter parts based on whether or not they are bound to grid levels (page 350). **1** = Yes, the part is bound to a grid level, **0** = No, the part is not bound to any grid level.
Pour phase	To filter parts based on their pour phase. For example, "Pour phase does not equal `0`."

Category: Component

Use the **Component** category to filter components (page 803) based on their common properties.

Property	Description
Name	To filter components based on their name. For example, "Component name equals "`shear plate simple`"."
Connection code	To filter components based on their connection code, which can be either a text string or a number. For example, "Component connection code equals `200_2`."
Running number	To filter components based on their unique running number. For example, "Component running number less than `150`."
Phase	To filter components based on their phase number. For example, "Component phase equals `2`."
Is conceptual	To filter components based on their type. Components can be either detailed or conceptual. **Yes** = conceptual, **No** = detailed. For example, "Component is conceptual equals `Yes`."

Category: Bolt

Use the **Bolt** category to filter bolts (page 363) based on their common properties.

Property	Description
Size	To filter bolts based on their diameter. For example, "Bolt size less than `20.00`."
Standard	To filter bolts based on their bolt assembly standard/grade. For example, "Bolt standard equals `7990`."
Site/Workshop	To filter bolts based on their assembly type. **Site** = 0, **Workshop** = 1. For example, "Bolt Site/Workshop equals `1`."

Property	Description
Phase	To filter bolts based on their phase number. For example, "Bolt phase equals `3 4`."
Length	To filter bolts based on their length. For example, "Bolt length greater than `50.00`."
Hole 1 type ... **Hole 5 type**	To filter bolts based on their hole types in the bolted parts 1...5. The options are **Slotted**, **Oversized**, **Tapped**, **No hole**, and **Regular**.

Category: Weld

Use the **Weld** property to filter welds (page 381) based on their common properties.

Property	Description
Size above line **Size below line**	To filter welds based on their size. For example, "Weld size above line equals `5.00`."
Reference text	To filter welds based on their reference text, which is a user-definable value in the **Weld** properties. For example, "Weld reference text contains `12345`."
Phase	To filter welds based on their phase number. For example, "Weld phase equals `3`."
Type above line **Type below line**	To filter welds based on their weld type (page 389). Select the type from the **Value** list.
Length above line **Length below line**	To filter welds based on their length value. For example, "Weld length greater than `0.00`."
Welding site	To filter welds based on where they should be made. The options are **Site** and **Workshop**.
Position number	To filter welds based on their unique position number. For example, "Weld position number is greater than `100`."
Angle above line **Angle below line**	To filter welds based on the angle of weld preparation, bevels, or groove. For example, "Weld angle below line greater than `0.000`."

Property	Description
Contour above line **Contour below line**	To filter welds based on their fill type contour. The options are **None**, **Flush**, **Convex**, and **Concave**. For example, "Weld contour above line does not equal `None`."
Effective throat above line **Effective throat below line**	To filter welds based on their weld size used in weld strength calculation. For example, "Weld effective throat above line equals `0.500`."
Finish above line **Finish below line**	To filter welds based on how they have been treated. The options are **None**, **Grind**, **Machine**, **Chip**, **Finished weld**, and **Smooth transition**.
Increment amount above line **Increment amount below line**	To filter welds based on their amount of increments. For example, "Weld increment amount above line greater than `0`."
Intermittent type	To filter welds based on their shape. The options are **Continuous**, **Intermittent**, and **Staggered intermittent**.
Pitch above line **Pitch below line**	To filter welds based on their spacing of weld increments.
Root face thickness above line **Root face thickness below line**	To filter welds based on their root face thickness, in other words, the height of the narrowest part inside the root opening.
Root opening above line **Root opening below line**	To filter welds based on the space found between the welded parts.
Size prefix above line **Size prefix below line**	To filter welds based on their weld size prefix. For example, "Weld size prefix above line equals `a`." The standard ISO 2553 prefixes are `a` (Design throat thickness), `s` (Penetration throat thickness), and `z` (Leg length).
User defined cross-section	To filter welds based on whether or not they contain user-defined cross-sections. The options are **Yes** and **No**.
Electrode classification	To filter welds based on their weld electrode classification. The options

Property	Description
	are (empty), **35**, **52**, **50**, **E60XX**, **E70XX**, **E80XX**, and **E90XX**.
Electrode strength	To filter welds based on their electrode strength. For example, "Weld electrode strength greater than `0.000`."
Electrode coefficient	To filter welds based on their electrode coefficient.
Process type	To filter welds based on their welding process type. The options are **SMAW**, **SAW**, **GMAW**, **FCAW**, **ESW**, and **EGW**.
NDT inspection	To filter welds based on their non-destructive testing and inspection level. The options are **A**, **B**, **C**, **D**, and **E**.
Is around weld	To filter welds based on whether only one edge or the entire perimeter of a face is welded. **No** = edge, **Yes** = around.

Category: Reinforcing bar

Use the **Reinforcing bar** category to filter reinforcing bars and other reinforcement objects (page 493) based on their common properties.

Property	Description
Name	To filter reinforcing bars based on their name. For example, "Reinforcing bar name equals `"STIRRUP"`."
Class	To filter reinforcing bars based on their class number. For example, "Reinforcing bar class equals `3`."
Size	To filter reinforcing bars based on their size. The size property depends on the environment and may contain letters and special characters. For example, in the US imperial environment, "Reinforcing bar size equals `#18`."
Diameter	To filter reinforcing bars based on their diameter. Diameter is the nominal diameter of the bar, not actual. For example, "Reinforcing bar diameter less than `12`."

Property	Description
Length	To filter reinforcing bars based on their total length. For example, "Reinforcing bar length greater than `5000.00`."
Material	To filter reinforcing bars based on their material grade. For example, "Reinforcing bar material does not equal `Undefined`."
Prefix	To filter reinforcing bars based on their numbering prefix. For example, "Reinforcing bar prefix equals `R`."
Start number	To filter reinforcing bars based on their start number. For example, "Reinforcing bar start number greater than `1`."
Numbering series	To filter reinforcing bars based on their numbering series information. For example, "Reinforcing bar numbering series equals `R/1`."
Position number	To filter reinforcing bars based on their position number. For example, "Reinforcing bar position number equals `R/3`."
Phase	To filter reinforcing bars based on their phase number. For example, "Reinforcing bar phase equals `2`."
Shape	To filter reinforcing bars based on their bending shape (page 617). For example, "Reinforcing bar shape does not equal `2_1`."
Joint type	
Bar positions	
Layer	To filter rebar set bars based on their bar layer information. For example, "Rebar set bar layer equals `B3`" (the third layer of bottom bars).
Frozen	To filter rebar sets based on whether or not they are frozen (page 571). **0** = Not frozen, **1** = Frozen, **2** = Partially frozen (some leg faces are frozen, some are not).

Category: Surface

Use the **Surface** category to filter surfaces (page 437) based on their common properties.

Property	Description
Name	To filter surfaces based on their name. For example, "Surface name equals `SURFACE`."
Type	To filter surfaces based on their type. The options are **Formwork** and **Concrete finish**.
Class	To filter surfaces based on their class number. For example, "Surface class does not equal `13`."
Phase	To filter surfaces based on their phase number. For example, "Surface phase equals `3 4`."

Category: Assembly

Use the **Assembly** category to filter assemblies, cast units (page 454), and rebar assemblies (page 643) based on their common properties.

Property	Description
Name	To filter assemblies and cast units based on their name. For example, "Assembly name does not contain `RAFTER`."
GUID	To filter assemblies based on their globally unique identifier (GUID). For example, "Assembly GUID equals `ID89F414A7-ECA6-4B14-99CB-6985B84E64CB`."
Prefix	To filter assemblies and cast units based on their numbering prefix. For example, "Assembly prefix equals `A`."
Start number	To filter assemblies and cast units based on their start number. For example, "Assembly start number greater than `1`."
Position number	To filter assemblies and cast units based on their position number. For example, "Assembly position number equals `A/13`."
Phase	To filter assemblies and cast units based on their phase number. For

Property	Description
	example, "Assembly phase does not equal `1`."
Assembly level	To filter assemblies and cast units based on their position in the assembly hierarchy. The larger the value, the lower the position in the assembly hierarchy. 0 is the top-most level, and 1 is the first sub-assembly level. For example, to check if the model contains sub-assemblies, use the filter rule "Assembly level greater than or equal to `1`."
Assembly type	To filter assemblies and cast units based on their type. • 0 = precast • 1 = cast in place • 2 = steel • 3 = timber • 4 = rebar • 6 = miscellaneous
Assembly series	To filter assemblies and cast units based on their numbering series information. For example, "Assembly series equals `C/1`."

Category: Construction object

Use the **Construction object** category to filter construction objects based on their common properties.

Property	Description
Phase	To filter construction objects based on their phase number. For example, "Construction object phase does not equal `1`."
Type	To filter construction objects based on their type. The options are **Line**, **Arc**, **Circle**, **Plane**, and **Polycurve**.

Category: Load

Use the **Load** category to filter loads based on their common properties.

Property	Description
Load group	To filter loads based on which load group they belong to. For example, "Load group does not equal `DefaultGroup`."
Load type	To filter loads based on their type. The options are **line**, **point**, **area**, **uniform**, and **temperature**. Note that wind loads are handled as area loads in filtering. Use the  **Select components** and  **Select objects in components** selection switches to select wind loads.
Phase	To filter loads based on their phase number. For example, "Load phase does not equal `1`."

Category: Template

Use the **Template** category to filter parts and other objects by using template attributes.

With this category, you can type the name of any template attribute or user-defined attribute directly in the **Property** box, even if it is not on the list. Use the `ASSEMBLY.`, `CAST_UNIT.`, or `POUR_UNIT.` prefix in front of the property name to access higher hierarchy level attributes, and the `USERDEFINED.` prefix to access user-defined attributes.

For example, to filter objects that are one hierarchy level below the cast-unit with the user-defined attribute **User field 1**, type `CAST_UNIT.USERDEFINED.USER_FIELD_1` in the **Property** box.

In some cases, you can filter objects based on properties of other objects from lower hierarchy levels. This is possible when there is only one lower level object to be accessed. For example, there is only one main part in each assembly or cast unit, so you can access the main part properties from the assembly or cast unit level using the `MAINPART` prefix. Similarly, there can only be one pour object in each pour unit, so you can access the pour object properties from the pour unit level using the `POUR_OBJECT` prefix.

For example, to filter objects in assemblies whose main part has a certain name, type `ASSEMBLY.MAINPART.NAME` in the **Property** box.

For example, to filter all reinforcing bars belonging to pour units with a certain pour object type, type `POUR_UNIT.POUR_OBJECT.POUR_TYPE` in the **Property** box.

TIP To check which unit Tekla Structures uses for a particular template attribute, use the **Select from model...** option in the **Value** list in the filtering dialog box.

Category: Reference assembly

Use the **Reference assembly** category to filter reference model assemblies based on their common properties.

Property	Description
Creation	
GUID	To filter reference model assemblies based on their globally unique identifier (GUID). For example, "Reference assembly GUID equals `IDA51E6BFF-DAB9-4A56-970C-7486EF17B7B7`."
Phase	To filter reference model assemblies based on their phase number. For example, "Reference assembly phase equals `2`."
Lot	To filter reference model assemblies based on their lot number. For example, "Reference assembly lot greater than `1`."
Description	To filter reference model assemblies based on their description, which is a user-definable value in the **Reference object** dialog box. For example, "Reference assembly description contains `"architect model"`."
Info text	To filter reference model assemblies based on their info text, which is a user-definable value in the **Reference object** dialog box. For example, "Reference assembly info text contains `revised`."
Locked	To filter reference model assemblies based on whether or not they are locked. **0** = No, **1** = Yes, **2** = Organization.
Logical name	To filter reference model assemblies based on their logical name,

Property	Description
	which is a user-definable value in the **Reference object** dialog box. For example, "Reference assembly logical name equals "`MEP heating system`"."

Category: Reference object

Use the **Reference object** category to filter reference model objects based on their common properties.

Property	Description
Creation	
GUID	To filter reference model objects based on their globally unique identifier (GUID).
Phase	To filter reference model objects based on their phase number. For example, "Reference object phase does not equal `1`."
Lot	To filter reference model objects based on their lot number. For example, "Reference object lot equals `1`."
Description	To filter reference model objects based on their description, which is a user-definable value in the **Reference object** dialog box. For example, "Reference object description contains "`architect model`"."
Info text	To filter reference model objects based on their info text, which is a user-definable value in the **Reference object** dialog box. For example, "Reference object info text contains `revised`."
Locked	To filter reference model objects based on whether or not they are locked. **0** = No, **1** = Yes, **2** = Organization.
Logical name	To filter reference model objects based on their logical name, which is a user-definable value in the **Reference object** dialog box. For

Property	Description
	example, "Reference object logical name contains "3rd floor"."

TIP You can filter reference model object attributes using the **Template** category and the EXTERNAL. prefix in the **Property** box. For example, "Reference object EXTERNAL.Material equals A572."

Category: Location breakdown structure

Use the **Location breakdown structure** category to filter objects based on their location categories defined in **Organizer** or based on their building hierarchy elements defined in **Building hierarchy**.

Building hierarchy is used when the XS_USE_INTEGRATED_BUILDING_HIERARCHIES advanced option is set to TRUE.

Property	Description
Site	To filter objects based on which site category they belong to. For example, "Location breakdown structure site equals "Site 2"."
Building	To filter objects based on which building category they belong to. For example, "Location breakdown structure building does not equal "Building A"."
Section	To filter objects based on which section category or building section they belong to. For example, "Location breakdown structure section equals Ramp."
Floor	To filter objects based on which floor or building storey they are located on. For example, "Location breakdown structure floor equals "Floor 4"."

Category: Pour object

Use the **Pour object** category to filter pour objects (page 475) based on their common properties.

Property	Description
Pour number	To filter pour objects based on their pour number. For example, "Pour number equals 5."

Property	Description
Pour type	To filter pour objects based on their type. For example, "Pour type equals `WALL`."
Concrete mixture	To filter pour objects based on the properties of their concrete mix, for example, maximum grain size of aggregate, and/or plasticity of fresh concrete.
Material	To filter pour objects based on their material grade. For example, "Material equals `C35/45`."
Pour phase	To filter pour objects based on their pour phase. For example, "Pour phase does not equal `0`."

Category: Pour break

Use the **Pour break** category to filter pour breaks (page 482) based on their common properties.

Property	Description
Creation	
ID number	To filter pour breaks based on their ID number. For example, "ID number equals `25237`."
Phase	To filter pour breaks based on their phase. For example, "Pour break phase equals `2 3`."
Pour break type	To filter pour breaks based on their type. For example, "Pour break type equals `"Watertight construction joint"`."

Category: Pour unit

Use the **Pour unit** category to filter pour units based on their common properties.

Property	Description
Name	To filter pour units based on their name. For example, "Pour unit name contains `beam`."
GUID	To filter pour units based on their globally unique identifier (GUID). For

Property	Description
	example, "Pour unit GUID contains `8505`."

Category: Task

Use the **Task** category to filter scheduled tasks based on their common properties.

Property	Description
Name	To filter scheduled tasks based on their name. For example, "Task name contains `floors`."
Planned start date	To filter scheduled tasks based on their planned start date. For example, "Task planned start date earlier than `Review date`."
Planned end date	To filter scheduled tasks based on their planned end date. For example, "Task planned end date later than or equal to `10/13/2026`."
Actual start date	To filter scheduled tasks based on their actual start date.
Actual end date	To filter scheduled tasks based on their actual end date.
Completeness	To filter scheduled tasks based on their completeness. The value is a percentage. For example, "Task completeness is `75`."
Critical	To filter scheduled tasks based on how critical they are. A task can be critical only if it has been imported from external software. **1** = Critical, **0** = Not critical. Note that this property is not visible in **Task manager**.
Local	To filter scheduled tasks based on whether they were created in **Task manager** or imported from external software. **1** = Created in Task Manager, **0** = Imported.
Contractor	To filter scheduled tasks based on the contractor. For example, "Task contractor equals `"Contractor A"`."

Property	Description
Scenario	To filter scheduled tasks based on the scenario they belong to. For example, "Task scenario equals "`Scenario 1`"."
Task type	To filter scheduled tasks based on their type. For example, "Task type does not equal "`A - Floor tiling`"."

Filtering techniques

By using conditions, brackets, and the **And/Or** options you can create filters that can be as complex as needed.

Conditions

Use conditions to define how filter criteria relate to one another. Note that when you create filters (page 172), you are always defining what should be **shown** (or be selectable) in the model or drawing. Therefore, if you enter "Component name does not contain `gusset`", you are telling Tekla Structures to show all components whose name **does not contain** the word "gusset". Tekla Structures then hides all components which have the word "gusset" in their name.

Condition	Description
Equals	Use this condition when the filter value must be matched exactly. For example, "Part name equals `BEAM`."
Does not equal	Filters out objects that contain the value you enter. For example, "Part profile does not equal `BL200*20`" means that Tekla Structures will hide (or not select) the objects whose profile is BL200*20. The rest of the objects will be shown (or selected).
Begins with	Finds all objects that start with the value you enter. For example, "Component name begins with `purlin`."
Does not begin with	Filters out objects that start with the value you enter. For example, "Component name does not begin with `shear`" means that Tekla Structures will hide (or not select) the

Condition	Description
	objects whose name begins with the word "shear". The rest of the objects will be shown (or selected).
Ends with	Finds all objects that end with the value you enter. For example, "Component name ends with `plate`."
Does not end with	Filters out objects that end with the value you enter. For example, "Component name does not end with `angle`" means that Tekla Structures will hide (or not select) the objects whose name ends with the word "angle". The rest of the objects will be shown or selected.
Contains	Finds all objects that include the value you enter. For example, "Component name contains `plate`" would find `base plate` and `shear plate simple`.
Does not contain	Filters out objects that include the value you enter. For example, "Component name does not contain `gusset`" means that Tekla Structures will hide (or not select) the objects whose name contains the word "gusset". The rest of the objects will be shown or selected.
Greater than	Finds all objects that exceed the value you enter. For example, "Template attribute `LENGTH` is greater than `5000`." This property can only be used with numerical data, such as the part start number, class, phase, or `LENGTH`.
Greater or equal	Finds all objects that match or exceed the value you enter. This property can only be used with numerical data, such as the part start number, class, phase, or `LENGTH`.

Condition	Description
Less than	Finds all objects that are less than the value you enter. This property can only be used with numerical data, such as the part start number, class, phase, or `LENGTH`.
Less or equal	Finds all objects that match or are less than the value you enter. This property can only be used with numerical data, such as the part start number, class, phase, or `LENGTH`.
Later than	Available for dates only. The date must be later than the one you have defined. For example, "Object's approval date later than `4/10/2017`."
Later than or equal	Available for dates only. The date must be later than or equal to the one you have defined.
Earlier than	Available for dates only. The date must be earlier than the one you have defined. For example, "Object's approval date earlier than `2/18/2017`."
Earlier than or equal	Available for dates only. The date must be earlier than or equal to the one you have defined.

And/Or options

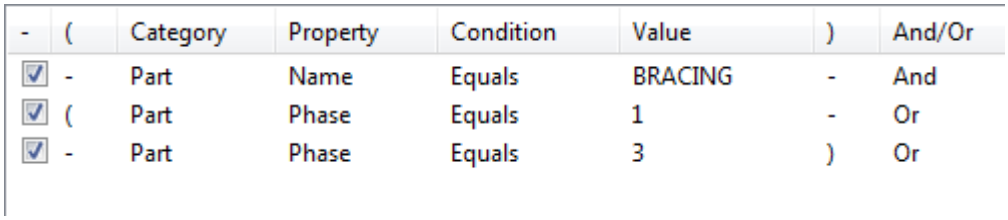
Use the **And/Or** options when you create filter rules that consist of several rows.

Option	Description
And	Use to find objects that match both values. When you create filter rules between objects whose **Category** settings differ, use the **And** option when possible to avoid potential problems with more complex rules.
Or	Use to find objects that match either value.
empty (= And)	Empty means the same as **And**.

Brackets

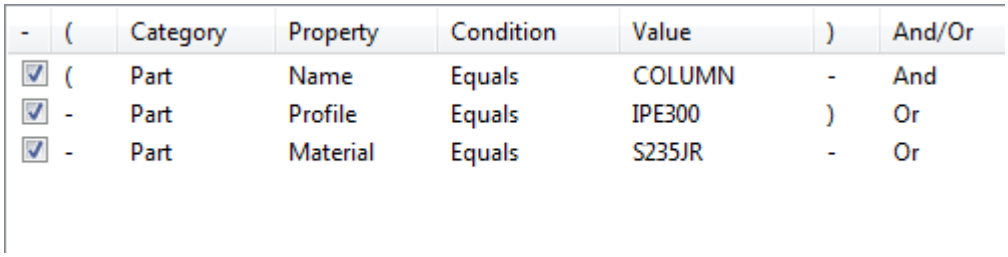
You can use single, double, and triple brackets to create more complex filter rules.

Example 1. Use the format "A and (B or C)" to find objects that match the first filter rule and **either** of the last two rules.



-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Part	Name	Equals	BRACING	-	And
☑	(	Part	Phase	Equals	1	-	Or
☑	-	Part	Phase	Equals	3	)	Or

Example 2. Use the format "(A and B) or C" to find objects that match both of the first two rules **or** the third.



-	(	Category	Property	Condition	Value	)	And/Or
☑	(	Part	Name	Equals	COLUMN	-	And
☑	-	Part	Profile	Equals	IPE300	)	Or
☑	-	Part	Material	Equals	S235JR	-	Or

Select values from the model

You can select object properties and dates directly from the model. This can be useful when creating view filters, selection filters, and object groups.

Before you start, create an empty view or selection filter, or an object group.

1. Create an empty view or selection filter (page 172), or an object group (page 692).
2. Click **Add row**.
3. Select options from the **Category** and **Property** lists.
4. In the **Value** list, select one of the options.

   The availability of options depends on your selection in the **Property** list. You can select dates from the model only if the property is a date.

   a. To select an object property, click **Select from model...** and then select an object.

   b. To select a date, click **Select date...** to open the **Select Date** dialog, and then select one of the options.

      You can either select a date from the calendar, select the review date, or define the number of days before or after the review date.

The review date is the same as **Review date** in the **Project Status Visualization** dialog box.

Wildcards

A wildcard is a symbol that stands for one or more characters. You can use wildcards to shorten strings, for example in filtering.

Wildcard	Description	Example
* (asterisk)	Matches any number of characters	`HE*` matches all parts with a profile name that begins with the characters "HE". You can also this symbol at the beginning of a word: `*BRAC*`.
? (question mark)	Matches a single character	`HE?400` matches parts with profile names such as HEA400, HEB400, and HEC400.
[ ] (square brackets)	Matches whatever is enclosed in the brackets	`L[78]X4X1/2` matches parts with the profile names L7X4X1/2 and L8X4X1/2.

NOTE The characters * and ? may also be used in object names in Tekla Structures. If the object name you want to filter contains * or ?, you need to enclose these characters in square brackets. For example, to find the profile P100*10, enter `P100[*]10` in the filter field.

See also

Filtering techniques (page 198)

Filter objects (page 168)

Examples of filters

Here are some examples of filters that you can create. The same filtering techniques can be used for view, selection, and drawing filters.

Filter parts based on their name

Create a filter that only shows parts with a certain name.

1. Create a new view filter. (page 172)
2. Click **Add row** three times to add three filter rules.
3. In the first filter rule, define that the object type must be part:
   a. In the **Category** list, select **Object**.

b. In the **Property** list, select **Object type**.

c. In the **Condition** list, select **Equals**.

d. In the **Value** list, select **Part**.

e. In the **And/Or** list, select **And**.

4. In the second and third filter rules, define that the part name must be BEAM or COLUMN:

a. In the **Category** list, select **Part**.

b. In the **Property** list, select **Name**.

c. In the **Condition** list, select **Equals**.

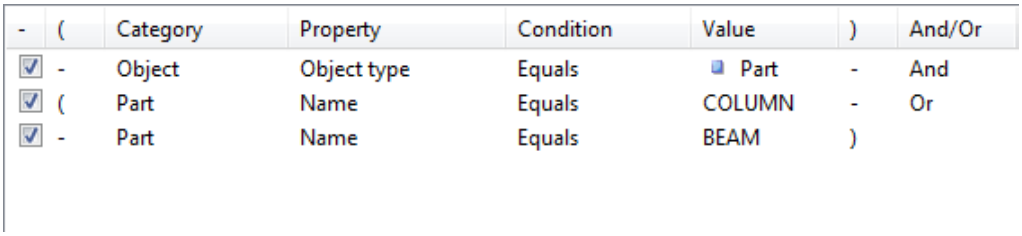
d. In the **Value** box, enter the part names, `BEAM` and `COLUMN`.

e. In the **And/Or** list, select **Or**.

5. Include the second and third filter rule in brackets. The filter is now looking for parts that are named either BEAM or COLUMN.

6. Enter a unique name in the box next to the **Save as** button.

7. Click **Save as**.



-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Object	Object type	Equals	Part	-	And
☑	(	Part	Name	Equals	COLUMN	-	Or
☑	-	Part	Name	Equals	BEAM	)	

Filter main parts

Create a filter that selects the main parts only.

1. Create a new selection filter. (page 172)

2. Click **Add row** twice to add two filter rules.

3. In the first filter rule, define that the object type must be part:

a. In the **Category** list, select **Object**.

b. In the **Property** list, select **Object type**.

c. In the **Condition** list, select **Equals**.

d. In the **Value** list, select **Part**.

e. In the **And/Or** list, select **And**.

4. In the second filter rule, define that you only want to include main parts:

a. In the **Category** list, select **Part**.

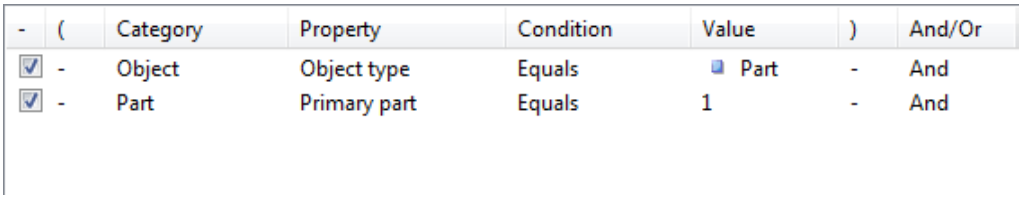
b. In the **Property** list, select **Primary part**.

c. In the **Condition** list, select **Equals**.

d. In the **Value** box, enter `1`.

In this context, `1` means the main parts, and `0` would mean the secondary parts.

5. Enter a unique name in the box next to the **Save as** button.
6. Click **Save as**.



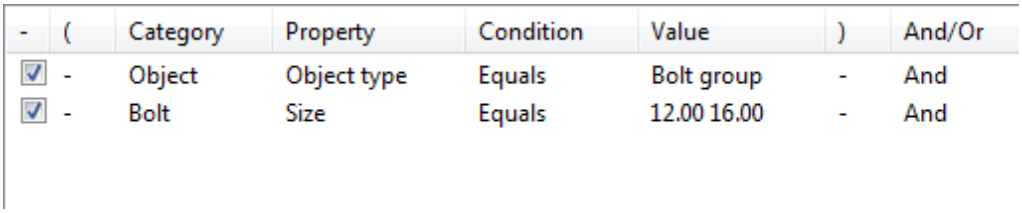
-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Object	Object type	Equals	Part	-	And
☑	-	Part	Primary part	Equals	1	-	And

Filter bolts based on their size

Create a filter that only shows bolts of certain sizes.

1. Create a new view filter. (page 172)
2. Click **Add row** twice to add two filter rules.
3. In the first filter rule, define that the object type must be bolt:
   a. In the **Category** list, select **Object**.
   b. In the **Property** list, select **Object type**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** list, select **Bolt group**.
   e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that the bolt size must be 12.00 or 16.00:
   a. In the **Category** list, select **Bolt**.
   b. In the **Property** list, select **Size**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** box, enter the bolt sizes, `12.00` and `16.00`.

   Separate the strings with a blank space.
5. Enter a unique name in the box next to the **Save as** button.
6. Click **Save as**.



-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Object	Object type	Equals	Bolt group	-	And
☑	-	Bolt	Size	Equals	12.00 16.00	-	And

Filter parts based on their assembly type

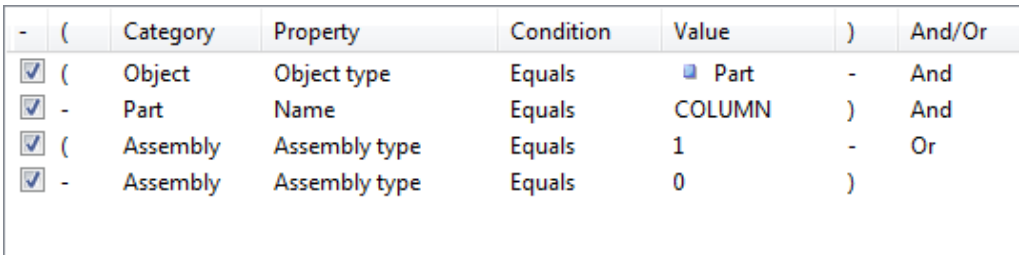
Create a filter based on assembly types. For example, you can create a filter that only shows cast-in-place and precast columns. Steel columns and any other columns or parts are hidden. The same filtering technique can be used for steel, concrete, timber, and miscellaneous parts.

1. Create a new filter. (page 172)
2. Click **Add row** four times to add four filter rules.
3. In the first filter rule, define that the object type must be part:
   a. In the **Category** list, select **Object**.
   b. In the **Property** list, select **Object type**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** list, select **Part**.
   e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that the part name must be COLUMN:
   a. In the **Category** list, select **Part**.
   b. In the **Property** list, select **Name**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** box, enter the part name, `COLUMN`.
   e. In the **And/Or** list, select **And**.
5. Include the first and second filter rule in brackets.
6. In the third and fourth filter rules, define that the assembly type must be precast or cast in place:
   a. In the **Category** list, select **Assembly**.
   b. In the **Property** list, select **Assembly type**.
   c. In the **Value** box, enter the assembly types, `0` and `1`.

Value	Assembly type
0	precast
1	cast in place
2	steel
3	timber
4	rebar
6	miscellaneous

   d. In the **And/Or** list, select **Or**.
7. Include the third and fourth filter rule in brackets. The filter is now looking for concrete parts that are named COLUMN.

8. Enter a unique name in the box next to the **Save as** button.
9. Click **Save as**.



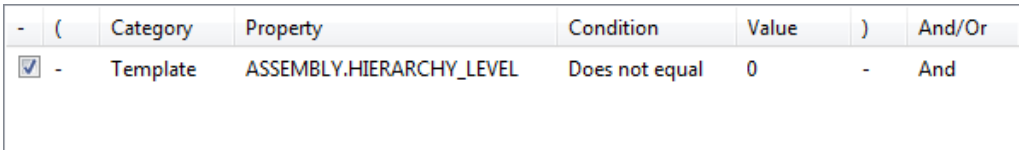
-	(	Category	Property	Condition	Value	)	And/Or
☑	(	Object	Object type	Equals	Part	-	And
☑	-	Part	Name	Equals	COLUMN	)	And
☑	(	Assembly	Assembly type	Equals	1	-	Or
☑	-	Assembly	Assembly type	Equals	0	)	

Filter sub-assemblies

Create a filter that only selects parts that belong to a sub-assembly.

1. Create a selection filter. (page 172)
2. Click **Add row** to add a new filter rule.
3. In the **Category** list, select **Template**.
4. In the **Property** list, select `ASSEMBLY.HIERARCHY_LEVEL`.
5. In the **Condition** list, select **Does not equal**.
6. In the **Value** box, enter `0`.

   In this context, `0` means that the part does not belong to any sub-assembly, and `1` would mean that the part does. The filter will show only those parts whose value is **not** `0`.
7. Enter a unique name in the box next to the **Save as** button.
8. Click **Save as**.



-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Template	ASSEMBLY.HIERARCHY_LEVEL	Does not equal	0	-	And

Filter objects based on their class

Create a filter based on object types and class. The following example filter can be used to select or show parts and reinforcement in certain classes.

1. Create a new view filter. (page 172)
2. Click **Add row** three times to add three filter rules.
3. In the first and second filter rules, define that the object type must be part or reinforcement.
   a. In the **Category** list, select **Object**.
   b. In the **Property** list, select **Object type**.

c. In the **Condition** list, select **Equals**.

d. In the **Value** list, select **Part** for the first rule and **Reinforcing bar** for the second rule.

e. In the **And/Or** list, select **Or** for the first rule and **And** for the second rule.

4. Include the first and second filter rule in brackets.
5. In the third filter rule, use the template attribute `CLASS_ATTR` to define that the object class must be less than or equal to 5:

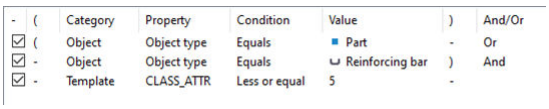
   a. In the **Category** list, select **Template**.

   b. In the **Property** list, type `CLASS_ATTR`.

   c. In the **Condition** list, select **Less or equal**.

   d. In the **Value** list, enter `5`.

   The filter is now looking for parts and reinforcement that belong to classes 0–5.
6. Enter a unique name in the box next to the **Save as** button.
7. Click **Save as**.



-	(	Category	Property	Condition	Value	)	And/Or
☑	(	Object	Object type	Equals	Part	-	Or
☑	-	Object	Object type	Equals	Reinforcing bar	)	And
☑	-	Template	CLASS_ATTR	Less or equal	5	-	

Filter reference model objects

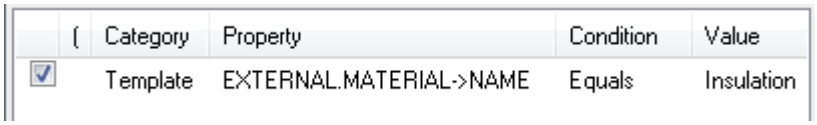
Create a filter based on reference model object properties.

1. Create an empty view or selection filter. (page 172)
2. Click **Add row** to add a new filter rule.
3. In the **Category** list, select **Template**.
4. In the **Property** list, select the desired template attribute or enter one of your own (page 181).

   TIP To find the attribute name used by the reference model, select a reference model object, right-click and select one of the **Inquire** commands. Find the property name in the **Inquire object** dialog and copy it.

5. Add the prefix `EXTERNAL.` in front of the template attribute name.
6. In the **Condition** list, select **Equals**.
7. In the **Value** box, enter the desired value or click **Select from model...** to select the object in the model.
8. Enter a unique name in the box next to the **Save as** button.

9. Click **Save as**.

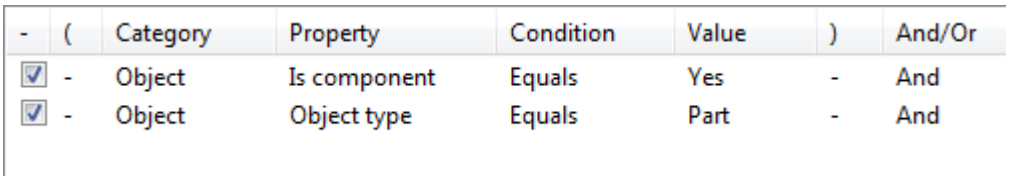


	(	Category	Property	Condition	Value
☑		Template	EXTERNAL.MATERIAL->NAME	Equals	Insulation

Filter parts within component

Create a filter that selects all parts within a component.

1. Create an empty selection filter. (page 172)
2. Click **Add row** twice to add two filter rules.
3. In the first filter rule, define that the object must be a component:
   a. In the **Category** list, select **Object**.
   b. In the **Property** list, select **Is component**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** list, select **Yes**.
   e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that the object type must be part:
   a. In the **Category** list, select **Object**.
   b. In the **Property** list, select **Object type**.
   c. In the **Condition** list, select **Equals**.
   d. In the **Value** list, select **Part**.
5. Enter a unique name in the box next to the **Save as** button.
6. Click **Save as**.



-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Object	Is component	Equals	Yes	-	And
☑	-	Object	Object type	Equals	Part	-	And

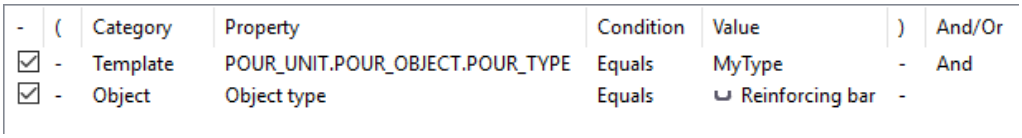
Filter reinforcement in pour units based on pour object type

Create a filter that only shows all reinforcement that belongs to pour units with a pour object of a certain type.

1. Ensure that you have XS_ENABLE_POUR_MANAGEMENT set to `TRUE`.
2. Calculate the pour units. (page 477)
3. Create a new view filter. (page 172)
4. Click **Add row** twice to add two filter rules.
5. In the first filter rule, define the pour object type.

a. In the **Category** list, select **Template**.
b. In the **Property** box, enter `POUR_UNIT.POUR_OBJECT.POUR_TYPE`.
c. In the **Condition** list, select **Equals**.
d. In the **Value** box, enter the pour object type, for example `MyType`, or click **Select from model...** to select the object in the model.
e. In the **And/Or** list, select **And**.

6. In the second filter rule, define that the object type must be reinforcement:
a. In the **Category** list, select **Object**.
b. In the **Property** list, select **Object type**.
c. In the **Condition** list, select **Equals**.
d. In the **Value** list, select **Reinforcing bar**.
7. Enter a unique name in the box next to the **Save as** button.
8. Click **Save as**.

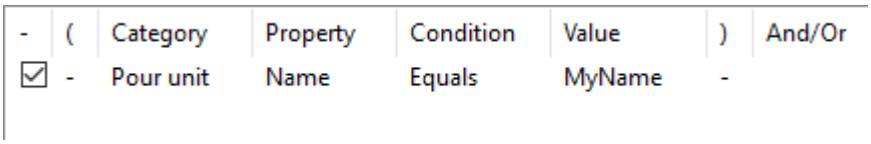


-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Template	POUR_UNIT.POUR_OBJECT.POUR_TYPE	Equals	MyType	-	And
☑	-	Object	Object type	Equals	⌣ Reinforcing bar	-	

Filter all contents of a pour unit

Create a filter that selects all contents of a pour unit with a certain name.

1. Ensure that you have XS_ENABLE_POUR_MANAGEMENT set to `TRUE`.
2. Calculate the pour units. (page 477)
3. Create a selection filter. (page 172)
4. Click **Add row** to add a new filter rule.
5. In the **Category** list, select **Pour unit**.
6. In the **Property** list, select **Name**.
7. In the **Condition** list, select **Equals**.
8. In the **Value** box, enter the pour unit name, for example `MyName`.
9. Enter a unique name in the box next to the **Save as** button.
10. Click **Save as**.



-	(	Category	Property	Condition	Value	)	And/Or
☑	-	Pour unit	Name	Equals	MyName	-	

Copy and remove filters

You can copy custom filters to another model by manually copying filter files to the `attributes` folder under the desired model folder. You can also manually remove unneeded filters from the same folder. If you want to make a filter available in all models, copy the file to your company's project or firm folder.

Copy a filter to another model

1. Select the filter you want to copy.

   The filters you have created are located in the `attributes` folder under the current model folder. You can recognize different filter types based on their file name extension:

File name extension	Filter type
`.VObjGrp`	Model view filter
`.SObjGrp`	Model selection filter
`.PObjGrp`	Object group filter
`.vf`	Drawing view filter
`.vnf`	Drawing view level neighbor part filter
`.wdf`	Single-part drawing filter
`.wdnf`	Single-part drawing neighbor part filter
`.adf`	Assembly drawing filter
`.adnf`	Assembly drawing neighbor part filter
`.cuf`	Cast unit drawing filter
`.cunf`	Cast unit drawing neighbor part filter
`.gdf`	General arrangement drawing filter
`.gdnf`	General arrangement drawing neighbor part filter
`.dsf`	Drawing selection filter

2. To make the filter available in another model, copy the file to the `attributes` folder of the destination model folder.
3. To make the filter available in all models, copy the file to your company's project or firm folder.
4. Restart Tekla Structures.

Remove a filter

1. Remove the filter file from the model's `attributes` folder.

2. Restart Tekla Structures.

1.10 Create model templates

Model templates allow you to start a model with predefined company templates and settings. This can be especially useful for sub-contractors.

Only single-user models can be created with model templates. If you wish to create a multi-user model using a model template, create the model in single-user mode and then switch to multi-user mode.

By default, the model template folder is saved in your environment folder. Use the advanced option XS_MODEL_TEMPLATE_DIRECTORY to define a different location.

NOTE Do not use the Tekla Structures Partner or the Tekla Structures Educational subscriptions to create model templates.

As a result, the template model does not work as expected and permanent watermarks are printed on drawings. The same will happen if you later use a commercial subscription to create a new model using a template created with the Tekla Structures Partner or the Tekla Structures Educational subscriptions.

Create a new model template

You can create your own model templates and use them for creating new models. You can select which catalogs, custom components, model sub-folders, drawing templates, and report templates from the model are included in the model template.

1. Create a new model.

   Always start by creating a new empty model. This is because old models that have been used in live projects cannot be completely cleaned. They may contain excess information that increases the size of the model even if you delete all objects and drawings from the model.

2. Add the desired part properties, drawing properties, profiles, materials, custom components, sketches, and so on, in the model.

   You can copy the needed attribute files from another model, for example.

3. On the **File** menu, click **Save as** --> **Save**.

   You need to save the model to include custom components in the `xslib.db1` file. If you do not save the model, custom components will not be included in the model template.

4. On the **File** menu, click **Save as** --> **Save as model template**.

5. Enter a name for the model template.

6. Select which catalogs, drawing templates, report templates, and model subfolders to include in the model template.

   You can only select files and folders that are available in the model folder. Catalogs are typically located in an `environment` folder and they are included in the model folder only if they have been modified.

7. If you want to open the destination folder after creating the model template, select the checkbox.
8. Click **OK**.

   You can now use the model template for creating new models.

9. When you create new models in **File** --> **New**, you can mark the important model templates as favorites, or hide the templates that you do not need.

   a. Select a model template in the list.

   b. Right-click and select **Favorite** or **Hidden**.

      If you marked a template as **Favorite**, it is placed on top of the template list. Alternatively, use the star icon on the template to mark it as **Favorite**, or to remove the marking.

      If you marked a template as **Hidden**, it is removed from the template list. Select the **Show hidden items** checkbox to show it again.

Modify an existing model template

To modify an existing template, save the model as a new template. Alternatively, you can modify the template by copying new or updated files directly to the model template folder.

1. Create a model using the existing model template.
2. Make the needed changes.
3. Save it as a new model template.

Download model templates

You can download, share, and store model templates using Tekla Warehouse.

Model template options

Use the **Save as model template** dialog to define which files and folders are included in the model template.

Setting	Files and folders included
Profiles	`profdb.bin` `profitab.inp`
Materials	`matdb.bin`

Components and sketches	`ComponentCatalog.txt`
	`ComponentCatalogTreeView.txt`
	`xslib.db1`
	`thumbnail_bitmap.arc`
	`*.dat` files
	`CustomComponentDialogFiles` folder

Attribute definitions	Includes all attribute definitions of the current model.
Bolts and bolt assemblies	`screwdb.db` `assdb.db`

Reinforcement	`rebar_database.inp`
	`RebarShapeRules.xml`
	`rebardatabase_config.inp`
	`rebardatabase_schedule_config.inp`

Meshes	`mesh_database.inp`
Options	Includes all options of the current model.
Drawing templates	`*.tpl` files
Report templates	`*.rpt` files
Include model subfolders	Lists all subfolders found in the model folder. The selected folders are included in the model template. The `attributes` folder, which contains part and drawing properties, and the `CustomComponentDialogFiles` folder are included by default.

1.11 Tips for large models

Modeling item	Tips
Coordinate system (page 54)	• Do not place the model far away from the origin. The further away from the origin you model, the less precise all the computations become. • Mark global coordinates as labels instead of actually using them during modeling. • If you need to use building site coordinates, drop the first digits if they are always the same. For example, instead of coordinate 758 375 6800, use 375 6800. • Base points allow you to use another coordinate system needed for interoperability and collaboration. You can use another coordinate system for inserting reference models and exporting IFC models. When you use base points, you can keep the coordinates small and locate the model wherever needed. You can create as many base points as you need, and select one of those to be the project base point. For more information, see Base points (page 60).
Work area (page 52) and visibility	• Keep the work area as small as possible. • Show only the required parts in views. • Use view filters to control the visibility of parts.
Views (page 31)	• Close unnecessary views. • Close all views when you save large models.
Selection switches	• Switch on the **Select reference models** selection switch only when necessary. The speed of zooming and rotating may be affected by this switch especially in

Modeling item	Tips
	large and complicated models that contain reference models.
Round objects	• Create holes with the **Create bolts** command instead of using part cuts with round beams. • Use studs to model small straight round objects instead of small round beams. • Model lifting hooks and other embeds with reinforcement bars instead of round polybeams.
Hollow core profiles	• Use simple fixed (non-parametric) profiles. • Use chamfers for curved corners.
Custom components (page 863)	• Do not create overly complex custom components. When used in great numbers they consume a lot of memory.
Numbering (page 752)	• Do not number the whole model in one go. Numbering all objects in large models may take a considerable amount of time.
Model database	• If your model file is getting large, repairing the model database can help to reduce the file size considerably and therefore help with memory problems.
Firm and Project folders	• Save `Firm` and `Project` folders locally on the hard drive of your computer instead of a network drive. This saves time if network speed is slow. When working in the multi-user mode, ensure that the folders are synchronized on all users' hard drives so that important data is not lost or changed.

1.12 How to model identical areas

Most structures contain identical areas, from simple frames to entire floors. Save time by modeling these areas once, then copying them throughout the model. For example, create a column with a base plate and cap plate, then copy the column to all the locations where it occurs in the model.

You can use this technique to create and reproduce any identical area. Depending on the project, you may even be able to add connections before copying the area of the building.

TIP For a project that has several identical floors, try modeling an entire floor, then copying it to several levels.

See also

Tips for copying and moving objects in models and in drawings (page 166)

2 Create parts, reinforcement, and construction objects

When you know the basic principles of how to create and modify different types of model objects in Tekla Structures, you can start working on more detailed level with your model.

First, to get your model started, you need to create some parts (page 217). Parts are the building blocks of the physical model. You can continue working with the parts by, for example, deforming their shape (page 332), or by adding some details to the parts (page 362), such as bolts, welds, or cuts or fittings.

When you use a workshop weld or bolt to join parts together, you will learn how to work with assemblies (page 447).

For concrete parts, each concrete part is considered a separate cast unit (page 454). For construction purposes, you may need to merge several concrete parts into one cast unit. If you model cast-in-place concrete structures, you may need to check how to work with pours (page 468). And once you have created a model of concrete parts, you will need to reinforce the parts (page 493) to gain higher strength for the parts.

Additionally, you may need to use points or construction objects (page 650) as modeling aids. Points and construction objects help you to place other objects in the model.

See also

Adjust how model objects are displayed (page 671)

Change the color and transparency of model objects by using object representation (page 686)

Check the model (page 697)

Number the model (page 752)

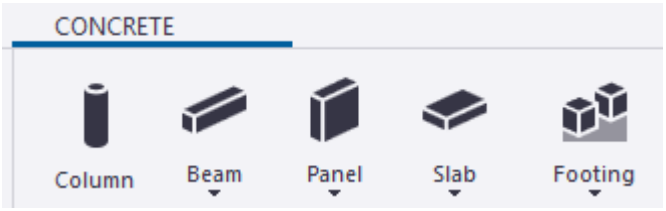
2.1 Create parts and modify part properties

In Tekla Structures, *part* is a basic building object that can be modeled and detailed further. Parts are the building blocks of the physical model.

You can create steel parts and concrete parts. A special type of parts are items, which are used to model objects that are difficult to create with basic Tekla Structures parts and commands, such as cutting.

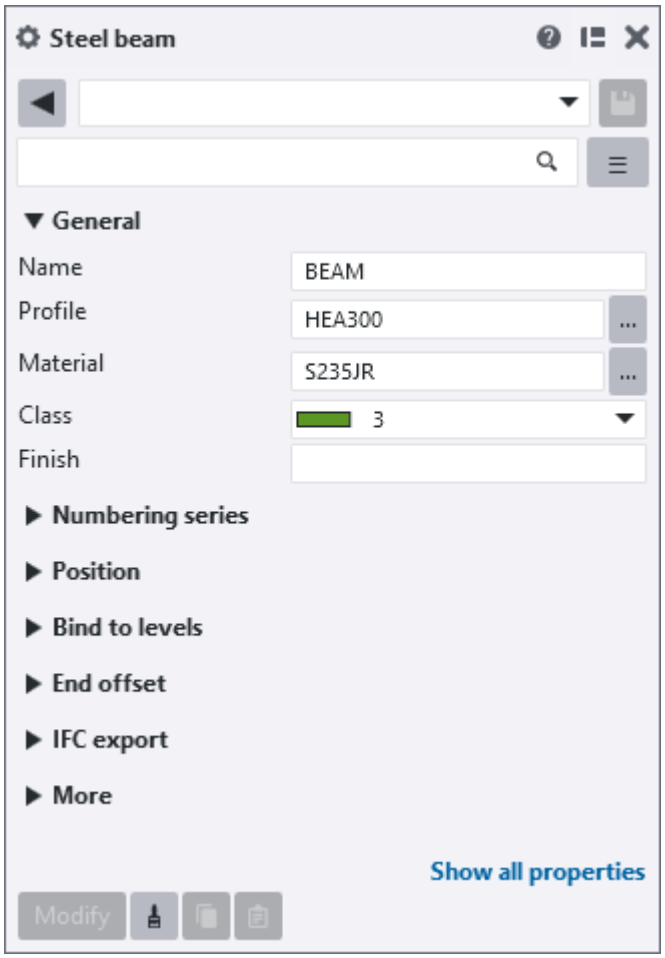
Create steel parts by using the commands on **Steel** tab on the ribbon. Create concrete parts by using the commands on the **Concrete** tab on the ribbon.





Each part has properties that define it, such as material, profile, and location. Also, parts have user-defined attributes that you can use to provide additional information about a part. You can use part properties in view filters and selection filters to, for example, select, modify, and hide parts. You can also include part properties and UDAs in drawing and report templates.

To view and modify the part properties, use the property pane. You can view and modify the properties of a one part type at a time, or the common properties of several, similar part types.



The basic steel parts are

- column (page 220)
- beam (page 223)
- polybeam (page 226)
- curved beam (page 229)
- twin profile (page 232)
- orthogonal beam (page 235)
- spiral beam (page 237)
- plate (page 241)
- bent plate (page 245)
- lofted plate (page 268)

The basic concrete parts are

- column (page 281)
- beam (page 284)

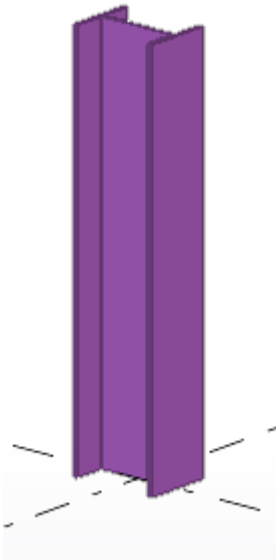
- polybeam (page 287)
- spiral beam (page 291)
- panel (page 295)
- slab (page 299)
- lofted slab (page 303)
- pad footing (page 314)
- strip footing (page 316)

Tools for timber detailing workflows are available in Tekla Warehouse, on the Timber tools page, as an extension package. For more information, see Create timber objects (page 331).

Create a steel column

1. On the **Steel** tab, click **Column** .
2. Pick a point.

   Tekla Structures creates the column using the **Steel column** properties in the property pane, and at the level defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.

2. In the property pane, click the **Object type list** button  and select **Steel column** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify steel column properties

1. If the property pane is not open, double-click the column to open the **Steel column** properties.
2. Change the properties as needed.
3. Click **Modify**.

Steel column properties

Use the **Steel column** properties in the property pane to view and modify the properties of a steel column. To open the properties, double-click the steel column. The file name extension of a steel column property file is `*.clm`.

If you have customized the property pane layout, the list of properties may be different.

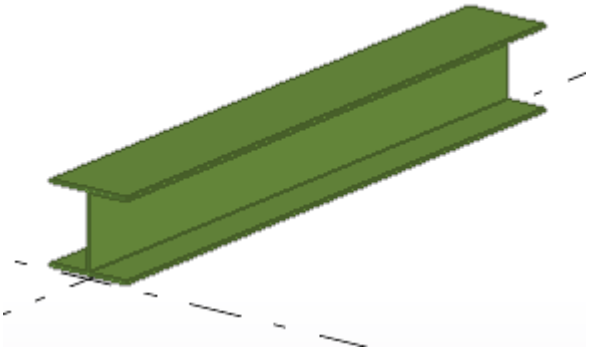
Setting	Description
General	
Name	User-definable name of the column. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the column.
Material	Material (page 334) of the column.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group columns. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).

Setting	Description
Assembly numbering	Assembly prefix and start number for the assembly position number (page 753).
Position	
Vertical	Vertical position (page 344) of the column, relative to column's reference point.
Rotation	Rotation (page 342) of the column around its axis on the work plane.
Horizontal	Horizontal position (page 346) of the column, relative to column's reference point.
Top	Position of the second end of the column in the global z direction.
Bottom	Position of the first end of the column in the global z direction.
Bind to levels	
Top level	Use to bind (page 350) the top face of the column to a selected grid level. Set the offset if needed.
Bottom level	Use to bind (page 350) the bottom face of the column to a selected grid level. Set the offset if needed.
Deforming	
Warping	Use to warp columns using deformation angles.
Cambering	Use to pre-camber (page 353) the column.
Shortening	Use to shorten the column in the model. The true length of the column is decreased in the drawing.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a steel beam

1. On the **Steel** tab, click .
2. Pick two points.

   Tekla Structures creates the beam between the points you picked, using the **Steel beam** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Steel beam** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify steel beam properties

1. If the property pane is not open, double-click the beam to open the **Steel beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Steel beam properties

Use the **Steel beam** properties in the property pane to view and modify the properties of a steel beam, steel polybeam, or curved beam. To open the properties, double-click the steel beam. The file name extension of a beam property file is `*.prt`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the beam.
Material	Material (page 334) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 753).
Position	
On plane	Beam's position on the work plane (page 340), relative to the beam's reference line.
Rotation	Rotation (page 342) of the beam around its axis on the work plane.
At depth	Position depth (page 343) of the beam. The position is always perpendicular to the work plane.
Bind to levels	
Start level	Use to bind (page 350) the start point of the beam to a selected grid level. Set the offset if needed.
End level	Use to bind (page 350) the end point of the beam to a selected grid level. Set the offset if needed.
End offset	

Setting	Description
Dx	Change the length of the beam (page 348) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 348) perpendicular to the beam's reference line.
Dz	Move the beam end (page 348) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 353) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
Subtype (IFC4)	
User-defined type (IFC4)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

If beams are created close to each other

When you create beams so that they are located very close to each other, Tekla Structures may consider them as a twin profile. To prevent this from happening, use the user attribute `MAX_TWIN_SEARCH_DIST` in the profile catalog.

1. On the **File** menu, click **Catalogs** --> **Profile catalog** to open the **Modify Profile Catalog** dialog box.

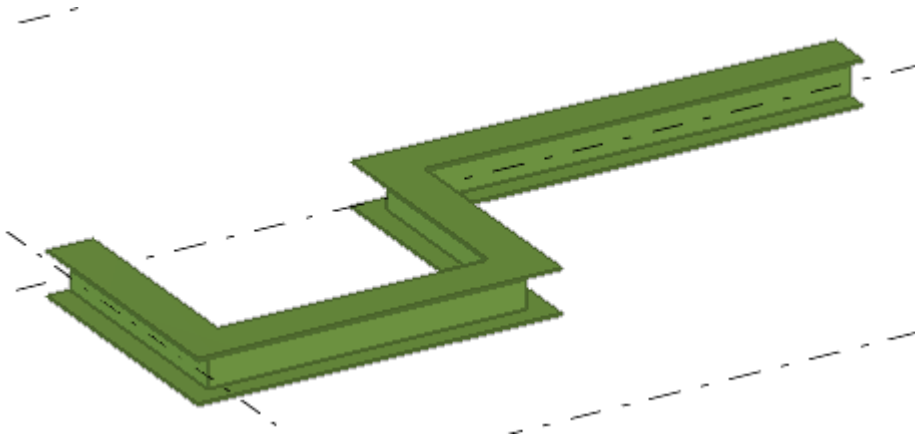
2. Select the desired profile in the profile tree.
3. Go to the **User attributes** tab and set the property **Twin profile detection distance** to a larger value than `0`, for example to `0.1`.
4. Click **OK**.
5. Create the beams using the profile.

Create a steel polybeam

A polybeam can contain straight and curved segments.

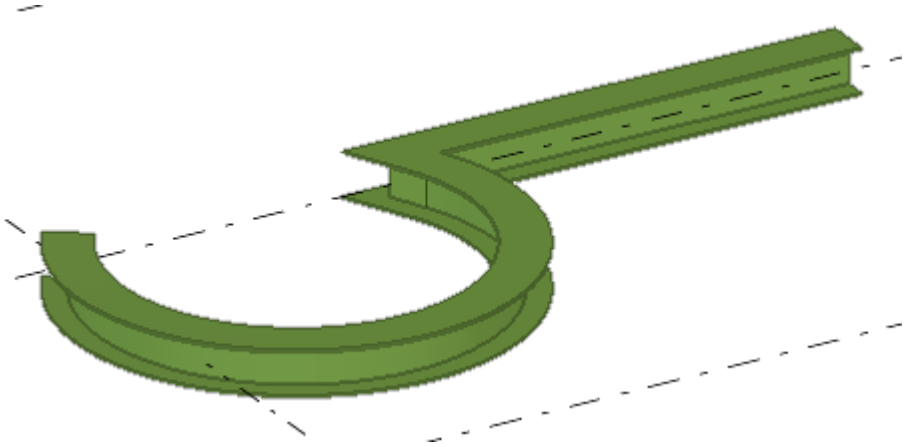
1. On the **Steel** tab, click **Beam** --> **Polybeam**.
2. Pick the points you want the beam to go through.
3. Click the middle mouse button.

   Tekla Structures creates the polybeam between the points you picked, using the **Steel beam** properties in the property pane. Note that you cannot create a closed polybeam.



4. If you want to create curved segments, chamfer the corners of the polybeam.

For example:



Modify steel polybeam properties

1. If the property pane is not open, double-click the polybeam to open the **Steel beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Steel beam properties

Use the **Steel beam** properties in the property pane to view and modify the properties of a steel beam, steel polybeam, or curved beam. To open the properties, double-click the steel beam. The file name extension of a beam property file is `*.prt`.

If you have customized the property pane layout, the list of properties may be different.

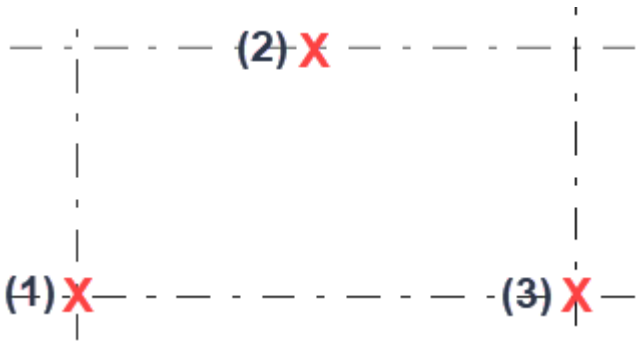
Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the beam.
Material	Material (page 334) of the beam.

Setting	Description
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 753).
Position	
On plane	Beam's position on the work plane (page 340), relative to the beam's reference line.
Rotation	Rotation (page 342) of the beam around its axis on the work plane.
At depth	Position depth (page 343) of the beam. The position is always perpendicular to the work plane.
Bind to levels	
Start level	Use to bind (page 350) the start point of the beam to a selected grid level. Set the offset if needed.
End level	Use to bind (page 350) the end point of the beam to a selected grid level. Set the offset if needed.
End offset	
Dx	Change the length of the beam (page 348) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 348) perpendicular to the beam's reference line.
Dz	Move the beam end (page 348) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.

Setting	Description
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 353) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
Subtype (IFC4)	
User-defined type (IFC4)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a curved steel beam

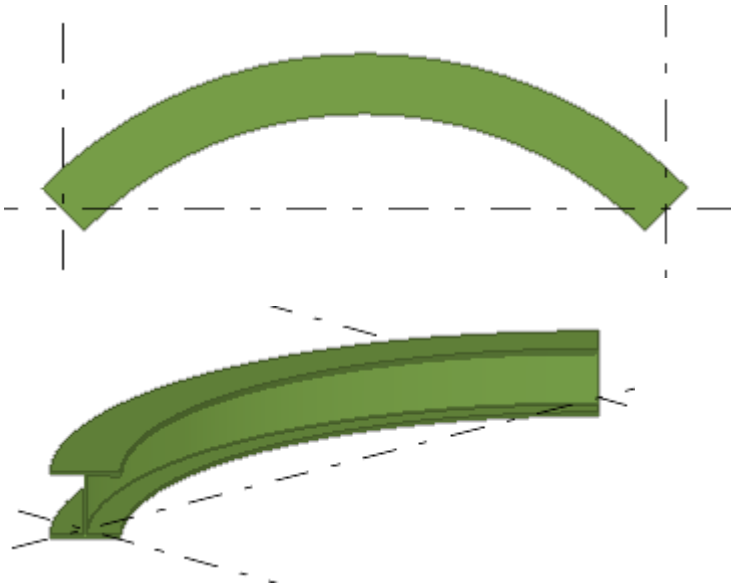
1. On the **Steel** tab, click **Beam** --> **Curved beam**.
2. Pick the start point (1).



3. Pick a point on the arc (2).
4. Pick the end point (3).

   Tekla Structures creates the beam between the points you picked, using the **Steel beam** properties in the property pane.

   The radius is defined by the points you pick.



Modify curved beam properties

1. If the property pane is not open, double-click the curved beam to open the **Steel beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Steel beam properties

Use the **Steel beam** properties in the property pane to view and modify the properties of a steel beam, steel polybeam, or curved beam. To open the properties, double-click the steel beam. The file name extension of a beam property file is `*.prt`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the beam.
Material	Material (page 334) of the beam.

Setting	Description
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 753).
Position	
On plane	Beam's position on the work plane (page 340), relative to the beam's reference line.
Rotation	Rotation (page 342) of the beam around its axis on the work plane.
At depth	Position depth (page 343) of the beam. The position is always perpendicular to the work plane.
Bind to levels	
Start level	Use to bind (page 350) the start point of the beam to a selected grid level. Set the offset if needed.
End level	Use to bind (page 350) the end point of the beam to a selected grid level. Set the offset if needed.
End offset	
Dx	Change the length of the beam (page 348) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 348) perpendicular to the beam's reference line.
Dz	Move the beam end (page 348) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.

Setting	Description
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 353) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
Subtype (IFC4)	
User-defined type (IFC4)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

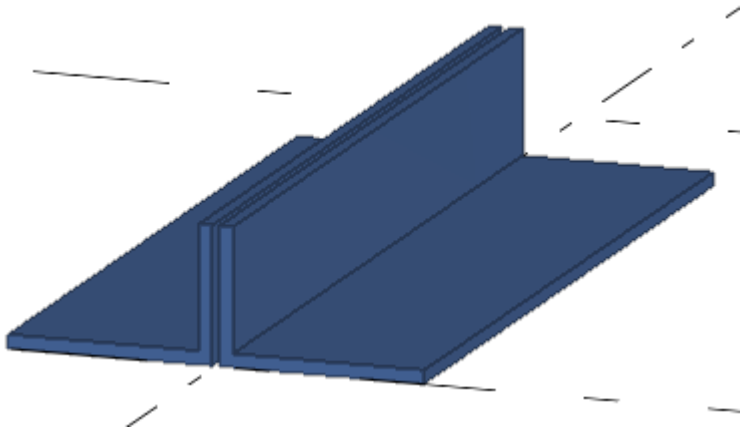
Create a twin profile

A twin profile consists of two parallel and identical beams. You define the positions of both beams by selecting the twin profile type and setting the clearance between the beams in two directions.

1. On the **Steel** tab, click **Beam** --> **Twin profile**.

2. Pick two points.

   Tekla Structures creates the twin profile between the points you picked, using the **Twin profile** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button and select **Twin profile** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify twin profile properties

1. If the property pane is not open, double-click either of the beams to open the **Steel beam** properties.
2. Change the properties as needed.
3. Click **Modify** .

Twin profile properties

Use the **Twin profile** properties in the property pane to view and modify the properties of a twin steel profile. The file name extension of a twin profile property file is `*.dia`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of a twin profile. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	The profile of both the beams in the twin profile.
Material	Material (page 334) of the beams.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group twin profiles. For example, you can display parts of different classes in different colors.
Twin profile type	Define how the profiles are combined.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number the assembly position number (page 753).
Position	
On plane	Twin profile's position on the work plane (page 340), relative to the twin profile's reference line.
Rotation	Rotation (page 342) of the twin profile around its axis on the work plane.
At depth	Position depth (page 343) of the twin profile. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the twin profile (page 348) by moving the twin profile end point along the twin profile's reference line.
Clearance between members	
Horizontal	The horizontal clearance between the profiles.
Vertical	The vertical clearance between the profiles.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend

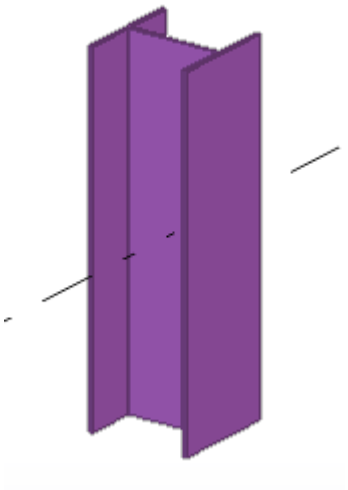
Setting	Description
Subtype (IFC4)	on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
User-defined type (IFC4)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create an orthogonal beam

Use the **Orthogonal beam** command when you want to create a steel part that is perpendicular to the current work plane. After you have created an orthogonal beam, you can modify it as if it were beam or column.

1. On the **Steel** tab, click **Beam** --> **Orthogonal beam**.
2. Pick a point.

   Tekla Structures creates the beam using the **Orthogonal beam** properties in the property pane, and at the level (page 349) defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.

2. In the property pane, click the **Object type list** button  and select **Orthogonal beam** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify orthogonal beam properties

1. If the property pane is not open, double-click the orthogonal beam to open the properties.
2. Change the properties as needed.
3. Click **Modify**.

Orthogonal beam properties

Use the **Orthogonal beam** properties in the property pane to view and modify the properties of an orthogonal steel beam. The file name extension of an orthogonal beam property file is `*.crs`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the beam.
Material	Material (page 334) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
Vertical	Vertical position (page 344) of the beam, relative to beam's reference point.

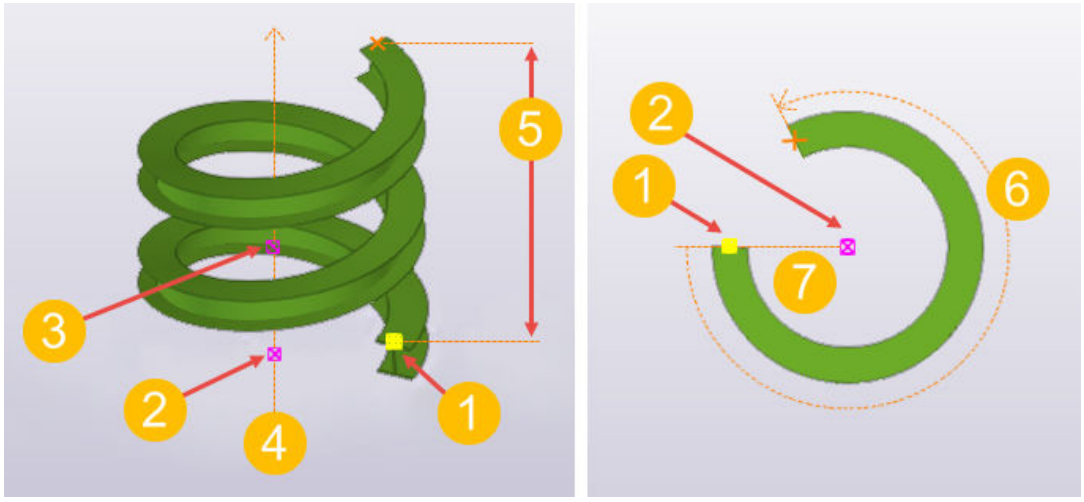
Setting	Description
Rotation	Rotation (page 342) of the beam around its axis on the work plane.
Horizontal	Horizontal position (page 346) of the beam, relative to beam's reference point.
Top	Position of the second end of the beam in the global z direction.
Bottom	Position of the first end of the beam in the global z direction.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number the assembly position number (page 753).
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
Subtype (IFC4)	
User-defined type (IFC4)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a steel spiral beam

Use the **Create steel spiral beam** command when you want to model spiral staircases and complex architectural shapes, for example.

Basic concepts related to spiral beams

The images below illustrate some basic concepts related to the creation of spiral beams. Note that if you change the positioning, the entire geometry of the spiral beam changes.



(1) Start point (the first point picked)

(2) Center point (the second point picked)

(3) Direction of the rotation axis (the optional third point picked)

(4) Center axis

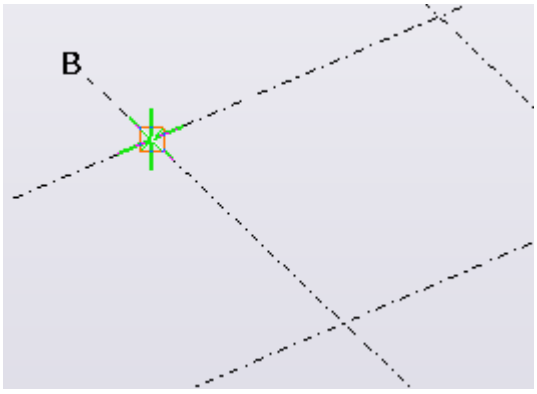
(5) Total rise: the distance from the start point to the end point, parallel to the center axis

(6) Rotation angle: the rotation angle of the spiral beam, given in degrees. Note: positive value = counter-clockwise rotation, negative value = clockwise rotation.

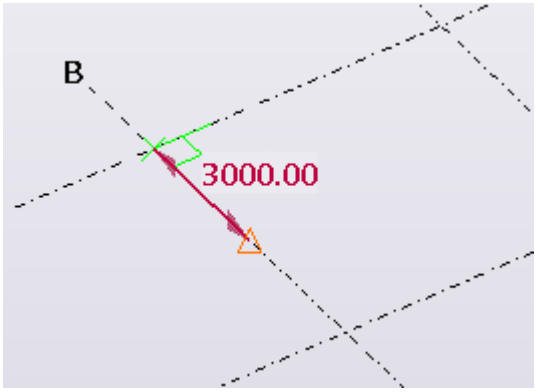
(7) Radius: the distance from the start point to the center point, perpendicular to the center axis

Create a spiral beam

1. On the **Steel** tab, click **Beam** --> **Spiral beam**.
2. Pick the start point.



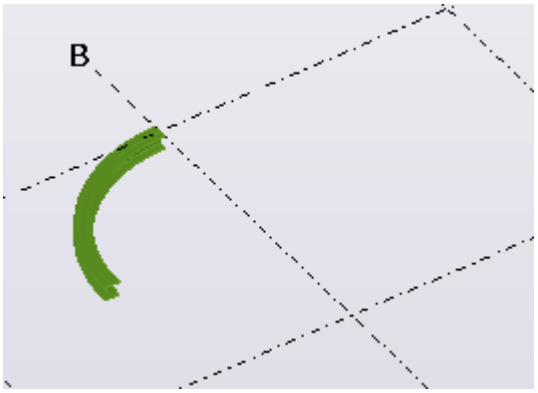
3. Pick the center point.



4. To set the rotation axis in the work plane +Z direction, click the middle mouse button to finish.

NOTE Alternatively, instead of clicking the middle mouse button, you can pick a second center axis point to define the direction of the rotation axis.

Tekla Structures creates the spiral beam. For example:



5. Click the spiral beam to select it.

   The contextual toolbar appears with the following options:



   (1) Rotation angle

   (2) Total rise

   (3) Twist angle at start

   (4) Twist angle at end

6. To add more rotation, enter a bigger value in the **Rotation angle** box.

7. To make the spiral more loose, enter a bigger value in the **Total rise** box.
8. To change the radius, move the start point or center point.

Limitations

- Spiral beam has a single, constant radius.
- Unfolding of spiral beams whose total rise is larger than 0.00 does not produce completely straight results in drawings. The amount of deviation in the part profile outlines and part length depends on several factors: the type, size, and length of the profile; the amount of total rise; and the amount of rotation angle and detailing used.
- Spiral beams are not always untwisted in unfolding. If unequal twisting is applied to the start end and end end, the unfolded drawing shows an unfolded but twisted part as a result.
- Connections and details may not work as expected with spiral beams.
- DSTV export of spiral beams may not produce a correct result.
- You cannot export spiral beams as parts in IFC export. If you are modeling cast-in-place structures with spiral beam, you can export the geometry to IFC as pour objects.

Steel spiral beam properties

Use the **Steel spiral beam** properties in the property pane to view and modify the properties of a steel spiral beam. To open the properties, double-click the spiral beam. The file name extension of a steel spiral beam property file is `*.sb`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the beam.
Material	Material (page 334) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.

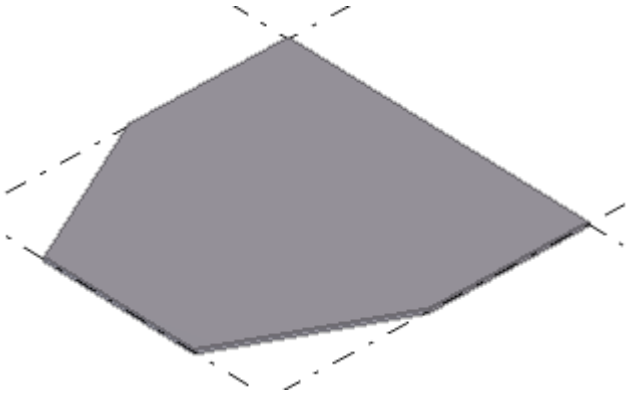
Setting	Description
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 753).
Position	
On plane	Beam's position on the work plane (page 340), relative to the beam's reference line.
Rotation	Rotation (page 342) of the beam around its axis on the work plane.
At depth	Position depth (page 343) of the beam. The position is always perpendicular to the work plane.
Geometry	
Rotation angle	Rotation angle of the spiral beam, given in degrees.
Total rise	Distance from the start point to the end point, parallel to the center axis.
Twist angle at start **Twist angle at end**	Twist +/- angle of the spiral beam at the start/end of the beam.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a contour plate

When you create a contour plate, the profile you select defines the thickness of the plate and the points you pick define the shape. The corners of the contour plate can be chamfered.

1. On the **Steel** tab, click .
2. Pick the corner points of the contour plate.
3. Click the middle mouse button.

   Tekla Structures creates the plate using the **Contour plate** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button and select **Contour plate** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Create a round contour plate

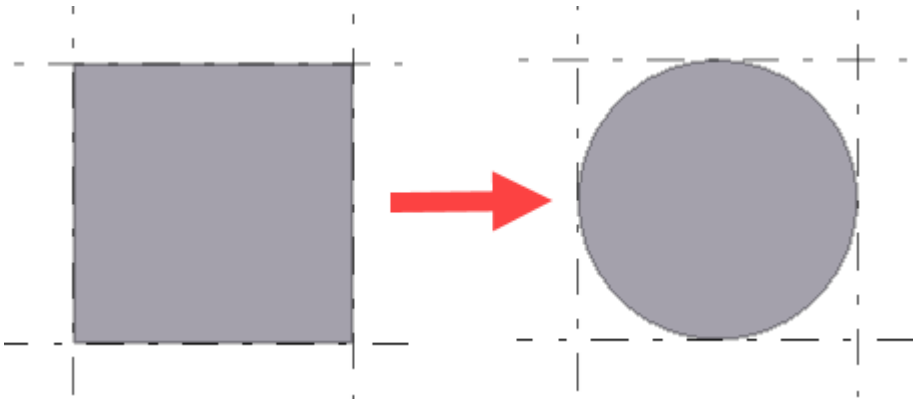
1. Create a square contour plate with four equal sides.
2. Select the plate.
3. Double-click a handle.

   To make it easier to select the handles of the contour plate corners, ensure that the Direct modification switch is **not** active.

   The **Contour plate corner chamfer** properties open in the property pane.
4. In the **Type** list, select **Rounding** .

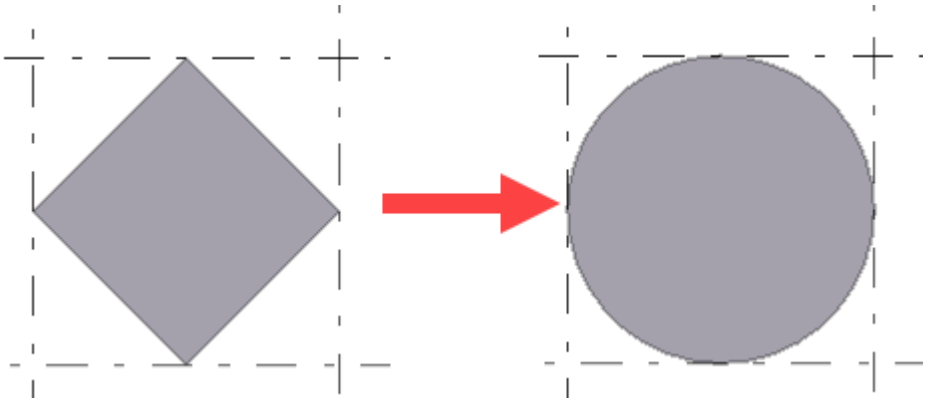
5. In the **Radius** box, enter the chamfer radius.

   The radius must be equal to half of the side of the square.
6. Click **Modify**.
7. Repeat the above steps for each corner you want to chamfer.



Alternative way of creating a round plate

1. Create a diamond-shaped plate with four equal sides.
2. To round the corners, use the **Arc point** chamfer type.



Modify contour plate properties

1. If the property pane is not open, double-click the plate to open the **Contour plate** properties.
2. Change the properties as needed.
3. Click **Modify**.

Contour plate properties

Use the **Contour plate** properties in the property pane to view and modify the properties of a contour plate. To open the properties, double-click the contour plate. The file name extension of a contour plate property file is `*.cpl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the contour plate. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the contour plate.
Material	Material (page 334) of the contour plate.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group contour plates. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number the assembly position number (page 753).
Position	
At depth	Position depth (page 343) of the contour plate. The position is always perpendicular to the work plane.
Bind to levels	
Level	Use to bind (page 350) the top face of the contour plate to a selected grid level. Set the offset if needed.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select
Subtype (IFC4)	
User-defined type (IFC4)	

Setting	Description
	USERDEFINED and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a conical or a cylindrical bent plate

You can create cylindrical or conical bent steel plates either by selecting two parts or two part faces. The parts that you use for creating a bent plate must be contour plates, or beams whose profile is a plate (for example, PL200*20). Position the parts so that there is some room on both sides, so that Tekla Structures can create a curved section between them.

After creating a cylindrical or conical bent plate, the individual parts no longer exist in the model. The bent plate gets its properties and coordinates from the first part that you selected when creating the bent plate. The first part is the main section of the bent plate. You can later change the main section, if needed.

Limitations

- Only the side faces of the part can be used for creating a bent plate.
- Chamfered or cut faces cannot be used for creating a bent plate.
- Curved beams and deformed parts cannot be used for creating a bent plate.
- Details (such as bolts, welds, cuts, chamfers, and preparations) in simple cases are supported on the curved section of the bent plate.

In addition to the cylindrical and conical bent plates, you can also create stand-alone bent plates (page 261) which do not require any input parts.

Create a cylindrical bent plate

You can create a cylindrical bent plate by selecting two steel parts or two part faces. Cylindrical bent plate has a radius that you can modify. The bent plate properties, such as the ID, thickness, class, and material of the plate, are determined by the first part you select.

You can create cylindrical bent plates also when the selected parts are intersecting.

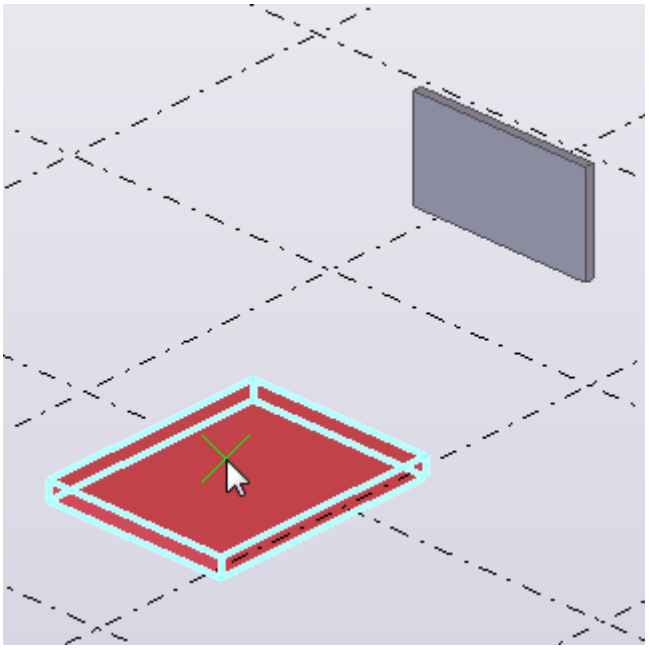
1. On the **Steel** tab, click **Plate** --> **Create cylindrical bent plate** .

2. On the bent plate toolbar, select whether you want to create the bent plate by selecting parts or by selecting part faces.

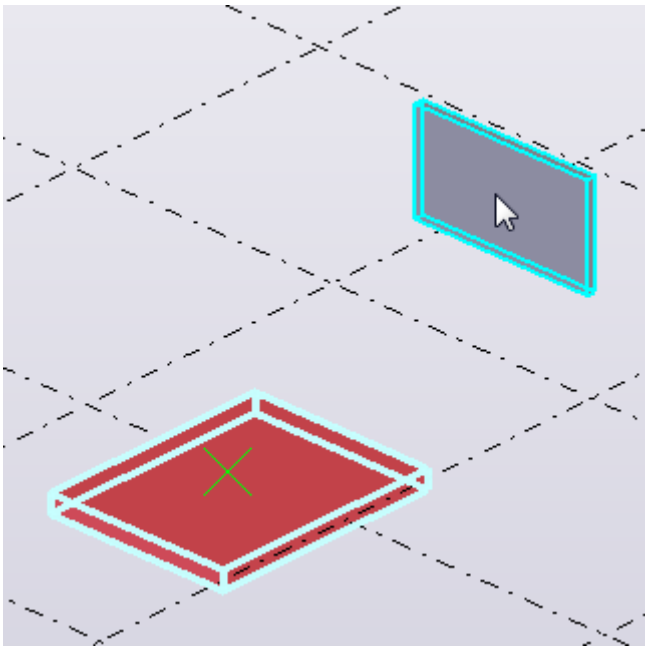
   Additionally, you can enter the radius of the cylindrical bent plate. If no radius is entered, Tekla Structures creates the bent plate using the default radius.

   - If you selected **By parts**:

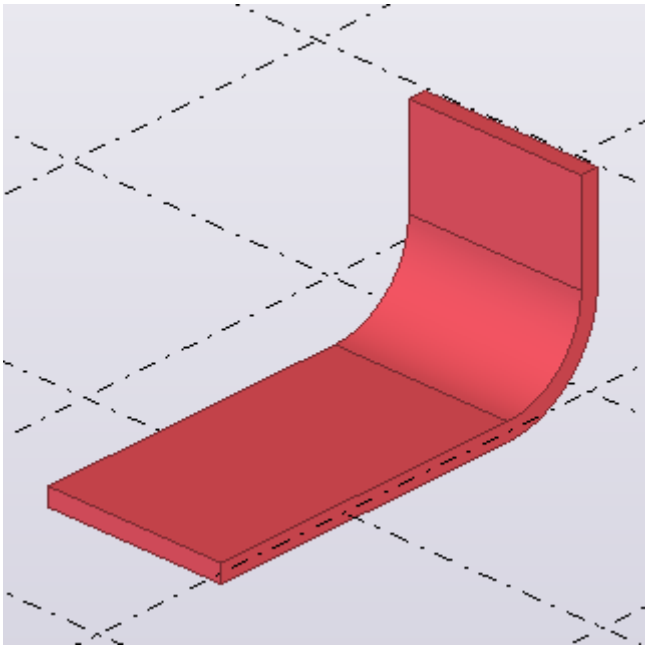
     a. Select the first part.



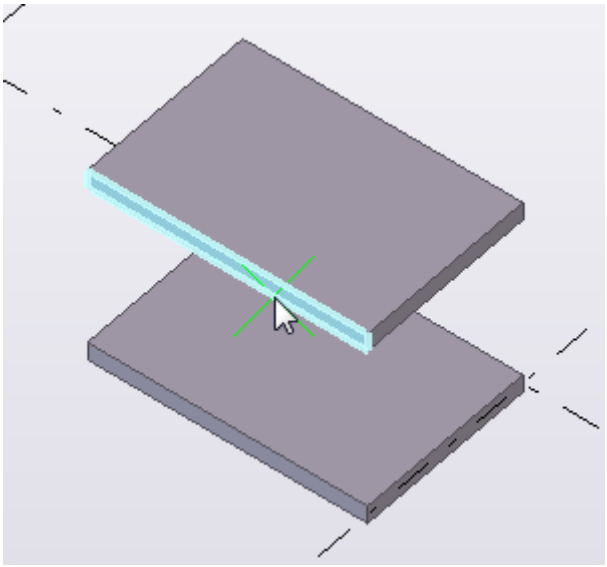
     b. Select the second part.



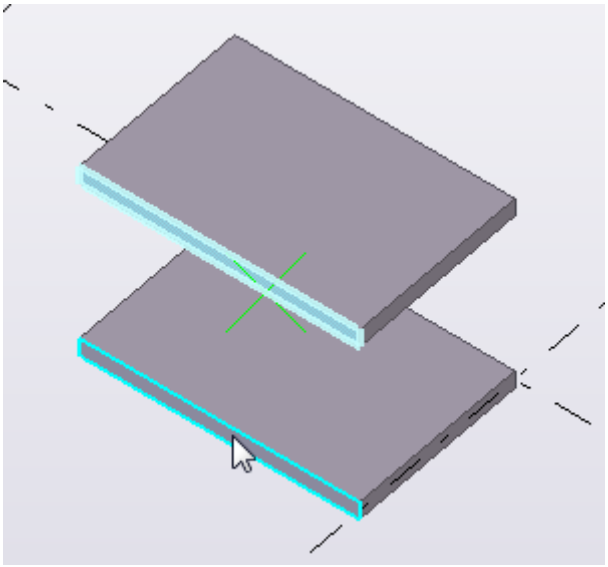
Tekla Structures creates the cylindrical bent plate.



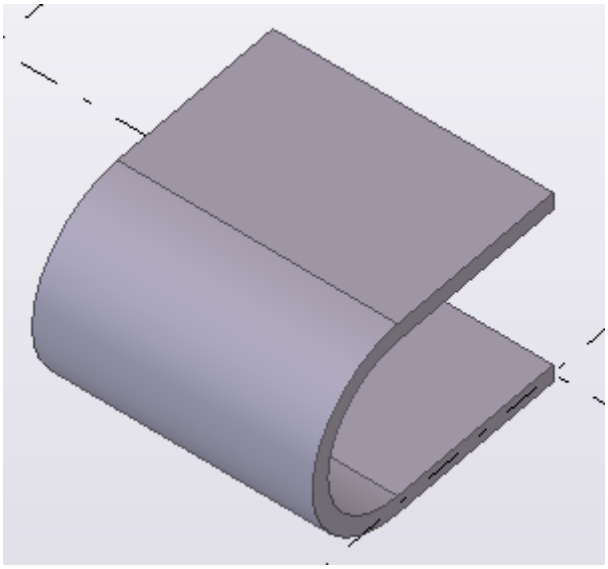
- If you selected **By faces**:
  a. Select the first part face.



b. Select the second part face.



Tekla Structures creates the cylindrical bent plate.



Create a conical bent plate

You can create a conical bent plate by selecting two steel parts or two part faces. Conical bent plate has two radiuses that you can modify. The bent plate properties, such as the ID, thickness, class, and material of the plate, are determined by the first part you select.

To be able to create conical bent plates, the selected parts or part faces need to be of a shape that creating a conical bent plate is possible. If the selected parts or part faces are such that creating a cylindrical bent plate is possible, a

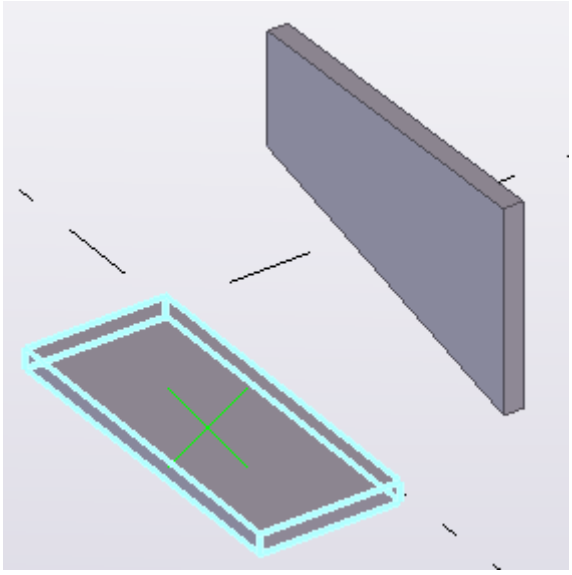
cylindrical bent plate is created. You can create different shapes of conical bent plates: inward bend, outward bend, or with 180 degrees opening angle.

1. On the **Steel** tab, click **Plate** --> **Create conical bent plate** .
2. On the bent plate toolbar, select whether you want to create the bent plate by selecting parts or by selecting part faces.

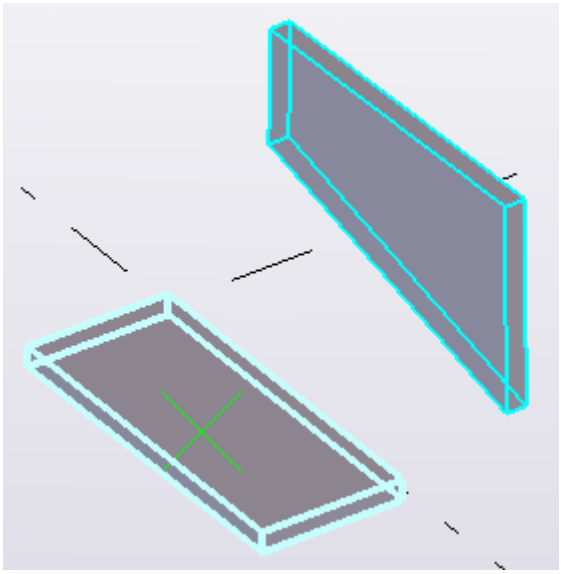
   You can enter two radiuses for the conical bent plate. If no radiuses are entered, Tekla Structures creates the bent plate using the default radiuses.

   - If you selected **By parts**:

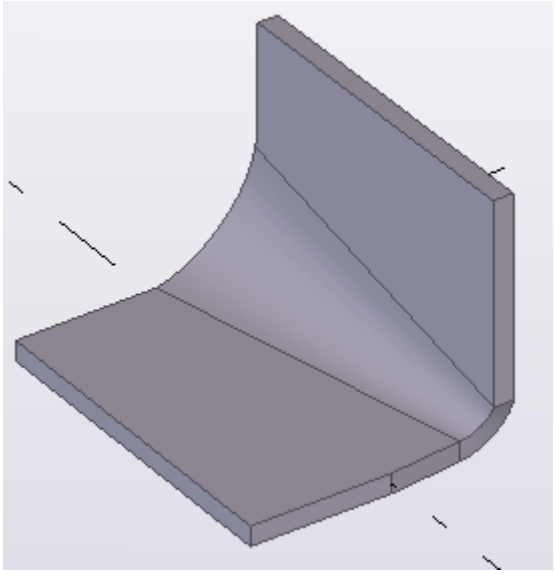
     a. Select the first part.

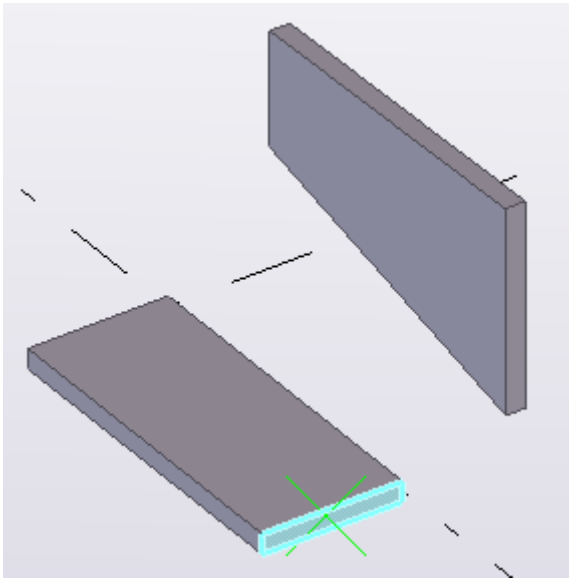
     b. Select the second part.

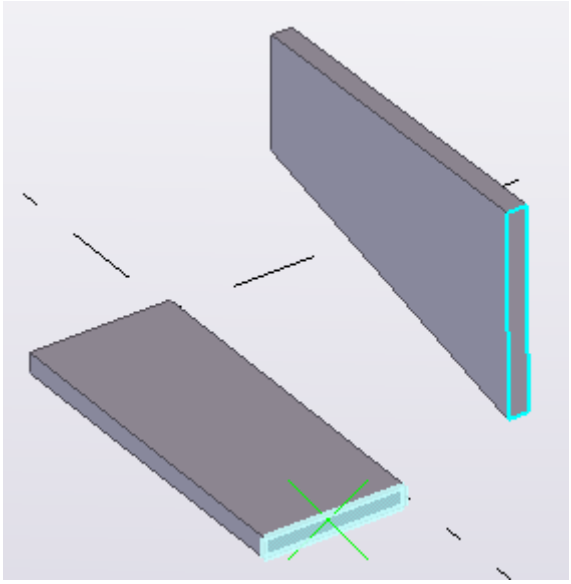
c. Tekla Structures creates the conical bent plate.



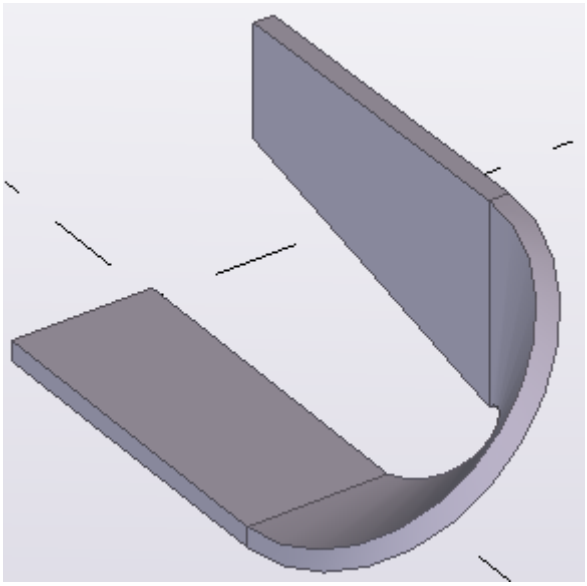
- If you selected **By faces**:
  a. Select the first part face.



b. Select the second part face.



c. Tekla Structures creates the conical bent plate.

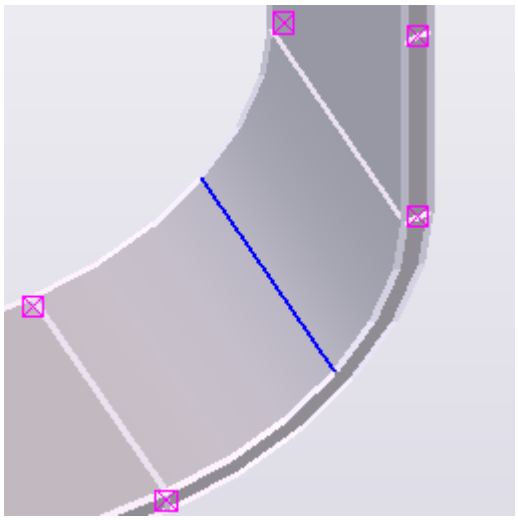


Modify the bend radius

When you create a cylindrical bent plate, you can enter a radius for the plate. For a conical bent plate you can enter two radiuses. If you do not enter any radius, Tekla Structures uses a default bend radius when creating bent plates. You can later change the bend radius to suit your needs.

1. Ensure that  **Direct modification** is switched on.
2. Select the bent plate.

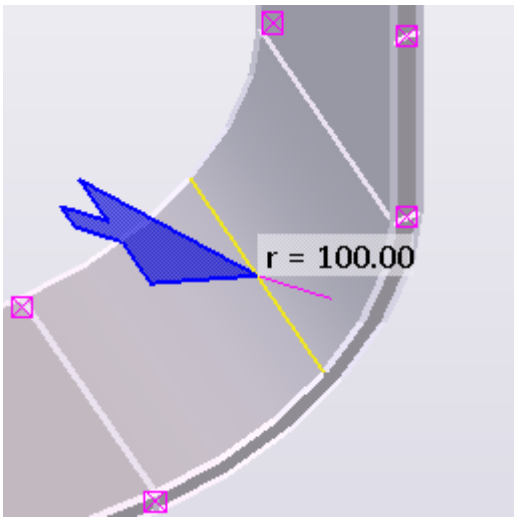
A blue line handle appears in the middle of the curved section.



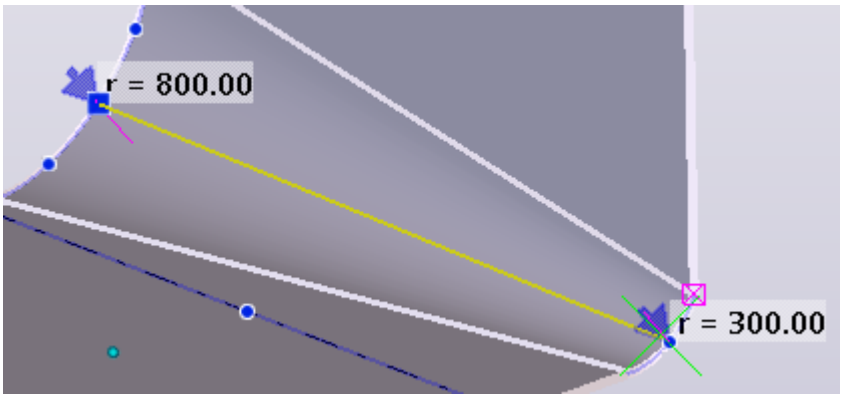
3. Select the line handle.

   Depending on the type of the bent plate, one (for cylindrical bent plate) or two (for conical bent plate) blue dimension arrows appear.

   - For cylindrical bent plate:

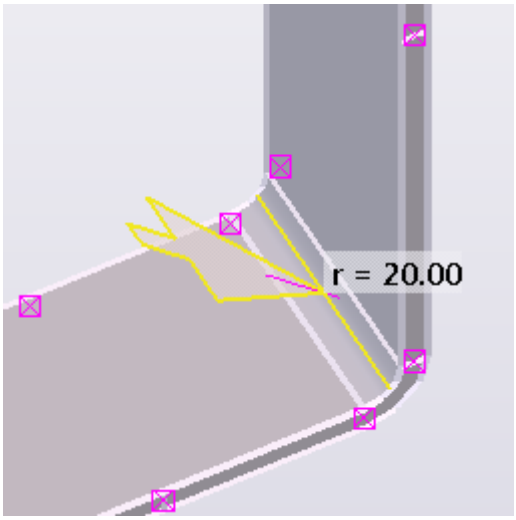


- For conical bent plate:



4. To modify the radius, do one the following:
   - Drag the arrow or arrows forward or backward along the magenta line.

     The dimension "r =" changes accordingly. When you release the arrow, the radius also changes in the model.



   - On the contextual toolbar, enter the radius or the radiuses.
   - Alternatively, you can select the arrow and type a dimension. When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog. Click **OK** to confirm the dimension.

Modify the shape of a bent plate

When you create a bent plate, Tekla Structures adds a curved section between the parts you select. You can modify the curved section by choosing one of the predefined options or by modifying the shape manually. You can also modify the flat sections, which are the original parts the bent plate was made of.

1. Ensure that  **Direct modification** is switched on.

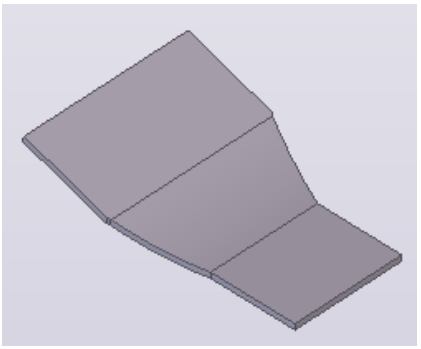
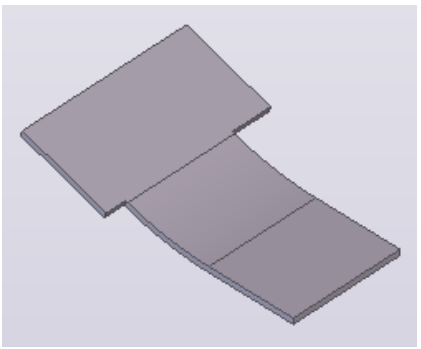
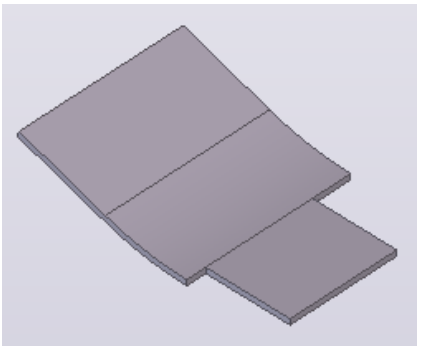
2. Select the bent plate.

   A blue line handle appears in the middle of the curved section.

3. Select the line handle.

   A contextual toolbar appears.

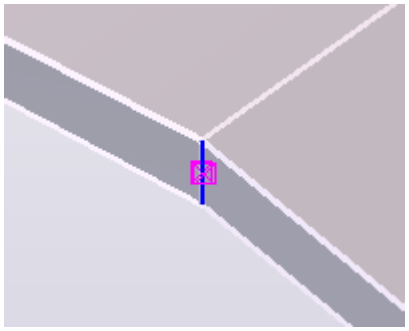
4. On the contextual toolbar, select one of the predefined shape options:

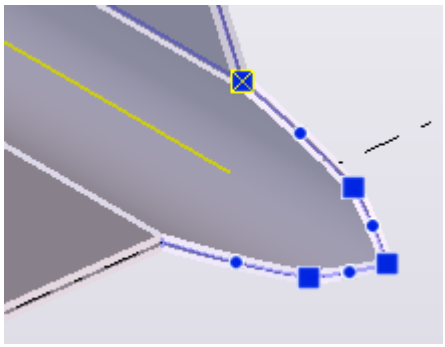
Option	Description	Example
Tapered bend 	A gradual decrease in the width between the parts. This is the default shape.	
Narrow bend 	Constant width between the parts. The width is determined by the **narrowest** part.	
Wide bend 	Constant width between the parts. The width is determined by the **widest** part.	

5. To modify the curved section manually:

   a. Select the blue line handle.

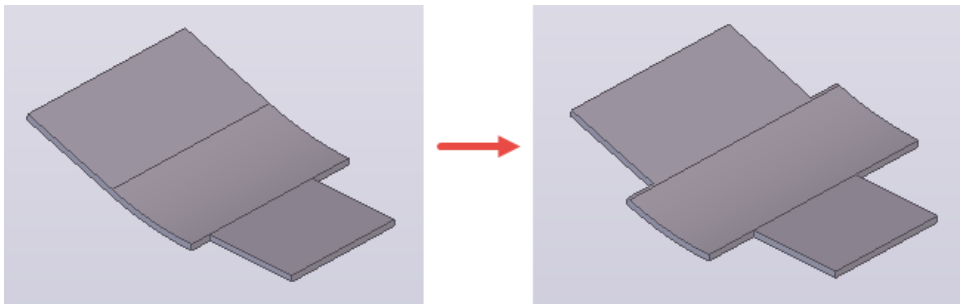
      You can modify the side boundaries or the lateral boundaries of the bends both in the cylindrical and in the conical bent plate. Tekla Structures displays the boundary handles in blue:

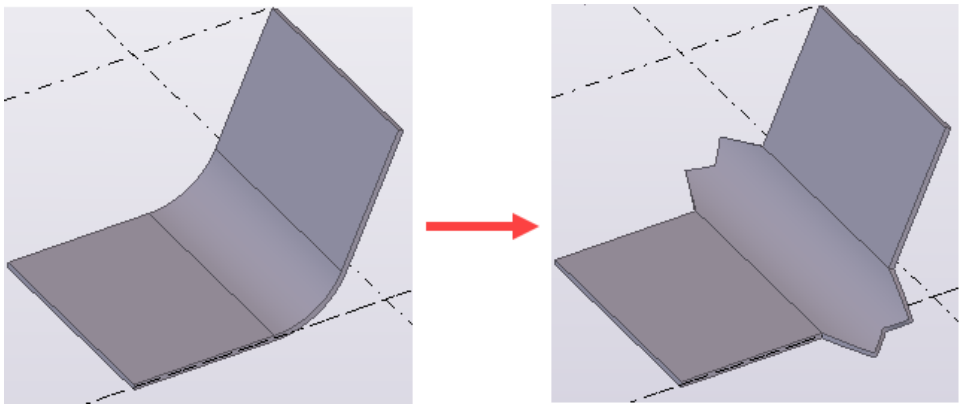




   b. Drag the handles to change the shape of the curved section.

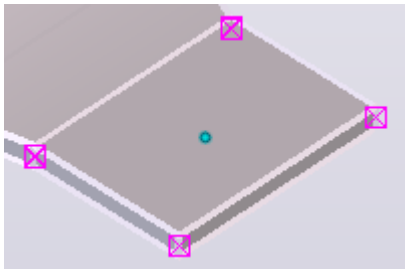
      For example:





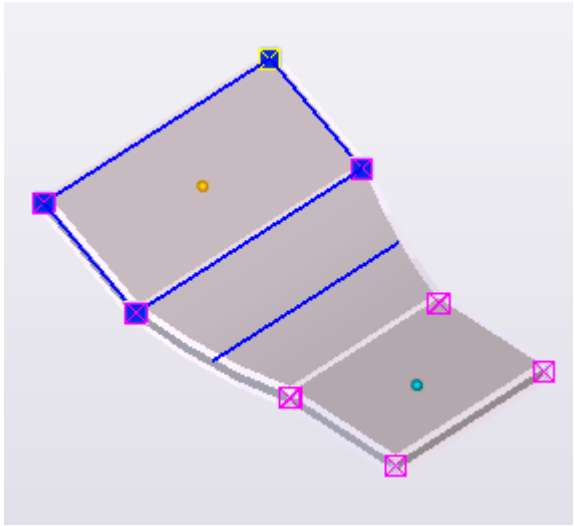
6. To modify the flat sections:
   a. Select the bent plate.

      Tekla Structures displays a green selection handle in the middle of each flat section:

   b. Click the selection handle of the section you want to modify.

      The direct modification handles of the selected section become visible:

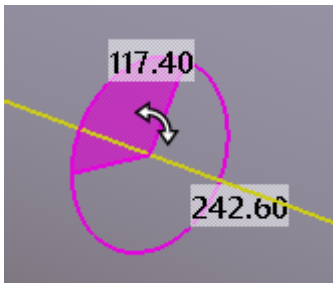
   c. Use the direct modification handles to change the shape of the flat section.

7. To modify the angle of the bent plate:
   a. Click the green selection handle in the middle of the flat section whose angle you want to modify.
   b. Select the line handle.

      A contextual toolbar appears.

   c. On the contextual toolbar, click **Enable angle manipulator**.

      The angle manipulator wheel appears in the model.

   d. Modify the angle using the wheel.

      If you want to modify the angle of another flat section, click the other green selection handle.
8. To change the main section of the bent plate:
   a. Click the green selection handle of the section that you want to set as the main section.

      A contextual toolbar appears.

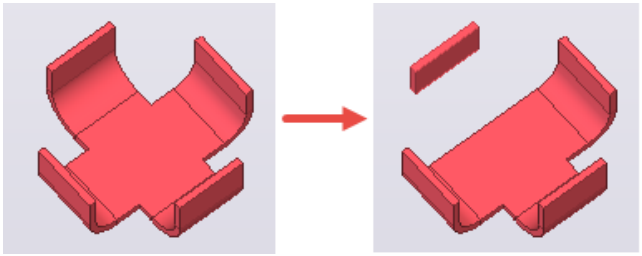
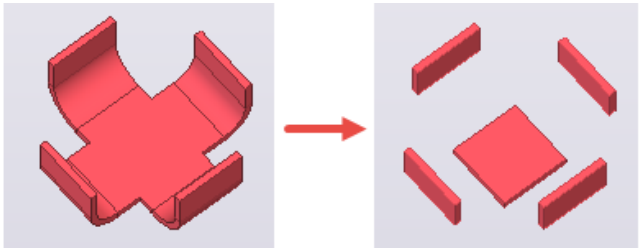
   b. On the contextual toolbar, click **Set main section**.

      The direct modification handles become active in the new main section. The main section and the coordinate system of the bent plate change accordingly, changing the bent plate orientation in an unfolded drawing.

Remove curved sections

You can return bent plates into individual objects, and then edit and use them as any other model object. If the bent plate consists of several curved sections that are all joined to the same part, you can either remove each curved section separately or explode the entire bent plate all at once.

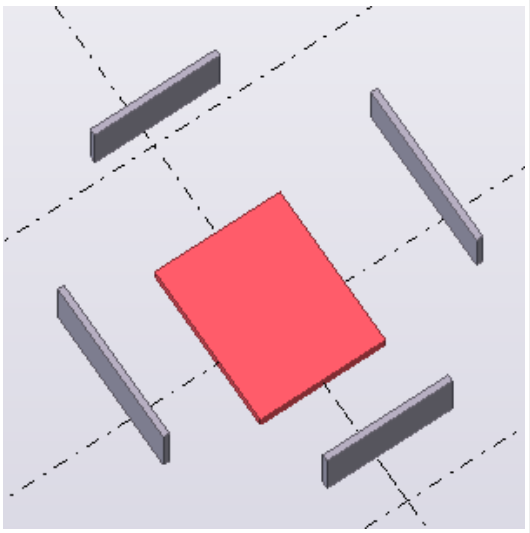
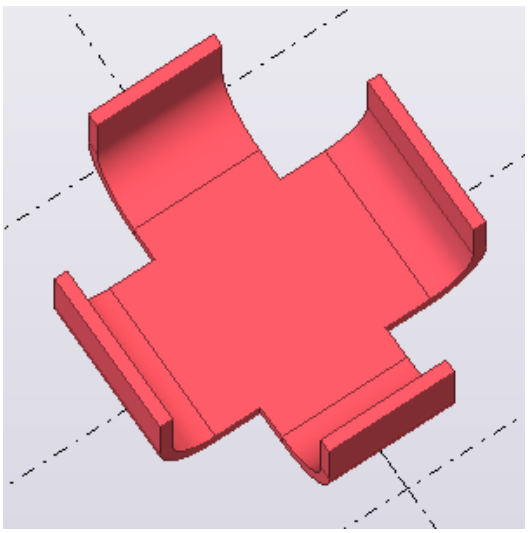
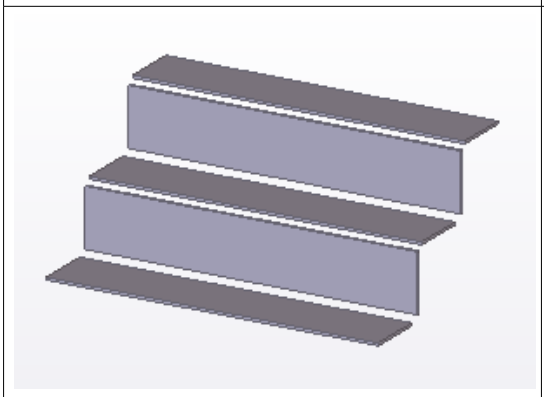
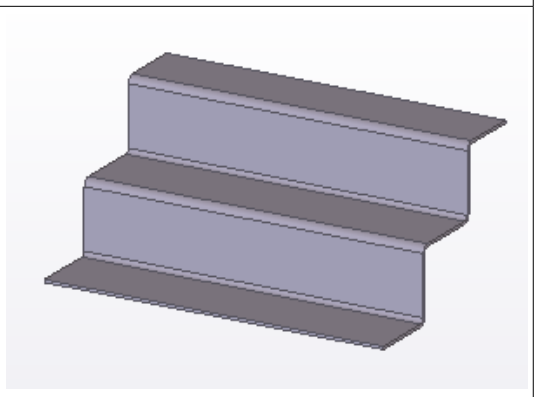
To	Do this
To remove individual curved sections	1. Ensure that **Direct modification** is switched on.

To	Do this
	2. Select the curved section you want to remove. A blue line handle appears. 3. Select the line handle. A contextual toolbar appears. 4. On the contextual toolbar, click  **Remove bend**. Tekla Structures removes the selected curved section. For example: 
To explode the entire bent plate	1. Select one of the curved sections. 2. Right-click and select **Explode**. Tekla Structures explodes the entire bent plate into individual objects. For example: 

If the bent plate solids that have been created using a previous version of Tekla Structures seem to be broken, you can remove or explode the bends. Then remodel the bends using the **Create cylindrical bent plate** command.

Examples

Here are some examples of bent plates that you can create:

Starting point	End result
	
	

Modify bent plate properties

1. If the property pane is not open, double-click the bent plate to open the **Bent plate** properties.
2. Change the properties as needed.
3. Click **Modify**.

Bent plate properties

Use the **Bent plate** properties in the property pane to view and modify the properties of a bent plate. To open the properties, double-click the bent plate. The file name extension of a bent plate property file is `*.bpl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of the plate. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the plate.
Material	Material (page 334) of the plate.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group plates. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 753).
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
Subtype (IFC4)	
User-defined type (IFC4)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a stand-alone bent plate

You can create stand-alone bent plates which do not require any input parts to be selected. Use stand-alone bent plates to model cylindrical and conical parts, such as hoods, hoppers, cones, and so on.

Limitations

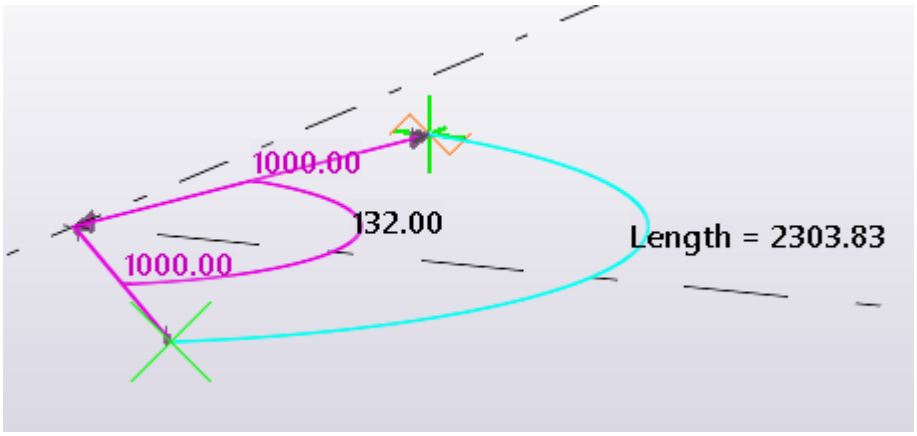
- Tekla Structures does not support full 360 degree stand-alone bent plates. However, you can create 359 degree plates instead.
- Use the local coordinate system in drawing creation.

In addition to the stand-alone bent plates, you can also create cylindrical and conical bent plates (page 245) either by selecting two parts or two part faces. The parts that you use for creating a bent plate must be contour plates, or beams whose profile is a plate.

Create a stand-alone bent plate

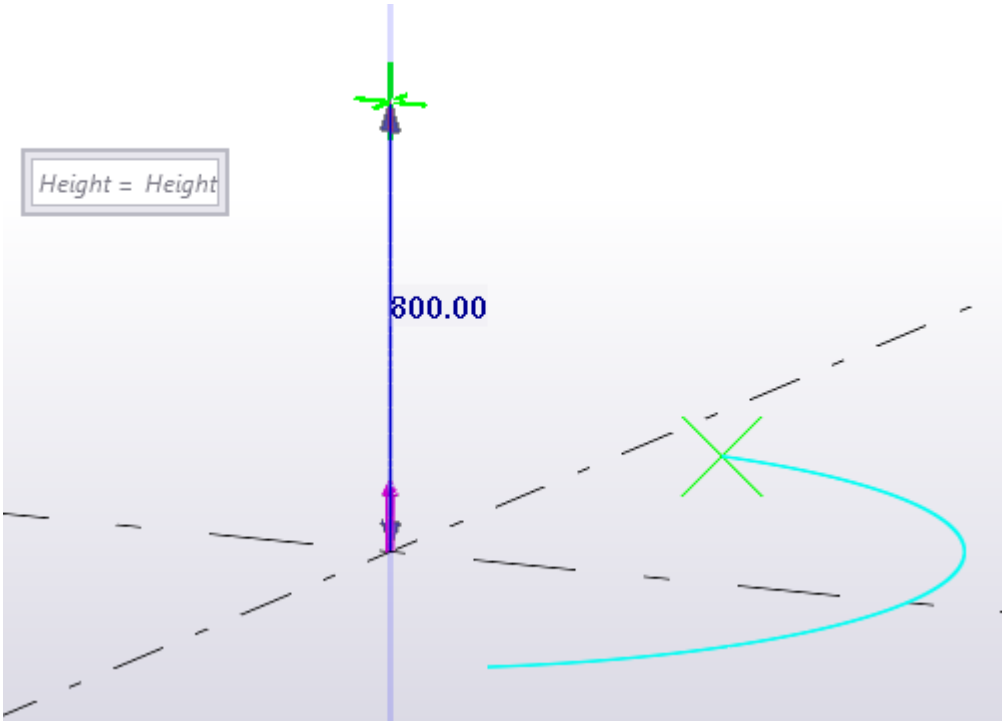
1. Ensure that  **Direct modification** is switched on.
2. On the **Steel** tab, click **Plate** --> **Create stand-alone bent plate** .
3. Define the first radius of the bend:
   a. Pick the center point.
   b. Pick the start point for the arc.
   c. Pick the end point for the arc.

   The picking order defines the up direction. For example, if you create an arc on the xy plane in the counterclockwise direction, the up direction points to the positive z axis, according to the right-hand rule.



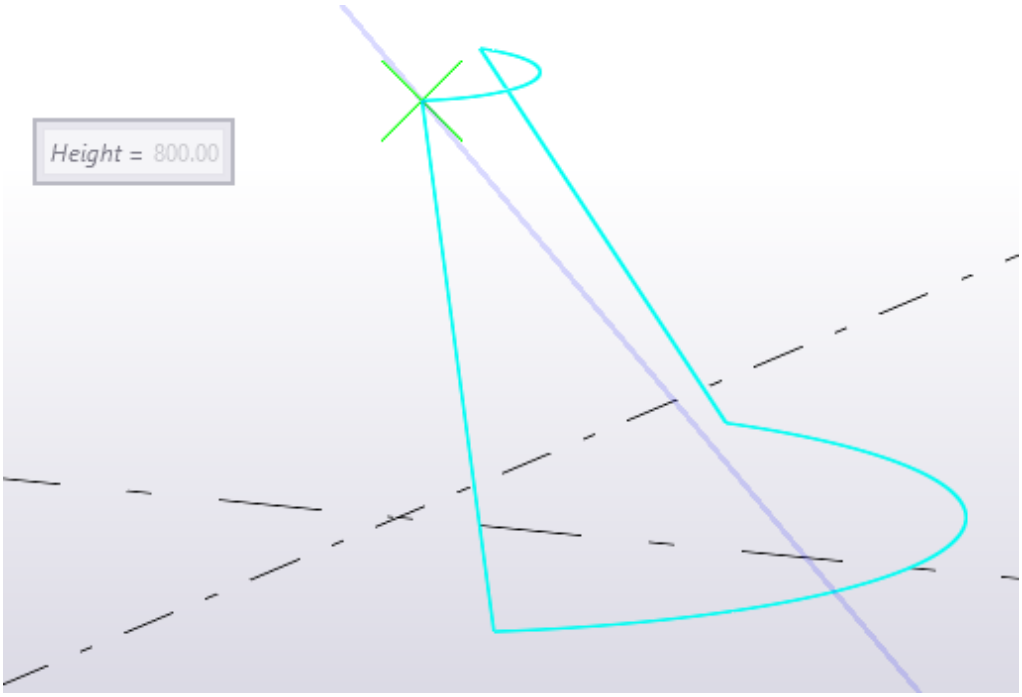
4. Pick a point to define the height of the bend.

Alternatively, you can enter the height in the bent plate contextual toolbar.

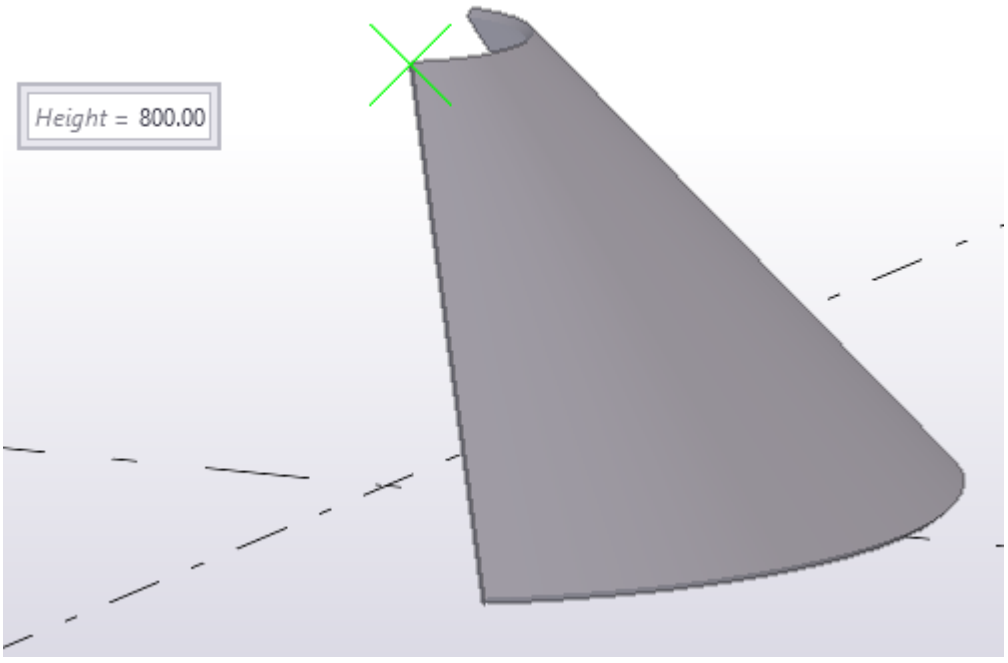


5. Define the second radius of the bend:
   a. Pick a point based on the preview of the plate.
   b. If you want to change the direction of the plate after you have picked a point, click the left mouse button.

      Alternatively, if you want to create a cylindrical bend, click the middle mouse button. In this case the radius 2=radius 1.

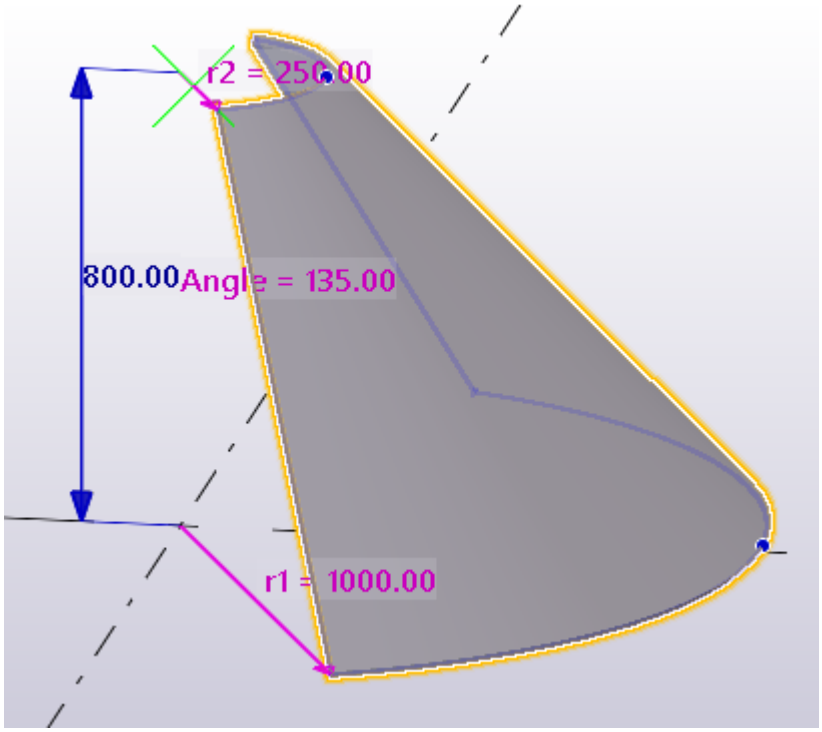


6. To complete the bent plate creation, click the middle mouse button.

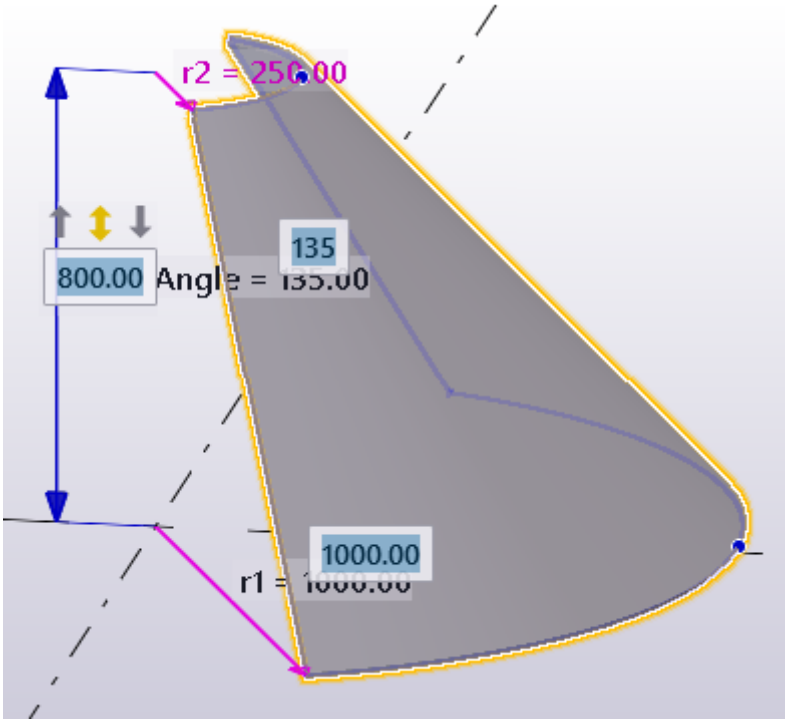


Modify the shape of a stand-alone bent plate

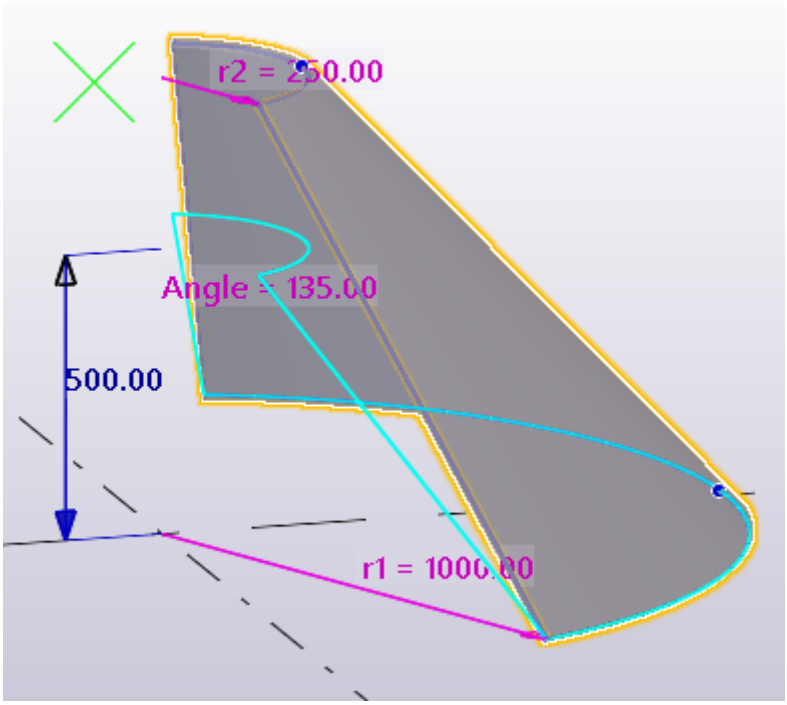
Use direct modification dimension values and handles to modify the bent plate shape.

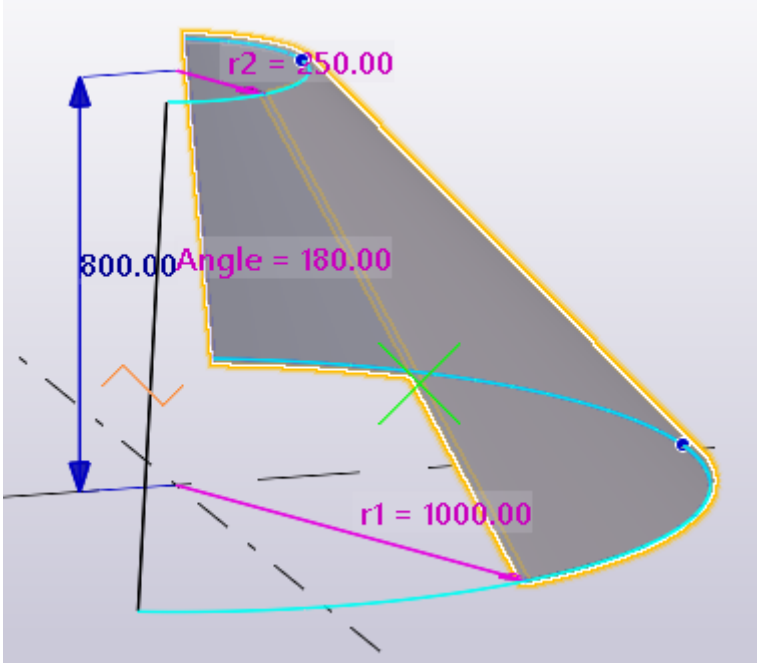


- You can change the angle, radiuses and height of the bend by entering new dimension values.

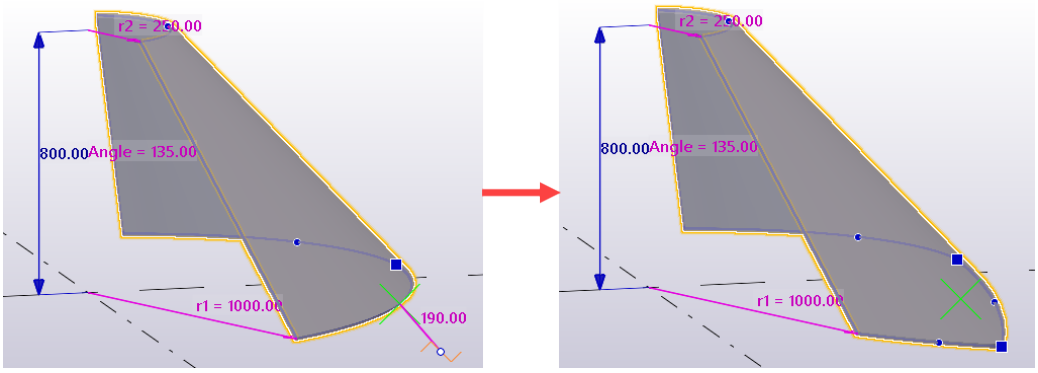


- You can drag and stretch the edges of the bent plate.





- You can add and delete intermediate points.



- You can create unfolded drawings of the stand-alone bent plates.

Modify bent plate properties

1. If the property pane is not open, double-click the bent plate to open the **Bent plate** properties.
2. Change the properties as needed.
3. Click **Modify**.

Bent plate properties

Use the **Bent plate** properties in the property pane to view and modify the properties of a bent plate. To open the properties, double-click the bent plate. The file name extension of a bent plate property file is `*.bpl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the plate. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the plate.
Material	Material (page 334) of the plate.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, for example, with anti-corrosive paint, hot galvanized, fire retardant coating, and so on.
Class	Use to group plates. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 753).
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
Subtype (IFC4)	
User-defined type (IFC4)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a steel lofted plate

With lofted plates you can create rolled plates with varying shapes and plates with double-curved form, for example.

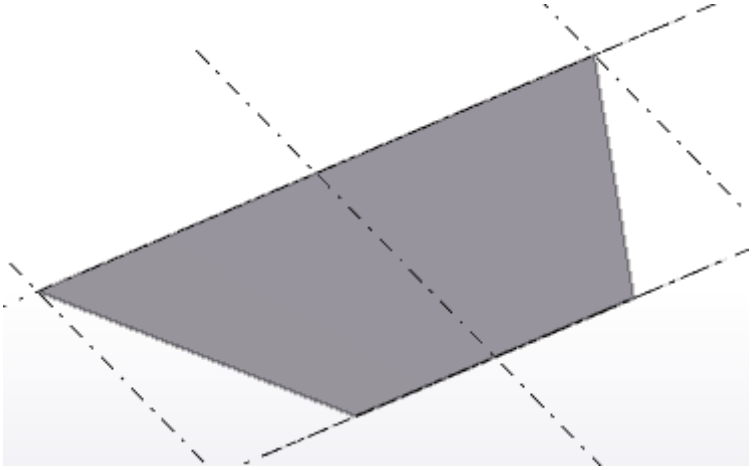
Prerequisites and examples of lofted plates

Before you can create lofted plates, you need to have construction objects (page 650) in your model. Tekla Structures creates the shape of the lofted plate according to the geometry of the used construction objects, by connecting the start point of the first construction object to the start point of the second construction object. The end points of construction objects are connected in similar way.

You can connect the following construction objects as a lofted plate:

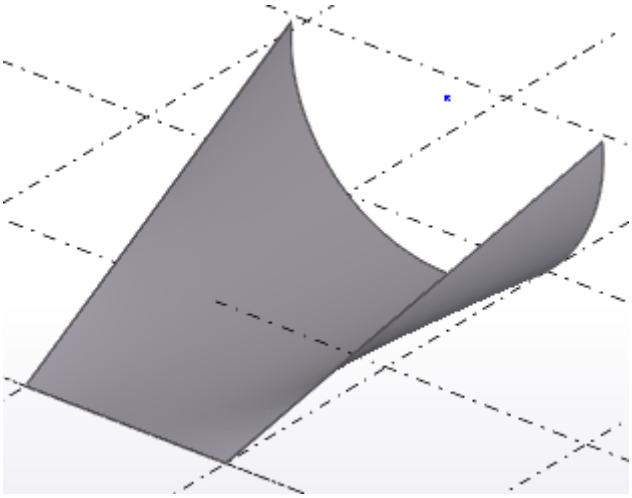
- construction line to construction line

  For example:



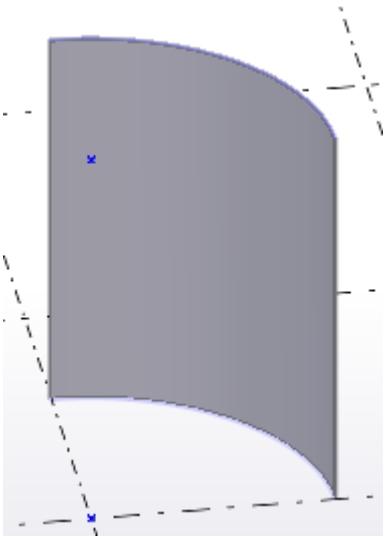
- construction line to construction arc

  For example:



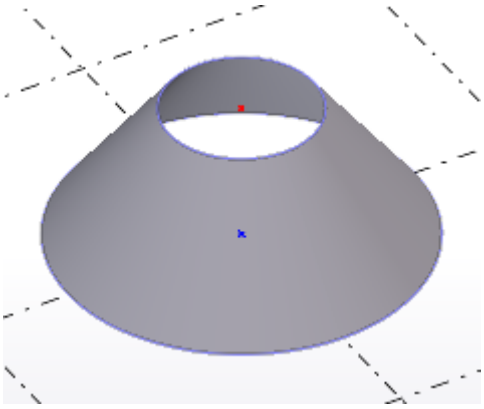
- construction arc to construction arc

  For example:

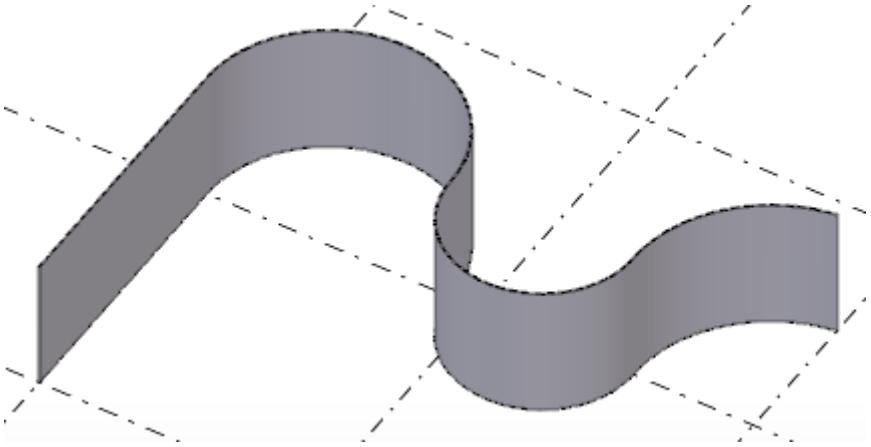


- construction circle to construction circle

  For example:



- construction polycurve to construction polycurve



Create a lofted plate

1. Create the needed construction objects in the model. The shape of the lofted plate is based on the construction objects' shape.

   You need to have

   - construction lines (page 651)
   - construction arcs (page 653)
   - construction circles (page 652)

     or

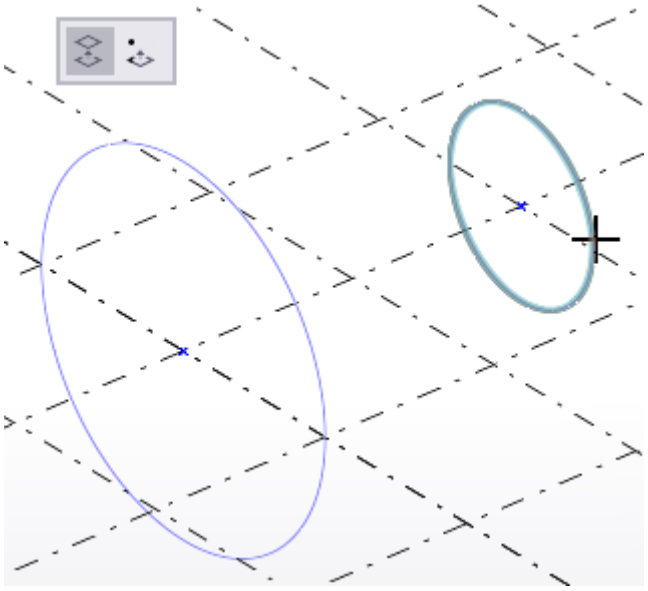
   - construction polycurves (page 655)

     With polycurves, use the **Create arc by tangent** or **Create tangent line** options  on the construction polycurve toolbar. To create polycurves only with straight segments, use the **Create line** option . Note that the construction polycurves do not need to have the same number of segments, as long as both of them are tangential.
2. When you have created the needed construction objects, go to the **Steel** tab and click **Plate** --> **Create lofted plate**.
3. On the toolbar that appears, click a button to specify whether to create the lofted plate by using two construction objects, or by using a construction object and a point.

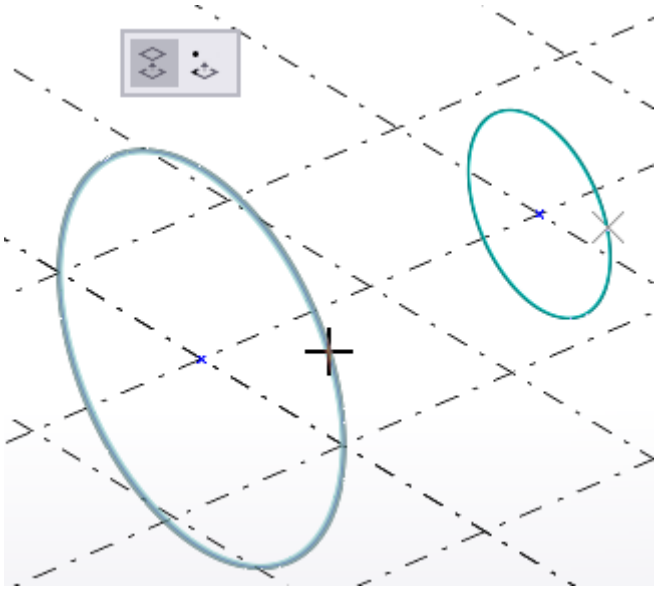
   - Use two construction objects  to create a lofted plate:

a. Select the first construction object: line, arc, circle, or polycurve.

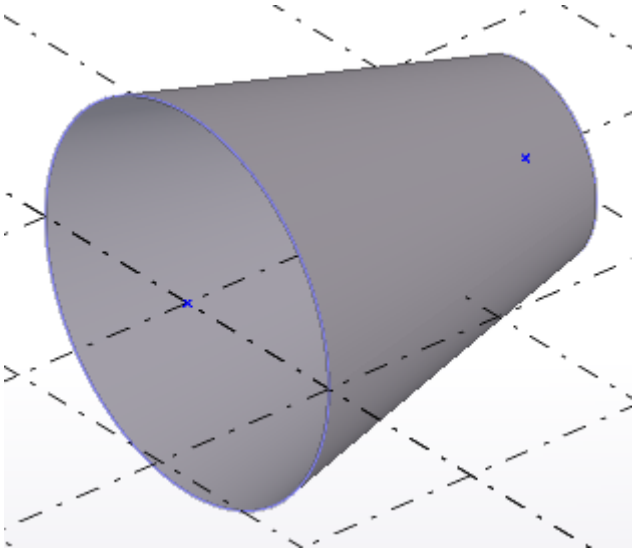
   For example, if you are using two construction circles to create a lofted plate.



b. Select the second construction object.

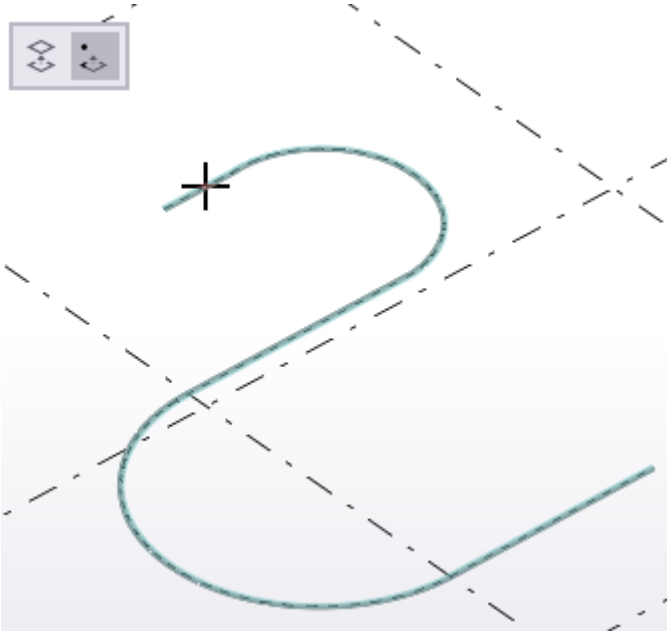


   Tekla Structures creates the lofted plate between the selected construction objects, using the **Lofted plate** properties in the property pane.



- Use one construction object and a point  to create a lofted plate:
  a. Select the first construction object: line, arc, circle, or polycurve.

     For example, if you are using a construction polycurve and a point to create a lofted plate.

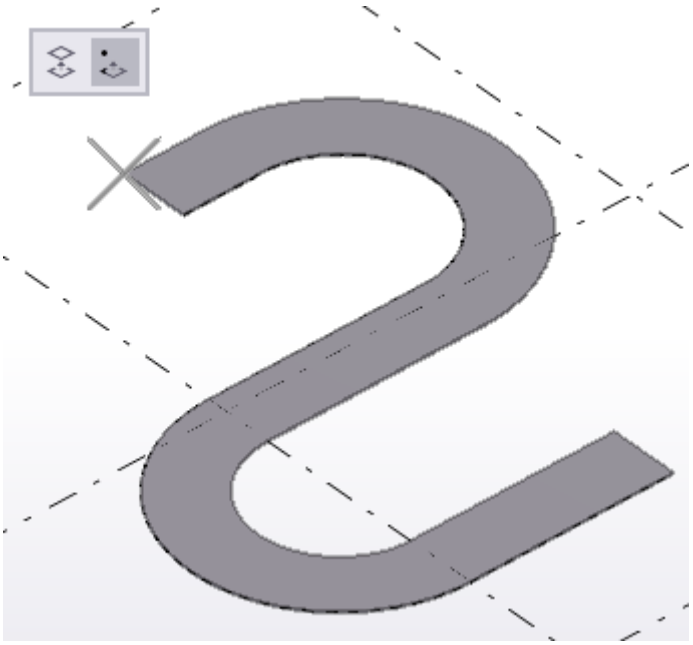
     

     Tekla Structures shows a preview of the part geometry. Use the preview to set the direction and size of the lofted plate.

b. Pick a point.



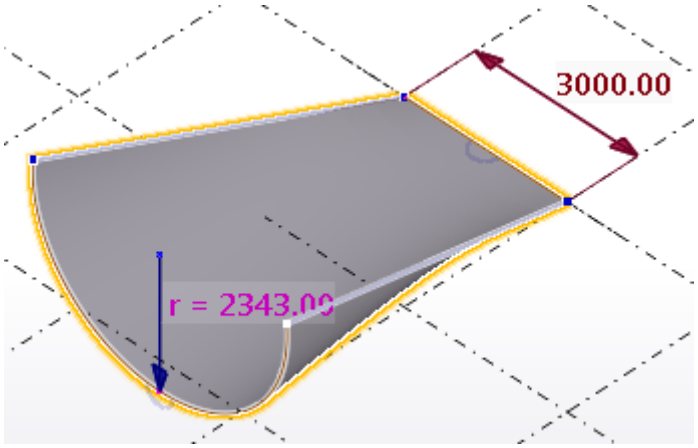
Tekla Structures creates the lofted plate based on the preview, using the **Lofted plate** properties in the property pane.

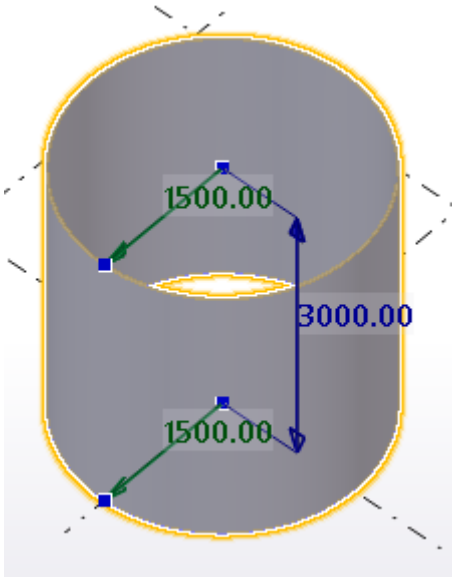


Modify the shape of a lofted plate

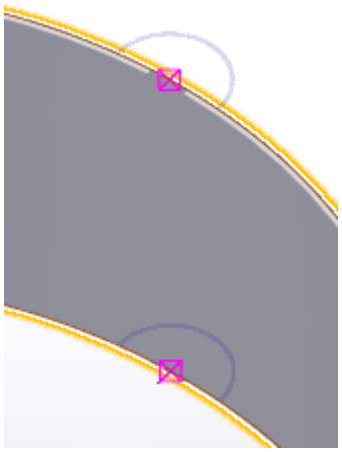
Use direct modification dimension handles and dimension values to modify the lofted plate shape.

- Modify, for example, the height and radius of the lofted plate.





- For lines and arcs: drag the arc symbol  at the midpoint of a line or an arc to modify the lofted plate shape.



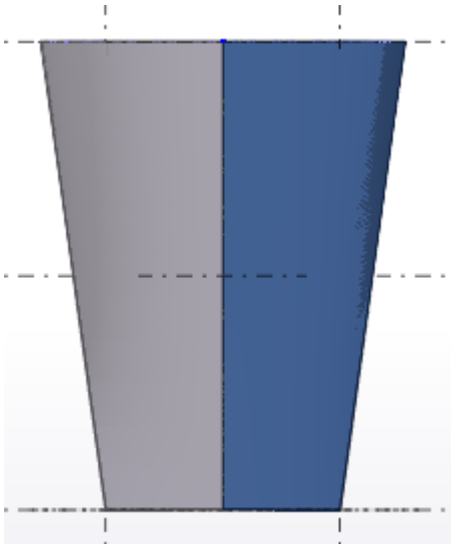
Split a lofted plate

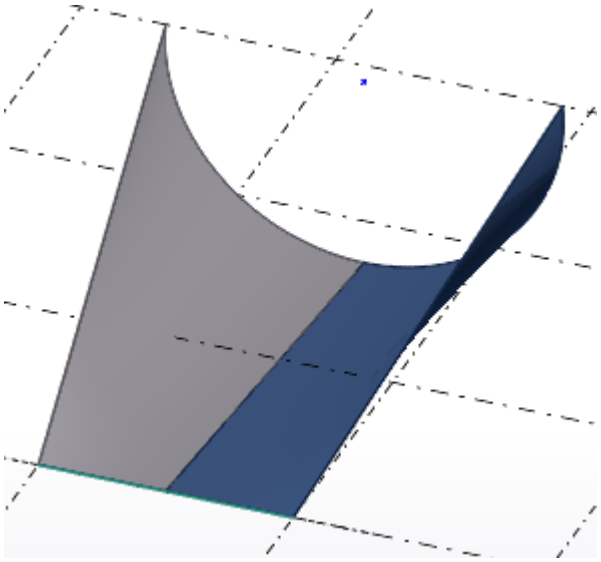
Note that you cannot split closed cylindrical or conical lofted plates.

1. On the **Edit** tab, click **Split**.
2. Select the lofted plate you want to split.
3. Pick a point for the dividing line.

   Tekla Structures splits the lofted plate.

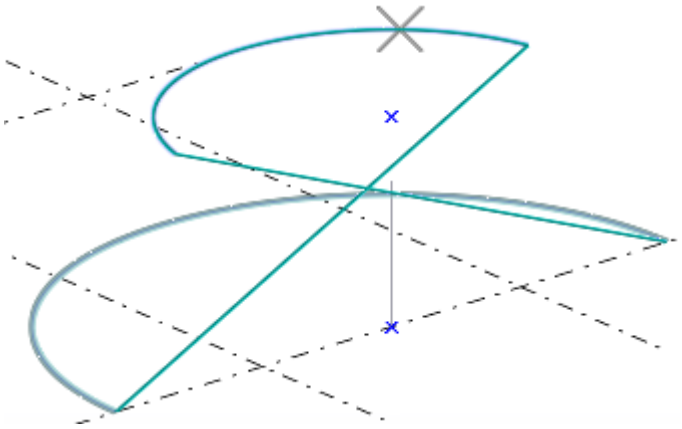
   For example:





Swap the end handle points to correct the geometry of a lofted plate

In some cases when you try to create a lofted plate, the geometry of the plate would become self-intersecting, meaning that the start and end points of the top and bottom construction object are opposite of each other. In these cases the plate is not created.



You can try to resolve the situation and create the lofted plate by changing the modeling direction of the construction lines or arcs.

1. Select the construction line or the construction arc.

   With construction lines, ensure that the **Direct modification** switch is **not** active. You can then see the yellow and magenta object handles.

2. On the contextual toolbar, click **Swap ends**.

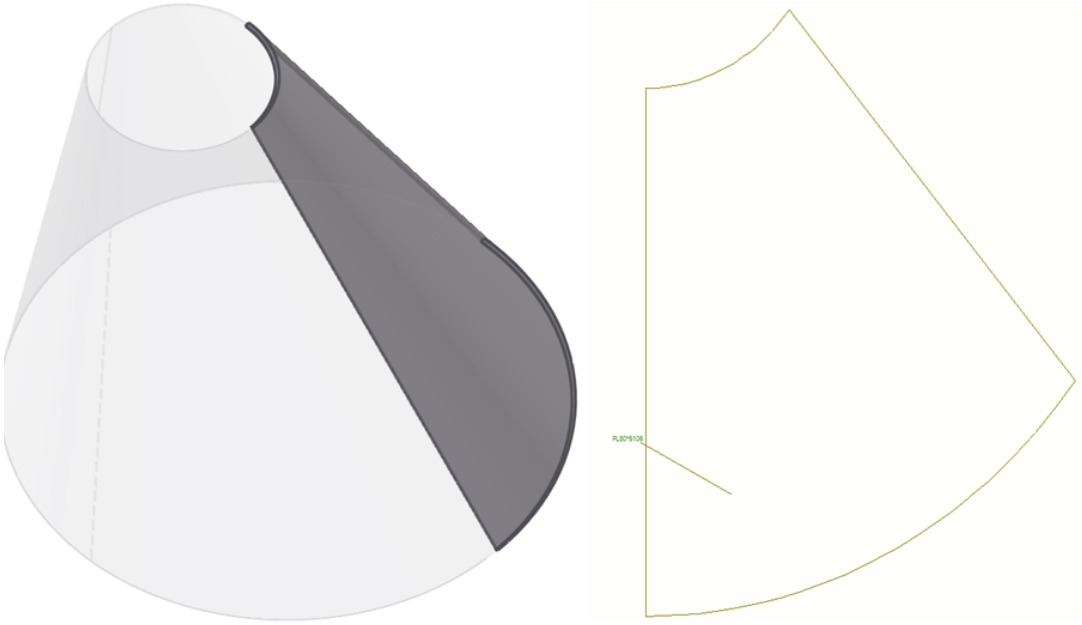
   Tekla Structures changes the modeling direction of the selected construction object, and the lofted plate can be created correctly.

With construction circles you can try to resolve the situation by moving either of the circles.

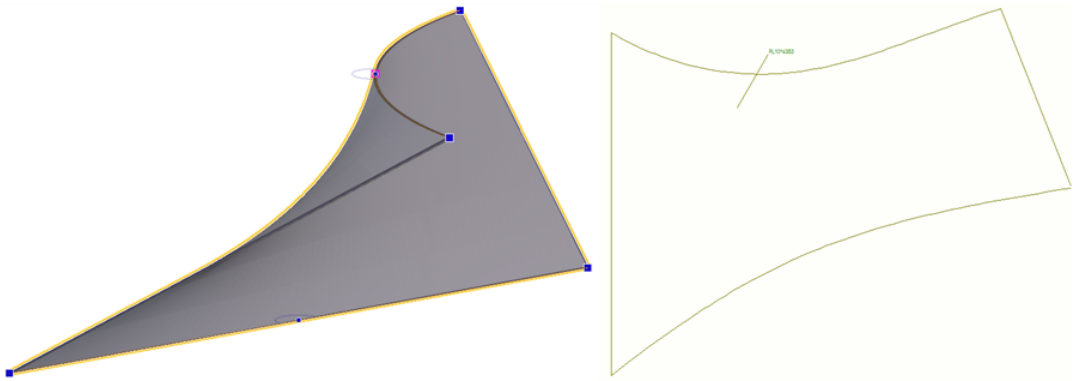
Unfold lofted plates

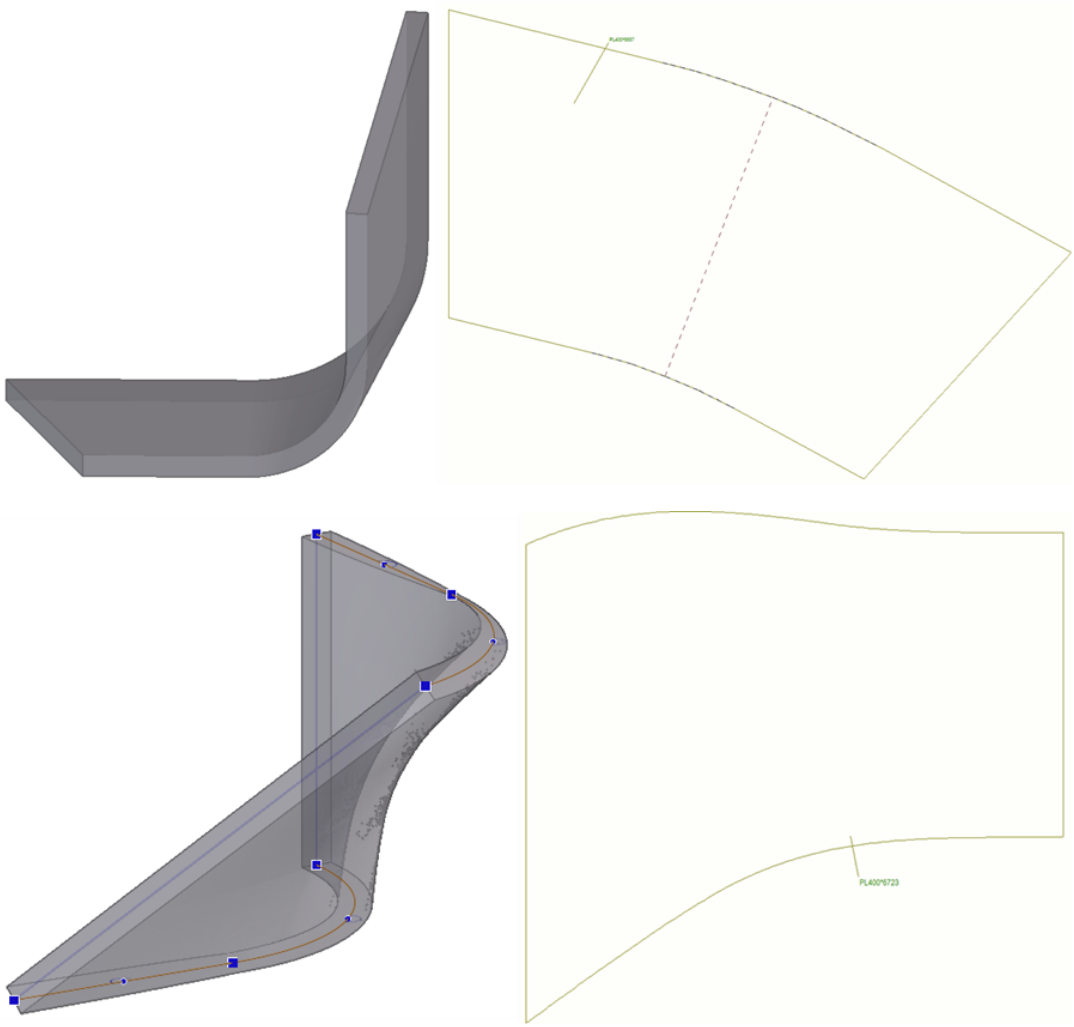
You can unfold lofted plates in single-part drawings. The unfolding works for lofted plates which have been created from single geometry to single geometry, and for tangential polycurve lofted plates.

Examples of unfolded lofted plates created from single geometry to single geometry:



Examples of tangential polycurve unfolded lofted plates:





Modify lofted plate properties

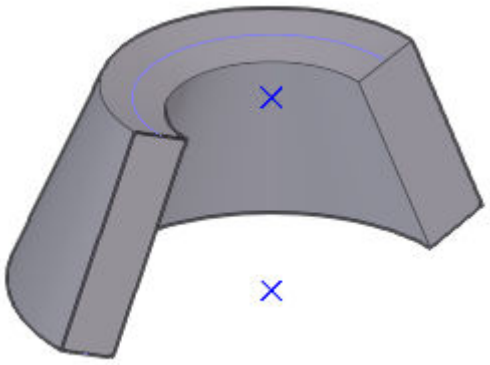
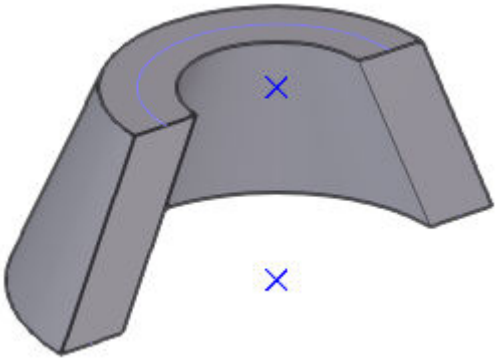
1. If the property pane is not open, double-click the lofted plate to open the **Lofted plate** properties.
2. Change the properties as needed.
3. Click **Modify**.

Lofted plate properties

Use the **Lofted plate** properties in the property pane to view and modify the properties of a steel lofted plate. To open the properties, double-click the steel lofted plate. The file name extension of a steel lofted plate property file is `*.lpl`.

If you have customized the property pane layout, the list of properties may be different.

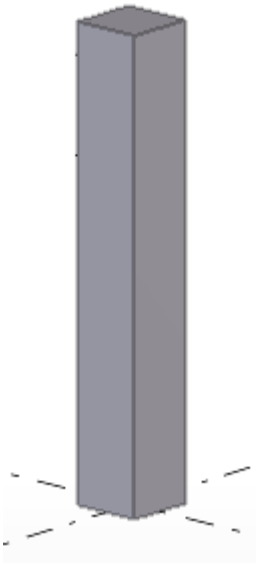
Setting	Description
General	
Name	User-definable name of the lofted plate. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the lofted plate.
Material	Material (page 334) of the lofted plate.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group lofted plates. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 753).
Face type	
Face type	Select whether the top and bottom faces of the plate are flush with the plane. **Perpendicular**:

Setting	Description
	 The top and bottom faces of the plate are non-linear.
	Bounded by curved planes:  The top and bottom faces of the plate are planar.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete column

1. On the **Concrete** tab, click **Column** .
2. Pick a point.

   Tekla Structures creates the column using the **Concrete column** properties in the property pane, and at the level defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Concrete column** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify concrete column properties

1. If the property pane is not open, double-click the column to open the **Concrete column** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete column properties

Use the **Concrete column** properties in the property pane to view and modify the properties of a concrete column. To open the properties, double-click the concrete column. The file name extension of a concrete column property file is `*.ccl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the column. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the column.
Material	Material (page 334) of the column.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group columns. For example, you can display parts of different classes in different colors.
Position	
Vertical	Vertical position (page 344) of the column, relative to column's reference point.
Rotation	Rotation (page 342) of the column around its axis on the work plane.
Horizontal	Horizontal position (page 346) of the column, relative to column's reference point.
Top	Position of the second end of the column in the global z direction.
Bottom	Position of the first end of the column in the global z direction.
Bind to levels	
Top level	Use to bind (page 350) the top face of the column to a selected grid level. Set the offset if needed.

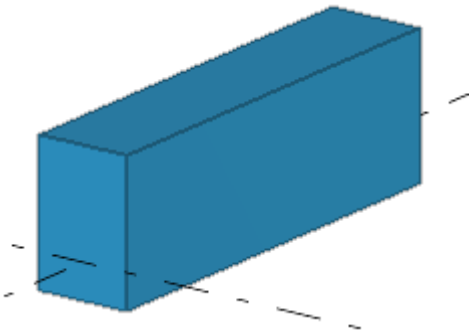
Setting	Description
Bottom level	Use to bind (page 350) the bottom face of the column to a selected grid level. Set the offset if needed.
Cast unit	
Cast unit numbering	Cast unit prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the column is precast or cast-in-place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Deforming	
Warping	Use to warp columns using deformation angles.
Cambering	Use to pre-camber (page 353) the column.
Shortening	Use to shorten the column in the model. The true length of the column is decreased in the drawing.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of

Setting	Description
	the part. UDAs provide additional information about the part.

Create a concrete beam

1. On the **Concrete** tab, click .
2. Pick two points.

   Tekla Structures creates the beam between the points you picked using the **Concrete beam** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Concrete beam** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify concrete beam properties

1. If the property pane is not open, double-click the beam to open the **Concrete beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete beam properties

Use the **Concrete beam** properties in the property pane to view and modify the properties of a concrete beam or polybeam. To open the properties, double-click the concrete beam. The file name extension of a concrete beam property file is `*.cbm`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the beam.
Material	Material (page 334) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
On plane	Beam's position on the work plane (page 340), relative to the beam's reference line.
Rotation	Rotation (page 342) of the beam around its axis on the work plane.
At depth	Position depth (page 343) of the beam. The position is always perpendicular to the work plane.
Bind to levels	
Start level	Use to bind (page 350) the start point of the beam to a selected grid level. Set the offset if needed.
End level	Use to bind (page 350) the end point of the beam to a selected grid level. Set the offset if needed.
End offset	
Dx	Change the length of the beam (page 348) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 348) perpendicular to the beam's reference line.
Dz	Move the beam end (page 348) in the z direction of the work plane.

Setting	Description
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Cast unit	
Cast unit numbering	Cast unit prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the beam is precast or cast in place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 353) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
Subtype (IFC4)	
User-defined type (IFC4)	
More	

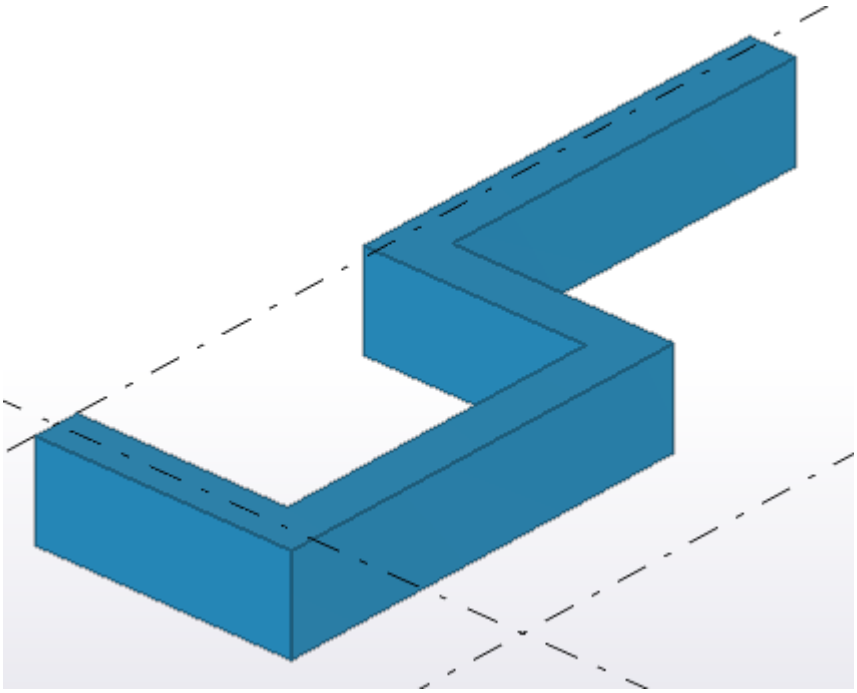
Setting	Description
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete polybeam

A polybeam can contain straight and curved segments.

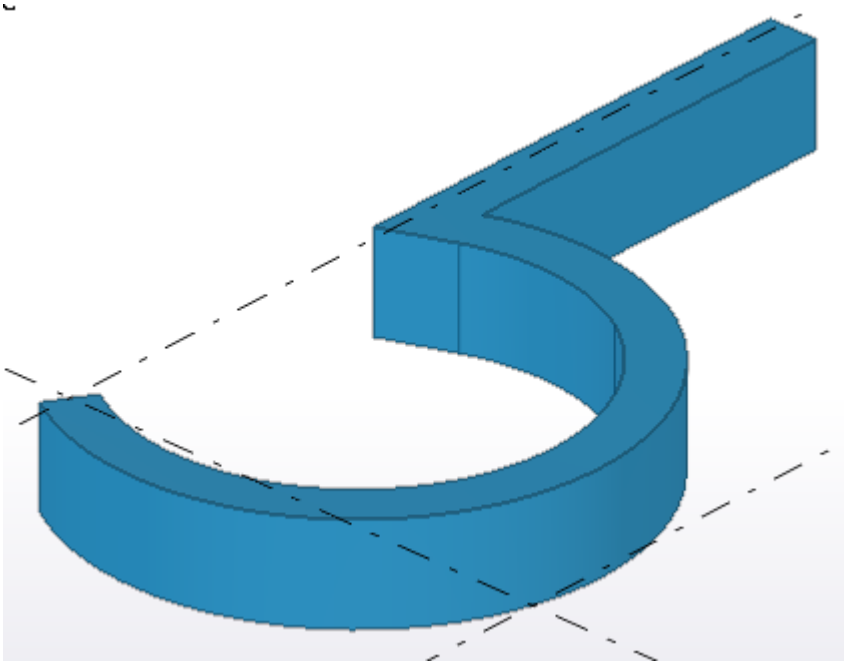
1. On the **Concrete** tab, click **Beam** --> **Polybeam**.
2. Pick the points you want the beam to go through.
3. Click the middle mouse button.

   Tekla Structures creates the beam between the points you picked, using the **Concrete beam** properties in the property pane. Note that you cannot create a closed polybeam.



4. If you want to create curved segments, chamfer the corners of the polybeam.

For example:



Modify concrete polybeam properties

1. If the property pane is not open, double-click the polybeam to open the **Concrete beam** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete beam properties

Use the **Concrete beam** properties in the property pane to view and modify the properties of a concrete beam or polybeam. To open the properties, double-click the polybeam. The file name extension of a concrete beam property file is `*.cbm`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the beam.
Material	Material (page 334) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
On plane	Beam's position on the work plane (page 340), relative to the beam's reference line.
Rotation	Rotation (page 342) of the beam around its axis on the work plane.
At depth	Position depth (page 343) of the beam. The position is always perpendicular to the work plane.
Bind to levels	
Start level	Use to bind (page 350) the start point of the beam to a selected grid level. Set the offset if needed.
End level	Use to bind (page 350) the end point of the beam to a selected grid level. Set the offset if needed.
End offset	
Dx	Change the length of the beam (page 348) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 348) perpendicular to the beam's reference line.
Dz	Move the beam end (page 348) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.

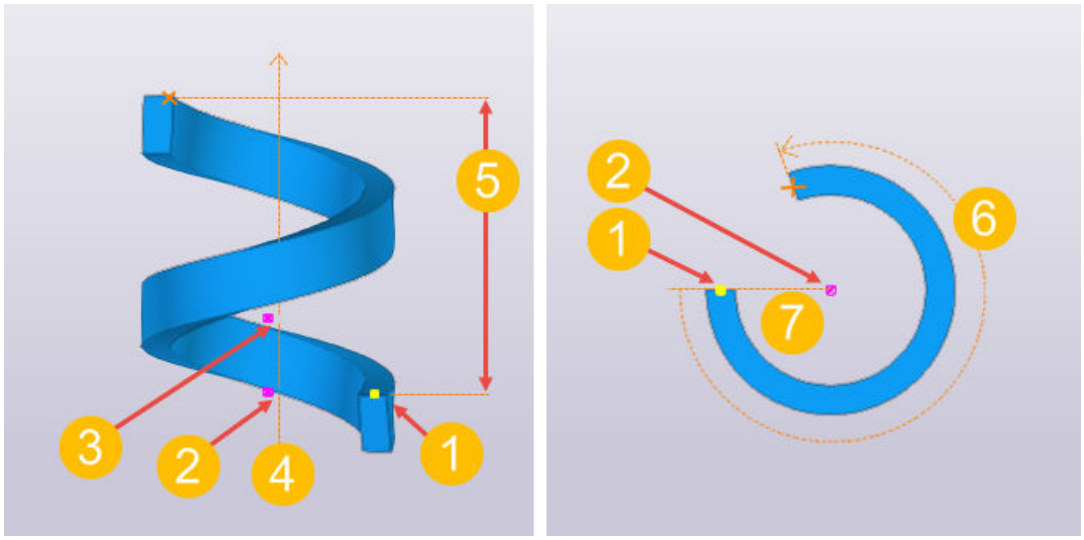
Setting	Description
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Cast unit	
Cast unit numbering	Cast unit prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the column is precast or cast in place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 353) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete spiral beam

Use the **Create concrete spiral beam** command when you want to model spiral staircases, parking ramps, and complex architectural shapes, for example.

Basic concepts related to spiral beams

The images below illustrate some basic concepts related to the creation of spiral beams. Note that if you change the positioning, the entire geometry of the spiral beam changes.



(1) Start point (the first point picked)

(2) Center point (the second point picked)

(3) Direction of the rotation axis (the optional third point picked)

(4) Center axis

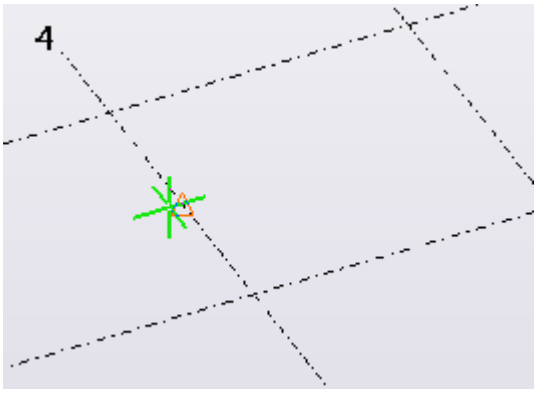
(5) Total rise: the distance from the start point to the end point, parallel to the center axis

(6) Rotation angle: the rotation angle of the spiral beam, given in degrees. Note: positive value = counter-clockwise rotation, negative value = clockwise rotation.

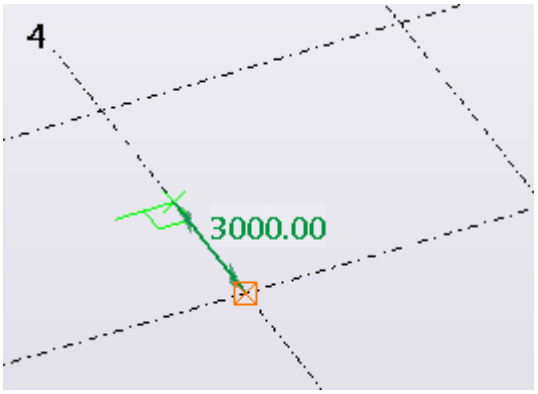
(7) Radius: the distance from the start point to the center point, perpendicular to the center axis

Create a spiral beam

1. On the **Concrete** tab, click **Beam** --> **Spiral beam**.
2. Pick the start point.



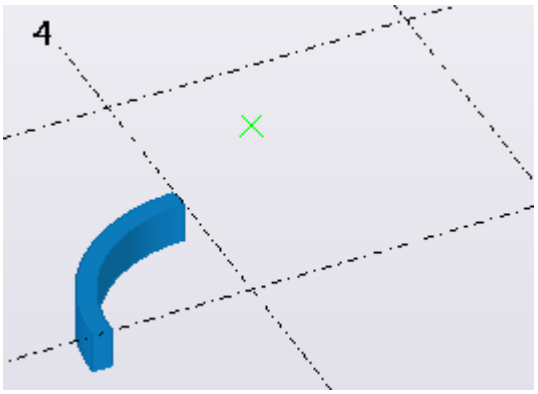
3. Pick the center point.



4. To set the rotation axis in the work plane +Z direction, click the middle mouse button to finish.

NOTE Alternatively, instead of clicking the middle mouse button, you can pick a second center axis point to define the direction of the rotation axis.

Tekla Structures creates the spiral beam. For example:



5. Click the spiral beam to select it.

The contextual toolbar appears with the following options:



(1) Rotation angle

(2) Total rise

(3) Twist angle at start

(4) Twist angle at end

6. To add more rotation, enter a bigger value in the **Rotation angle** box.
7. To make the spiral more loose, enter a bigger value in the **Total rise** box.
8. To change the radius, move the start point or center point.

Limitations

- Spiral beam has a single, constant radius.
- Unfolding of spiral beams whose total rise is larger than 0.00 does not produce completely straight results in drawings. The amount of deviation in the part profile outlines and part length depends on several factors: the type, size, and length of the profile; the amount of total rise; and the amount of rotation angle and detailing used.
- Spiral beams are not always untwisted in unfolding. If unequal twisting is applied to the start end and end end, the unfolded drawing shows an unfolded but twisted part as a result.
- Connections and details may not work as expected with spiral beams.
- DSTV export of spiral beams may not produce a correct result.
- You cannot export spiral beams as parts in IFC export. If you are modeling cast-in-place structures with spiral beam, you can export the geometry to IFC as pour objects.

Concrete spiral beam properties

Use the **Concrete spiral beam** properties in the property pane to view and modify the properties of a concrete spiral beam. To open the properties, double-click the spiral beam. The file name extension of a concrete spiral beam property file is `*.csb`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the beam.
Material	Material (page 334) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
On plane	Beam's position on the work plane (page 340), relative to the beam's reference line.
Rotation	Rotation (page 342) of the beam around its axis on the work plane.
At depth	Position depth (page 343) of the beam. The position is always perpendicular to the work plane.
Geometry	
Rotation angle	Rotation angle of the spiral beam, given in degrees.
Total rise	Distance from the start point to the end point, parallel to the center axis.
Twist angle at start **Twist angle at end**	Twist +/- angle of the spiral beam at the start/end of the beam.
Cast unit	
Cast unit numbering	Cast unit prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the column is precast or cast in place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets	

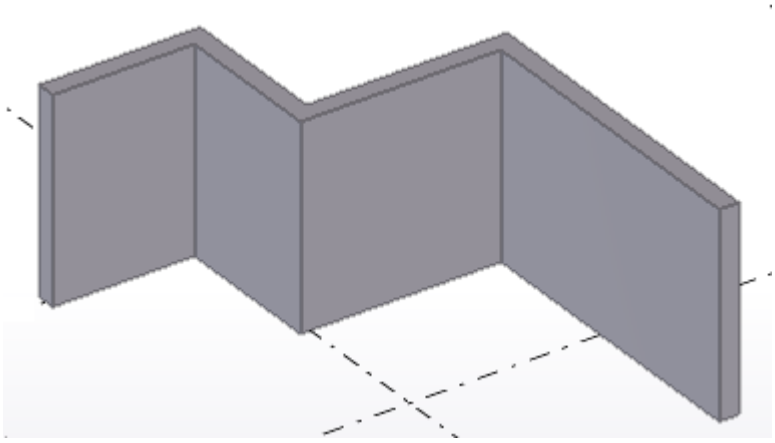
Setting	Description
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete panel or wall

You can create a concrete panel or wall that passes through the points you pick.

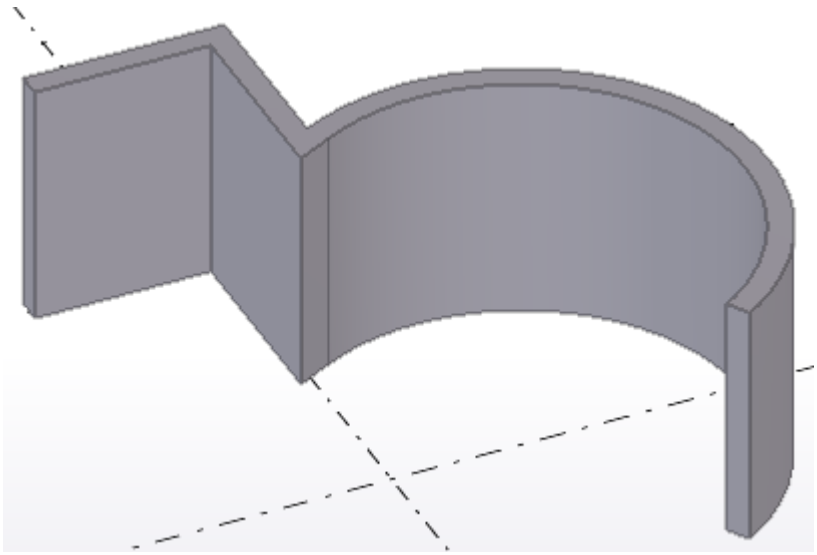
1. On the **Concrete** tab, click **Panel** .
2. Pick the points you want the panel or wall to go through.
3. Click the middle mouse button.

Tekla Structures creates the panel or wall using the **Concrete panel** properties in the property pane. Note that you cannot create a closed panel or wall structure.



4. If you want to create curved segments, chamfer the corners of the panel or wall.

   For example:



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button and select **Concrete panel** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify concrete panel or wall properties

1. If the property pane is not open, double-click the panel or the wall to open the **Concrete panel** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete panel or wall properties

Use the **Concrete panel** properties in the property pane to view and modify the properties of a concrete panel or a wall. To open the properties, double-click the panel or the wall. The file name extension of a concrete panel property file is `*.cpn`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the panel. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the panel (thickness × height of the wall).
Material	Material (page 334) of the panel.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group panels. For example, you can display parts of different classes in different colors.
Position	
On plane	Panel's position on the work plane (page 340), relative to the panel's reference line.
Rotation	Rotation (page 342) of the panel around its axis on the work plane.
At depth	Position depth (page 343) of the panel. The position is always perpendicular to the work plane.
Bind to levels	

Setting	Description
Top level	Use to bind (page 350) the top face of the panel to a selected grid level. Set the offset if needed.
Bottom level	Use to bind (page 350) the bottom face of the panel to a selected grid level. Set the offset if needed.
End offset	
Dx	Change the length of the panel (page 348) by moving the beam end point along the panel's reference line.
Dy	Move the panel end (page 348) perpendicular to the panel's reference line.
Dz	Move the panel end (page 348) in the z direction of the work plane.
Cast unit	
Cast unit numbering	Cast unit prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the panel or wall is precast or cast in place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Bending	
Plane	Plane of curvature.
Radius	Radius of the curved panel.
Number of segments	Number of segments Tekla Structures uses when drawing the curved panel.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
IFC export	

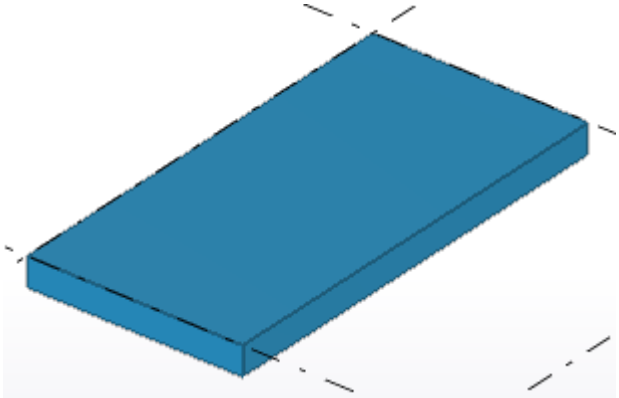
Setting	Description
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
Subtype (IFC4)	
User-defined type (IFC4)	
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete slab

When you create a concrete slab, the profile you select defines the thickness of the slab and the points you pick define the shape. The corners of the slab can be chamfered.

1. On the **Concrete** tab, click **Slab** .
2. Pick the corner points of the slab.
3. Click the middle mouse button.

   Tekla Structures creates the slab, using the **Concrete slab** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.

2. In the property pane, click the **Object type list** button and select **Concrete slab** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

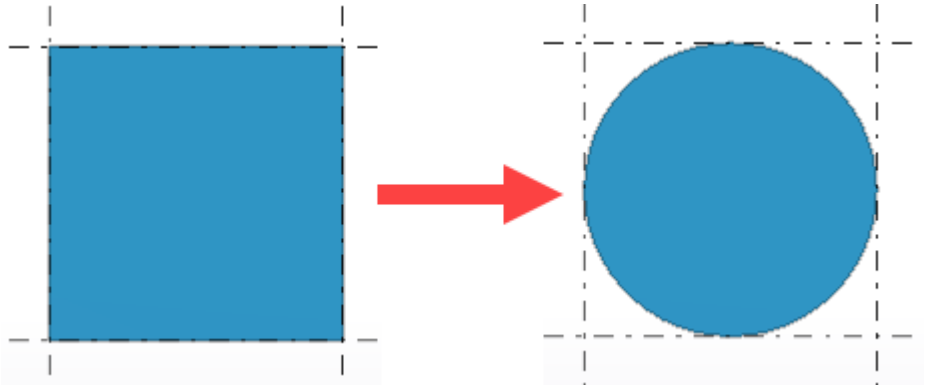
Create a round concrete slab

1. Create a square slab with four equal sides.
2. Select the plate.
3. Double-click a handle.

   To make it easier to select the handles of the slab corners, ensure that the Direct modification switch is **not** active.

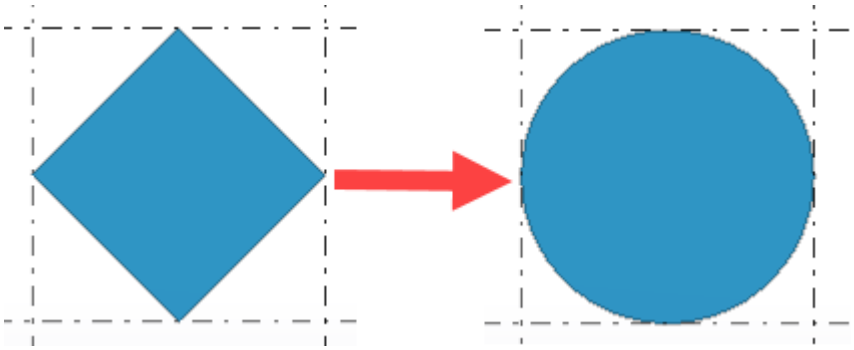
   The **Concrete slab corner chamfer** properties open in the property pane.
4. In the **Type** list, select **Rounding** .
5. In the **Radius** box, enter the chamfer radius.

   The radius must be equal to half of the side of the square.
6. Click **Modify**.
7. Repeat the above steps for each corner you want to chamfer.



Alternative way of creating a round slab

1. Create a diamond-shaped slab with four equal sides.
2. To round the corners, use the **Arc point** chamfer type.



Modify concrete slab properties

1. If the property pane is not open, double-click the slab to open the **Concrete slab** properties.
2. Change the properties as needed.
3. Click **Modify**.

Concrete slab properties

Use the **Concrete slab** properties in the property pane to view and modify the properties of a concrete slab. To open the properties, double-click the concrete slab. The file name extension of a concrete slab property file is `*.csl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the slab. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Thickness	Thickness of the slab.
Material	Material (page 334) of the slab.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group slabs. For example, you can display parts of different classes in different colors.
Position	

Setting	Description
At depth	Position depth (page 343) of the concrete slab. The position is always perpendicular to the work plane.
Bind to levels	
Level	Use to bind (page 350) the top face of the concrete slab to a selected grid level. Set the offset if needed.
Cast unit	
Cast unit numbering	Cast prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the slab is precast or cast in place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete lofted slab

With lofted slabs you can create curved and double-curved slabs or walls, for example.

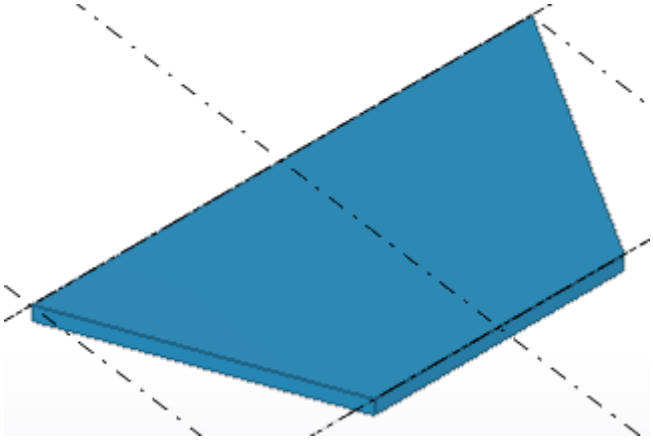
Prerequisites and examples of lofted slabs

Before you can create lofted slabs, you need to have construction objects (page 650) in your model. Tekla Structures creates the shape of the lofted slab according to the geometry of the used construction objects, by connecting the start point of the first construction object to the start point of the second construction object. The end points of construction objects are connected in similar way.

You can connect the following construction objects as a lofted slab:

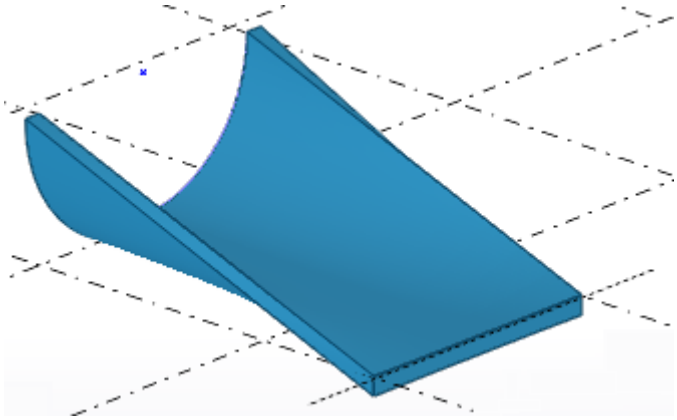
- construction line to construction line

  For example:



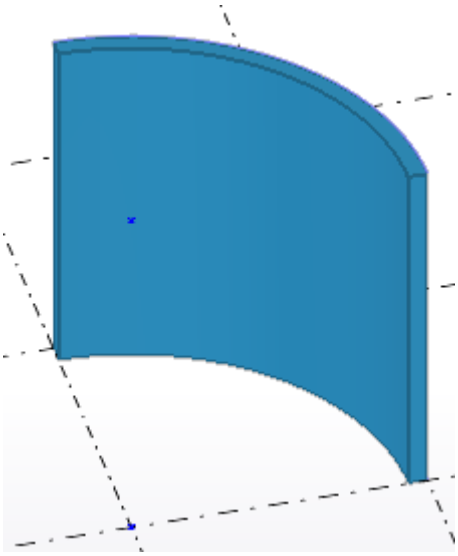
- construction line to construction arc

  For example:



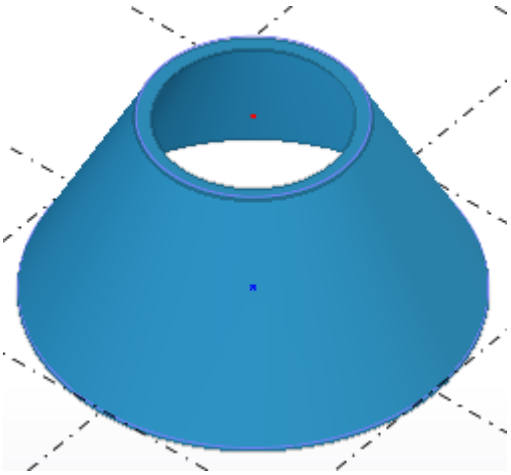
- construction arc to construction arc

  For example:



- construction circle to construction circle

  For example:



- construction polycurve to construction polycurve



Note that rebar sets do not work with lofted slabs.

Create a lofted slab

1. Create the needed construction objects in the model. The shape of the lofted slab is based on the construction objects' shape.

   You need to have

   - construction lines (page 651)
   - construction arcs (page 653)
   - construction circles (page 652)

     or

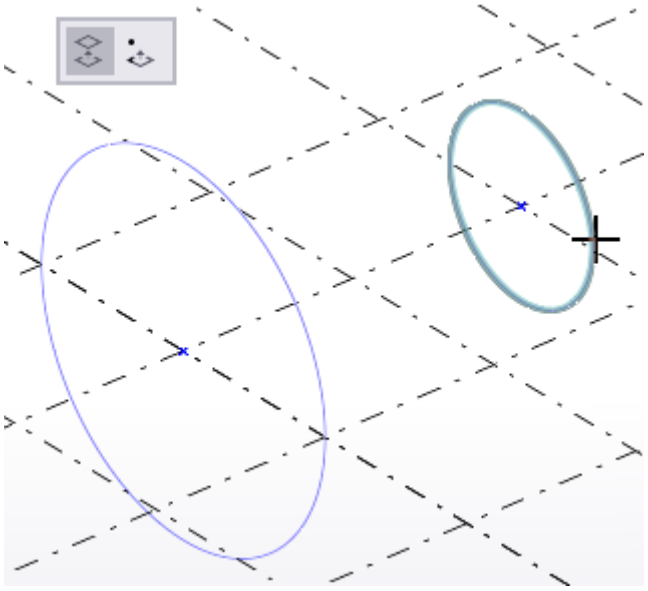
   - construction polycurves (page 655)

     With polycurves, use the **Create arc by tangent** or **Create tangent line** options  on the construction polycurve toolbar. To create polycurves only with straight segments, use the **Create line** option . Note that the construction polycurves do not need to have the same number of segments, as long as both of them are tangential.
2. When you have created the needed construction objects, go to the **Concrete** tab and click **Slab** --> **Create lofted slab**.
3. On the toolbar that appears, click a button to specify whether to create the lofted slab by using two construction objects, or by using a construction object and a point.

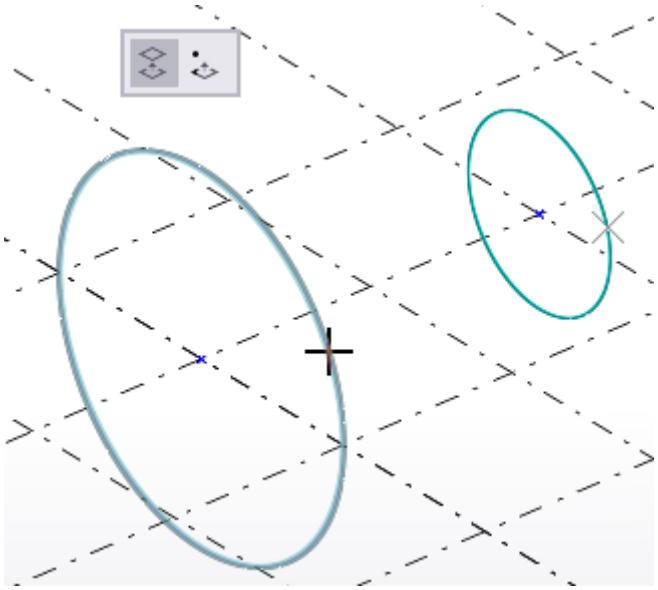
   - Use two construction objects  to create a lofted slab:

a. Select the first construction object: line, arc, circle, or polycurve.

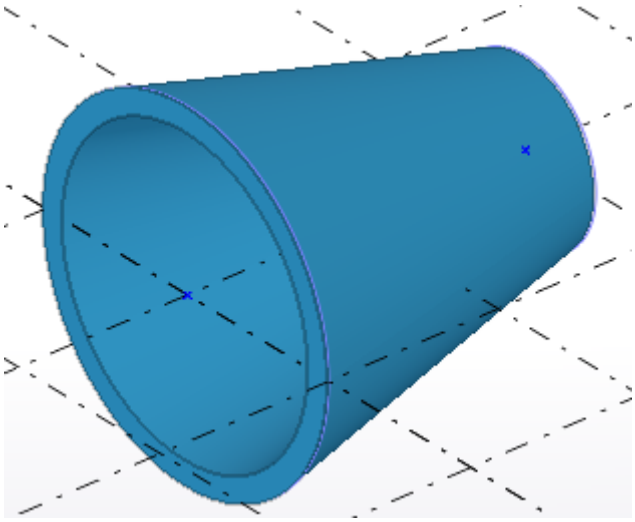
   For example, if you are using two construction circles to create a lofted slab:



b. Select the second construction object:

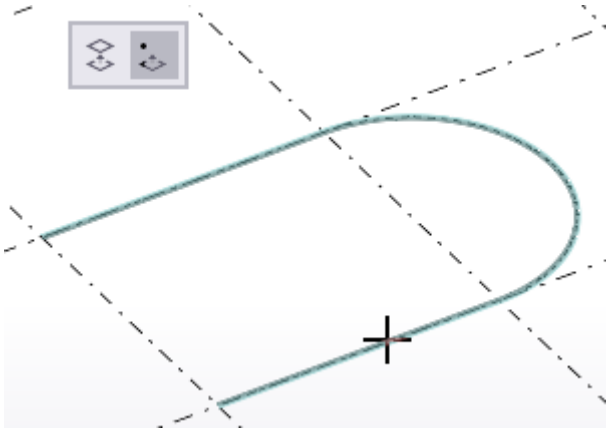


   Tekla Structures creates the lofted slab between the selected construction objects, using the **Lofted slab** properties in the property pane.



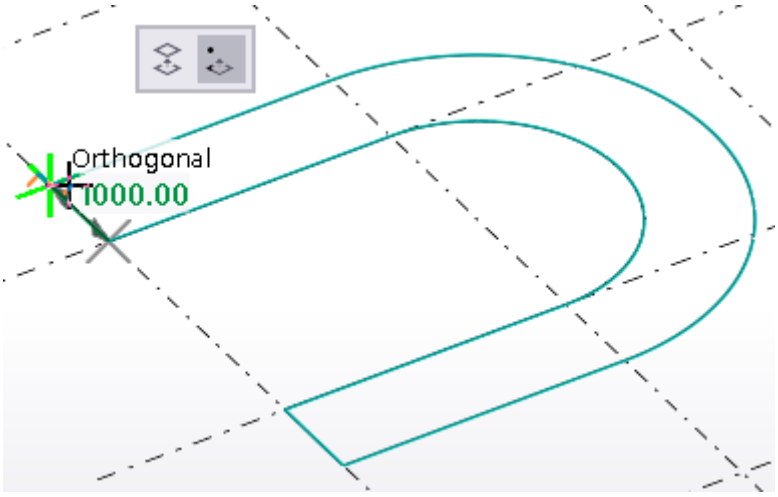
- Use one construction object and a point  to create a lofted slab:
  a. Select the first construction object: line, arc, circle, or polycurve.

     For example, if you are using a construction polycurve and a point to create a lofted slab:

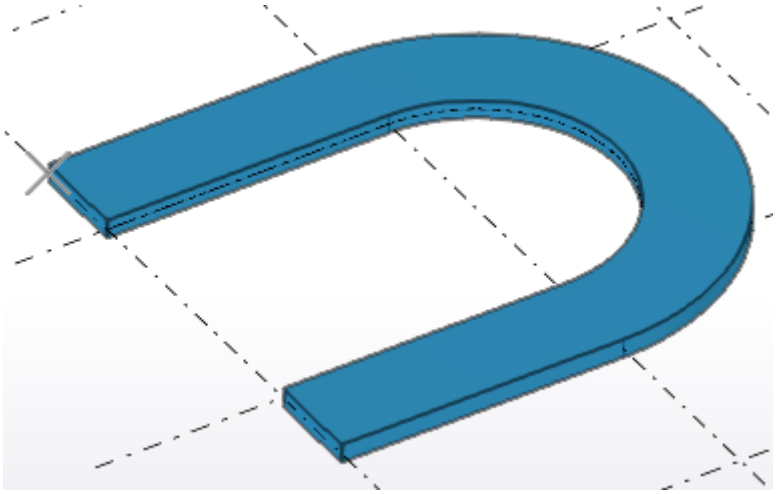
     

     Tekla Structures shows a preview of the part geometry. Use the preview to set the direction and height of the lofted slab.

b. Pick a point.



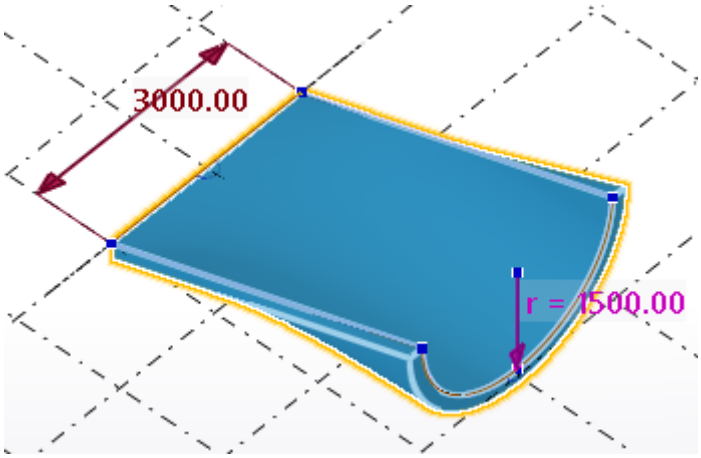
Tekla Structures creates the lofted slab based on the preview.

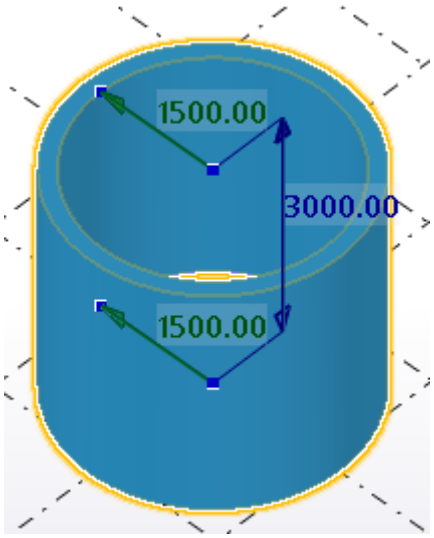


Modify the shape of a lofted slab

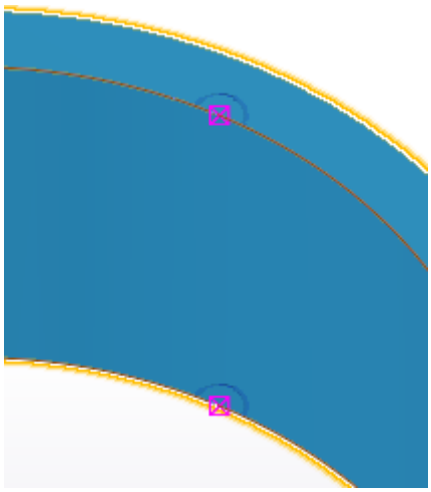
Use direct modification dimension handles and dimension values to modify the lofted slab shape.

- Modify, for example, the height and radius of the lofted slab.





- For lines and arcs: drag the arc symbol  at the midpoint of a line or an arc to modify the lofted slab shape.



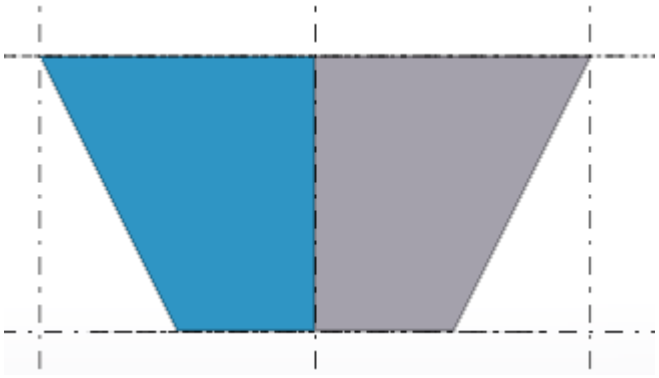
Split a lofted slab

Note that you cannot split closed cylindrical or conical lofted slabs.

1. On the **Edit** tab, click **Split**.
2. Select the lofted slab you want to split.
3. Pick a point for the dividing line.

   Tekla Structures splits the lofted slab.

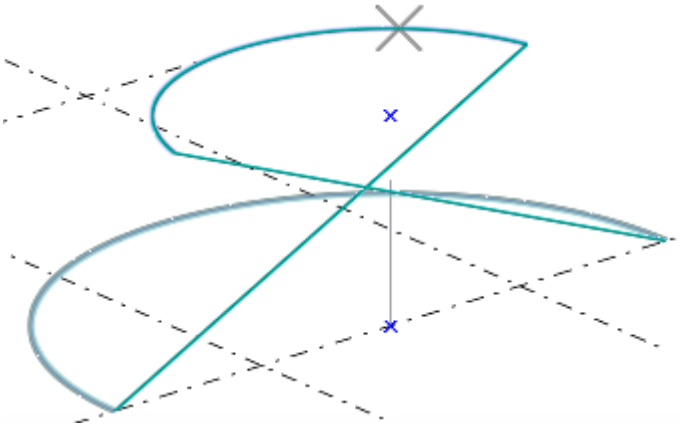
   For example:



Swap the end handle points to correct the geometry of a lofted slab

In some cases when you try to create a lofted slab, the geometry of the slab would become self-intersecting, meaning that the start and end points of the

top and bottom construction object are opposite of each other. In these cases the slab is not created.



You can try to resolve the situation and create the lofted slab by changing the modeling direction of the construction lines or arcs.

1. Select the construction line or the construction arc.

   With construction lines, ensure that the **Direct modification** switch is **not** active. You can then see the yellow and magenta object handles.

2. On the contextual toolbar, click **Swap ends**.

   Tekla Structures changes the modeling direction of the selected construction object, and the lofted slab can be created correctly.

With construction circles you can try to resolve the situation by moving either of the circles.

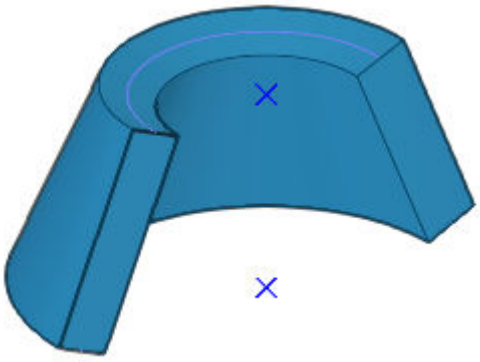
Modify concrete lofted slab properties

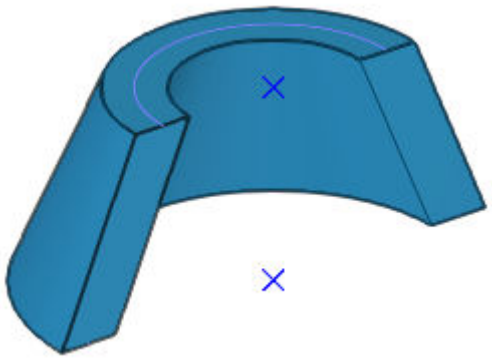
1. If the property pane is not open, double-click the lofted slab to open the **Lofted slab** properties.
2. Change the properties as needed.
3. Click **Modify**.

Lofted slab properties

Use the **Lofted slab** properties in the property pane to view and modify the properties of a concrete lofted slab. To open the properties, double-click the concrete lofted slab. The file name extension of a concrete lofted slab property file is `*.lsl`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the lofted slab. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Thickness	Thickness of the slab.
Material	Material (page 334) of the lofted slab.
Finish	Type of finish.
Class	Use to group lofted slabs. For example, you can display parts of different classes in different colors.
Face type	
Face type	Select whether the top and bottom faces of the slab are flush with the plane. **Perpendicular**:  The top and bottom faces of the slab are non-linear.

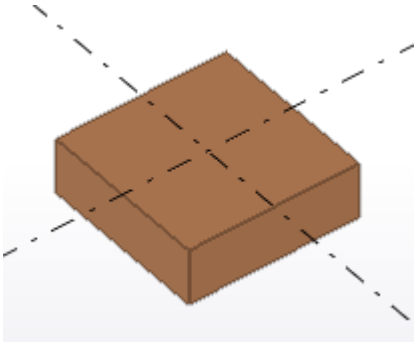
Setting	Description
	Bounded by curved planes:  The top and bottom faces of the slab are planar.
Cast unit	
Cast unit numbering	Cast unit prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the slab is precast or cast in place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
IFC export	
IFC entity	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
User-defined type (IFC4)	
More	

Setting	Description
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a pad footing

1. On the **Concrete** tab, click .
2. Pick a point.

   Tekla Structures creates the pad footing using the **Pad footing** properties in the property pane, and at the level (page 349) defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Pad footing** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify pad footing properties

1. If the property pane is not open, double-click the pad footing to open the **Pad footing** properties.
2. Change the properties as needed.

   For example, to create a circular pad footing, select a circular section for **Profile**.
3. Click **Modify**.

Pad footing properties

Use the **Pad footing** properties in the property pane to view and modify the properties of a pad footing. To open the properties, double-click the pad footing. The file name extension of a pad footing property file is `*.cpf`.

If you have customized the property pane layout, the list of properties may be different.

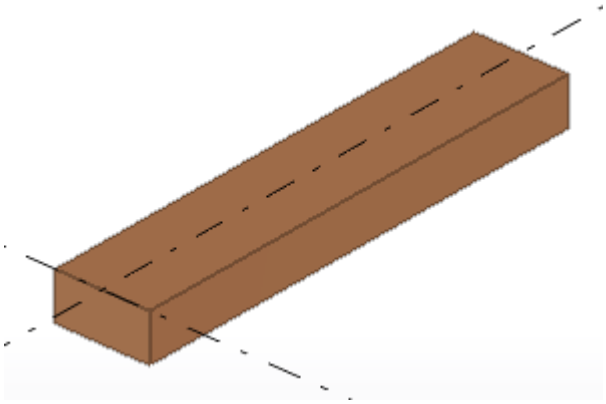
Setting	Description
General	
Name	User-definable name of the pad footing. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the pad footing.
Material	Material (page 334) of the pad footing.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group pad footings. For example, you can display parts of different classes in different colors.
Position	
Vertical	Vertical position (page 344) of the pad footing, relative to pad footing's reference point.
Rotation	Rotation (page 342) of the pad footing around its axis on the work plane.
Horizontal	Horizontal position (page 346) of the pad footing, relative to pad footing's reference point.
Top	Position of the top surface of the pad footing in the global z direction.
Bottom	Position of the bottom surface of the pad footing in the global z direction.
Bind to levels	
Level	Use to bind (page 350) the top face of the pad footing to a selected grid level. Set the offset if needed.
Cast unit	

Setting	Description
Cast unit numbering	Cast unit prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the pad footing is precast or cast-in-place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create a strip footing

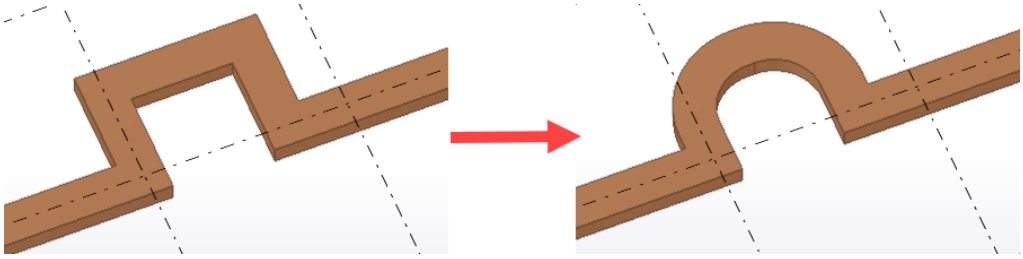
1. On the **Concrete** tab, click **Footing** --> **Strip footing**.
2. Pick the points you want the footing to go through.
3. Click the middle mouse button.

Tekla Structures creates the strip footing between the points you picked using the **Strip footing** properties in the property pane. Note that you cannot create a closed strip footing.



4. If you want to create curved segments, chamfer the corners of the footing.

   For example:



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button and select **Strip footing** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

Modify strip footing properties

1. If the property pane is not open, double-click the strip footing to open the **Strip footing** properties.
2. Change the properties as needed.
3. Click **Modify**.

Strip footing properties

Use the **Strip footing** properties in the property pane to view and modify the properties of a strip footing. To open the properties, double-click the strip footing. The file name extension of a strip footing property file is `*.csf`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the strip footing. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Profile	Profile (page 332) of the strip footing.
Material	Material (page 334) of the strip footing.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group strip footings. For example, you can display parts of different classes in different colors.
Position	
On plane	Strip footing's position on the work plane (page 340), relative to the footing's reference line.
Rotation	Rotation (page 342) of the strip footing around its axis on the work plane.
At depth	Position depth (page 343) of the strip footing. The position is always perpendicular to the work plane.
Bind to levels	
Level	Use to bind (page 350) the top face of the strip footing to a selected grid level. Set the offset if needed.
End offset	
Dx	Change the length of the strip footing (page 348) by moving the footing end point along the footing's reference line.
Dy	Move the strip footing end (page 348) perpendicular to the beam's reference line.

Setting	Description
Dz	Move the strip footing (page 348) in the z direction of the work plane.
Cast unit	
Cast unit numbering	Cast unit prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the strip footing is precast or cast-in-place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Bending	
Plane	Plane of curvature.
Radius	Radius of the curved strip footing.
Number of segments	Number of segments Tekla Structures uses when drawing the curved strip footing.
Concrete covers for rebar sets	
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the part. The available subtypes depend on the selected IFC entity. You can start typing the name of the IFC entity and the matching entities are shown in the list. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Create items

In Tekla Structures, the term *item* refers to the parts that have a 3D *shape*. Shapes are created in an external modeling software, or in Tekla Structures, and they are available in the Tekla Structures shape catalog.

Items are similar to other parts (page 217), such as beams and columns. The main difference between items and other types of parts is that a 3D shape defines the geometry of an item, whereas a part has a 2D profile that is extruded to create the length of the part.

You can use items to model objects that would otherwise be difficult to model using basic Tekla Structures parts and commands, such as cutting. You can also use items to model objects that use shapes modeled in an external software or by a manufacturer.

Every item has properties that define it, such as shape, material, and location. If you want to use item properties in view and selection filters or in drawing and report templates, you need to use the template attributes of parts and profiles. If you want to separate items from parts, use the IS_ITEM template attribute.

Limitations to items

- Items cannot be mirrored.
- Items cannot be split or combined. Splitting an imported item creates a duplicate to the splitting position.
- Items can only be cut or attached to another part if they have a solid shape.
- The gross weight value of an imported item may be different from that of an identical Tekla Structures part modeled with cuts. This is because the cuts are not taken into account when calculating the gross weight of parts.

Create an item or a concrete item

1. Ensure that  **Direct modification** is switched on.

   This makes it easier to adjust the location and rotation of the item in the model.
2. Depending on the material of the item you want to create, do one of the following:

   - On the **Steel** tab, click **Item** .
   - On the **Concrete** tab, click **Item** .

TIP Alternatively, you can start the command in the property pane.

a. Ensure that you have nothing selected in the model.

b. In the property pane, click the **Object type list** button and select **Item** or **Concrete item** from the list.

   Tekla Structures starts the command and shows the properties in the property pane.

   In the property pane, you can modify the item properties and select the shape, for example.

3. Move the mouse pointer over object faces and edges in the model, and see how the item turns over and adjusts to the object faces.

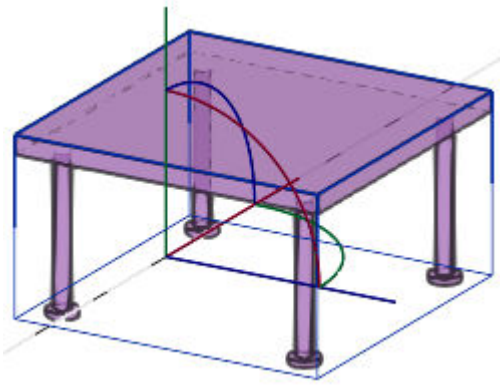
   If you are adding the item to another object (such as part or pour object), Tekla Structures also shows location dimensions to the nearest object edges.

4. Pick the first point of the item.

5. To indicate the direction of the item, pick another point.

   Tekla Structures places the item between the points you picked starting from the first point (yellow handle) towards the direction of the second point (magenta handle), using the **Item** or **Concrete item** properties in the property pane.

   Tekla Structures shows the coordinate axes, rotation handles, and location dimensions that you can use to fine-tune the location and rotation of the item. The handles are red, green, and blue, according to the local coordinate system of the item.



6. To move the item along any of its coordinate axes, drag the relevant axis handle to a new location.

7. To rotate the item around any of its coordinate axes, drag the relevant rotation handle to a new location.

   Press **Tab** to rotate the item in 90-degree steps in the direction of the selected rotation handle.

8. To move or rotate the item by specifying a distance or angle:
   a. Select an axis handle, a rotation handle, or a dimension arrowhead.
   b. Type the value by which you want the dimension to change.

      When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog.

   c. Click **OK** to confirm the new dimension.
9. If you want to add more items to the model, click the middle mouse button and repeat steps 3–8.
10. To stop adding items, press **Esc**.
11. If needed, you can modify the item geometry (page 325) further in the **Geometry editing** mode.

Modify item or concrete item properties

1. If the property pane is not open, double-click the item or the concrete item to open the **Item** or the **Concrete item** properties.
2. Change the properties as needed.
3. Click **Modify**.

Change the shape of an item

When you create or modify an item, you can select the shape from the list that contains all the shapes available in the shape catalog.

Before you start, ensure that you have the required shape available in the shape catalog.

1. Double-click an item to open the item properties in the property pane.
2. Click the **...** button next to the **Shape** box to open the **Select shape** dialog.
3. On the left side of the dialog, select a group or sub-group.

   TIP The latest shapes you have used can be found in the **Recent** group, and the shapes you have marked as important are in the **Starred** group.

4. If needed, use the **Filter** box to search for a shape in the selected group.
5. Select a shape from the list.

6. Click **OK** to close the **Select shape** dialog.
7. Click **Modify** in the property pane to apply the changes.

Convert a part to an item

You can change existing parts in the model to items and at the same time add corresponding shapes to the shape catalog.

When you change a part to an item, Tekla Structures deletes the original part and replaces it with the newly created item in the model. The name, material, finish, class, pour phase, and the numbering properties of the original part are saved as the corresponding item properties. Other part type specific properties and user-defined attributes are not saved. The objects that are attached to the original part, such as reinforcement and surfaces, are deleted.

1. Create the parts (page 217) that you want to change to an item.
2. If you want to include more than one part in the item, attach the parts to each other (page 359).
3. On the **Edit** tab, click  **Convert part to item**.
4. Select the part.

   Alternatively, you can first select the part, right-click, and then select **Convert part to item**.

Tekla Structures changes the part to an item and adds a new shape to the shape catalog.

The shape name is generated using the part name and part location in the format <grid location>_<elevation>_<part name>. For example:

- 1/D_+0_FOOTING
- 3/C_+0-+3600_COLUMN
- 1-2/A-B_+3600_SLAB

If there is already a shape with the same name in the shape catalog, Tekla Structures adds two underscore characters and a running number at the end of the new shape name. For example, 1/D_+0_FOOTING__1.

Item and concrete item properties

Use the **Item** and the **Concrete item** properties in the property pane to define, view, and modify the properties of an item. To open the properties, double-click the item or the concrete item.

The file name extension of an item property file is `*.ips`.

The file name extension of a concrete item property file is `*.ipc`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of an item. The name can contain a maximum of 61 characters. Tekla Structures uses part names in reports and in **Document manager**, and to identify parts of the same type.
Shape	Shape of an item. To select a shape from the shape catalog, click the **...** button next to the **Shape** box. To show the item shape in reports and drawing tables, use the `PROFILE` template attribute.
Material	Material (page 334) of the item.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group items. For example, you can display parts of different classes in different colors.
Numbering series (available for items)	
Part numbering	Part prefix and start number for the part position number (page 753).
Assembly numbering	Assembly prefix and start number for the assembly position number (page 753).
Position	
On plane	Item's position on the work plane (page 340), relative to the item's reference line.
Rotation	Rotation (page 342) of the item around its axis on the work plane.
At depth	Position depth (page 343) of the item. The position is always perpendicular to the work plane.
End offset	
Dx	Move the item along (page 348) its reference line.
Dy	Move the item perpendicular (page 348) to its reference line.

Setting	Description
Dz	Move the item in the z direction (page 348) of the work plane.
Cast unit (available for concrete items)	
Cast unit numbering	Cast unit prefix and start number for the cast unit position number (page 753).
Cast unit	Indicate whether the item is precast or cast-in-place.
Pour phase	Pour phase (page 474) of the cast-in-place parts. Use to separate pour objects from one another.
Concrete covers for rebar sets (available for concrete items)	
Coordinate system	Select whether the concrete cover thickness (page 588) of the rebar sets in the part is defined in the global coordinate system, or in the part's local coordinate system. The default global and local cover thickness values are defined in the **Options** dialog. If you select the empty option, Tekla Structures uses the global values.
Top, **Bottom**, **Sides**, **Front** , **Back**, **Start**, **End**	To override global or local default values from the **Options** dialog, define the cover thickness at each required part face.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (page 356) (UDAs) of the part. UDAs provide additional information about the part.

Modify items

You can fine-tune the geometry of the model by modifying items. This is how you can create sloping slabs, for example.

TIP You can create item shapes using existing geometry and parts in the model.

When you modify an item's geometry, you can move item vertexes (corner points), edges, and faces. However, the faces always need to stay planar, so you cannot move all vertexes or edges freely. You may first need to add new edges to split existing faces. These edges act as hinges or turning joints between the faces.

When you have modified an item's geometry and want to save the changes, you can choose to update the item's current shape, or to create a new shape. Tekla Structures also stores temporary shapes in the shape catalog in case you

need to undo actions. The unused temporary shapes will be deleted from the shape catalog when you save the model.

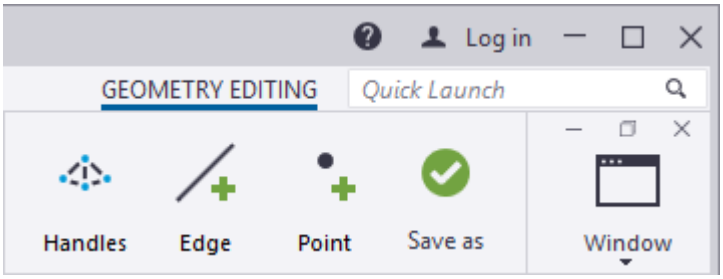
Start geometry editing

Before you start, create items (page 320).

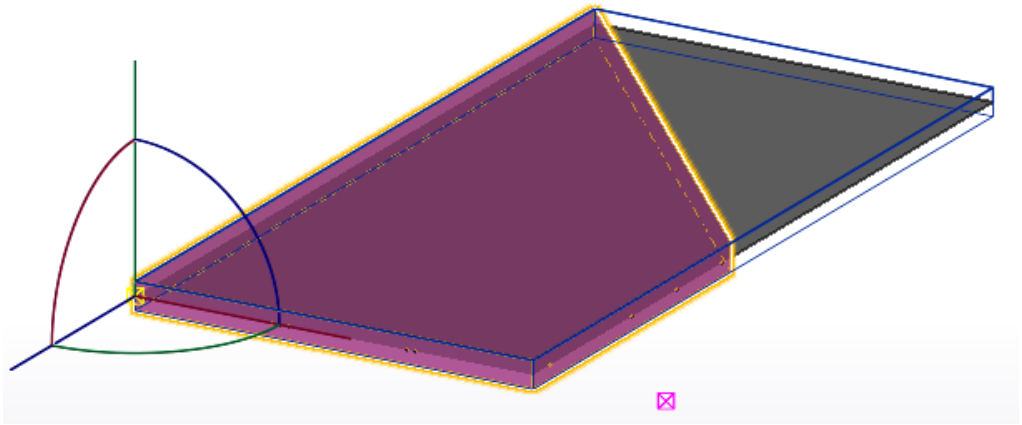
1. Ensure that  **Direct modification** is switched on.
2. Select the item whose geometry you want to modify.

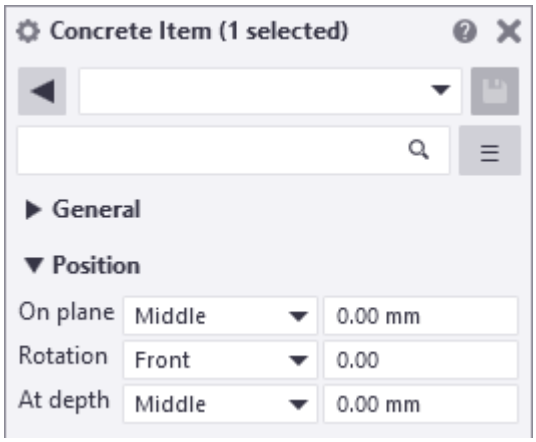
   You can only modify items that have a solid shape.

   The **Geometry editing** tab appears at the right end of the ribbon:

   Tekla Structures shows the direct modification axis and rotation handles for the selected item. For example:

   
3. Modify the position settings in the item properties (page 323) so that both **On plane** and **At depth** are set to **Middle** with offset 0.



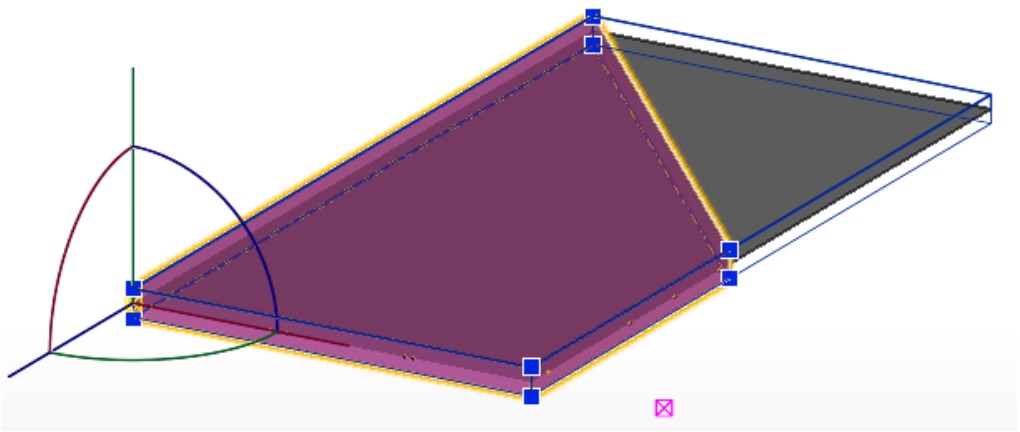
This ensures that the later geometry modifications are accurate.

4. If you need to adjust the item location in the model, drag the direct modification axis handles or location dimension arrowheads.

Modify the geometry of an item

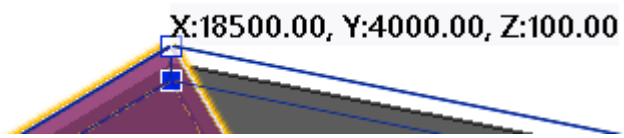
Before you start, ensure that you have the **Geometry editing** tab open and the correct item selected.

1. Click **Handles** to display handles for the selected item. For example:



2. Move the mouse pointer over the selected item to highlight its faces, edges, and vertexes.
3. To check the coordinates of a vertex, select the vertex.

Tekla Structures shows the absolute, work plane based coordinates in the model view. For example:



4. To move a face, edge, or vertex, drag it to a new location.

   Alternatively, you can use numeric snapping (page 103) and the **Enter a Numeric Location** dialog.

   To move a selected vertex to an exact location, you must enter all three coordinates (x, y, and z) of the new location, for example `400,200,0`.

   Note that you can use a special character (page 107) in front of the coordinates to temporarily override the default (relative) snapping mode, for example `!` for global coordinates. With relative and absolute coordinates, you can also use coordinate axis prefixes `x`, `y`, and `z` to move a vertex or dragged edge in the prefixed directions only, for example `@z500` or `$y6000,z-500`.

5. To delete an edge or vertex, select it and press **Delete**.

   You can only delete edges whose both adjacent faces are in the same plane, and vertexes whose all adjacent faces are in the same plane.

   If the adjacent faces of an edge or vertex are not in the same plane, you can drag the edge or vertex on top of another one to combine them.

6. If you need to undo actions, press **Ctrl+Z**.

Add an edge to an item

You can add edges to the selected item, and so split existing faces. Create the new edges so that they do not cross over any existing edges.

Before you start, ensure that you have the **Geometry editing** tab open and the correct item selected.

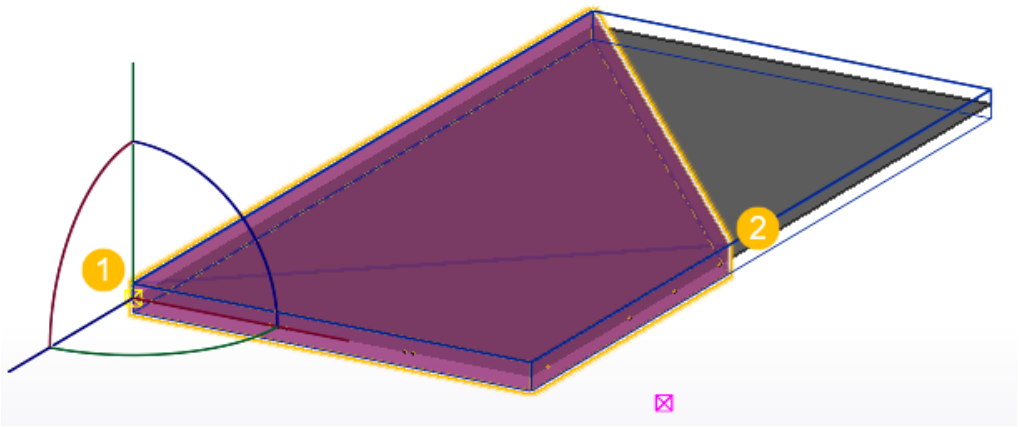
1. On the **Geometry editing** tab, click  **Edge**.
2. Pick the start point of the edge.

   The start point must be located on an existing edge or vertex.

3. Pick the end point of the edge.

   The end point must be located on a vertex or on the same face as the start point, but on another edge. If you try to pick an invalid point, Tekla Structures shows a preview of the edge in red.

   Tekla Structures creates the edge between the points you picked. For example:



   (1) Start point

   (2) End point

4. If you want to add more edges, repeat steps 2 and 3.
5. To stop adding edges, press **Esc**.

Add a vertex to an item

You can add vertexes and related edges to the faces of the selected item. You can use the vertexes as drainage points when you create sloping slabs and plan drainage, for example.

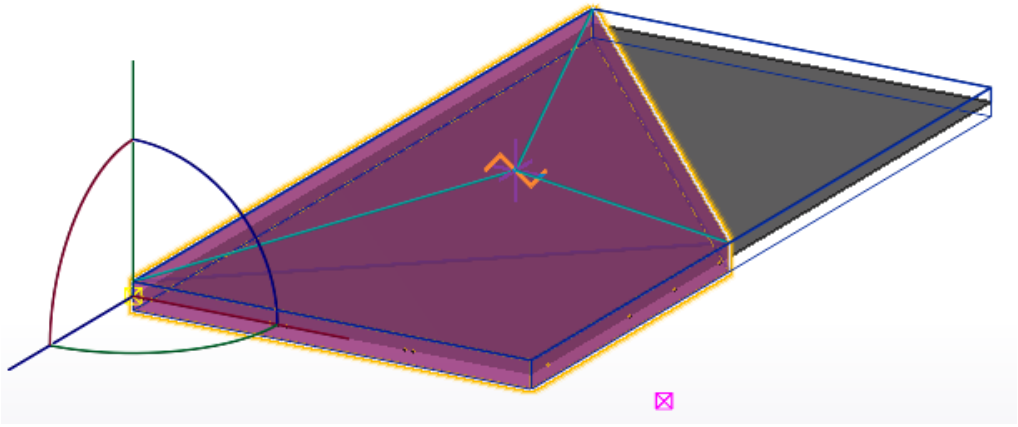
Note that you cannot add vertexes to faces that have openings or recesses if these openings are part of the item **shape**. However, if only the item in the model has been cut, and not its shape, you can add vertexes to these faces.

Before you start, ensure that you have the **Geometry editing** tab open and the correct item selected.

1. On the **Geometry editing** tab, click  **Point**.
2. Move the mouse pointer over the faces of the selected item.

   In convex-polygonal areas, where all internal angles at face corners are less than 180 degrees, Tekla Structures shows a preview of the edges that

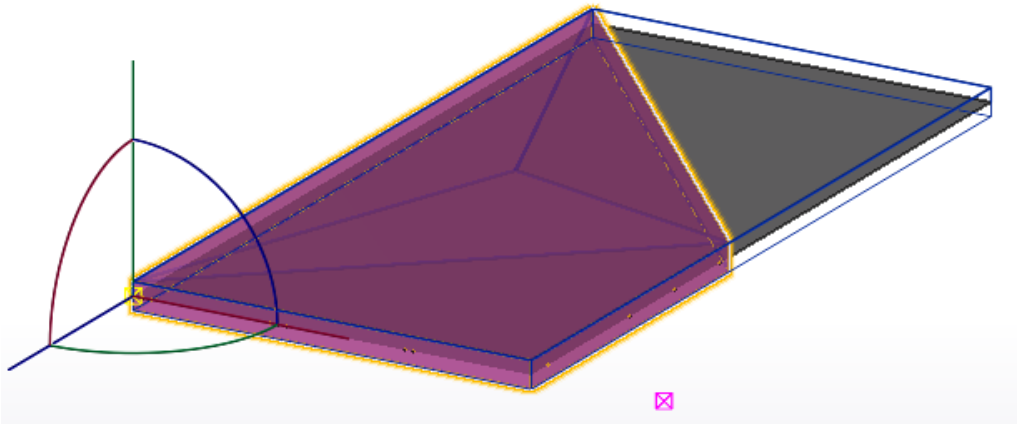
start from each face corner and join at the current location of the mouse pointer. For example:



If the item faces have concave-polygonal areas, where at least one internal angle is larger than 180 degrees, Tekla Structures automatically shows and adds edges that separate the concave-polygonal areas from the convex-polygonal areas.

3. Pick a point on the desired face.

   Tekla Structures creates the vertex and related edges that join the new vertex to all existing vertexes of the same face. For example:



4. If you want to add more vertexes, pick more points.
5. To stop adding vertexes, press **Esc**.

Save a modified item and shape

When you save modified items in the model, you can choose to update the selected item's current shape, or to create a new shape for later use.

Before you start, ensure that you have the **Geometry editing** tab open and the correct item selected.

1. On the **Geometry editing** tab, click  **Save as**.

   The **Save as** dialog opens.

2. Select one of the following options:

   - **Update current shape to use this geometry**

     to update the current shape, the modified item, and all the items using that shape in the model.

     If all items are not updated in the model, save and re-open the model.

   - **Create a new shape in the shape catalog with the name**

     to create a new shape in the shape catalog, and to update the modified item in the model to use the new shape.

3. If you chose to create a new shape, enter a name for the shape.
4. Click **Save**.

   Tekla Structures updates the items in the model and saves the shape in the shape catalog.

The temporary shapes that were created during the editing process will be deleted from the shape catalog when you save the model if the shapes are not used by any item.

Create timber objects

Tools for timber detailing workflows are available in Tekla Structures as an extension package.

Download the extension package from Tekla Warehouse, on the Timber tools page. The extension package contains timber objects, screws, and marker tool.

Timber objects

The extension includes the basic timber objects, including timber beams, timber columns, timber plates, timber walls, and timber slabs as a ribbon tab. Each timber object type is modeled similarly to its steel or concrete counterpart, with properties and parameters specific to timber objects.

Screws

The extension includes screw groups for screw objects. Each screw group is modeled similarly to its bolt counterpart, with properties and parameters

specific to screw objects. The package also includes screw presentation for drawings, providing an exact representation of the location and threading of the screw.

Marker tool

Use the marker tool to create 3D symbols and text on objects. You can communicate the lamella direction of mass timber panels with a 3D symbol to ensure alignment with the CNC operator, or indicate the bearing direction of slabs or metal grating.

Modify parts and part properties further

Every part has properties that define it, such as material, profile, and location. Also, parts have user-defined attributes (UDAs) that you can use to provide additional information about a part. These properties can be set when the parts are created, or they can be modified later.

Also, you can modify part shape further by using the part properties in the property pane, such as using the deforming options to warp and camber parts.

Additionally, you can split and combine parts.

Select and change the profile or material of a part (page 332)

Adjust part position (page 334)

Camber a part (page 353)

Warp a part (page 354)

Examples of user-defined attributes (UDAs) for parts (page 356)

Create curved beams, panels, or strip footings (page 357)

Combine parts (page 358)

Attach parts to each other (page 359)

Split parts (page 361)

Select and change the profile or material of a part

Each part has a profile and a material, which are selected using the profile catalog and the material catalog.

Select and change the profile of a part

There are two types of profiles available for parts in Tekla Structures:

- *Fixed profiles*

  Fixed profiles are profiles that can be obtained pre-manufactured. The properties of the fixed profiles conform to industry standards, and you should not modify them unless you are an administrator. Fixed profiles are environment-specific.

- *Parametric profiles*

  Parametric profiles are partly user-definable: they have a predefined shape but you can change their cross section dimensions using one or more parameters. Tekla Structures calculates the cross section shape each time you open the model.

You can use the fixed or parametric profiles available in the Tekla Structures profile catalog, or you can customize the profile catalog in several ways.

Change the profile of a part

When you create or modify a part (page 217), you can select the profile of a part from a list that contains all the profiles available in the profile catalog.

1. Double-click a part to open the part properties in the property pane.
2. Click the **...** button next to the **Profile** box.

   The **Select Profile** dialog appears.

   By default, only the profile types that are relevant to the material of the part are shown. For example, if you are changing the profile of a steel part, only the profile types that are associated with steel are shown.
3. Select a profile from the list.
4. If the profile is parametric, define its dimensions on the **General** tab.

   Click the **Value** box and replace the existing value with a new one.
5. Click **OK** to close the **Select Profile** dialog.
6. Click **Modify** in the property pane.

Alternatively, if you know the name of the profile, you can enter it directly in the **Profile** box in the property pane or in the contextual toolbar.

Use standardized values for profile dimensions

You can use standardized values for the dimensions of parametric profiles.

1. Double-click a part to open the part properties in the property pane.

2. Click the **...** button next to the **Profile** box.

   The **Select Profile** dialog appears.

3. Select a parametric profile.

   If standardized values have been defined for this profile, the **Use industry standardized values only** checkbox appears on the **General** tab under the profile properties.

4. Select the **Use industry standardized values only** checkbox.

5. Select the profile dimensions from a list in the **Value** column.

Select and change the material of a part

When you create or modify a part (page 217), you can select the material and grade of a part from a list that contains all the materials available in the material catalog.

1. Double-click a part to open the part properties in the property pane.

2. Click the **...** button next to the **Material** box.

   The **Select Material** dialog appears.

3. If needed, define what material information you want to see.

   - To include aliases for material grades in the list, select the **Show aliases** checkbox.

     Aliases are alternative names. For example, they can be former names, or names used in different countries or standards. Tekla Structures automatically changes aliases into the standard name when you select a material grade.

   - To see all the properties of materials, select the **Show details** checkbox.

4. Select a material from the list.

5. Click **OK** to close the **Select Material** dialog.

6. Click **Modify** in the property pane.

Alternatively, if you know the name of the profile, you can enter it directly in the **Profile** box in the property pane or in the contextual toolbar.

TIP If needed, you can customize the material catalog.

Adjust part position

When you create parts in Tekla Structures, understanding how parts are positioned helps to model consistently. Several aspects affect the positioning of parts.

Part reference points and part reference lines

When you create a part, you position the part by picking points. These points are the part reference points. If you pick two points to position a part, the points form a part reference line, and handles appear at the line ends. The position of a part is always relative to its reference points.

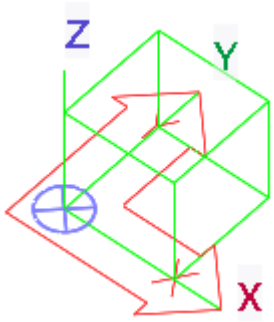
Changing the **Position** values in the part properties moves the profile around the reference line; it does not move the location of the reference line in the model.

All of the **Position** properties are based on the reference line looking from the start reference point towards the end reference point.

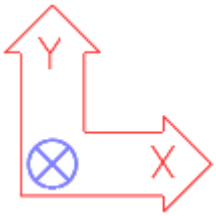
Coordinate systems

There are two different types of coordinate systems in Tekla Structures: the global and the local coordinate system.

Global coordinate system is the main reference for positiong objects in Tekla Structures. It is represented by a green cube symbol at the global point of origin (x=0, y=0, z=0), in the bottom corner of a new model. The global coordinate system is static and it cannot be changed.



The local coordinate system is also known as the current active work plane. It is represented both as a red coordinate arrow with the X and Y directions and as the symbol in the lower right corner of the model view. You can use the symbol in the lower right corner also for rotating the model view from 3D to a 2D plane by clicking the navigation control axes in the symbol.





The local coordinate system has a direct effect on the positioning values in the beam and column properties in the property pane. A value, which is used when the local coordinate system is set one way, gives completely different results if the local coordinate system is moved or changed.

In addition, the part internal coordinate system can have an effect to the part positioning. You can use the **Show coordinate system** macro in the **Applications & components** catalog to show the coordinate system for a part that you select.

Part faces

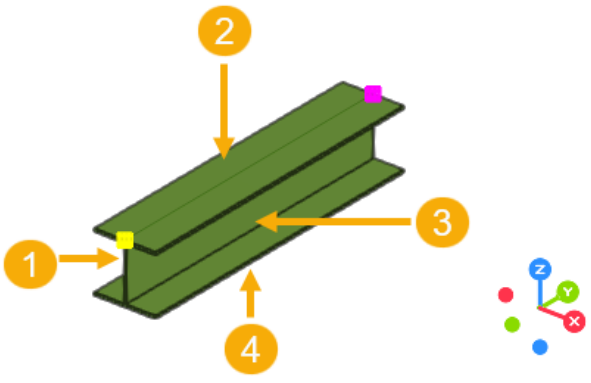
Tekla Structures parts have four faces: top, front, back, and below.

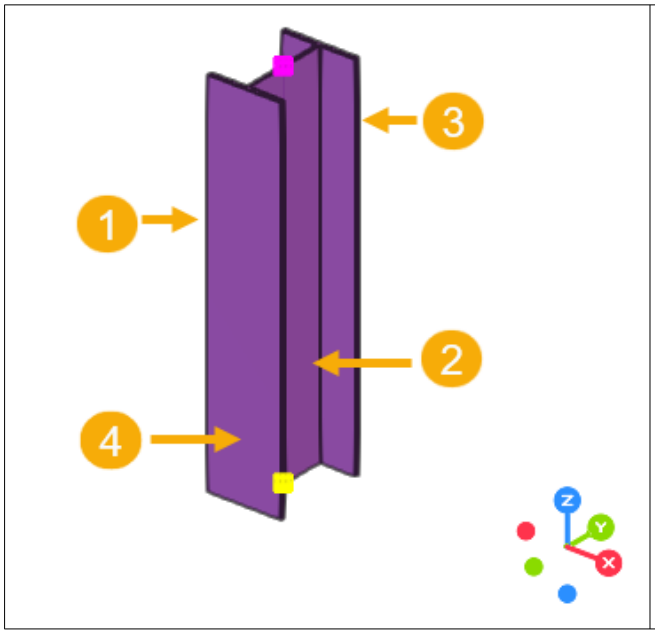
Part faces are represented as if looked from the yellow start handle towards the magenta end handle.

The beam and column properties in the property pane have the **Rotation** setting. The options for rotating the part around its axis on the work plane are **Top**, **Front**, **Back**, and **Below**.

With a default coordinate system where **Z** direction points up, the beam top face should face the positive **Z** direction.

Note that the faces do not change if the parts are modeled in differently: if you roll a beam over on its side, the top is still the top, it is just no longer facing up.

	**(1)**: **Back** **(2)**: **Top** **(3)**: **Front** **(4)**: **Below**

	**(1)**: **Front** **(2)**: **Back** **(3)**: **Top** **(4)**: **Below**

Beam properties

For parts that are modeled using the **Beam** command, all the positioning settings relate to the currently active **Z** direction.

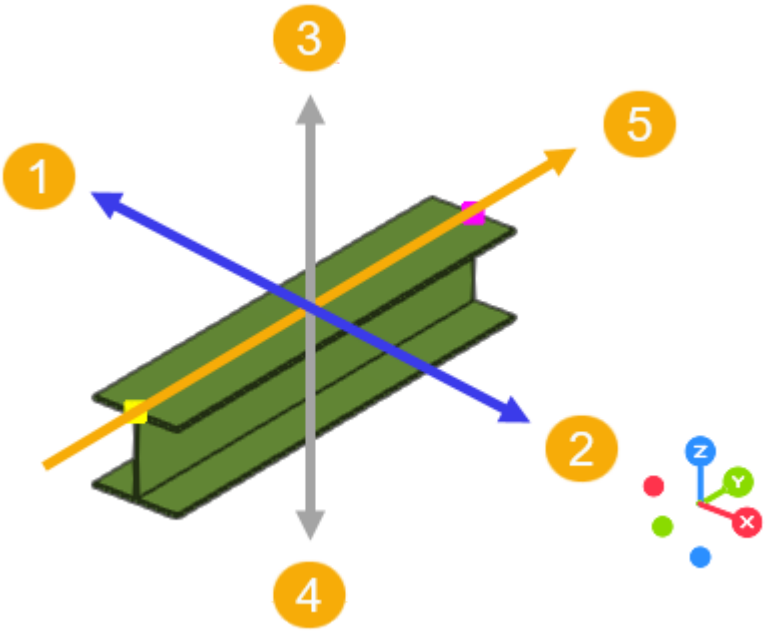
The beam properties have the following positioning options:

- **On plane** controls the right-left position, relative to the part reference line.

  The options are **Middle**, **Right**, or **Left**.
- **Rotation** controls which side faces the positive Z direction.

  The options are **Top**, **Front**, **Back**, and **Below**.
- **At depth** controls the up-down position, relative to the part reference line.

  The options are **Middle**, **Front**, and **Behind**.



(1): **On plane**: **Left**

(2): **On plane**: **Right**

(3): **At depth**: **Front**

(4): **At depth**: **Behind**

(5): reference line

Column properties

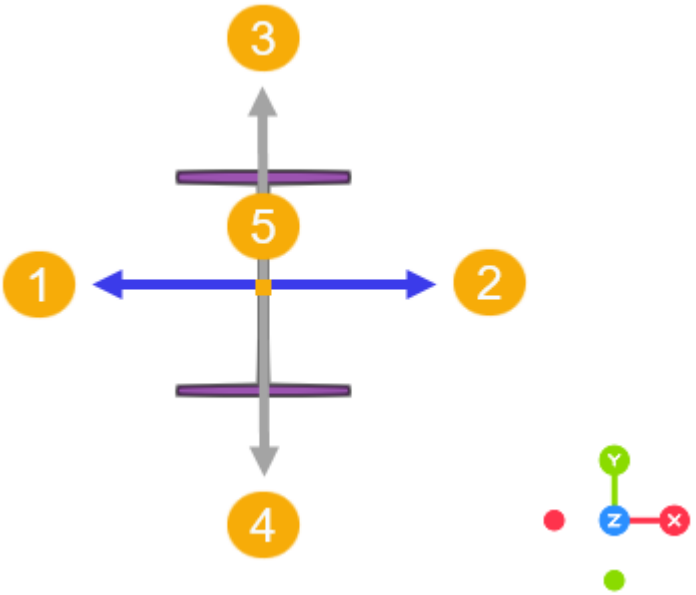
The column properties have the following positioning options:

- **Vertical** controls the up-down position, relative to the part reference point.

  The options are **Middle**, **Down**, or **Up**.
- **Rotation** controls which side faces the negative X direction.

  The options are **Top**, **Front**, **Back**, and **Below**.
- **Horizontal** controls the left-right position, relative to the part reference point.

  The options are **Middle**, **Left**, and **Right**.



(1): **Horizontal**: **Left**

(2): **Horizontal**: **Right**

(3): **Vertical**: **Up**

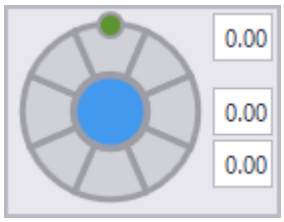
(4): **Vertical**: **Down**

(5): reference point

Modify the position of a part

Use the property pane and its **Position** and **End offset** sections to modify the part position, or use the options on the contextual toolbar.

To	Do this
Modify part position using the property pane	1. Double-click a part to open the part properties in the property pane. 2. In the **Position** or in the **End offset** section, modify the desired position settings, such as part rotation (page 342) or part vertical position (page 344). For example, you can define the part to be positioned 200 units above its handles. 3. Click **Modify**.
Modify part position using the contextual toolbar	1. Click  on the contextual toolbar. 2. Modify the settings. The object moves in the model accordingly. • To change the overall position of a beam, column, panel, or footing, use the round

To	Do this
	selection dial. Click a sector in the dial to select a position. • To change the rotation angle, click and drag the green rotation angle knob. The rotation angle knob snaps to every 5 degrees. Hold down **Shift** to override this. • To change the **Angle**, **Plane offset**, or **Depth offset**, enter a value in the corresponding box.  • To change the position of a plate or slab, select an option and enter a value in the **Depth offset** box. 

See also

Part reference lines and part handles (page 116)

Part position on the work plane (page 340)

Part rotation (page 342)

Part position depth (page 343)

Part vertical position (page 344)

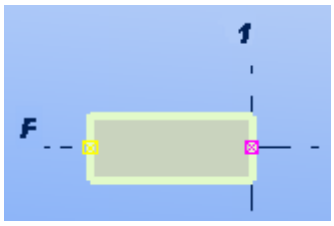
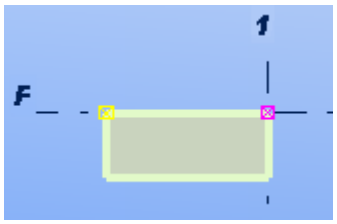
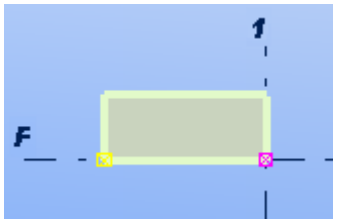
Part horizontal position (page 346)

Part end offsets (page 348)

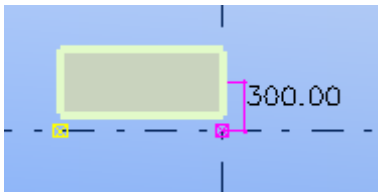
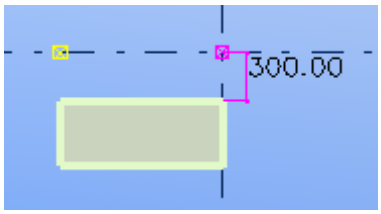
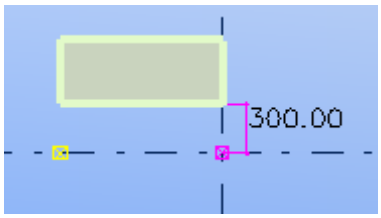
Part position on the work plane

Use the property pane and its **On plane** setting to view and change the part's position on the work plane, or use the options on the contextual toolbar or the keyboard shortcuts.

The part's position on the work plane is always relative to part's reference line.

Option	Description	Example
Middle	The reference line is in the middle of the part.	
Right	The part is positioned underneath the reference line.	
Left	The part is positioned above the reference line.	

Examples

Position	Example
Middle 300	
Right 300	
Left 300	

See also

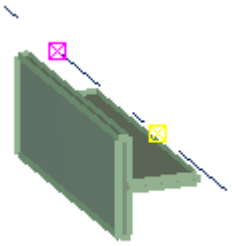
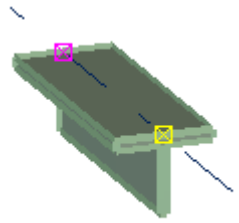
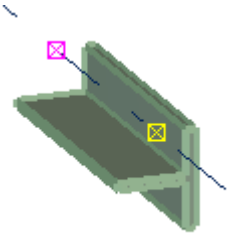
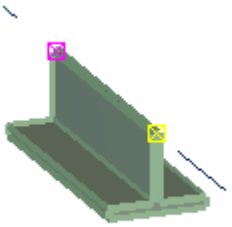
Adjust part position (page 334)

Part reference lines and part handles (page 116)

Part rotation

Use the property pane and its **Rotation** setting to view and change the rotation of the part around its axis on the work plane, or use the options on the contextual toolbar or the keyboard shortcuts.

You can also define the angle of rotation. Tekla Structures measures positive values clockwise around the local x axis.

Option	Description	Example
Front	The work plane is parallel to the front plane of the part.	
Top	The work plane is parallel to the top plane of the part.	
Back	The work plane is parallel to the back plane of the part.	
Below	The work plane is parallel to the bottom plane of the part.	

See also

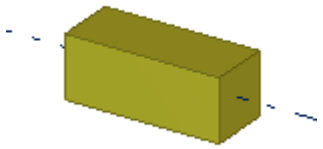
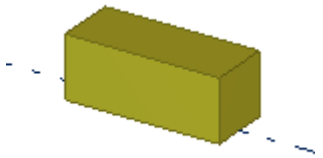
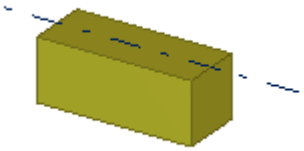
Adjust part position (page 334)

Part reference lines and part handles (page 116)

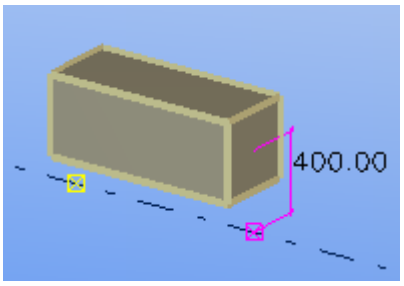
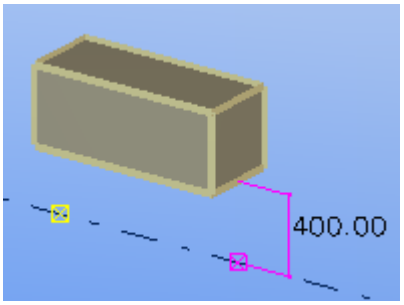
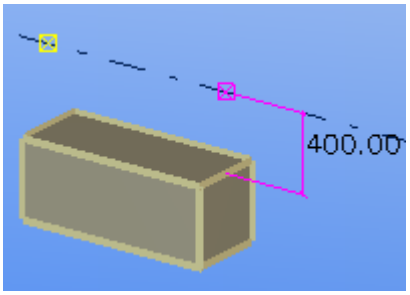
Part position depth

Use the property pane and its **At depth** setting to view and change the position depth of the part, or use the options on the contextual toolbar or the keyboard shortcuts.

The position is always relative to the part reference line between the part handles.

Option	Description	Example
Middle	The part is positioned in the middle of the reference line.	
Front	The part is positioned above the reference line.	
Behind	The part is positioned underneath the reference line.	

Examples

Position	Example
Middle 400	
Front 400	
Behind 400	

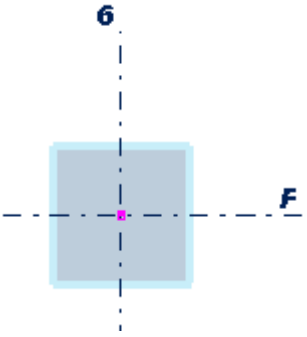
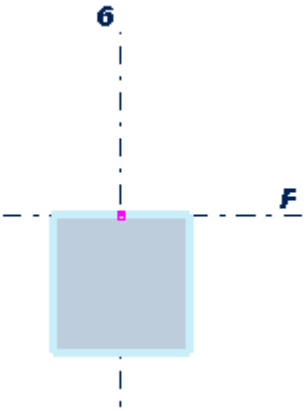
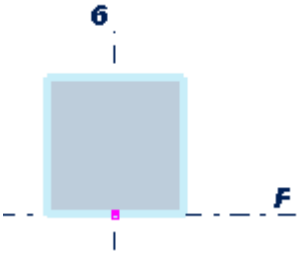
See also

Adjust part position (page 334)

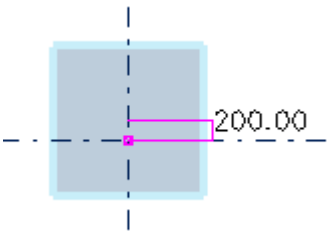
Part reference lines and part handles (page 116)

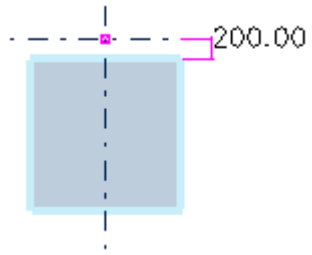
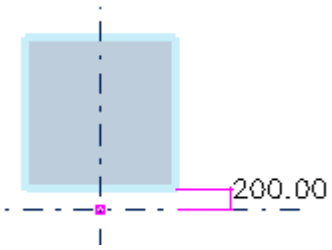
Part vertical position
Use the property pane and its **Vertical** setting to view and change the vertical position of the part, or use the options on the contextual toolbar or the keyboard shortcuts.

The position is always relative to the part's reference point.

Option	Description	Example
Middle	The reference point is in the middle of the part.	
Down	The part is positioned underneath the reference point.	
Up	The part is positioned above the reference point.	

Examples

Position	Example
Middle 200	

Position	Example
Down 200	
Up 200	

See also

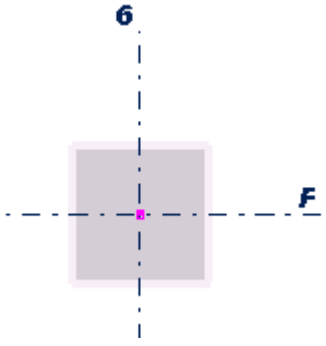
Adjust part position (page 334)

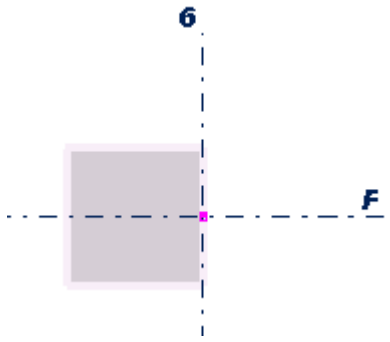
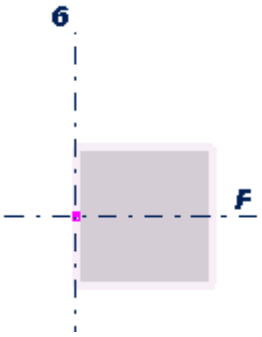
Part reference lines and part handles (page 116)

Part horizontal position

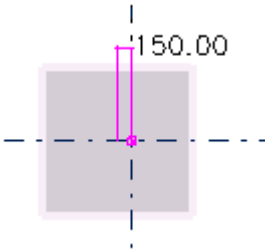
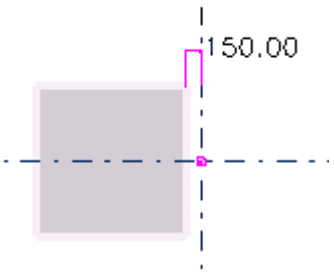
Use the property pane and its **Horizontal** setting to view and change the horizontal position of the part, or use the options on the contextual toolbar or the keyboard shortcuts.

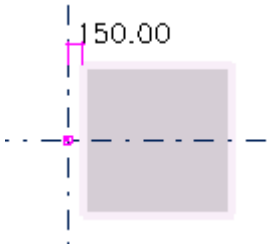
The position is always relative to the part's reference point.

Option	Description	Example
Middle	The reference point is in the middle of the part.	

Option	Description	Example
Left	The part is positioned on the left side of the reference point.	
Right	The part is positioned on the right side of the reference point.	

Examples

Position	Example
Middle 150	
Left 150	

Position	Example
Right 150	

See also

Adjust part position (page 334)

Part reference lines and part handles (page 116)

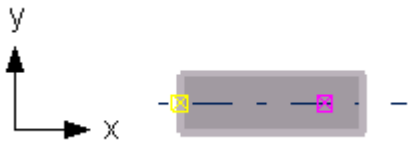
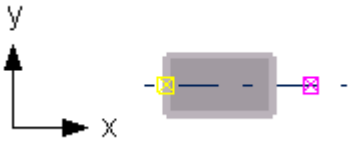
Part end offsets

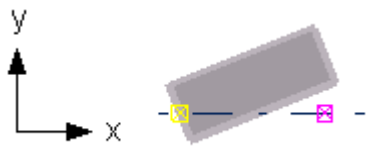
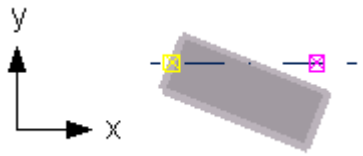
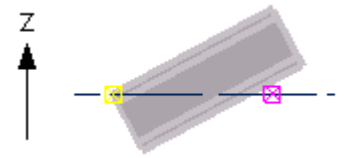
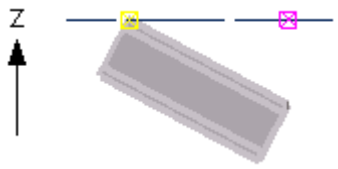
Use the property pane and its **Dx**, **Dy** and **Dz** end offset settings to move the ends of a part.

The offset is relative to part's reference line. You can enter positive and negative values.

Option	Description
Dx	Changes the length of the part by moving the part start or the part end point along the reference line.
Dy	Moves the part start or the part end perpendicular to the reference line.
Dz	Moves the part start or the part end in the z direction of the work plane.

Examples

Position	Example
Dx End point: 200	
Dx End point: -200	

Position	Example
Dy End point: 300	
Dy End point: -300	
Dz End point: 400	
Dz End point: -400	

See also

Adjust part position (page 334)

Part reference lines and part handles (page 116)

Position columns, pad footings, and orthogonal beams

For parts that you create by picking only one point (such as columns), you can define the top and bottom level of the part in the global z direction. The part is created at the defined level, **not** at the level you picked in the model. This can be useful when creating multi-story structures, as you can define exact levels for each part you create.

To define the top and bottom levels of a part:

1. Create a part that requires you to pick only one point.

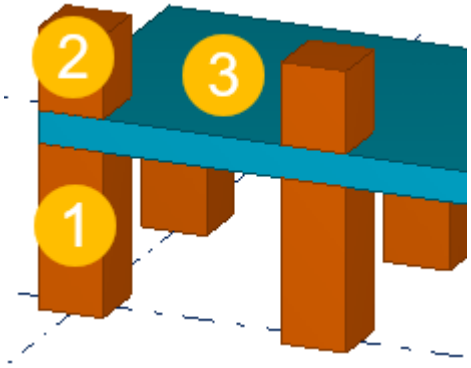
   For example, a column.
2. Double-click the part to open the part properties.
3. Go to the **Position** settings area.
4. Modify the top and bottom levels of the part.
   - **Top**: Use to define the top level of the part.

- **Bottom**: Use to define the bottom level of the part.

5. Click **Modify**.

Example

In this example, the concrete columns form a two-story structure. To position the upper columns correctly, you must modify their bottom level position.



(1) Top level = 1000, Bottom level = 0

(2) Top level = 1700, Bottom level = 1200

(3) Slab thickness = 200

See also

Create a concrete column (page 281)

Create a pad footing (page 314)

Create an orthogonal beam (page 235)

Bind parts to grid levels

Bind steel and concrete parts to various z-direction elevations that are taken from the project grids. By linking parts directly to defined grid elevations, manual work and errors are minimized. If there are any changes in the part levels in the z-direction, all bound parts update to match the new z-direction elevation. This ensures that any changes to grid levels or story elevations, if you are using building hierarchy, automatically reposition the bound parts.

For example, if the z-coordinate increases by 1 meter, the height of the bound object is automatically adjusted by the same amount. Bindings can be further refined with positive or negative offset values that correspond to the bound levels. Additionally, you have the flexibility to adjust, add, remove, or rename levels as needed. To find the bound parts in a model, you can create a filter or a report, and once the bindings are not needed anymore, you can unbind all bound parts at once.

Parts are bound to grid levels in the following ways:

Part	How the part is bound
Columns and walls	Bind to grid levels from the top and bottom part face.
Beams and polybeams	Bind to grid levels from the start and end points of the part. Note: When start and end points of a polybeam are bound, any intermediate points scale proportionally to maintain the relative geometry of the part.
Slabs, footings and plates	Bind to a single level from the top face only.

To bind parts to grid levels:

1. Start by ensuring your grids are suitably named.

   This will help you to identify different grids and make it easier to select the correct level for a part in question.

   a. In the grid properties, go to the **Name** property to name the grid.

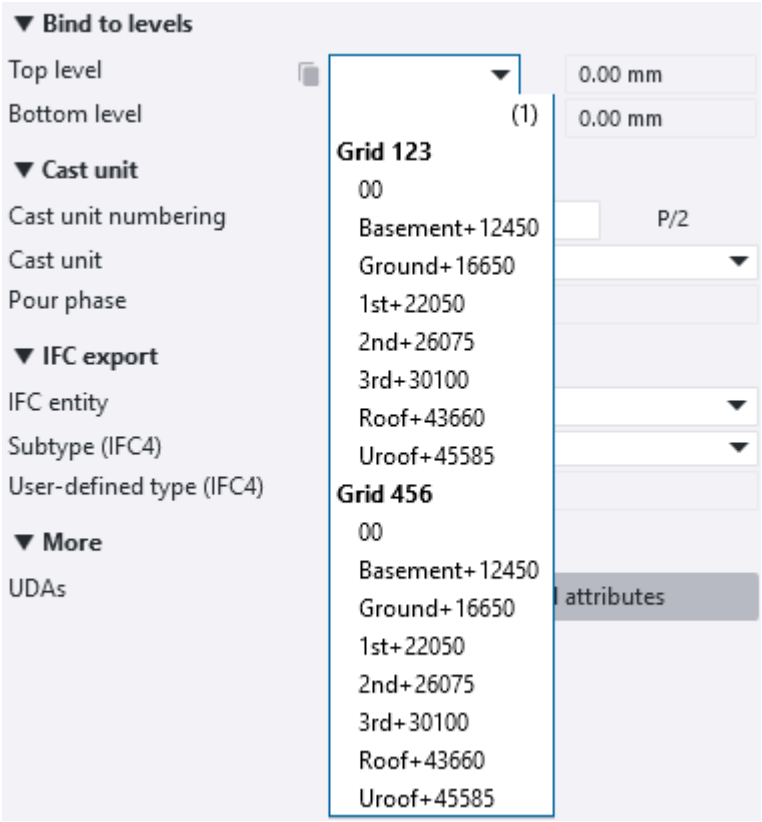
      When a new grid is created, the grids get automatically named to GRID, GRID 1, and so on.

   b. Change the current grid name if needed.

2. Bind a part to a selected grid level.

   a. Select a part in the model that you want to bind to a grid level.

b. In property pane of the selected part, go to the **Bind to levels** property.



c. Select the grid level to which you want to bind the selected part.

   You can bind the selected part fully or partially. If you bind the part only partially, leave the other end unbound. This allows you to manually set the position value for that end.

   - For a column or wall, you can bind either the top or bottom part face using the **Top level** and **Bottom level** lists.
   - For a beam or polybeam, you can bind either the start or end point using the **Start level** and **End level** lists.

     For polybeams:

     - If you enter only one point, the second point will either stay unbound if left empty, or be set based on the picked point's coordinate.
     - When the polybeam start and end points are bound, any intermediate points scale proportionally to maintain the relative geometry of the polybeam.
     - When you have a flat polybeam, where the start and end level are the same, the polybeam stays flat even when the levels change, and all the intermediate points move together by the same amount.

- For a plate, a slab, or a footing, you can only bind the top face to a single level using the **Level** list.

d. If needed, you can enter a positive or negative offset value to refine the position of the bound end relative to the level.

e. Click **Modify** to set the binding.

If you resize or reshape the bound parts

If you later use direct modification to modify the part, the part will not be unbound from its level. Instead, changes made using direct modification or changing the part profile adjust the offset value relative to the bound level.

If you copy or move the bound parts

- If you move a part and its new z-coordinates align with the z-level grid, its top and bottom levels are updated to these new z-levels. This only works if the new level difference matches the original difference.
- If only one end of the part, either top or bottom, aligns with the new z-level, that end connects to the new level and the other end is unbound.
- If the part is positioned between levels and the new level difference does not match the original, it loses its binding and returns to its original position.
- If the part moves to a position without any z-level match, the part loses its binding and returns to its default position.

  Note that building hierarchy works the same way when you change a building story's height.

3. Once the objects are bound to levels, use the `BIND_TO_LEVELS` template attribute in reports to report on all bound objects in your model.

   To filter parts based on whether or not they are bound to levels, use the property **Bind to levels** in the selection filter in the **Part** category.

4. When you do not need the bindings anymore, you can remove the bindings. Select an empty value in the list of levels in the property pane, or click **Edit** --> **Parametric modeling** and select **Unbind all** or **Unbind selected objects from levels**.

Camber a part

You can use cambering to pre-camber parts, in other words, to curve long heavy sections that will settle on site and become flat. Use cambering to show the natural camber of a prestressed part in a model. Cambering affects the position of cuts, skews, and embeds in the model.

1. Double-click the part to open the part properties.

2. Go to the **Deforming** section.
3. In the **Cambering** box, define the degree of camber.
4. Click **Modify**.

   Tekla Structures cambers the part in the local z direction.



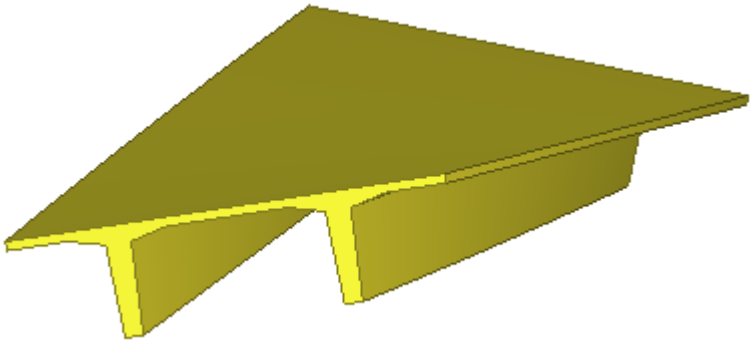
Warp a part

You can warp steel and concrete beams and columns, and concrete slabs. The warping functionality is available only in the Tekla Structures Diamond and Tekla Structures Graphiteconfigurations.

Warp a beam or a column using deformation angles

1. Double-click a beam or a column to open the properties.
2. Go to the **Deforming** section.
3. In the **Warping Start** box, enter the angle of the beam at its start point, relative to the part handles.
4. In the **Warping End** box, enter the angle of the beam at its end point, relative to the part handles.

   For example, to warp the beam 10 degrees at the end point, enter `0` in the **Start** angle box and `10` in the **End** angle box.
5. Click **Modify** to warp the beam.



Warp a concrete slab by moving chamfers

Before you start, create a concrete slab by using the **Slab** command on the **Concrete** tab.

1. Double-click a chamfer to open the **Corner chamfer** properties.
2. Modify the chamfer properties.

   Do not modify the chamfers so that the slab faces are no longer planar.

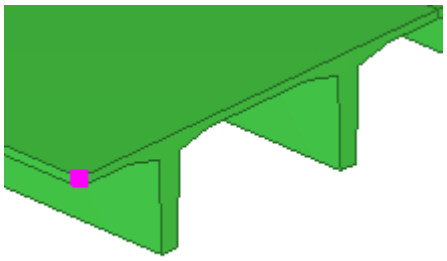
   - To move the upper corner of the chamfer, modify the **Dz1** value.
   - To move the lower corner of the chamfer, modify the **Dz2** value.
3. Click **Modify** to warp the slab.

Warp a Floor Bay (66) slab

Before you start, create a concrete slab by using the Modeling of floor bay (66) component.

1. Ensure that the **Select components** selection switch is on.
2. Select the chamfer you want to move.

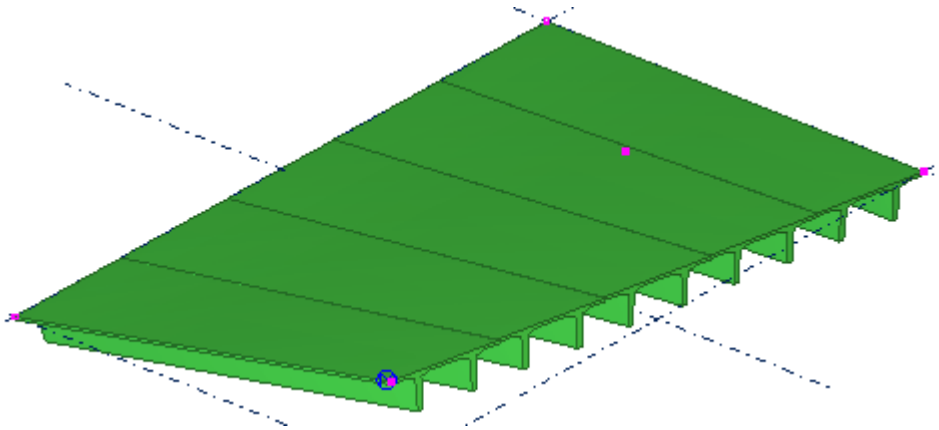
   For example, select the corner point of a slab component to warp that end of the slab:



3. Right-click and select **Move special** --> **Linear...** .
4. In the **Move - Linear** dialog, enter a value in the appropriate direction box.

   For example, enter `100` in the **dZ** box to lift that corner up 100 mm.
5. Click **Move**.

Tekla Structures moves the point in the direction you selected, which warps the slabs.



6. Right-click and select **Interrupt**.
7. Ensure that the **Select objects in components** selection switch is on.
8. To see the warping angle of a single slab, double-click a slab to open the **Concrete beam** properties, and go to the **Deforming** section.

   The **Warping** start and end values show the warping angle at the start point and at the end point of the part.

Examples of user-defined attributes (UDAs) for parts

User-defined attributes (UDAs) provide additional information about a part. UDAs can consist of numbers, text, or lists. If needed, you can define and update user-defined attributes.

The following table gives some examples of what you can use part UDAs for:

Attribute	Can be used...
Comment	In part and weld marks in Tekla Structures drawings, or in projects.
Shorten	When drawings of the parts are created, Tekla Structures decreases the true length of the part by this value. This is useful when creating assembly drawings of bracing bars that should always be under tension.
Camber	In part marks in Tekla Structures drawings.
Preliminary mark	To obtain preliminary numbers for parts in reports.
Locked	To protect objects from being accidentally changed.
Shear, Tension, and Moment	To save reaction forces for AutoDefaults. You can enter forces separately for each end of a part.

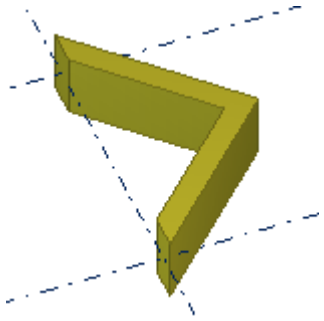
Attribute	Can be used...
User field 1...4	User-defined fields. You can change the names of these fields and add new user-defined fields.
Connection code	When importing information on connection types into Tekla Structures. You can then use the connection codes as rules in AutoConnection and AutoDefaults. Each end of a part can have a different connection code.
Moment connection	For selecting whether to show moment connection symbols in drawings or not.

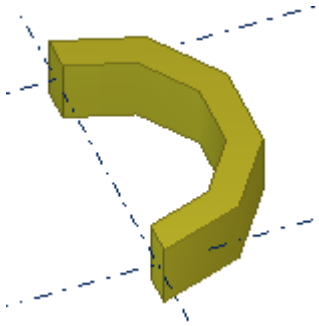
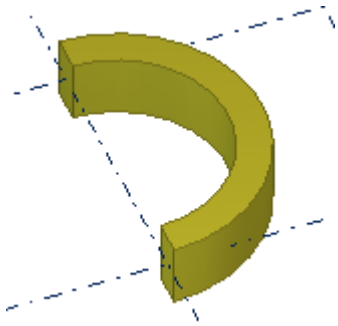
Create curved beams, panels, or strip footings

You can create curved parts by defining a radius and the number of segments for a part. The number of segments determines how realistic the curved part looks: the more segments, the less angular the part appears.

1. Create a part that can be bent: a beam, panel, or strip footing.
2. Double-click the part to open the part properties.
3. Go to the **Curved beam** or **Bending** settings area, depending on the part type.
4. In the **Radius** box, enter the radius.
5. In the **Number of segments** box, enter the number of segments you want to use.
6. If needed, define the plane of curvature, which is relative to the current work plane.
7. Click **Modify** to bend the part.

Examples

Number of segments: 2	

Number of segments: 5	
Number of segments: 15	

See also

Create a concrete beam (page 284)

Create a concrete panel or wall (page 295)

Create a strip footing (page 316)

Combine parts

You can combine two parts into one. This can be useful when you want to model complex parts (such as folded plates) that are otherwise difficult to model, or when you want to model prefabricated parts that are delivered to the workshop already attached to profiles.

When you combine parts, Tekla Structures retains the attached objects and connections. Tekla Structures does not recreate connections in the part that was selected first.

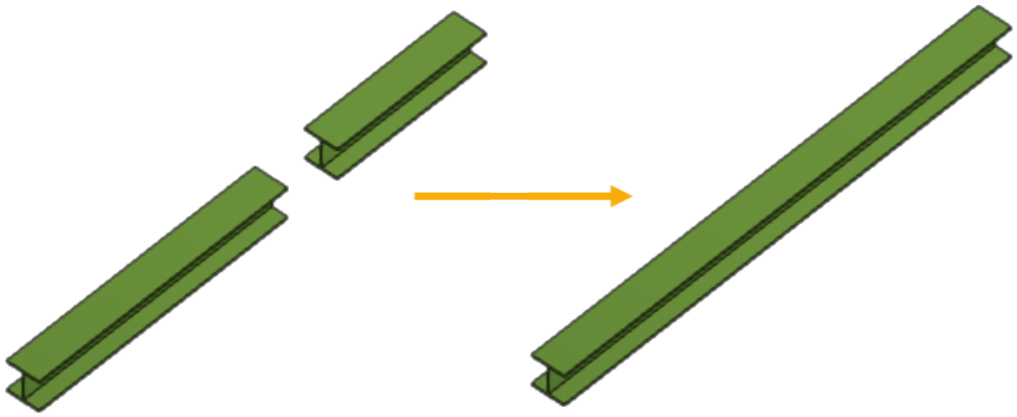
Limitations:

- Combining does not work for contour plates, polybeams or slabs.

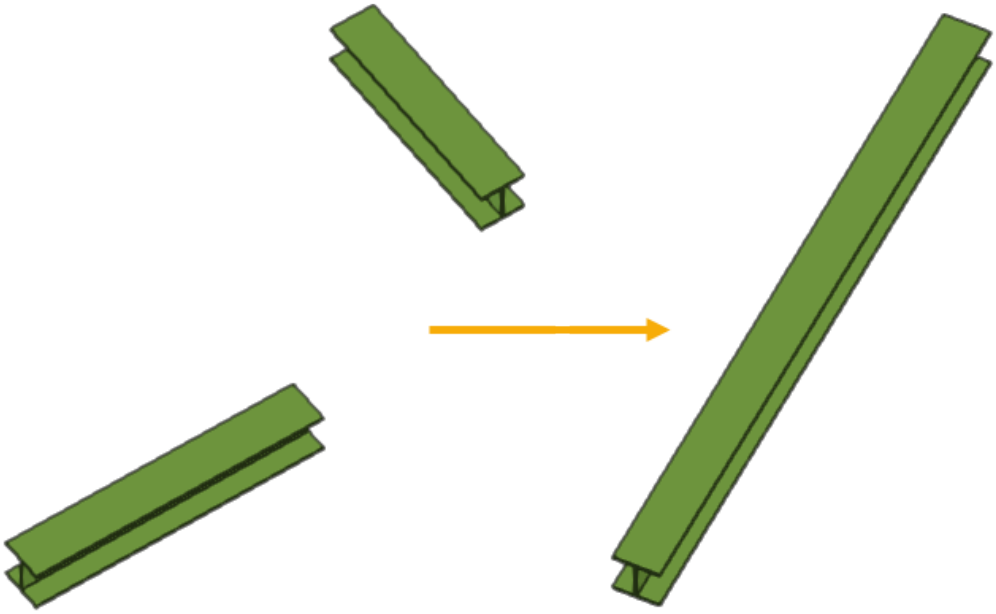
1. On the **Edit** tab, click **Combine**.
2. Select the first part.

   The properties of the first selected part will be used for the combined part.
3. Select the second part.

The parts are combined into one.



If the center lines of the parts are not in line with each other, Tekla Structures combines them by taking the largest distance between the start and end points from both parts. For example:



See also

Attach parts to each other (page 359)

Split parts (page 361)

Create parts and modify part properties (page 217)

Attach parts to each other

You can attach one or more parts to another part, or detach or explode the attached parts using the **Added material** commands.

When you modify the properties of attached parts, note that some of the part properties are taken from the main part. These properties are not shown in the properties of the attached part. You can inquire the properties of the whole part and the properties of each attached part separately. The attached parts are taken into account when calculating area, volume, and weight:

- **Weight (Gross)** compares the weight with fittings and without fittings, and shows the biggest weight result without cuts and with attached parts.
- **Weight (Net)** shows the weight with cuts and attached parts based on the geometry volume of the modeled part.
- **Weight** shows the net weight.

Limitations

- Connections must be added to the part to which other parts have been attached. You cannot add connections to an attached part.
- All reinforcement components may not work correctly with parts that have been attached to each other using the **Added material** commands. The geometry of the parts is not always kept suitable for adding a component. For example, the reference points of the attached part may be lost and therefore the orientation information needed for adding the reinforcement is not known anymore.

Attach a part to another part

1. Double-click the view to open the view properties, click the **Display...** button, and ensure that the **Cuts and added material** option is selected in the display settings.
2. On the **Edit** tab, click **Added material** --> **Attach to part**.
3. Select the part to attach to.
4. Select the part you want to attach.

   You can attach more than one part at a time.
5. Click the middle mouse button to attach the part.

Detach an attached part

1. Double-click the view to open the view properties, click the **Display...** button, and ensure that the **Cuts and added material** option is selected in the display settings.
2. On the **Edit** tab, click **Added material** --> **Detach from part**.
3. Select the attached part you want to detach.

   You can detach more than one part from several different parts at a time. Select the parts either by clicking them or by using area selection.

4. Click the middle mouse button to detach the part.

   The detached part keeps the color it had as an attached part.

Explode attached parts

You can explode a part that has attached parts.

1. Double-click the view to open the view properties, click the **Display...** button, and ensure that the **Cuts and added material** option is selected in the display settings.
2. On the **Edit** tab, click **Added material** --> **Explode part**.
3. Select the part you want to explode.
4. Click the middle mouse button to explode the part.

Split parts

Use splitting to split a part into two. You can use splitting with straight parts, polybeams and curved beams without offsets. You can also split contour plates and concrete slabs.

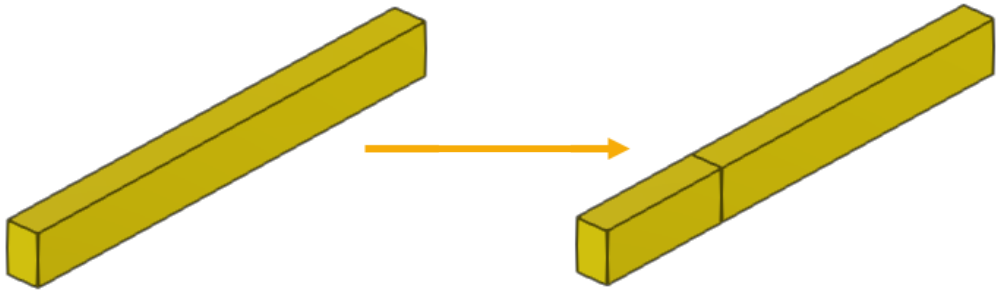
When you split parts, Tekla Structures retains the properties and attached objects when the parts are split.

Limitations:

- You cannot split surface treatment.
- You cannot split bent plates, lofted plates, or lofted slabs.
- If the part to be split contains a bolt group, and the part is split at a position where there are bolts, you need to also split the bolt group.
- If you have already carried out numbering, the part and assembly number is assigned to the part that was at the original start point. The other part gets a new number.

Split a straight or curved part or polybeam

1. On the **Edit** tab, click **Split**.
2. Select the part you want to split.
3. Pick a point for the splitting line.
4. If you split a polybeam, check that the following are correct:
   - The position and orientation settings of the split polybeams
   - The components related to the split polybeams



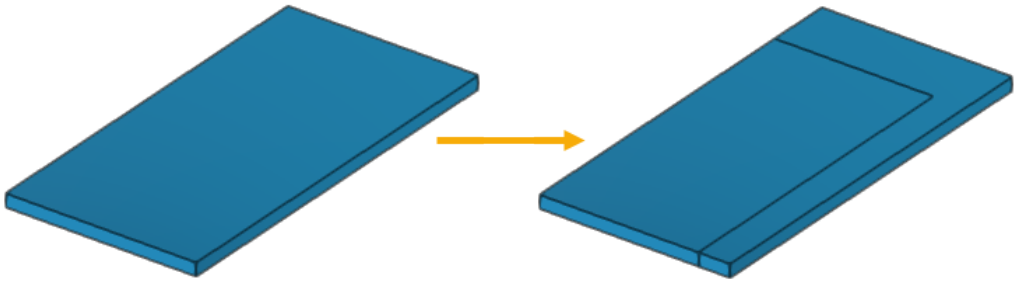
New part start and end points are placed at the position of the split.

Split a plate or slab

1. On the **Edit** tab, click **Split**.
2. Select the contour plate or concrete slab you want to split.

   Note that bent plates, lofted plates, or lofted slabs cannot be split.
3. Pick the points you want the splitting line to go through.

   The first picked point and the last picked point need to be on an edge of the plate or slab.



New part start and end points are placed at the position of the split.

NOTE Areas with chamfers, cuts, or openings cannot be split. If you try to pick points in these areas, the preview of the splitting line turns red, indicating that splitting is not possible.

See also

Combine parts (page 358)

Create parts and modify part properties (page 217)

Split a reinforcing bar group (page 581)

Split a rebar set (page 568)

2.2 Add details to parts

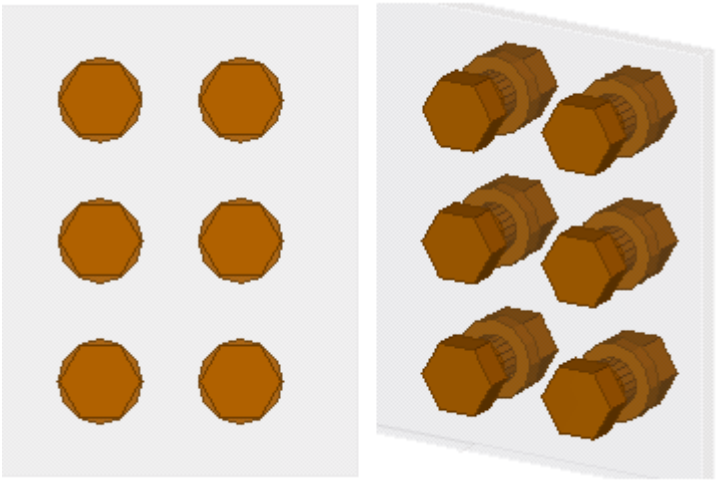
This section explains how to create and view details using Tekla Structures. It also gives you some techniques for fine-tuning the part shape.

You can create and view details, for example, the following ways:

- create bolts (page 363), studs (page 375) and holes (page 376)
- create welds (page 381) and define how they appear in the model (page 393)
- create fittings (page 397), cuts (page 412) and part chamfers (page 420) to refine the shape of a part, and to fit beams and columns (page 400) and also contour plates, slabs, and panels (page 403) to each other
- add surface treatment to parts (page 425) and surfaces to part faces (page 437)
- modify adaptivity (page 438) of the details
- show all the detailing that belongs to a part (page 439)

Create bolts

To create bolts, you can either create a single bolt group or apply a component that automatically creates bolt groups.



Tekla Structures uses the same command for creating bolts, studs (page 375) and holes (page 376). If you want to create only holes, do not use any bolt elements (such as bolts, washers, and nuts).

If you want to create bolts without creating holes, use the **No hole** option for **Special hole type** in the **Bolt** properties.

You can create different marks for bolts and holes in drawings.

Create a bolt group

1. On the **Steel** tab, hold down **Shift** and click **Bolt** .

   The **Bolt** properties open in the property pane, and the following contextual toolbar appears:

2. Modify the **Bolt** properties, if needed.

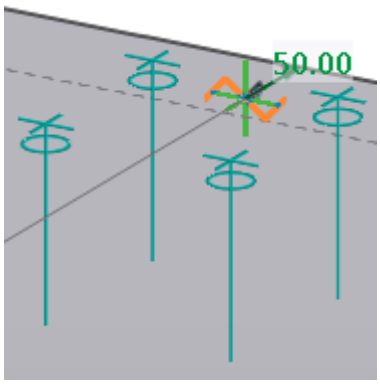
   For example, the **Bolt group** settings affect the end result: the shape of the bolt group, the number of bolts in the group, and the spacing of the bolts. See the **Bolt group shape** section below for examples.

   To define the hole properties, use the **Holes** settings in the **Bolt** properties.

3. Move the mouse pointer over the parts in the model to see a preview of the bolts.

   With the options and where you use a part face, the edge of the part face that is nearest to the mouse pointer is selected as the basis for the preview. The bolt group direction will be perpendicular to this edge.

   Bolt heads are shown with crosses in the preview. For example:

4. To adjust whether dimensions are shown in the preview, click on the contextual toolbar. Then click **Internal dimensions** to show or hide the dimensions between the bolts in the group.

5. If you want to snap to part center lines when you place the bolts, click on the contextual toolbar. Then select whether you want to show the center lines for profiles, such as beams and columns, or for plates, or for both.

6. Choose how you want to create and place the bolts; click a button on the contextual toolbar to select one of the three methods, **By face**, **By parts and face**, or **By parts and points**. Then follow the related steps below.

   When you pick locations, you can use various snapping methods, such as numeric snapping (page 103), or you can create temporary reference points (page 109) by holding down **Ctrl**.

   If you need to change the direction of the bolts by 180°, click on the contextual toolbar. Clicking only flips the bolt direction, but maintains the other bolt properties, such as the order of special holes and the hole locations.

To	Do this
Create bolts using a part face	a. On the contextual toolbar, click . b. Pick a location on a part face. To temporarily lock the edge direction, hold down **Alt** while you pick. Tekla Structures uses the **Cut length** value in the **Bolt** properties to select the parts to be connected. The part with the largest volume will be the main part, and the secondary parts will be in order that they are found.
Create bolts based on the parts to be connected and using a part face	a. On the contextual toolbar, click . b. Select the main part, to which the secondary parts will be bolted. c. Select the secondary parts. d. Click the middle mouse button to finish selecting parts. e. Pick a location on a part face. To temporarily lock the edge direction, hold down **Alt** while you pick. Tekla Structures calculates the bolt cut length based on the selected parts and corrects the **Cut length** value in the **Bolt** properties.
Create bolts based on the parts to be connected and by picking two points	a. On the contextual toolbar, click . b. Select the main part, to which the secondary parts will be bolted.

To	Do this
	c. Select the secondary parts. d. Click the middle mouse button to finish selecting parts. e. Pick a point to indicate the bolt group origin. f. Pick a second point to indicate the direction of the bolt group x axis. Tekla Structures determines the location of the bolt group using the following values: the bolt group x axis and the work plane. Dimensions are relative to the bolt group origin, which is the first point picked. Tekla Structures sets the x direction of the bolt group using the second point picked. It is important that the points you pick to create the bolt group are close enough to the parts you want to connect.

Create a single bolt

1. On the **Steel** tab, hold down **Shift** and click **Bolt**  to open the **Bolt** properties.
2. Under **Bolt group**, select **Array** from the **Shape** list.
3. In the **Bolt dist X** and **Bolt dist Y** boxes, enter `0`.
4. Create the bolt the same way you would create a bolt group. See the instructions in the table above.

Create bolts using the Auto bolt component

Use the **Auto bolt** component to bolt parts and nearby parts, shim plates, splice plates, or other plates. **Auto bolt** follows the part rotation and finds the best rotation so that you do not need to set the work plane. With **Auto bolt** one bolt group can span many parts, for example, manage a splice as a single group.

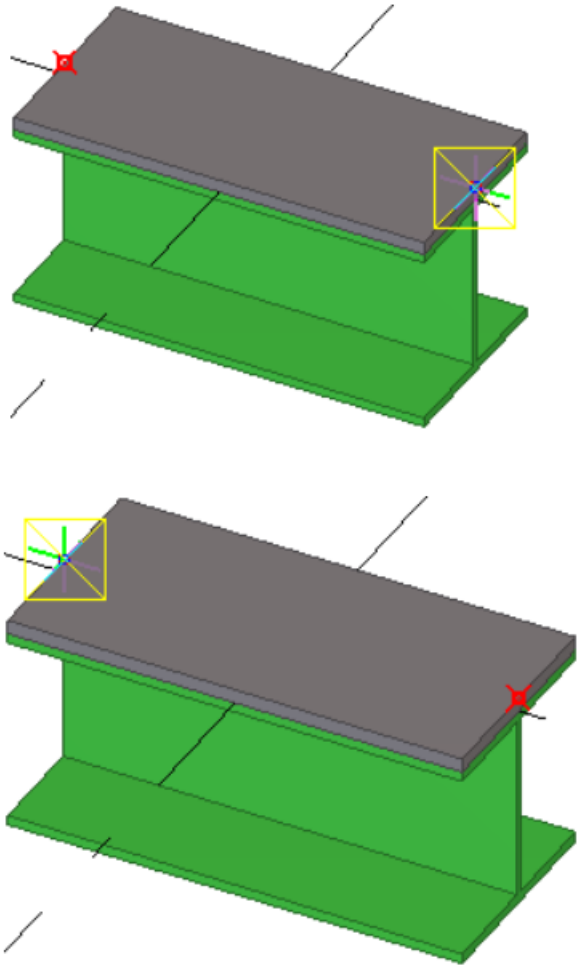
1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Start typing `auto bolt` in the search box.
3. Double-click **Auto bolt** in the catalog to open the **Auto bolt** dialog.
4. Define the bolt properties.
5. If needed, you can show cut length as temporary lines to view where the bolts should be placed even if they are not created.

- Select in the list at the bottom of the dialog to not show the temporary lines.
- Select in the list at the bottom of the dialog to show the temporary lines.

To delete the temporary lines, right-click the view and select **Redraw view**.

6. Click **Apply**.
7. Select the main part.

   Auto bolt uses this part to identify the best rotation. This part will be the main part of the assembly.
8. Select the secondary part.
9. Click the middle mouse button.
10. Pick the first and the second position to define the bolt group direction.



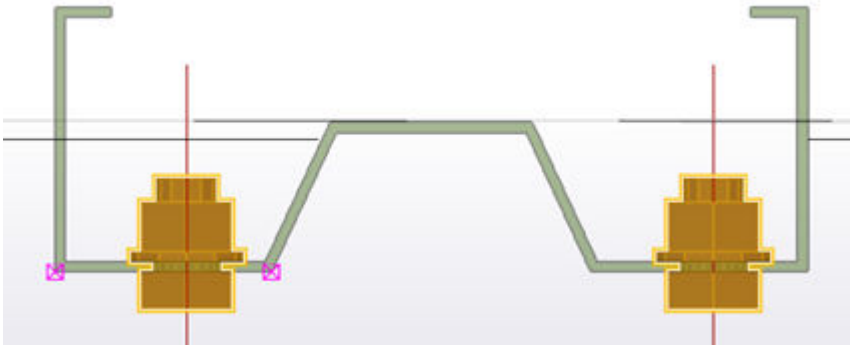
The positions you pick define a line that is used to find all parallel part faces and the optimal plane and locations for the bolts. The plane is chosen as follows:

a. The face with the longest projection of the line

b. If there are two faces with equally long projections, the face which is closer to the line

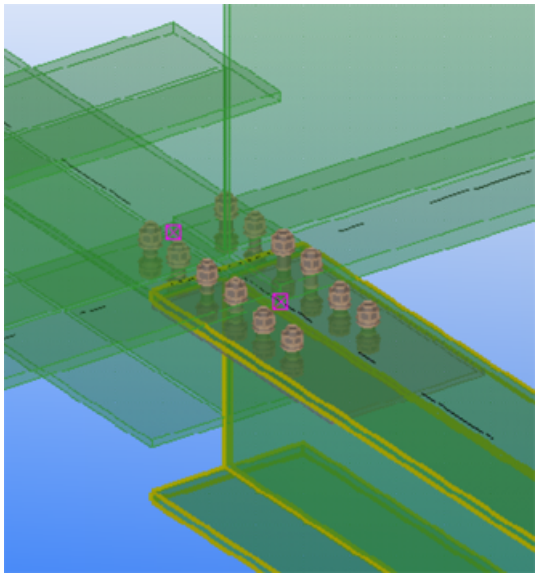
c. If there are two equally close faces, the larger face

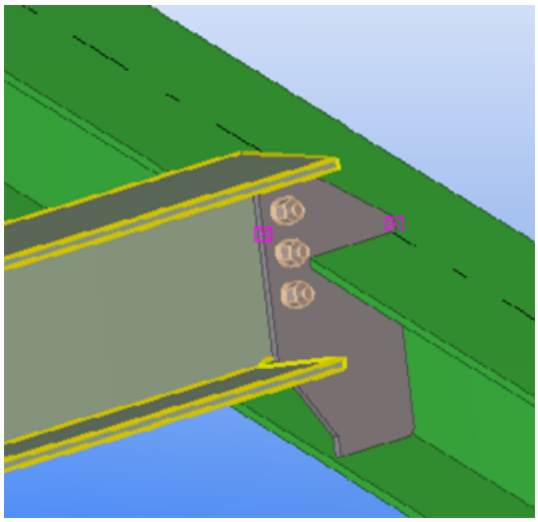
For example, with special part profiles, selecting points close to the wanted face ensures that the bolts are placed correctly:

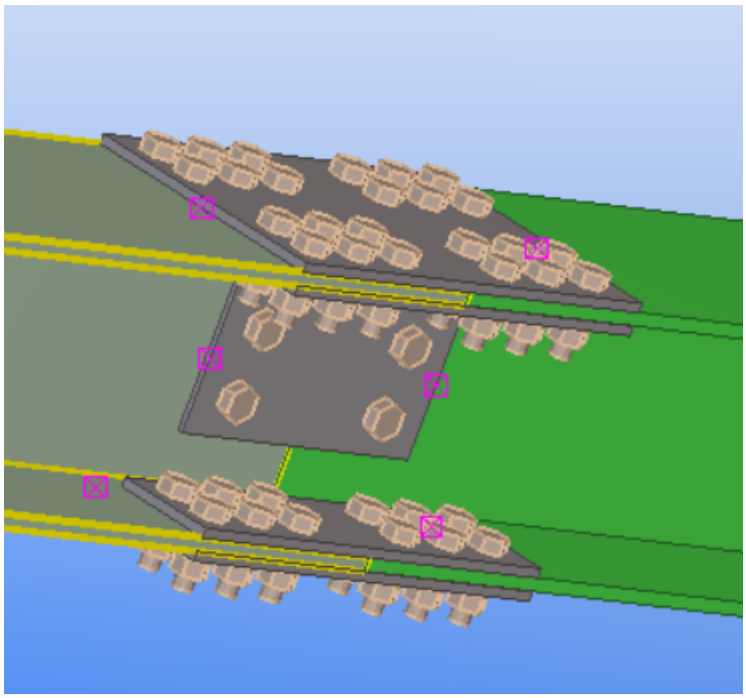


Examples

Examples of parts bolted using the **Auto bolt** component are shown below. The main parts and the selected points are highlighted.







Create a bolt group by exploding a component

An alternative way to create bolts is to first apply a component that includes bolt groups, and then explode the component.

1. Apply a component that includes bolt groups.

   For example, connect two beams, or a beam to a column, using a bolted end plate.

2. Explode (page 866) the component.

   a. Select the component to explode.

b. Right-click and select **Explode component**.

Tekla Structures separates the objects in the component.

3. Modify the bolt group.
   a. Select the bolt group and double-click it to open the properties.
   b. Modify the properties.
   c. Click **Modify**.

Change or add bolted parts

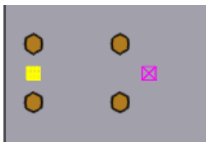
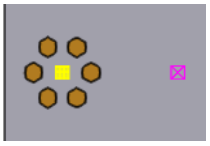
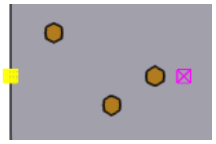
You can change the parts a bolt group connects to.

1. On the **Steel** tab, click **Bolted parts**.
2. Select the bolt group.
3. Reselect the main and secondary parts.
4. Click the middle mouse button to finish selecting the parts.

Bolt group shape

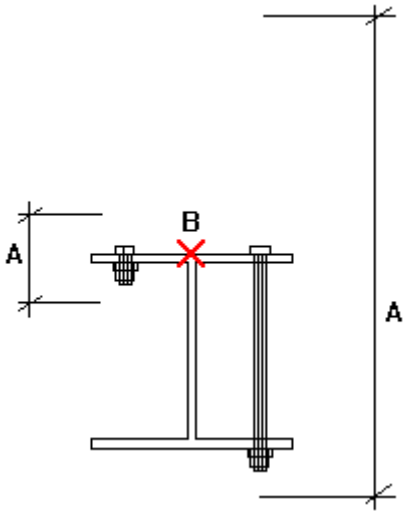
Use the **Bolt group** settings in the **Bolt** properties to select the shape of a bolt group and to determine how many bolts the bolt group contains.

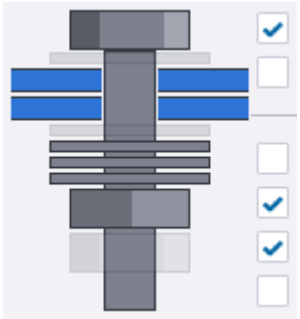
In the images below, the yellow handle indicates the origin of the bolt group, and the magenta handle indicates the x direction of the bolt group.

Shape	Other bolt group settings	Example values	Result
Array	**Bolt dist X** Spacing between bolts, in the x direction of the bolt group.	150	
	Bolt dist Y Spacing between bolts, in the y direction of the bolt group.	100	
Circle	**Number of bolts**	6	
	Diameter of the bolt group	100	
List	**Bolt dist X** x coordinate of each bolt, from the bolt group point of origin.	75 175 250	
	Bolt dist Y y coordinate of each bolt, from the bolt group point of origin.	75 -50 0	

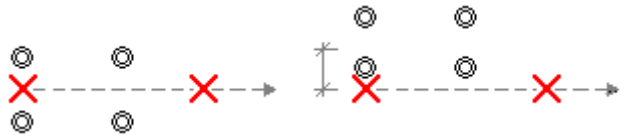
Bolt properties

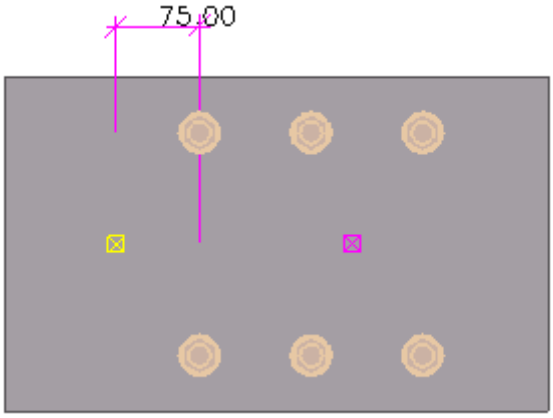
Use the **Bolt** properties to view or modify the properties of a bolt group. Use the **Holes** section to define the properties of bolt holes. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.

Setting	Description
Bolt	
Size	Bolt diameter.
Standard	Bolt assembly standard/grade.
Bolt type	Define whether the bolts are assembled on-site or in the shop.
Connect as	Indicate whether you are bolting a secondary part or a sub-assembly.
Thread in material	Indicate if the thread of the bolt can be inside the bolted parts. Tekla Structures does not use this value when calculating the length of full-threaded bolts.
Cut length	Indicate which parts the bolt connects. The value defines the area Tekla Structures should search for parts that belong to the bolt group. Using cut length you can determine whether the bolt will go through one flange or two. Tekla Structures searches for parts using half the cut length value, in both directions from the bolt group plane. In the illustration below, A is the cut length and B is the bolt origin. Tekla Structures calculates the search area as A/2 in both directions from point B. 

Setting	Description
	Tekla Structures warns you if the cut length is too small (i.e. the bolt group contains no parts) and makes the bolt length 100 mm. If there are large gaps between the connected parts, the gap is added to the length of the bolt. Tekla Structures calculates bolt length using the total distance between the first and last surfaces. **NOTE:** If you want to force a bolt to be a certain length, enter a negative value for cut length (e.g. -150). **NOTE:** If holes or blind holes cannot be created, increase the cut length.
Extra length	Additional bolt length. Increases the material thickness that Tekla Structures uses when calculating bolt length. For example, you might need extra bolt length to allow for painting. You can also build additional lengths into bolt assemblies.
Assembly	Select whether to create washers and nuts with the bolt. Select the **Bolt** checkbox to create the bolt, and then select the needed washer and nut checkboxes. For example:  If you want to create only holes (page 376) without any bolts, clear all the checkboxes.
Bolt group	
Shape	Bolt group shape. You have the following options: • **Array** for rectangular • **Circle** for circular • **List** for any shape
Bolt dist X	Bolt spacing or coordinate, depending on the bolt group shape (**Array** or **List**).

Setting	Description
Bolt dist Y	Bolt spacing or coordinate, depending on the bolt group shape (**Array** or **List**).
Number of bolts	Number of bolts in a circular bolt group.
Diameter	Diameter of a circular bolt group.
Holes	
Tolerance	Tolerance = Hole diameter - Bolt diameter
Plain hole type	Select **Through** to create holes that are open throughout the part. Select **Blind** to create partial-depth holes that do not extend completely through parts.
Hole depth	Depth of a blind hole measured from the bolt/hole reference points (yellow and magenta handles). Note that you may also need to adjust the **Cut length** value.
Parts with special holes	If you want to create oversized, slotted, or tapped holes, or if you want to omit holes, select the desired checkboxes to indicate which plies of the connection get special holes.
Use the same settings for all special holes	Select this checkbox to create similar special holes in each of the connected parts. The properties of the special hole that is closest to the bolt head will be used for all special holes in the bolt group. If you clear this checkbox, you can define special hole properties separately for each of the parts.
Special hole type	Oversized, slotted, tapped, or no holes. This option becomes active when you select one or more **Special hole** checkboxes next to **Parts with special holes**.
Oversized	Allowance of an oversized hole.
Slotted hole X	x allowance of a slotted hole. Zero for a round hole. To create slotted holes with an offset from the bolt center in the x direction, enter an offset value in the second box (**Slot offset**).
Slotted hole Y	y allowance of a slotted hole. Zero for a round hole. To create slotted holes with an offset from the bolt center in the y direction, enter an offset value in the second box (**Slot offset**).

Setting	Description
Core hole size	Size of a predrilling hole. **NOTE:** The core hole size does not affect the DSTV output. This value is only for marking and representation of the hole in the model and drawings.
The following two settings are available only in the **Auto bolt** component:	
Rotate slots	If the bolt connects several parts, you may want to rotate alternate holes by 90 degrees. This allows the bolt to move in different directions.
Show cut length as temporary lines	Shows where the bolts should be placed even if they are not created. • Select to not show the temporary lines. • Select to show the temporary lines.
Position	
On plane	Move the bolt group perpendicular to the bolt group x axis. 
Rotation	Define how far the bolt group is rotated around the x axis, relative to the current work plane. For example, you can use this box to indicate on which side of the connected parts you want the bolt head to be. 
At depth	Move the bolt group perpendicular to the current work plane.
Offset from	
Dx, Dy, Dz	Offsets that move the bolt group by moving the bolt group x axis. Use to change the position of the bolt group. The start point values **Dx**, **Dy** and **Dz** move the first end of the bolt group, relative to the bolt group x axis. The end point values move the second end of the bolt group. • A positive **Dx** value moves the start point towards the end point.

Setting	Description
	• **Dy** moves the end point perpendicular to the bolt group x axis on the current work plane. • **Dz** moves the end point perpendicular to the current work plane. An example bolt group with the **Dx** start point set to 75: 
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the bolt. UDAs provide more information about the bolts.

Create studs

A stud is special type of bolt that is welded to steel parts to transfer loads between steel and concrete.

Tekla Structures uses the same command for creating bolts (page 363), studs and holes (page 376). When you create studs, select the stud assembly standard in the **Bolt** properties. You can create a group of studs or a single stud.

You can also create studs by using the **Shear Stud (1010)** component.

1. Ensure that the needed studs are added to the bolt catalog and bolt assembly catalog.
2. On the **Steel** tab, hold down **Shift** and click **Bolt**  to open the **Bolt** properties.
3. In the **Standard** list, select the bolt assembly standard for the studs.
4. Under **Bolt group**, do one of the following:

- To create a group of studs, define **Shape** and the related properties as desired.
- To create a single stud, select **Array** from the **Shape** list and type `0` in the **Bolt dist X** and **Bolt dist Y** boxes.

5. Modify the other properties as desired.
6. Select the main part.
7. Click the middle mouse button to finish selecting parts.
8. Pick a point to indicate the origin of the stud or stud group.
9. Pick a second point to indicate the direction of the stud group x axis.

Create bolt holes

Tekla Structures uses the same command for creating bolts, studs and holes. Before creating holes, you need to change some of the properties in the **Bolt** properties. If you want to create only holes, do not use any bolt elements (such as bolts, washers, and nuts).

You can create the following types of holes:

- Round
- Oversized
- Slotted
- Tapped

When several parts are connected together with a bolt group or single bolt, you can create different holes in each of the parts, or you can use the same properties for the special holes in all the parts.

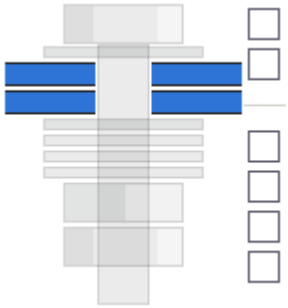
You can also select whether the holes you create are open throughout the parts, or partial-depth or blind holes that do not extend completely through the parts.

The hole properties are available in the **Holes** section in the **Bolt** properties (page 371).

Create round holes

You can create a group of round holes, or a single round hole. Tekla Structures calculates the diameter of a round hole as the sum of **Size** and **Tolerance**.

1. On the **Steel** tab, hold down **Shift** and click  **Bolt** to open the **Bolt** properties.
2. If you do not want to create any bolts, clear all the **Assembly** checkboxes.



3. If needed, modify the hole properties.

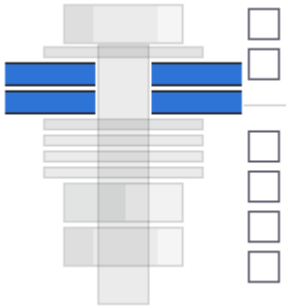
   For example, to create partial-depth holes, set **Plain hole type** to **Blind** and enter a value in the **Hole depth** box.

4. Create the holes the same way you would create a bolt group (page 363).

Create oversized holes

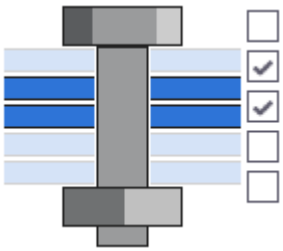
You can create a group of oversized holes.

1. On the **Steel** tab, hold down **Shift** and click **Bolt** to open the **Bolt** properties.
2. If you do not want to create any bolts, clear all the **Assembly** checkboxes.



3. Next to **Parts with special holes**, indicate which plies of the connection get oversized holes by selecting the desired **Special hole** checkboxes.

   For example:



4. If you want to create similar holes in each of the parts with oversized holes, select the **Use the same settings for all special holes** checkbox.

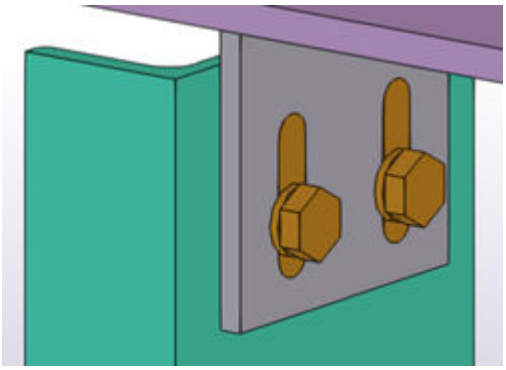
The holes will be created using the properties of the holes in the first part from the bolt head.

If you clear this checkbox, you can define hole properties separately for each of the parts with oversized holes.

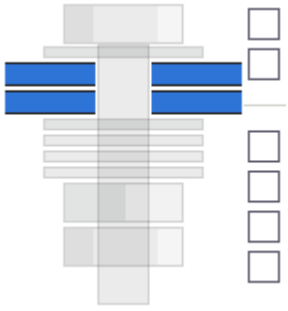
5. In the **Special hole type** list, select **Oversized**.
6. In the **Oversized** box, enter the allowance for the oversized hole.
7. If needed, modify the other hole and bolt properties.
8. Create the holes the same way you would create a bolt group (page 363).

Create slotted holes

You can create a group of slotted holes. You can also create slotted holes with offsets from the bolts.

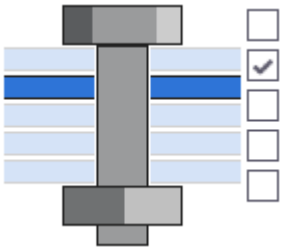


1. On the **Steel** tab, hold down **Shift** and click **Bolt** to open the **Bolt** properties.
2. If you do not want to create any bolts, clear all the **Assembly** checkboxes.



3. Next to **Parts with special holes**, indicate which parts should be slotted by selecting the desired **Special hole** checkboxes.

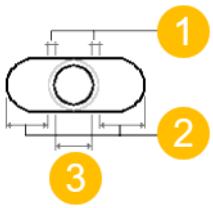
Tekla Structures counts the pieces of steel from the head of the bolt down. For example, if you select the second checkbox from the head of the bolt, Tekla Structures slots the second piece of steel from the head of the bolt.



4. If you want to create similar holes in each of the slotted parts, select the **Use the same settings for all special holes** checkbox.

   The holes will be created using the properties of the holes in the first slotted part from the bolt head.

   If you clear this checkbox, you can define hole properties separately for each of the slotted parts.
5. In the **Special hole type** list, select **Slotted**.
6. Enter the allowance for the slotted hole in the x and y directions of the hole group using the **Slotted hole X** and **Slotted hole Y** boxes.



   (1) Tolerance

   (2) Slotted hole X or Y

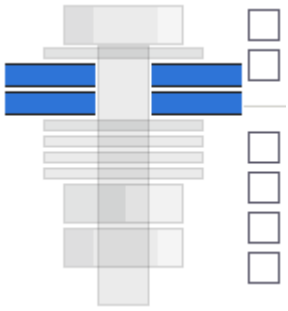
   (3) Bolt size
7. If you want to offset the slotted holes from the bolts so that the bolts are not in the middle of the holes, enter an offset value in the **Slot offset** box next to **Slotted hole X** and **Slotted hole Y**.
8. If needed, modify the other hole and bolt properties.
9. Create the holes the same way you would create a bolt group (page 363).

Create tapped holes

You can create tapped holes that have threads drilled in them.

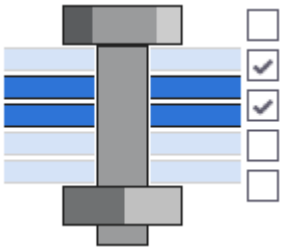


1. On the **Steel** tab, hold down **Shift** and click **Bolt** to open the **Bolt** properties.
2. If you do not want to create any bolts, clear all the **Assembly** checkboxes.



3. Next to **Parts with special holes**, indicate which plies of the connection get tapped holes by selecting the desired **Special hole** checkboxes.

   For example:



4. If you want to create similar holes in each of the tapped parts, select the **Use the same settings for all special holes** checkbox.

   The holes will be created using the properties of the holes in the first tapped part from the bolt head.

   If you clear this checkbox, you can define hole properties separately for each of the tapped parts.
5. In the **Special hole type** list, select **Tapped**.
6. In **Core hole size**, enter the predrilling hole size.

   Note that the core hole size does not affect the DSTV output. This value is only for marking and representation of the hole in the model and drawings.
7. If needed, modify the other hole and bolt properties.
8. Create the holes the same way you would create a bolt group (page 363).

Create different holes with one bolt group

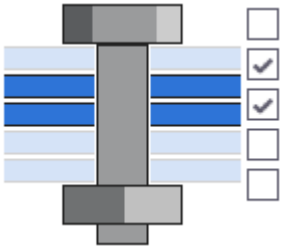
When several parts are connected together with a bolt group or single bolt, you can create different holes in each of the parts. For example, the holes in

one part can be slotted, and the holes in the other parts can be either special holes or regular round holes.

Up to five parts can get special bolt holes with one bolt. The rest of the connected parts (the sixth and so on) will get regular round holes.

1. On the **Steel** tab, hold down **Shift** and click **Bolt** to open the **Bolt** properties.
2. Next to **Parts with special holes**, indicate which parts get special holes by selecting the desired **Special hole** checkboxes.

   For example:

3. Clear the **Use the same settings for all special holes** checkbox.
4. For each of the parts with special holes, select an option in the **Special hole type** list.
5. Depending on the type of the special holes in each of the parts, enter the needed hole dimensions and other properties.
6. Create the holes the same way you would create a bolt group (page 363).

Create welds

You can either create a weld manually, or use a component that automatically creates welds.

By default, Tekla Structures places the welds to the arrow side using the **Above line** properties according to the ISO standard. You can change this to **Below line** to comply with the AISC standard with the advanced option XS_AISC_WELD_MARK.

Create a weld between parts

Weld two parts together using the weld position defined in the **Weld** properties. The length of the weld depends on the length of the connection between the welded parts.

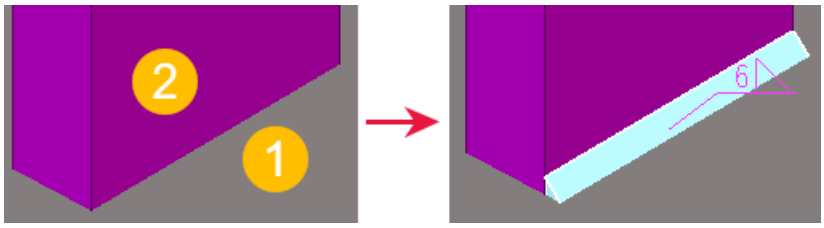
1. On the **Steel** tab, click **Weld** --> **Create weld between parts**.

2. Select the part to weld to.

   If you are creating a workshop weld, this is the main part of the assembly.

3. Select the part to be welded.

   If you are creating a workshop weld, this is the secondary part of the assembly.



(1) Main part

(2) Secondary part

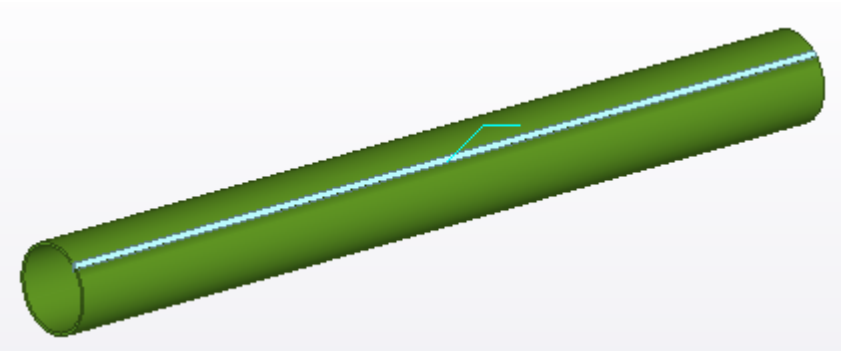
Create a weld to a part

You can create a weld to a single part, without connecting any other parts.

1. On the **Steel** tab, click **Weld** --> **Create weld to part**.
2. Select the part that you want to weld.
3. Pick the starting and end point, or alternatively, pick the points you want the weld to go through.
4. Click the middle mouse button to create the weld.

Example

Use the **Create weld to part** command to weld seams in tubular sections:



TIP To model tubular sections with visible seams, use the SPD profile.

Create a polygon weld

Create polygon welds when you want to define the exact position of the weld by picking the points you want the weld to traverse.

If you want to create double-sided polygon welds, define both the **Above line** and **Below line** properties.

1. On the **Steel** tab, click **Weld** --> **Create polygon weld**.
2. Select the part to weld to.

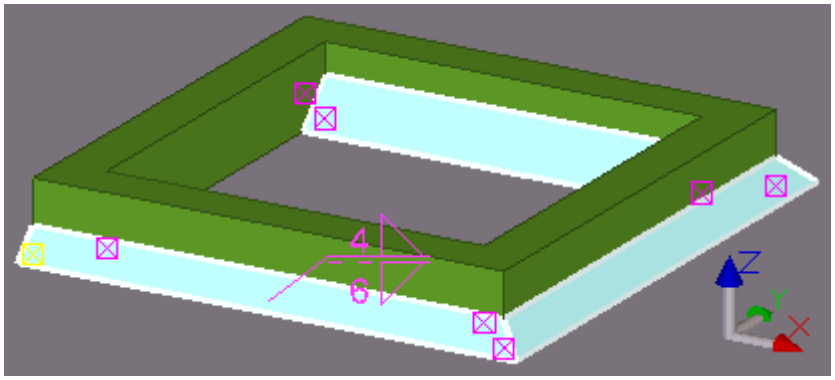
   If you are creating a workshop weld, this is the main part of the assembly.
3. Select the part to be welded.

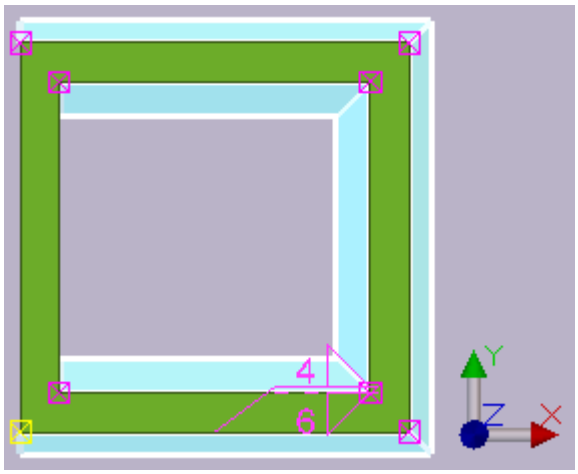
   If you are creating a workshop weld, this is the secondary part of the assembly.
4. Pick the starting and end point, or alternatively, pick the points you want the weld to go through.

   To create a double-sided polygon weld, pick the polygon points on one side of the part to be welded. Tekla Structures automatically searches for the corresponding points on the other side of the part.
5. Click the middle mouse button to create the weld.
6. If needed, modify the weld by dragging the handles.

Example

This example shows a double-sided polygon weld along three (outer and inner) edges of a rectangular hollow profile:



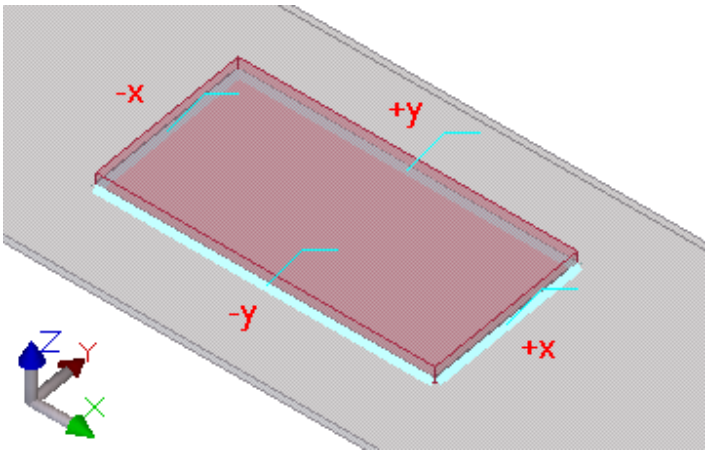


Weld properties

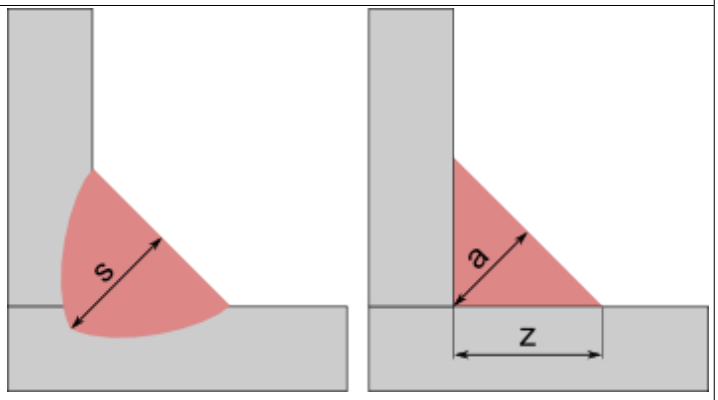
Use the **Weld** properties to view or modify the properties of a weld. The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.

NOTE Some of the properties are only displayed in reports, not in drawings.

Setting	Description
Common attributes	
Edge/Around	Indicates whether only one edge or the entire perimeter of a face should be welded. **Edge**:  **Around**: 
Workshop/Site	Indicates where the weld should be made. This setting affects assemblies and drawings. **Workshop**: **Site**: 
Position	Not available for polygon welds. Defines the position of a weld relative to the work plane. The type and position of the parts to be welded affect the position of the weld. The options for weld position are: • **+ x** • **- x**

Setting	Description
	• **+ y** • **- y** • **+ z** • **- z** In most cases, Tekla Structures creates the weld on the face or side of the part that faces in the selected direction (x, y, or z). Also the following factors may affect the position of the weld: • perpendicularity of the part edge towards the selected direction (x, y or z) • length of the part edge • distance of the part edge in the selected direction (x, y or z) The following image shows welds in different positions: 
Shape	The shape of the weld can be: •  (Regular, continuous weld) •  (Intermittent weld) •  (Staggered, intermittent weld)
Connect as	See Use welds to create assemblies (page 449).
Placement	Defines how the weld is placed in relation to the assembly parts. The options are:

Setting	Description
	• **Auto** The weld placement adapts to the typical situation of the weld type. Square-, V-, and U-groove welds are located in the middle of the main and secondary parts. Single-bevel and J-groove welds are located on the secondary part side. This is the default option. • **Main part** The weld is located fully on the main part side. Does not affect V- or U-groove welds. • **Secondary part** The weld is located fully on the secondary part side. Does not affect V- or U-groove welds.
Preparation	Defines which assembly parts, if any, are automatically prepared for welding. The options are: • **None** Parts are not prepared for welding. This is the default option. • **Auto** Parts are prepared for welding according to the weld type. • **Main part** The main part is prepared for welding. • **Secondary part** The secondary part is prepared for welding.
Weld	
Prefix	The weld size prefix. Shown in drawings, but only if the weld size is also defined. The standard ISO 2553 prefixes are: • **a** - Design throat thickness • **s** - Penetration throat thickness • **z** - Leg length

Setting	Description
	 Note that if the last character of the prefix is `s`, Tekla Structures creates the solid weld object according to the right-hand image so that `a` equals weld size.
Type	See the list of weld types (page 389) below.
Size	The size of the weld. If you enter a negative weld size, Tekla Structures creates the weld, but does not display it in drawings. If you enter a zero weld size, Tekla Structures creates the weld, and in drawings displays the weld reference line and the weld type symbol without the weld size. For compound weld types V+bevel and II+bevel, you can enter two size values.
Angle	The angle of weld preparation, bevels, or groove. Enter a positive value for bevel and groove welds. Tekla Structures displays the angle between the weld type symbol and the fill type contour symbol.
Contour	The fill type contour of a weld can be: • None • Flush — • Convex ⌒ • Concave ⌣ This setting does not affect solid weld objects.
Finish	Tekla Structures displays the finish symbol above the weld type symbol in drawings. The options are: • **G** (Grind) • **M** (Machine)

Setting	Description
	• **C** (Chip) • (Flush finished weld) • (Smooth blended weld face) This setting does not affect solid weld objects.
Root face	Root face thickness is the height of the narrowest part inside the root opening. Root face values do not appear in drawings, but you can use the `WELD_ROOT_FACE_THICKNESS` template attribute in reports to show the root face dimension in the weld list.
Effective throat	The weld size used in weld strength calculation.
Root opening	The space between the welded parts. Enter a positive value for square-groove welds.
# of incr.	The amount of increments in an intermittent weld. Only use with the ISO standard.
Length	Defines the length value that is shown in the weld mark. For intermittent welds, defines the length of an increment. Does not affect continuous solid weld objects.
Pitch	If the advanced option XS_AISC_WELD_MARK is set to `TRUE`, the center-to-center spacing of weld increments in an intermittent weld. If the advanced option XS_AISC_WELD_MARK is set to `FALSE`, the space between the weld increments in an intermittent weld. Tekla Structures uses the – character by default to separate weld length and pitch, for example 50–100. To change the separator to @, for example, set the advanced option XS_WELD_LENGTH_CC_SEPARATOR_CHAR to `@`.
	Use these buttons to copy and to link together the **Above line** and **Below line** property values. Click the and buttons to copy values between the **Below line** and **Above line** columns. Click the button to switch the linking on or off.

Setting	Description
	The middle button is blue  when the values are linked. This means that if you change a value in either of the columns, the corresponding value in the other column also changes.
Tail information	
NDT inspection level	Defines the non-destructive testing and inspection level.
Electrode classification	Defines the weld electrode classification.
Electrode strength	Defines the electrode strength.
Electrode coefficient	Defines the electrode strength coefficient.
Process type	Defines the process type.
Reference text	Additional information to appear in the weld mark. For example, information about the weld specification or process. The maximum number of characters that can be shown is 80, including one character for each line of text. To show long reference texts in reports, also adjust the template field length accordingly. Note that the special characters are shown in weld marks in model views only if the Arial font supports the special characters.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the weld. UDAs provide additional information about the weld.

List of weld types

Use the weld properties to define the weld type. Some weld types also automatically prepare the parts to be welded. The following table shows the available weld types:

Number	Type	Name	Optional automatic weld preparation	Supported solid weld object
0		None	No	No
10		Fillet weld	No	Yes

Number	Type	Name	Optional automatic weld preparation	Supported solid weld object
3		Bevel-groove (single-V butt weld)	Yes	Yes
4		Bevel-groove (single-bevel butt weld)	Yes	Yes
2		Square-groove (square butt weld)	Yes	Yes
5		Single-V butt weld with broad root face	Yes	Yes
6		Single-bevel butt weld with broad root face	Yes	Yes
7		U-groove weld (single-U butt weld)	Yes	Yes
8		J-groove weld (single-J butt weld)	Yes	Yes
16		Flare V-groove weld	No	No
15		Flare-bevel-groove weld	No	No
1		Edge-flange weld	No	No
17		Corner-flange weld	No	No
11		Plug weld	No	No
9		Bevel backing weld	No	No
12		Spot weld	No	No
13		Seam weld	No	No
14		Slot weld	No	No

Number	Type	Name	Optional automatic weld preparation	Supported solid weld object
18		Partial penetration weld (single-bevel butt + fillet)	No	Yes
19		Partial penetration weld (square groove + fillet)	No	Yes
20		Melt-through weld	No	No
21		Steep-flanked single-V butt weld	Yes	Yes
22		Steep-flanked single-bevel butt weld	Yes	Yes
23		Edge weld	No	No
24		Surfacing weld	No	No
25		Fold joint	No	No
26		Inclined joint	No	No

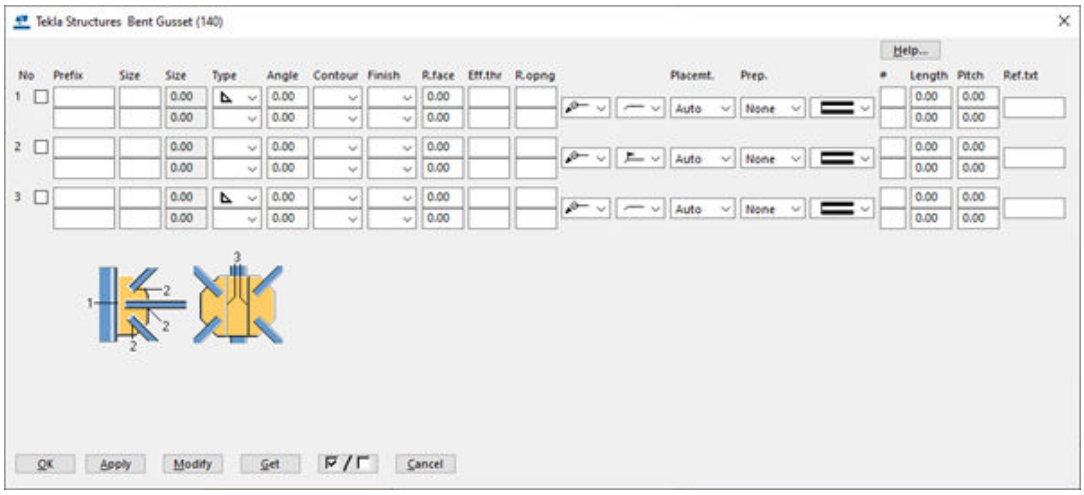
Welds in components

You can define the properties of the welds used in components. Tekla Structures displays the appropriate weld dialog when you click the **Welds** button in the component properties dialog.

The example image shows each weld definition using a number for the **Bent gusset (140)** connection. For each weld definition, use the upper row to define the above-line properties of the weld, and the lower row for the below-line properties.

You can select the weld type from the **Type** list.

For compound welds  and , you can enter two size values.



See also

Create welds (page 381)

Weld preparation

When parts are prepared for welding, their edges can be beveled to produce a groove for the weld. You can define the angle of bevels and grooves.

You can either prepare a part for welding manually, or apply a component that does it automatically, or you can use the **Preparation** options in the **Weld** properties or in the component weld properties.

Note the following:

- When you use components' weld preparation options, the supported weld types (page 389) are placed correctly in the model. If you use cuts for preparing the part edges, the welds may not be placed correctly.
- Weld preparation bevels and grooves are not dimensioned in drawings. Bevels and grooves are dimensioned only if they have been created using cut commands (page 412) on the **Edit** tab.

Prepare a part for welding with a polygon

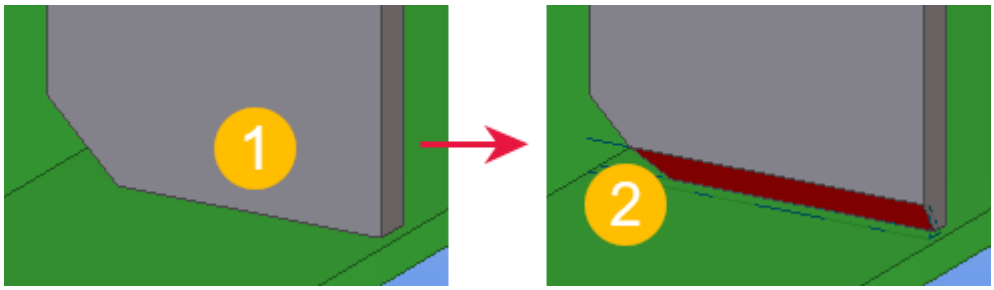
You can manually prepare a part for welding by cutting it with a polygonal shape.

Before you start, ensure that the work plane (page 56) is on the plane you are cutting on.

1. On the **Steel** tab, click **Weld** --> **Prepare part for welding with polygon**.
2. Select the part that you want to cut.
3. Pick positions to outline the polygon to be used for cutting.

   Extend the polygon outside the part, so that it is clear that the part edge should be cut away.

4. Click the middle mouse button to close the polygon and to cut the part.



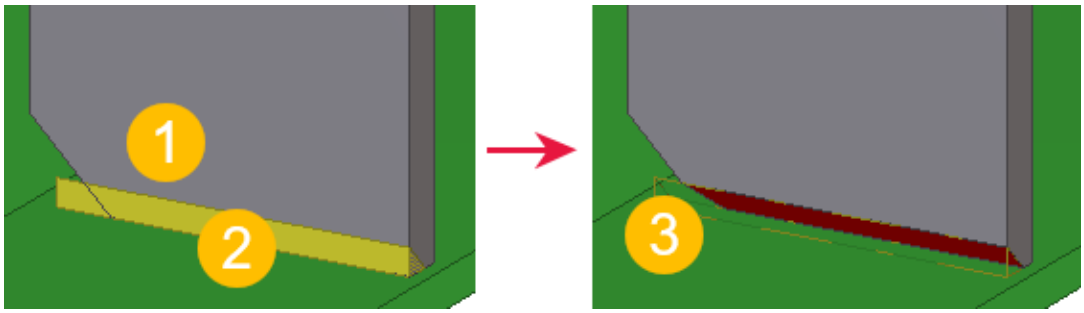
(1) The part to be cut

(2) Cuts are displayed using dash-and-dot lines

Prepare a part for welding with another part
You can manually prepare a part for welding by cutting it with another part. The cutting part will subsequently be deleted.

Before you start, create a cutting part and position it through the part you want to cut.

1. On the **Steel** tab, click **Weld** --> **Prepare part for welding with another part**.
2. Select the part that you want to cut.
3. Select the cutting part.



(1) The part to be cut

(2) The cutting part

(3) Cuts are displayed using dash-and-dot lines

Set the visibility and appearance of welds
Modify the display settings to define how welds appear in the model.

1. Double-click the view to open the **View properties** dialog.
2. Click **Display...** to open the **Display** dialog.
3. Ensure that the **Welds** checkbox is selected.

4. Select a representation option for welds:
   - **Fast**

     Use this option to show the weld symbols only.

   - **Exact**

     Use this option to show welds as solid objects with weld symbols, and to display weld marks when you select welds.

   - **Exact - no weld mark**

     Use this option to show welds as solid objects without weld symbols. Weld marks will not be displayed when you select welds.

     
5. Ensure that the view is being selected.
6. Click **Modify** to apply the changes.

NOTE If the representation option is **Exact** and you still cannot see the weld object in the model, check that the following properties have been defined for the weld in question:

- **Size**
- **Type**
- **Angle**
- **Root opening**

See also

Create welds (page 381)

Adjust the display settings (page 676)

Change a weld to a polygon weld

You can change existing welds to polygon welds if the existing welds have been created by using the **Create weld between parts** command or by a component. The new polygon welds will traverse the same points as the original welds did.

When converting double-sided weld segments to a polygon weld, Tekla Structures may not be able to create the polygon weld. If the welds to be converted consist of more than one polygon, or if there is a different number of weld segments on the sides of the welded part, Tekla Structures does not create the double-sided polygon weld but separate single-sided polygon welds.

1. Select the weld you want to change.

   To select multiple welds, hold down the **Ctrl** or **Shift** key.
2. On the **Steel** tab, click **Weld** --> **Convert to polygon weld** .

See also

Create welds (page 381)

Split a polygon weld

You can split a double-sided polygon weld into two single-sided polygon welds.

1. Select the double-sided polygon weld that you want to split.
2. Right-click and select **Split**.

See also

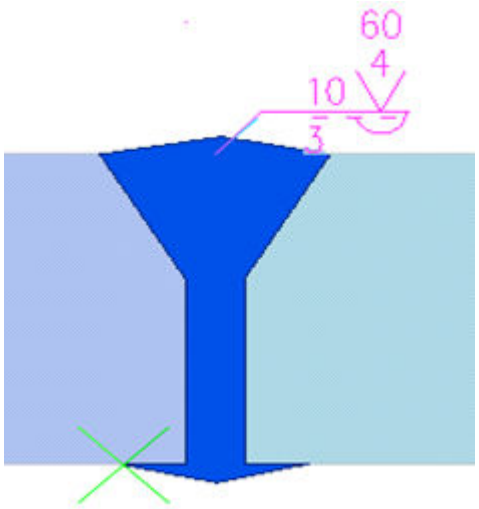
Create welds (page 381)

Change a weld to a polygon weld (page 395)

Create user-defined cross sections for welds

You can define special cross sections for model welds. This is useful when you need weld cross sections that are not predefined in Tekla Structures.

For example, you can create bevel backing welds:



To find the welds in the model that have a user-defined cross section, set **Category** to **Weld** and **Property** to **User-defined cross section** in the selection or view filter, or in the color and transparency settings.

Define a user-defined cross section for a weld

1. Select the weld you want to modify.
2. Right-click and select **Define Cross Section**.
3. In the weld cross section editor view:
   a. Pick points to indicate the weld cross section corners.
   b. Click the middle mouse button to finish picking.

Remove a user-defined cross section from a weld

You can remove user-defined cross sections from model welds and revert to the previous standard cross sections.

1. Select a weld that has a user-defined cross section.
2. Right-click and select **Remove Cross Section**.

   Tekla Structures removes the user-defined cross section and uses the previous standard cross section and properties for the weld.

Limitations

- User-defined weld cross sections are reported using only the above-line properties.

- User-defined weld cross sections do not create automatic weld preparation.

Fit part ends

Use fittings to adjust the end of a beam or a column, not the sides, by creating a straight fitting line between two points you pick. For example, you can cut a corner off a beam end to make it skew.

Fittings can also be used to extend or shorten the part end. However, do not extend or shorten the part considerably as this may lead to unexpected results. If you need to modify the part length, move the part handles or use direct modification instead.

When you use fittings to adjust the part end, Tekla Structures automatically selects the side of the part that is removed. The side to be removed depends on which side of the part midpoint the fitting is. The shorter side is cut off.

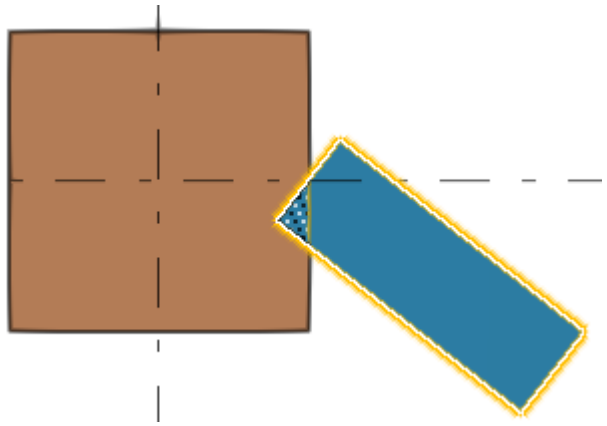
Note that fittings affect the gross length of a part. If you export information to NC files, fittings affect the length of the beam in the NC file. The overall length of the beam will be the fitted net length of the beam. This means that Tekla Structures always takes the fitting into account when calculating the beam length.

Create fittings

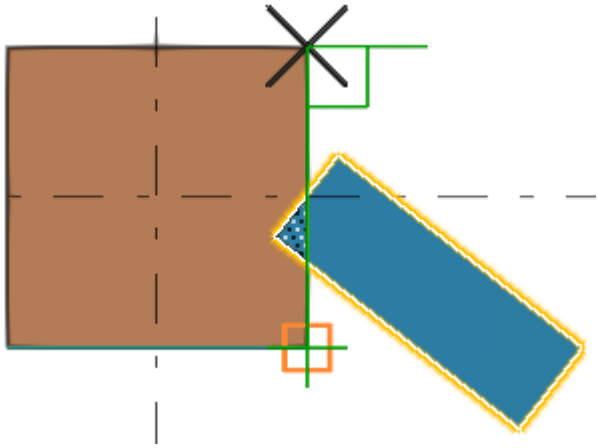
Before you start, ensure that you are working in a view with a suitable view plane. This makes it easier to pick the points for the fitting line.

- In many cases you can create a view with a suitable view plane by using the **New view** --> **Default part views** command. The command creates views which have view planes along the main axes (x,y,z) of the selected part.
- If you need the view plane on some other angle which is not along the part axis, you can first define the work plane and then use the **New view** --> **On work plane** command.

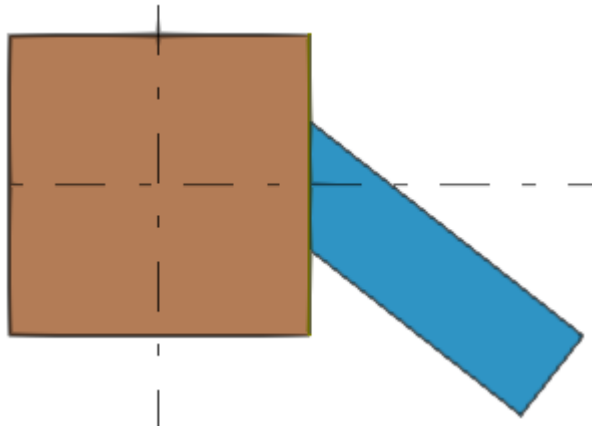
1. On the **Edit** tab, click **Fit part end**.
2. Select the part that you want to adjust with a fitting.



3. Pick the first point of the fitting line.
4. Pick the second point of the fitting line.



Tekla Structures creates a fitting plane, perpendicular to the view plane that passes through the two points you pick. Tekla Structures automatically removes the shortest side.



Fit part ends

Use fittings to adjust the end of a beam or a column, not the sides, by creating a straight fitting line between two points you pick. For example, you can cut a corner off a beam end to make it skew.

Fittings can also be used to extend or shorten the part end. However, do not extend or shorten the part considerably as this may lead to unexpected results. If you need to modify the part length, move the part handles or use direct modification instead.

When you use fittings to adjust the part end, Tekla Structures automatically selects the side of the part that is removed. The side to be removed depends on which side of the part midpoint the fitting is. The shorter side is cut off.

Note that fittings affect the gross length of a part. If you export information to NC files, fittings affect the length of the beam in the NC file. The overall length of the beam will be the fitted net length of the beam. This means that Tekla Structures always takes the fitting into account when calculating the beam length.

Should I use fittings or line cuts?

Both the fittings and line cuts can be used to adjust the part end.

With fittings

- you can adjust the part end, or to shorten or to extend the part moderately, or make the end of a part skew
- the gross length of the part is affected
- Tekla Structures automatically selects the side of the part that is removed (the shorter side is cut off)
- the part adjusting operation is faster and more lightweight in terms of Tekla Structures performance than creating line cuts

With line cuts

- you can cut the end of a beam or a column, or to cut a plate, a slab, an item, or a rebar set
- you cannot extend a part
- the gross length of the part is not affected
- you need to select the side of the part that you want to be removed

Add one fitting per one part end. However, you can use a fitting and a line cut to modify the same part end. If the end of the part needs to be inclined in two directions, first use the **Fit part end** command and then the **Line cut** command.

If you apply a second fitting to the same part end, Tekla Structures will ignore the first fitting, and use the second one.

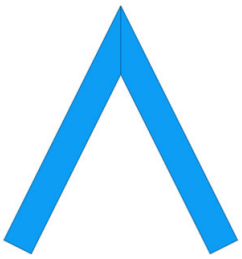
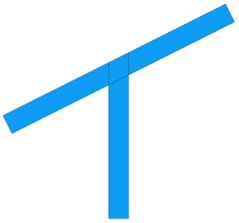
If you want have more options on how to create and adjust fittings, use the Fit beams and columns (page 400) or the Fit objects (page 403) component.

Fit beams and columns

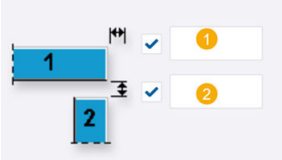
You can use the **Fit beams and columns** connection to connect two beams, two columns, or a beam and a column with fittings and cuts only.

The fittings and cuts that connect the parts can be placed in several ways. You can define that a gap is created between the parts, and use welds to connect them.

The table below shows examples of the connections.

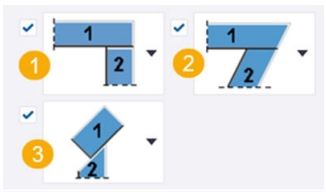
Example	Description
	Mitre connection
	Fit both profiles
	One of the two profiles is not cut

1. Select the **Fit beams and columns** component in the **Applications & components** catalog.
2. To change the default properties, open the **Fit beams and columns** component dialog and modify the properties on the **Settings** tab.
3. Define the **gap size** between the main part and the secondary part. If you do not enter any value, a gap is not created.



	Description
1	Cut offset of the main part
2	Gap between the main part and the secondary part

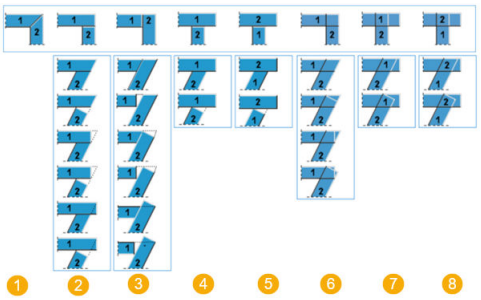
4. Select the **fitting type**.



The type you select in list **(1)** defines which fitting methods are available in list **(2)**. For the fitting type options, see the image below.

(3) When both the main part and the secondary part are skewed, select whether the fitting is created in the direction of the web or the flange.

The fitting type options are as follows:



	Description
1	Mitre
2	Secondary part is fitted to the main part, both parts are fitted.
3	Main part is fitted to the secondary part, both parts are fitted.
4	Secondary part is fitted to the main part, the main part is not fitted.
5	Main part is fitted to the secondary part, the secondary part is not fitted.
6	Secondary part is fitted with an overlap on the main part, both parts are fitted.
7	Secondary part is fitted with an overlap on the main part, the main part is not fitted.
8	Main part is fitted with an overlap on the secondary part, the secondary part is not fitted.

5. Define the **maximum gap size** between the main part and the secondary part. To do this, select the following fitting type:



Select one of the following:

- **Use default**

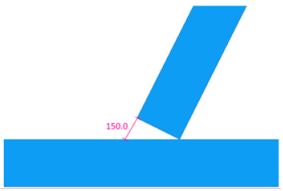
  The secondary part is fitted perpendicularly.



- **Method by threshold**

  Define the gap threshold distance. If the distance between the main part and the secondary part is shorter than the value you define, the secondary part is fitted perpendicularly.

  In the example image below, the distance is `150.0`, while the value you have defined is `200.0`.



  If the distance is greater than the threshold value you define, the fitting on the secondary part is aligned with the main part.

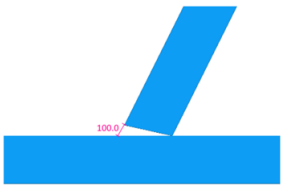
  In the example image below, the defined distance is `0`.



- **Define gap**

  The value you define is used as the distance to fit the secondary part.

  In the example image below, the defined distance is `100.0`.



6. Select whether the **Main part (1)** and **Secondary part (2)** are fitted, or cut with a line cut.
7. In **Connect parts**, select how the parts are connected:
   - **No**: Parts are not connected.
   - **Cast unit**: The secondary part is added to the main part cast unit.
   - **Sub assembly**: The secondary part is added to the main part cast unit as a sub-assembly.
   - **By welding**: The secondary part is welded to the main part.

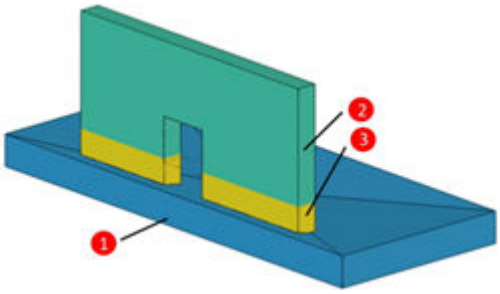
     When you select **By welding**, you can define the weld properties on the **Welding** tab.
8. On the **Welding** tab, define the weld properties.

   For detailed information on the weld properties, see the **Weld properties** section in Create welds (page 381).
9. Click **OK**.
10. In the model, select the main part (beam or column).
11. Select the secondary part (beam or column).

    The connection is automatically created when you select the secondary part.

Fit objects

You can use the **Fit objects** component to fit contour plates, slabs, beams, columns, and panels in any order.



(1) Main part

(2) Secondary part

(3) Additional material

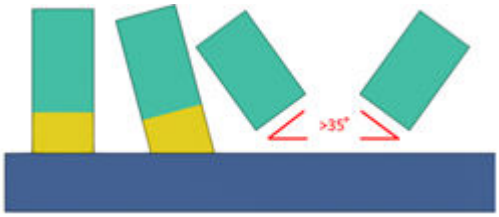
For example, in concrete constructions, you can use the **Fit objects** component to fit walls to sloping slabs and keep them connected. The walls are fitted to follow the slab geometry, and they adapt to any changes made in the slab.

You can also use the **Fit objects** component to fill the gap between inclined slabs, such as ramps and horizontal floor slabs. Filling these gaps exactly without small gaps or overlaps creates steady pours of these cast-in-place parts.

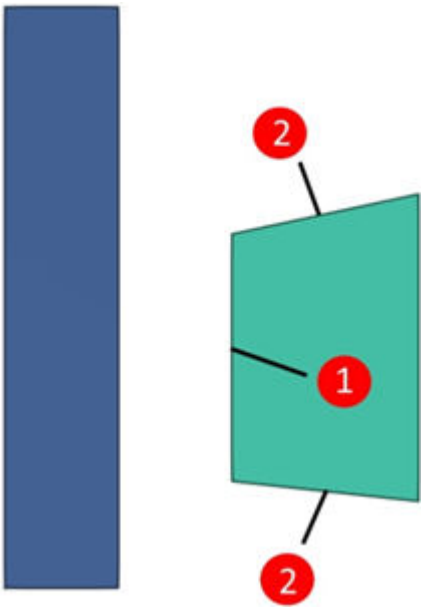
Connection logic

The **Fit objects** component fits the secondary part to the main part by creating cuts where necessary and filling the gaps with additional parts.

The component only fits secondary part faces that create an angle between -35° and 35° degrees with the main part. The example image below shows that additional material is not created for secondary parts with an angle more than 35°.



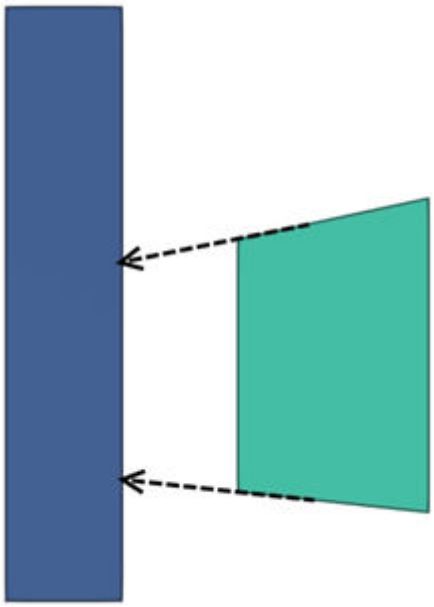
Once the main part and secondary part faces to be fitted have been identified, the secondary part face is fitted in the direction of its adjacent faces.



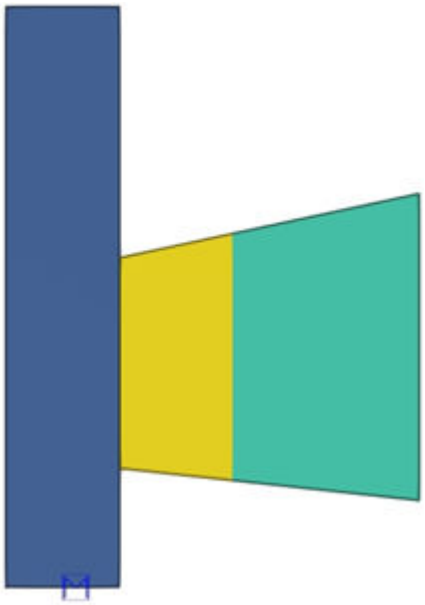
(1) Face to be fitted

(2) Adjacent faces

Planes are extended from the adjacent faces until they intersect with the main part.



Additional material is created to fill the space between the extended planes, the main part, and the secondary part.



Limitations

- Curved panels, polywalls, and polybeams are not supported.
- **Fit to main part** setting: With the **Face projection** option, only the closest face of the main part is projected. Multiple faces cannot be projected.

- Additional material is not always created from the same object type as the secondary part. For example, for walls, the additional material is created as a slab.
- Parts created with sketched profiles are not supported as secondary parts.

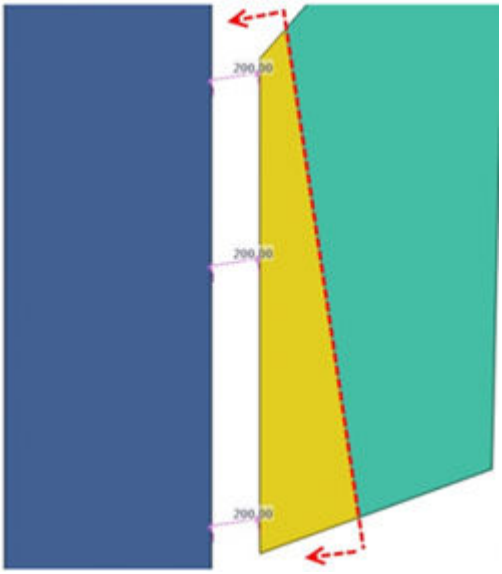
Fit objects in the model

We recommend that you first define the properties that you want to use when fitting the objects. When you have defined the properties, select the parts that you want to fit, and create the fitting in the model.

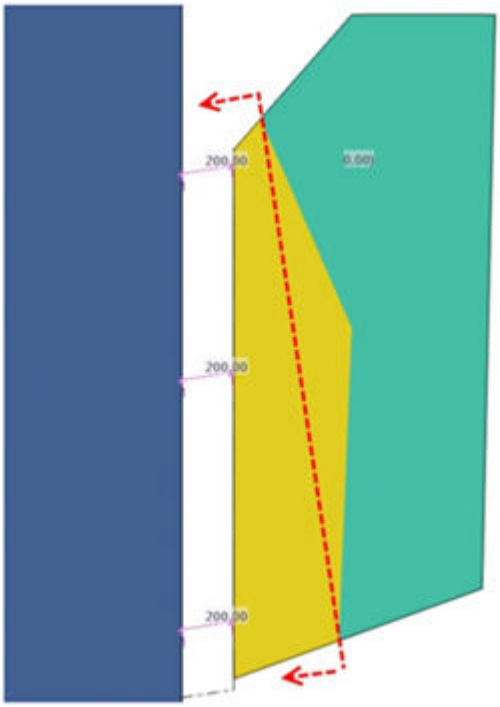
1. Select the **Fit objects** component in the **Applications & components** catalog.
2. To change the default properties, open the **Fit objects** dialog and modify the properties.
3. **Create gap**: Define the gap distance between the main part and the additional material or the secondary part. To create a fitting that is level with the main part, leave the value as `0.0`.



The gap distance is measured in the direction perpendicular to the secondary part face, shown with the dashed line in the image below. After the gap direction is established, the same gap is applied along the main part face. In the example, the gap distance is `200 mm`.



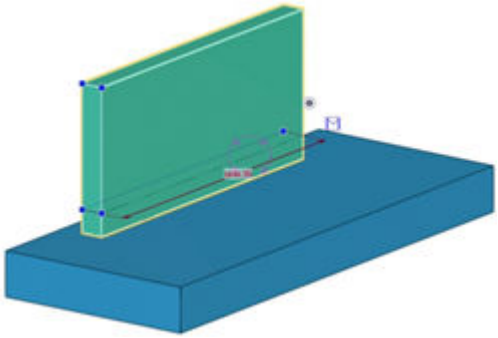
When more than one face is fitted, the gap direction is perpendicular to an imaginary line between the outer faces of the object. To use the same example, when there is only one face between the outer faces, the direction of the gap distance remains the same.



4. **Create additional material**: Select what kind of additional material is created.

   - **As added material**

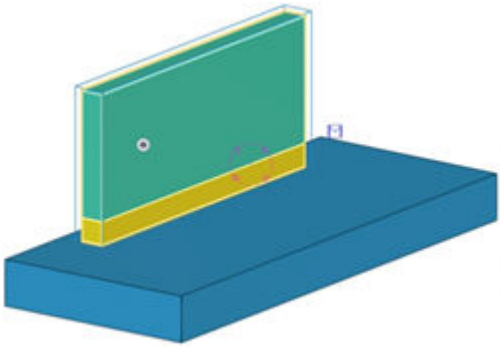
     Tekla Structures automatically attaches the added material (page 359) to the secondary part to form one single combined part.

     This is a useful option if you are creating quantity reports based on parts, because quantities such as `HEIGHT` take into account the added material and attempt to take the full height of the combined part.

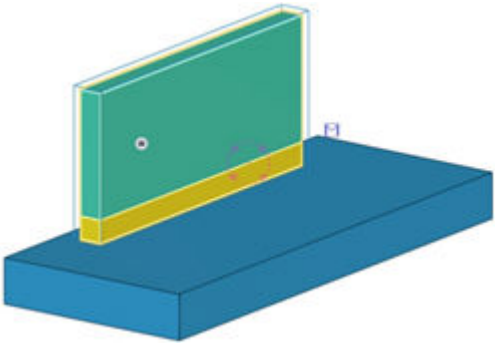
- **As parts added to assembly/cast unit**

  Additional material is added to the cast unit (concrete) or assembly (other material) of the fitted part.



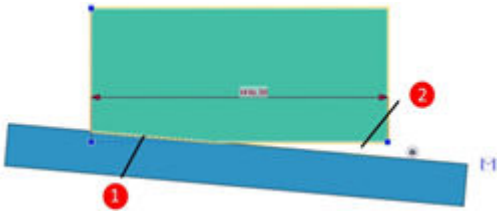
- **As loose parts**

  Additional material is created as a separate individual part.



- **No**

  Additional material is not added to fill the gap between the part to be fitted and the main part. Note that if the part to be fitted overlaps with the main part, cuts are used to remove the overlapping material.



  (1) Wall fitted only by cutting material

  (2) Additional material not added

5. **Create to**: Select whether the part is fitted to the nearest face or to all faces within a certain distance.

   - **Nearest face** only fits the nearest face to the main part.

     Fit objects considers each secondary part face one at a time to determine which secondary part face is closest to the main part. For each face, **Fit objects** checks the following:

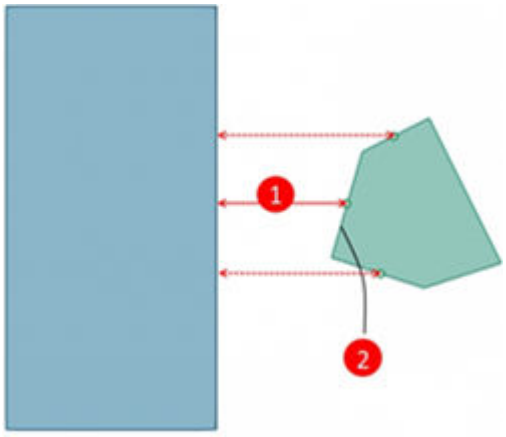
     a. Identifies the center point of the secondary part face.

     b. Creates a line from the center point of the secondary part face towards the main part, perpendicular to the main part face, until the line intersects with the main part.

     c. Measures the line.

     d. Repeats steps **a - c** for all secondary parts and the main part.

     The shortest line indicates the two closest faces. The face corresponding to the secondary part is the **Nearest face**.

     (1) Shortest distance

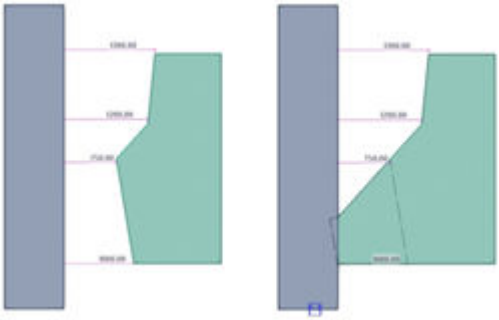
     (2) Face to be fitted (= nearest face)

     The result is shown in the image below:

   - **Faces closer than** fits all the faces that are closer than the specified distance.

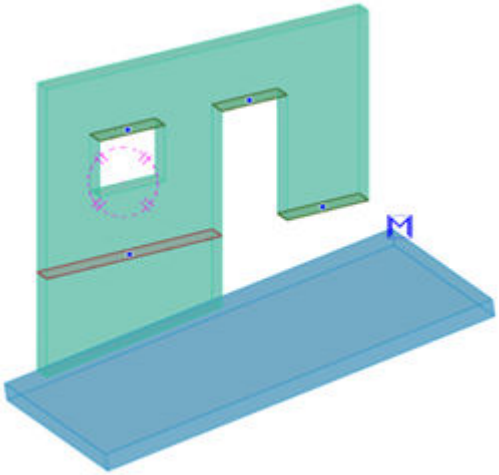
     The distance is measured similarly as in **Nearest face**, except that all points of a face need to be below the defined distance for that face to be fitted.



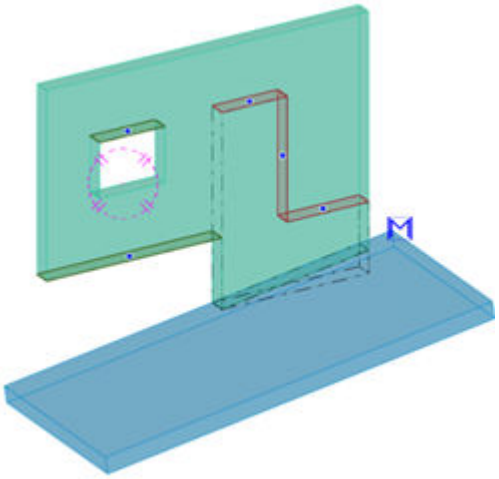
In the example above, the used limit distance is `1000 mm`. The only face that has all its vertexes closer than `1000 mm` is the face near the bottom, and it is therefore the only face fitted.

- Use **Direct modification** to manually select which secondary part faces are to be fitted.

  In the model, select the component. The green and red direct modification handles that appear on the secondary part faces indicate the faces that can be fitted.



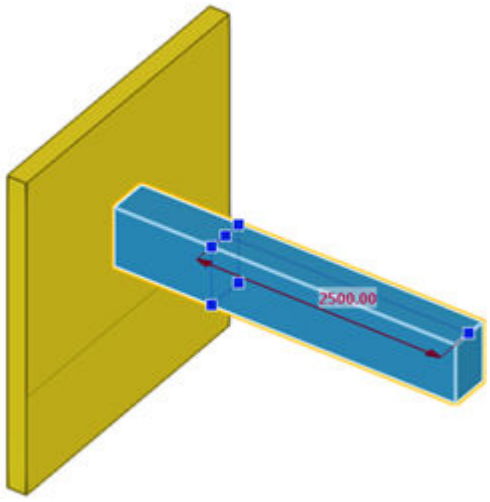
Red and green faces indicate the faces that have been and that have not been fitted, respectively. Click a green face to fit that face. Click a red face to stop that face from getting fitted, as shown in the example image below.



6. **Use fit part end if possible**: Add fittings to parts in the same way as with the Fit part end command (page 397).

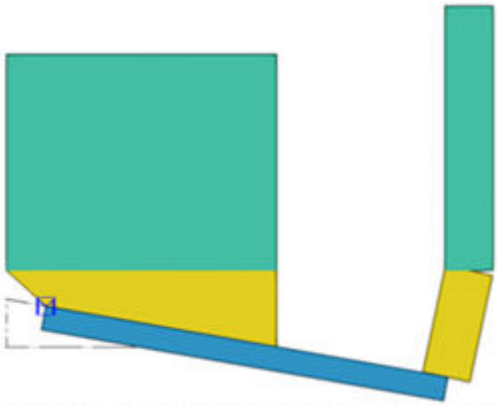
   This option only applies to those objects that can be fitted with the **Fit part end** command: beams, columns, and footings. Slabs, plates, and walls cannot be fitted in this way, so this option does not affect them.

   - **Yes**: If possible, parts are fitted by using the **Fit part end** command.

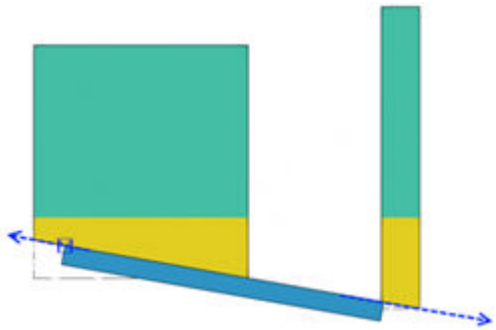


   - **No**: The **Fit part end** command is not used, and parts are fitted using the option you select in **Create additional material**.

7. **Fit to main part**: Define how the main part is used to fit the secondary part.
   - **Actual face**: The secondary parts are fitted exactly to the geometry of the main part. The image below shows an example of **Actual face** applied to a wall and a column.



   - **Face projection**: The secondary parts are fitted to the plane that is coplanar with the main part face. The image below shows an example of **Face projection** applied to a wall and a column.



8. Click **Modify** to save the properties that you have defined.
9. Select the main part (part to be fitted to).
10. Select the secondary part (part to be fitted).

    The part is fitted when you select the secondary part.

Cut objects with a line, a polygon, or a part

Use line cuts, polygon cuts, or part cuts to shape a part in a model. If you need to modify the part length, do not use cuts but move the part handles or use direct modification instead.

Limitations

- Do not use cuts to cut a part into two. Numbering, material lists and drawings will consider the part as one, not as two separate parts.

- Do not use cuts to shorten steel parts because cuts are not taken into account in LENGTH_GROSS, which is often used in steel material lists.

Line cuts

Use line cuts to shape a beam or a column, or to cut a plate, a slab, an item, or a rebar set.

A line cut creates a straight cutting line between two points you pick, perpendicular to the view plane that passes through the two picked points.

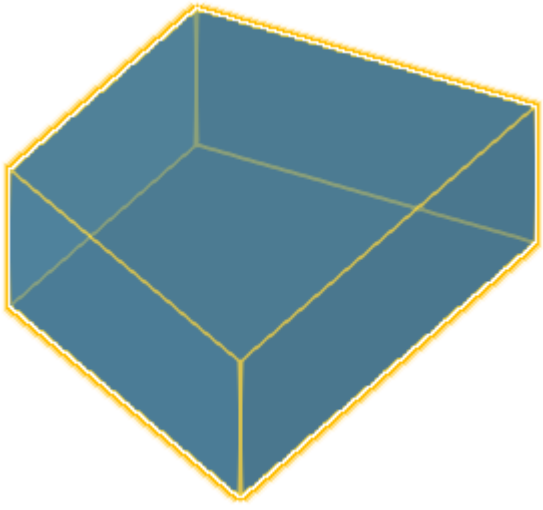
Use line cuts to cut the end of a part or the side of a part. For example, make a part narrower or cut a corner off from a beam end. You can have several line cuts to one part end.

Before you start, ensure that you are working in a view with a suitable view plane. This makes it easier to pick the points for the cutting line.

- In many cases you can create a view with a suitable view plane by using the **New view** --> **Default part views** command. The command creates views that have view planes along the main axes (x,y,z) of the selected part.
- If you need the view plane at some other angle, not along the part axis, you can first define the work plane and then use the **New view** --> **On work plane** command. Alternatively, you can use the **3D cut** component from the **Applications & components** catalog.

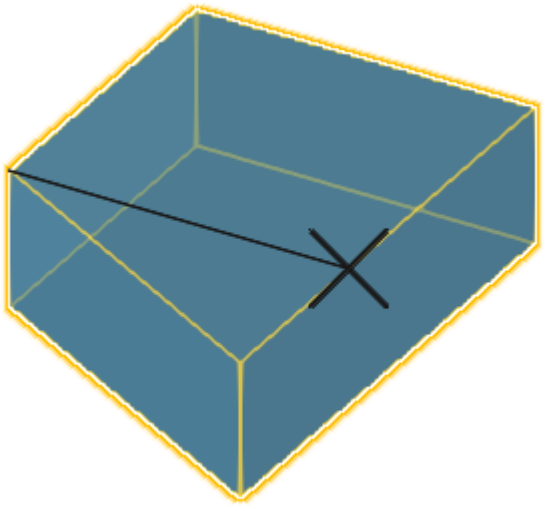
Cut objects with a line

1. On the **Edit** tab, click **Line cut**.
2. Select the object that you want to cut.

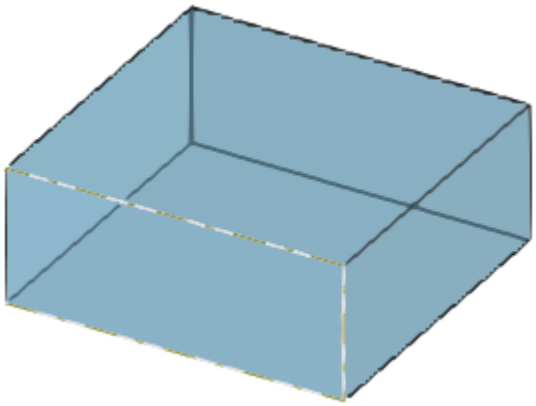


3. Pick the first point of the cutting line.

4. Pick the second point of the cutting line.



5. Pick the side that you want to remove.



Tekla Structures displays the cut using dash-and-dot lines.

You can hide the cuts, if needed.

Polygon cuts

Use a polygon cut to cut a part, or a rebar set, with a closed polygon, perpendicular to the plane that is defined by the points you pick.

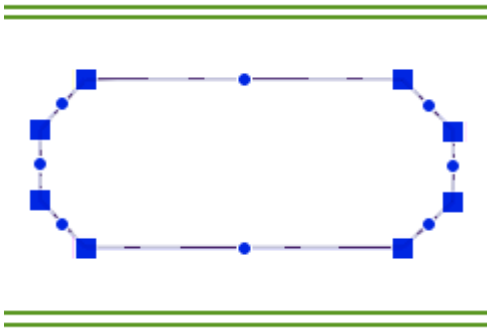
Before you start, ensure that you are working in a view with a suitable view plane. This makes it easier to pick the points for the polygonal shape.

In many cases you can create a view with a suitable view plane by using the **New view** --> **Default part views** command which creates views that have view planes along the main axes (x,y,z) of the selected part.

Cut objects with a polygon

1. On the **Edit** tab, click **Polygon cut**.

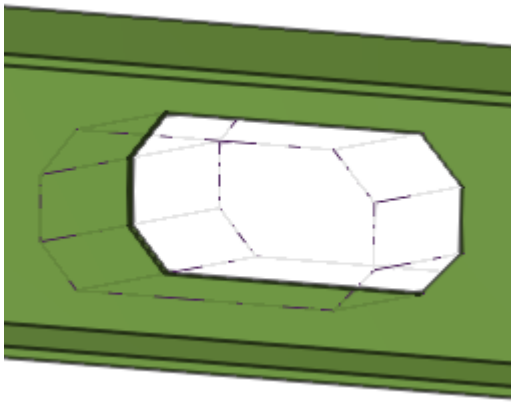
2. Select the object that you want to cut.
3. Pick points to outline the polygon to be used for cutting.



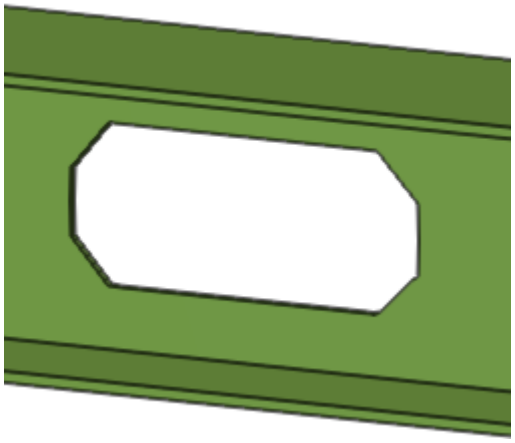
When defining the polygon, do not place the cut edges exactly at the same location as the edge of the part to be cut, as it can be unclear whether the edge should be cut away or not.

The depth of the cut is calculated automatically based on the dimensions of the object to be cut. The cut is automatically extended slightly (3 mm) outside of the part face in depth direction.

4. Click the middle mouse button to close the polygon and to cut the object.



Tekla Structures displays the cut using dash-and-dot lines. You can hide the cuts, if needed.



If you need to modify the cut shape, use part handles or direct modification to add or remove points or to move an edge.

You can add chamfers to the corners of the polygon cut to fine tune the shape. Chamfers have no effect on NC files.

Polygon cuts have properties that you can modify in the property pane.

Part cuts

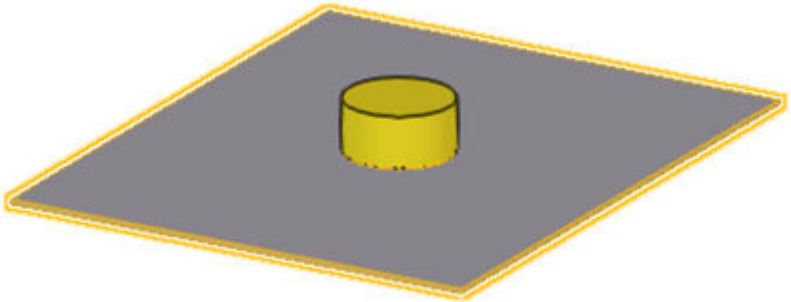
Cut a part, or a rebar set, using another part.

You can also cut parts that already have cuts. This can be useful, for example, when you want to create more sophisticated cut shapes.

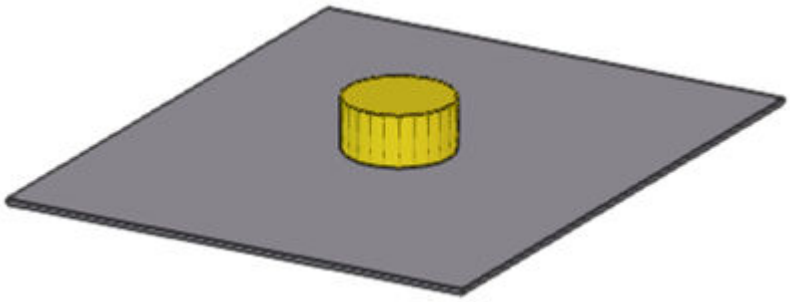
Cut objects with a part

1. If you do not have a part to cut with, create a cutting part and position it through the part you want to cut.

   Do not place the cutting part so that the cut edges or vertices are exactly at the same location as the edges or vertices of the part to be cut. It may be unclear what should be cut away.
2. On the **Edit** tab, click **Part cut**.
3. Select the object that you want to cut.



4. Select the cutting part.



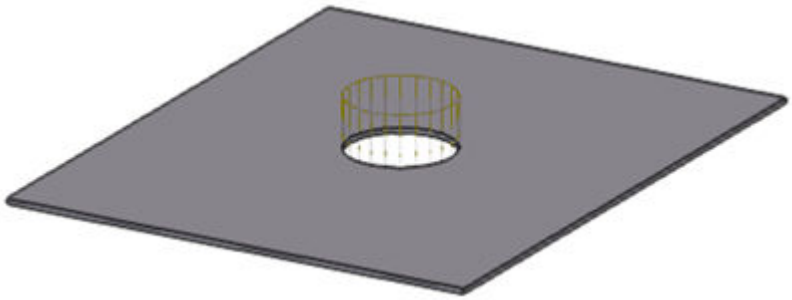
Using the cutting part, Tekla Structures creates the cut to the selected object. The part cut does not affect other objects.

Tekla Structures displays the cut using dash-and-dot lines.

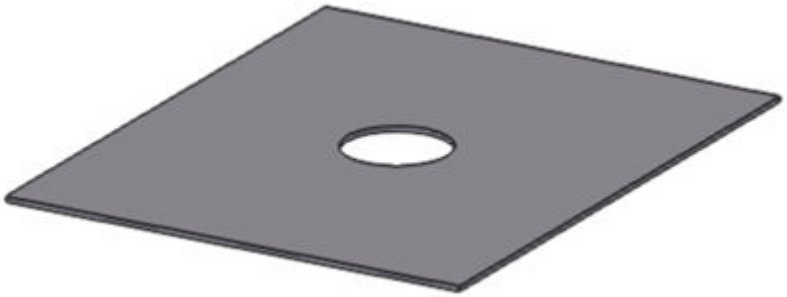
5. Delete the cutting part.

   a. Ensure that the  **Select cuts and added materials** selection switch is off.

   b. Select the cutting part and press **Delete**.



You can hide the cuts, if needed.



Cutting parts have properties that you can modify in the property pane.

Hints and tips for efficient cutting

- **Consider how to place the cut edges**

  Avoid placing cut edges, cut vertices, or other cuts cutting the same part exactly at the same location as the edges or vertices of the part to be cut. Extend the cut outside of the part or other cut at least 0.3 mm. This practice helps you to avoid solid errors.

- **If a cut causes a solid error**

  If a cut causes a solid error, Tekla Structures is unable to render part faces and the part becomes transparent, only some edge lines being visible. An error notification is printed in the session history log stating which part and which cut caused the failure.

  To locate the failure in the model, click a row that contains a GUID identifier in the session history log. Tekla Structures selects the corresponding part and cut in the model.

  To fix the error, move the problematic cut slightly (0.3 mm) to a different direction. If the cut is intersecting with other cuts, you can try to move also the other cuts.

- **Use edge chamfers**

  Whenever possible, use edge chamfers (page 421) instead of small cuts, especially in components.

- **Flange cuts of steel profiles**

  When cutting a flange, if the cutting part cuts very slightly the web as well (at least 0.3 mm), the cut is more likely to succeed. For example, if you are cutting a beam that has roundings, it may be useful to cut even further onto the web in depth direction than just the flange thickness.

- **Cuts for round tubes**

  Use the Round tube (23) component for round tube cuts. The component automatically rotates the cutting part until a successful cut position is found. If the component fails, rotate the cutting part slightly until you find a successful cut position.

Hide cuts in a model view

1. Double-click the view background to open the **View properties** dialog.
2. Click **Display...** to open the **Display** dialog.
3. Ensure that the **Cuts and added material** option is **not** selected in the display settings.
4. Click **Modify**.
5. If you want to temporarily display the hidden cuts in a model view:
   a. Select the part.

b. Click **Display detailing** on the contextual toolbar.

All the cuts of the selected part are displayed. To hide them again, redraw the view (page 49).

Polygon cut properties

Use the **Polygon cut** properties in the property pane to view and modify the properties of a polygon cut.

Note that the polygon cut properties are available in the property pane only after a polygon cut has been created and selected. You cannot access or modify the cut properties before the cut is created.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	Name of the polygon cut.
Profile	Profile of the polygon cut, by default parametric profile `BL`.
Material	Material of the polygon cut, by default `ANTIMATERIAL`. The cut material cannot be changed.
Class	Use to group polygon cuts.
Position	
At depth	Position of the polygon cut in depth direction.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the cut. UDAs can be used to store more information for each cut, for example, to control the dimension creation in drawings.

Part cut properties

Part cut uses the properties of the cutting part. For example, if the cutting part is a steel beam, part cut uses the **Steel beam cut** properties. The default part cut properties depend on the used cutting part.

If you want to prevent the copying of user-defined attributes (UDAs) from the original cutting part to the anti-material cutting part, use the XS_PART_CUT_INHERIT_UDAS_FROM_CUTTING_PART advanced option. When set to `FALSE`, the UDAs are not copied to the anti-material part.

If the cutting part is a hollow section, the cut is created with a similar, but non-hollow section, so that the inside of the cut is removed as well.

Note that the part cut properties are available in the property pane only after a part cut has been created and selected. You cannot access or modify the cut properties before the cut is created.

Chamfer part corners and edges

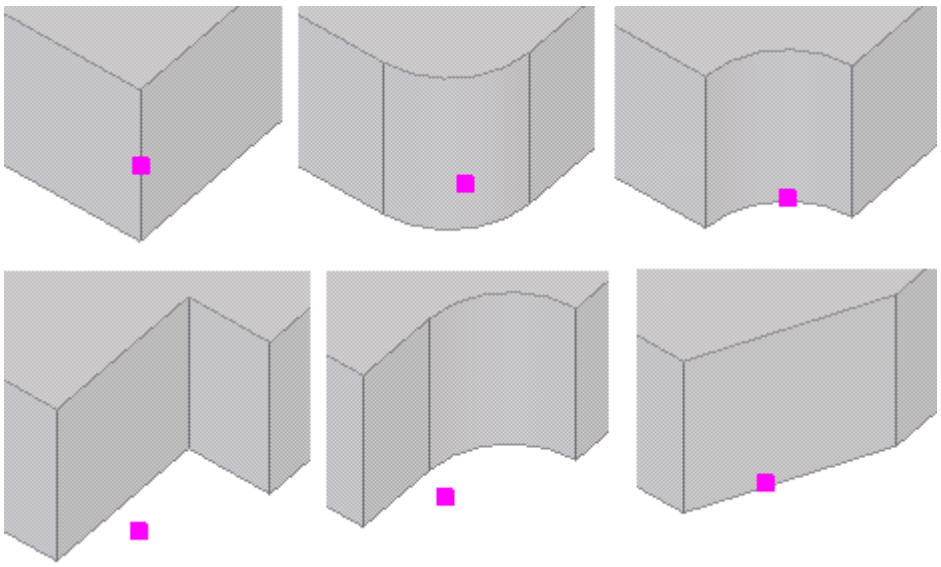
Chamfers are modeling details that can be used to refine the shape of parts for aesthetic, practical, and manufacturing reasons. In Tekla Structures, you can chamfer part corners and part edges.

Limitations:

- Only the following parts have corner chamfers: contour plates, concrete slabs, strip footings, steel and concrete polybeams, and concrete panels.
- The end points of a part do not have corner chamfers. The handles that you select must be at corner points or between two segments of a part.

Chamfer part corners

When Tekla Structures creates a part, by default it has a rectangular chamfer at each corner, which does not change the geometry of the part. You can modify the default chamfers.



TIP To make it easier to select the handles of part corners, ensure that the **Direct modification** switch  is **not** active.

1. Select the part.
2. Double-click the handle of a part corner.

   The **Corner chamfer** properties open.
3. Modify the chamfer properties.
4. Select the handles of the part corners you want to modify.
5. Click **Modify**.

Chamfer part edges

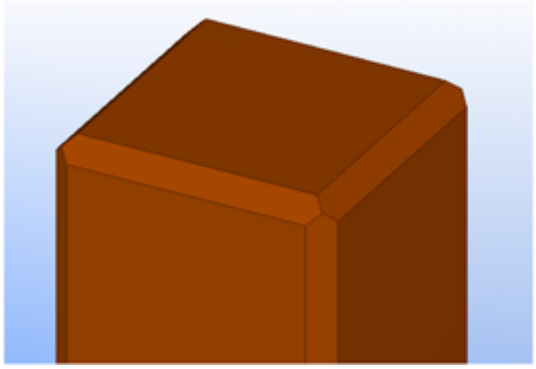
1. Double-click the view to open the **View properties** dialog, click the **Display...** button, and ensure that the **Cuts and added material** option is **not** selected in the display settings.
2. On the **Edit** tab, click **Chamfer edge**.

   Alternatively, you can start the command in the **Object type list** in the property pane.
3. Select the part you want to chamfer.
4. Pick a point where you want the chamfer to start on the part edge.
5. Pick a second point where you want the chamfer to end on the part edge.

   Tekla Structures displays the chamfer in light blue.
6. If needed, you can modify the chamfer.
   a. Double-click the chamfer to open the **Edge chamfer** properties.
   b. Change the chamfer properties.
   c. Click **Modify**.

   TIP Alternatively, use the contextual toolbar to modify the edge chamfer.
7. Right-click the view and select **Redraw view**.

   Tekla Structures removes the chamfered edge.



Corner chamfer properties

Use the **Corner chamfer** properties in the property pane to view and modify the properties of a corner chamfer. To open the properties, double-click the handle of chamfered corner.

The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.

Setting	Description
Shape	
Type	Shape of the chamfer. See **Corner chamfer types and dimensions** for more information.
X / **Distance X** / **Radius**	Dimensions of the chamfer. The dimension depends on the chamfer type.
Y / **Distance Y** / **Radius**	
Dz1	Only used for contour plates and concrete slabs. Moves the top or bottom surface of the part corner in the part's local z direction. Use these options, for example, to give plates varying thicknesses.
Dz2	

Corner chamfer types and dimensions

The table below describes the available corner chamfer types and dimensions. The chamfer type numbers can be used in sketches and custom components. Straight chamfers can have different dimensions in two directions. Curved chamfers only use one dimension.

Number	Type	Symbol	Dimensions
0	**None**		x: not used y: not used
1	**Line**		x:the distance in the x coordinate direction from the corner y: the distance in the y coordinate direction from the corner
2	**Rounding**		x: the radius y: not used
3	**Arc**		x: the radius y: not used

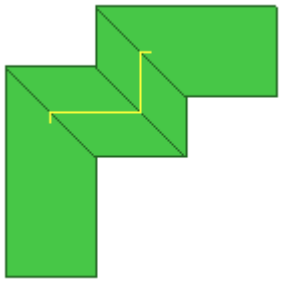
Number	Type	Symbol	Dimensions
4	**Arc point**		x: not used y: not used
5	**Square**		The chamfer is perpendicular to the edges. x: the distance in the x coordinate direction from the corner y: the distance in the y coordinate direction from the corner
6	**Square parallel**		The chamfer is parallel to the opposite edge. x: the distance in the x coordinate direction from the corner y: the distance in the y coordinate direction from the corner
7	**Line and arc**		x (if smaller than y): the arc radius x (if bigger than y): the distance in the x coordinate direction from the corner y (if smaller than x): the arc radius y (if bigger than x): the distance in the y coordinate direction from the corner

Status of corner chamfers in polybeams

To see the corner chamfer lines of polybeams, set the advanced option XS_DRAW_CHAMFERS_HANDLES to `CHAMFERS` or to `CHAMFERS_AND_HANDLES`.

Tekla Structures shows the status of polybeam chamfers using the following colors:

Color	Description	Example
Magenta	Correct chamfer	

Color	Description	Example
Yellow	Correct chamfer that cannot be unfolded	
Red	Incorrect chamfer	

Edge chamfer properties

Use the **Edge chamfer** properties in the property pane to view and modify the properties of an edge chamfer. To open the properties, double-click the edge chamfer when the chamfer if visible in the model. The file name extension of an edge chamfer property file is `*.cha`.

The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.

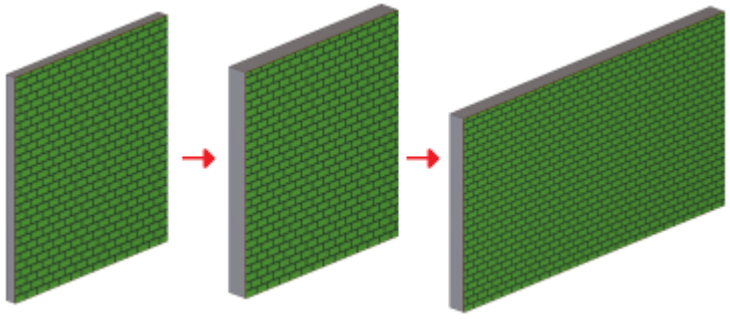
Setting	Description	More information
Shape on edge		
Type	Shape of the chamfer.	
Distance X	Defines how far away from the chamfered edge the chamfer will end in the x direction.	
Distance Y	Defines how far away from the chamfered edge the chamfer will end in the y direction.	
Shape at end		
First end type	The shape and position of the first end point.	The options are: • **Full**: The end point is positioned at the end

Setting	Description	More information
Second end type	The shape and position of the second end point.	of the part (moving along the nearest edge), and the shape is straight. • **Straight**: The end point is positioned at the point you pick, and the shape is straight. • **Bevelled**: The end point is positioned at the point you pick, and the shape is angled.
Distance	The distance between the (picked) end point and the bevelled points.	
General		
Name	Name of the chamfer.	

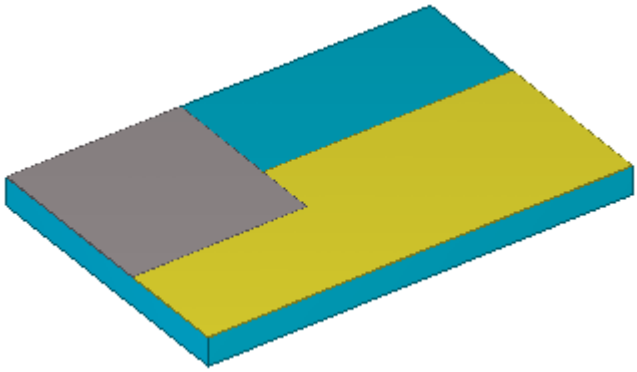
Add surface treatment to parts

Use the surface treatment tools to add surface treatment to parts. Surface treatment for concrete parts include flat finishes, surface mixes, and tiles. Surface treatment for steel parts include fire-proofing and unpainted areas, for example.

When you modify the shape or size of a part, Tekla Structures automatically modifies the surface treatment to fit the part (page 438).



When you create overlapping surface treatments, the smaller surface treatment overrides the larger one. The overlapping area is recognized in reports: only the topmost (visible) surface treatment is calculated.



Add surface treatment to an entire part face

1. On the **Edit** tab, click **Surfaces** --> **Surface treatment to part face** .
2. Pick the origin of the surface treatment.
3. Pick a point to indicate the direction of the surface treatment.
4. Select the part to apply the surface treatment to.
   a. Move the mouse cursor over a part. Tekla Structures highlights the faces that you can select.
   b. Select the part face.

Add surface treatment to a selected area on a part face

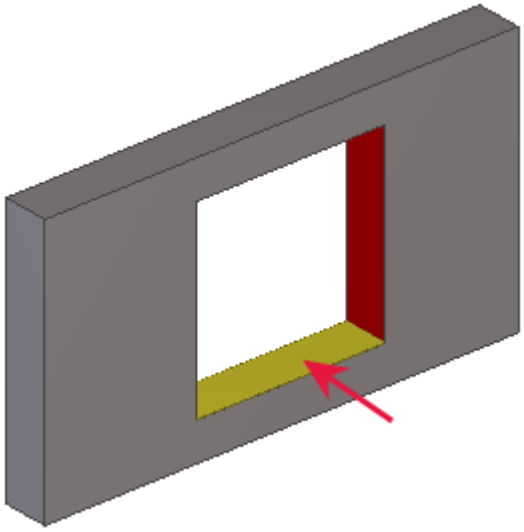
1. On the **Edit** tab, click **Surfaces** --> **Surface treatment to selected area** .
2. Pick the origin of the surface treatment.
3. Pick a point to indicate the direction of the surface treatment.
4. Select an area of the part face to apply the surface treatment to.
   a. Move the mouse pointer over a part. Tekla Structures highlights the faces that you can select.
   b. Select the part face.
   c. Pick three or more points on the part face to define a polygonal area.

Add surface treatment to all faces of a part

1. On the **Edit** tab, click **Surfaces** --> **Surface treatment to all faces of part** .
2. Select the part to apply the surface treatment to.

Add surface treatment to cut faces

1. On the **Edit** tab, click **Surfaces** and then either **Surface treatment to part face** or **Surface treatment to selected area**.
2. Pick the origin of the surface treatment.
3. Pick the direction.
4. Select the cut face to apply the surface treatment to:



5. If you are using the **Surface treatment to selected area** command, pick the points to define the area of the surface treatment.

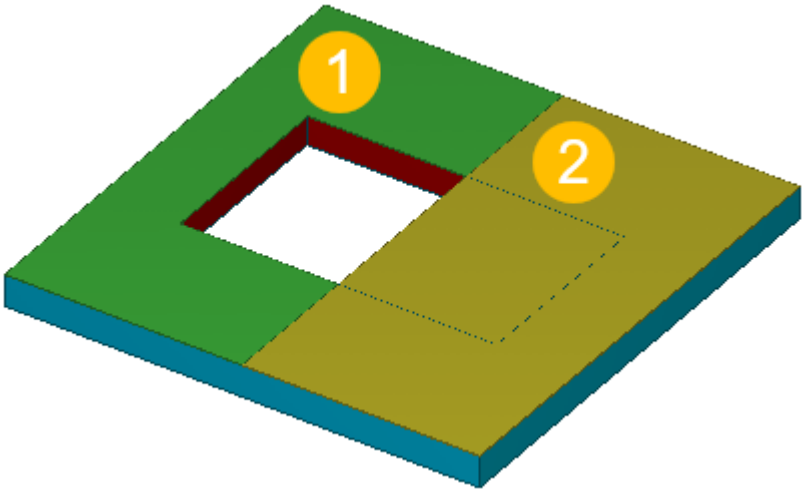
Surface treatment on chamfered parts

Take these things into account when adding surface treatment to chamfered parts:

- Surface treatment does not work on sketched profiles with chamfers.
- Add surface treatment before chamfering the part. If surface treatment is applied to a chamfered part, the surface treatment chamfer cannot be modified later on.
- The chamfers for the main part and surface treatment are separate. Modifying the main part chamfer does not affect the surface treatment chamfer.
- The orientation of unsymmetrical chamfers depends on the face where it was created (such as top, bottom, left, or right). To change the orientation of an unsymmetrical chamfer, you must swap the chamfer's x and y values.

Surface treatment on parts with openings and recesses

To force Tekla Structures to consider openings and recesses in parts when adding surface treatment, set the **Cut by father part cuts** to **Yes** in the **Surface treatment** properties.



(1) The green surface treatment has the **Cut by father part cuts** set to **Yes**.

(2) The tiled surface treatment is not cut by the cut in the part: **Cut by father part cuts** is set to **No**.

NOTE If you use the **Surface treatment to all faces of part** command and set the **Cut by father part cuts** to **Yes**, Tekla Structures automatically adds surface treatment also to the cut faces.

Modify surface treatment properties

1. If the property pane is not open, double-click the surface treatment to open the **Surface treatment** properties.
2. Change the properties as needed.
3. Click **Modify**.

Note that if you modify the **Pattern** properties, you need to first click **Modify** in the property pane and then redraw the view to make the changes visible.

Surface treatment properties

Use the **Surface treatment** properties in the property pane to view and modify the properties of a surface treatment. To open the properties, double-

click the surface treatment. The file name extension of a surface treatment property file is `*.srf`.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the surface treatment. The name can contain a maximum of 61 characters.
Type	Select the type of surface treatment.
Subtype	Select the subtype of a specific surface treatment.
Material	Depending on the type of the surface treatment, select the material of the surface treatment.
Color	Depending on the type of the surface treatment, set the color of the surface treatment.
Thickness	Depending on the type of the surface treatment, enter the thickness of the surface treatment.
Cut by father part cuts	To force Tekla Structures to consider openings and recesses in parts when adding surface treatment, set this to **Yes**.
Position	
In depth	Select the location of the surface treatment, and set the **Depth offset** value.
Pattern (for tiled surface treatment)	
Pattern	If you have set **Tile surface** as the **Type** of the surface treatment, select the pattern of the tiled surface treatment.
Tile width **Tile height**	Define the tile width and height.
Mortar height **Mortar width**	Define the mortar height and width.

Setting	Description
Tile color **Mortar color**	If needed, use the color picker to pick the tile and the mortar color.
More	
UDAs	Click the **User-defined attributes** button to open the user-defined attributes (UDAs) of the surface treatment. UDAs provide more information about the surface treatment.

Define new surface treatment subtypes

You can add new options to the **Subtype** list in the **Surface treatment** properties by editing the `product_finishes.dat` file.

1. Copy the `product_finishes.dat` file to your firm, project, or model folder. The file is located under the `\ProgramData\Trimble\Tekla Structures\<version>\environments` folder. The exact file location may vary depending on the folder structure of your environment files.
2. Open the copied file using a text editor, such as Microsoft Notepad.

   The first section of the file explains the available types of surface treatment. The surface treatment types are hard-coded, so do not modify this section:

   ```
   // Product finishes
   // ------------------------
   //
   // Type          : Type of surfacing
   //                     1 = concrete finish
   //                     2 = special mix
   //                     3 = tile surface
   //                     4 = steel finish
   ```

3. Go to the sections that define the options for each type of surface treatment:

   ```
   // ========================================
   // *** Concrete Finish
   // ========================================
   // WET FINISH
   // ----------
   1            MF       "Magnesium Float"
   1            SMF      "Smooth Magnesium Float"
   1            WT       "Wet Trowel"
   ```

4. Add rows to define new options.

a. Define the surface treatment type. For example, `1` for concrete finish.

b. Define a code for the surface treatment option. For example, `MF` for Magnesium Float.

c. Define the full name of the surface treatment option. For example, `Magnesium Float`. Remember to enclose the name in double quotes `" "`.

5. Save the file.

See also

Add surface treatment to parts (page 425)

Tiled surface treatment

Tekla Structures includes complex tile and brick surface treatment options, such as basketweave and herringbone patterns. Tiled surface treatment options are based on repeating tile patterns that are stored in XML format.

The tile pattern surface treatment are available in **Surface treatment** properties if you have set the **Type** to **Tile surface**.

Define a new tile pattern

1. Copy the `TilePatternCatalog.xml` file to your firm, project, or model folder. The file is located under the `\ProgramData\Trimble\Tekla Structures\<version>\environments` folder. The exact file location may vary depending on the folder structure of your environment files.
2. Open the copied file using a text editor, such as Microsoft Notepad.
3. Add a new `<TilePattern>` element to the file.

   The `<TilePattern>` element must have `<HOffset>` and `<VOffset>` elements and at least one `<Tile>` element. Other elements are optional.

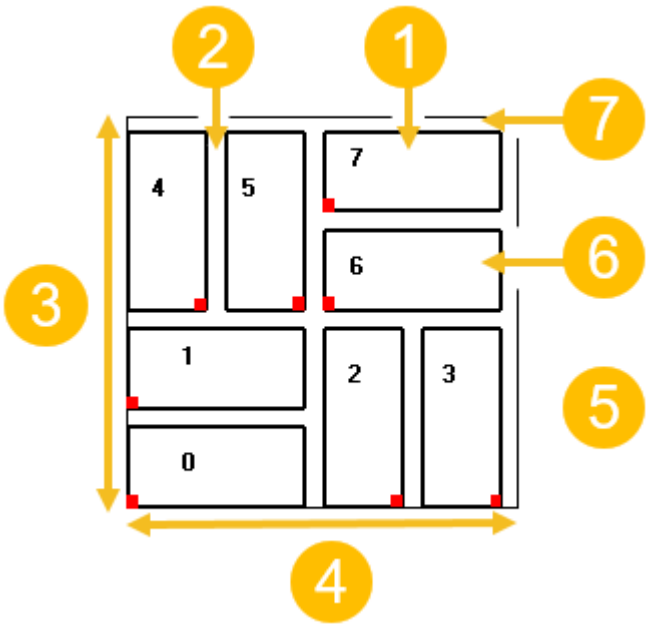
   TIP You may find it easier to copy one of the existing elements, and then modify it to suit your needs.

4. Repeat adding `<TilePattern>` elements for all the patterns you want to define.
5. Save the `TilePatternCatalog.xml` file.

Example tile pattern definition

This example explains how the **Basketweave** tile pattern is defined in the `TilePatternCatalog.xml` file.

The **Basketweave** pattern block is made up of eight tiles:



(1) Tile width

(2) Mortar width

(3) `VOffset`

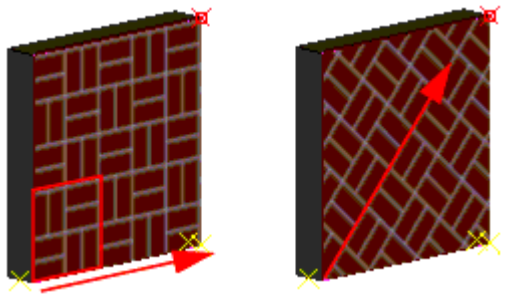
(4) `HOffset`

(5) Red marks indicate `TileOrigin`. Angle value for vertical tiles is 90.

(6) Tile height

(7) Mortar height

The pattern is repeated in the x and y direction of the surface treatment, starting from the origin of the surface treatment. You can run the pattern in different x directions:



In the `TilePatternCatalog.xml` file, the pattern is defined as follows:

```
<TilePattern Name="Basketweave">
        <Parameter Name="W" DefaultValue="220">
                <Label> _Tile_Width </Label>
        </Parameter>
        <Parameter Name="H" DefaultValue="100">
                <Label> _Tile_Height </Label>
        </Parameter>
        <Parameter Name="TH" DefaultValue="100">
                <Label> _Tile_Thickness </Label>
        </Parameter>
        <Parameter Name="MH" DefaultValue="20">
                <Label> _Mortar_Height </Label>
        </Parameter>
        <Parameter Name="MW" DefaultValue="20">
                <Label> _Mortar_Width </Label>
        </Parameter>
        <HOffset>
                <Vector2D X="W+2*H+3*MW" Y="0" />
        </HOffset>
        <VOffset>
                <Vector2D X="0" Y="W+2*H+3*MH" />
        </VOffset>
        <Tile Angle="0" Width="W" Height="H" Thickness="TH">
                <TileOrigin>
                        <Vector2D X="0" Y="0" />
                </TileOrigin>
```

(1) Name of the pattern

(2) Size of the pattern block in the x direction, after which the pattern repeats

(3) Size of the pattern block in the y direction, after which the pattern repeats

The definition file uses the same symbols as the **Surface treatment** properties:



Tile pattern definitions

The predefined tile patterns that are available in the **Surface treatment** properties are stored in the following files:

File	Description
`TilePatternCatalog.xml`	• Contains the tile pattern definitions. • Located under the `\ProgramData\Trimble\Tekla Structures\<version>\environments` folder.
`TilePatternCatalog.dtd`	• Document Type Declaration (DTD) file that defines the elements allowed in the `TilePatternCatalog.xml` file. • Located in the same folder as the `TilePatternCatalog.xml` file.
Thumbnail images	• The images that appear in the **Pattern** section in the **Surface treatment** properties. • Located in the `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps` folder. • File names identify the pattern types. For example, `herringbone.bmp` illustrates the herringbone pattern type.

Tile pattern elements

The `TilePatternCatalog.xml` file can contain the following elements:

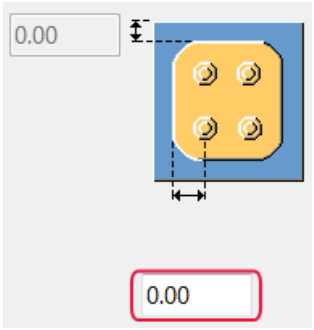
Element	Description
`TilePatternCatalog`	The container for tile patterns. Required.
`TilePattern`	Tile pattern element. Required. This element can contain the following elements listed in this table.
`HOffset`	Horizontal offset of the tile pattern. Required.
`VOffset`	Vertical offset of the tile pattern. Required.
`Tile`	The individual tiles used in a tile pattern. At least one required.
`Color`	Color of the tile or mortar, defined by the RGB values (0–255). Optional.
`Parameter`	Creates an attribute for any element in the `TilePattern`. Optional.
`Label`	The label that identifies a parameter in the **Surface treatment** properties. Optional.

Element	Description
`TileOrigin`	The origin of an individual tile, defined from the origin of the pattern. Optional.

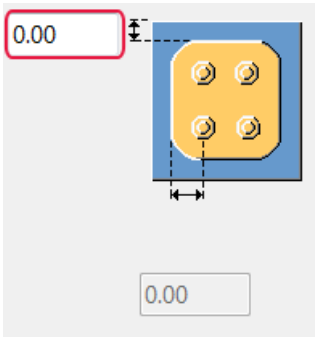
Create an unpainted area using the No paint area component

You can create an unpainted area between bolted steel parts using the **No paint area** component.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for **No paint area**.
3. On the **General** tab:
   a. Click the **Load bolt standards** button to display the available bolt standards, and select the relevant standards.
   b. Select the clearance location from the **Create for** list.
      - Define the hole tolerance.



      - Define the contact area offset.



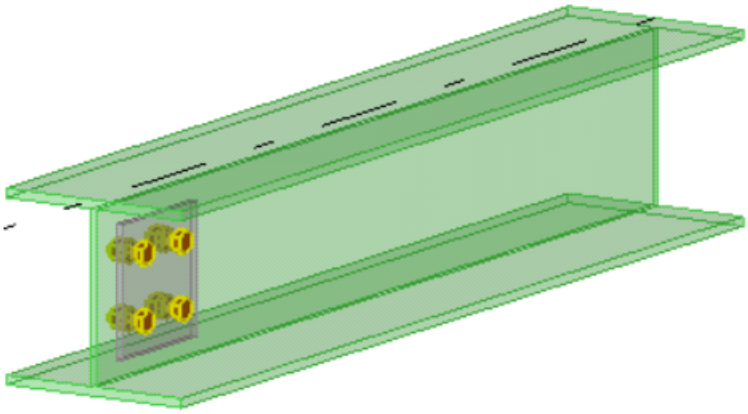
   c. In the **Allowable gap** box, enter the maximum distance that can exist between two plates so that the surface treatment can be created.
4. On the **Surfacing attributes** tab:
   a. Select one of the following **Surfacing attributes**.
      - `standard` surface treatment properties file

- A custom surface treatment properties file

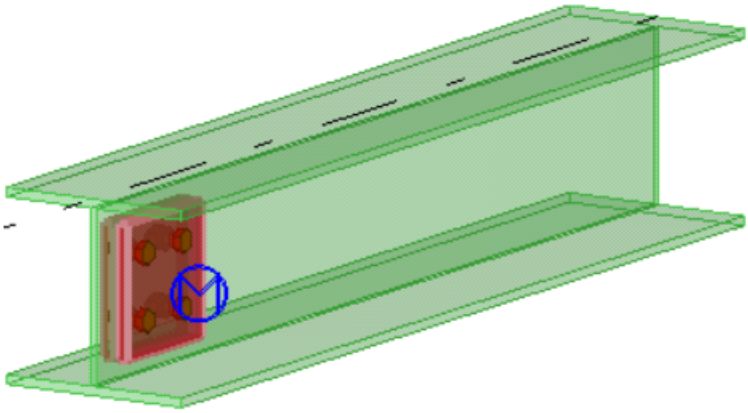
  You can create your own property files in the **Surface treatment** properties. The **Type** has to be **Steel finish** and the **Subtype** has to be **UP - Unpainted**.
- . . .

  Define the custom attributes and the position for the surface treatment.

5. Click **OK**.
6. Select a bolt group in the model.



The unpainted area is created between the bolted parts.



See also

Add surface treatment to parts (page 425)

Add surfaces to part faces and pour object faces

You can add surfaces to the faces of parts and pour objects in the model. You can add surfaces to any face geometry, for example curved faces. You can use

surfaces for calculating surface areas, such as formwork areas, or for defining face-specific concrete covers for rebar sets.

Each surface is added to a single flat or curved face. Surfaces do not extend over fillet edges between flat and curved faces.

Surfaces are associated with the objects they are attached to. Surfaces cannot exist as standalone objects. A surface can be associated either with a cast-in-place part or a pour object, but not with both. Surfaces associated with parts or pours can be shown in reports as belonging to those objects.

If you modify the geometry of the associated part, the surface will adapt to the changes. If you copy, move, or delete the associated part, the surface will follow. If a part and one or more of its surfaces are selected and copied or moved, all the surfaces of the part will follow.

If a surface is added to a pour object, it will not automatically adapt to changes that only affect the pour object, such as inserting pour breaks. The surface will not be copied if the parts that form the pour object are copied.

You can also copy and move surfaces separately from the objects they are attached to, but only within the same object type, i.e. from a part to another part, or from a pour object to another pour object. The source object face and the target object face need to be similar enough and facing in the same direction. The point picked when the source surface was created must also land on the target face.

Limitations:

- Surfaces do not recognize faces created by displaying objects with high accuracy, such as curved profile fillets.
- Surfaces do not have handles, so their geometry cannot be modified separately from the associated object.
- Surfaces are not shown in drawings.

Add a surface to a face

1. On the **Edit** tab, click **Surfaces** --> **Add surface to face**.
2. Depending on whether you want to create the surface on a part or on a pour object, use a part view or a pour view (page 471).

   To switch between part and pour views, click **Pour view** on the **Concrete** tab.
3. Select the part face or pour object face to which you want to add the surface.

Tekla Structures adds the surface using the **Surface** properties in the property pane.

If you modify the properties, Tekla Structures uses the new properties the next time you create an object of the same type.

If you try to add a surface to a part face or pour object face that already has a surface of the same type, Tekla Structures does not add the new, duplicate surface.

Modify surface properties

1. If the property pane is not open, double-click the surface to open the **Surface** properties.
2. Change the properties as needed.

   For example, you can define the surface type and whether you want the surface to be cut by the holes in the part or pour object.

   If you want to use the surface for defining a specific concrete cover thickness (page 588) for rebar sets at this part face or pour object face, set **Type** to **Concrete cover** and then enter the value in the **Concrete cover** box in the **Rebar set** section.
3. Click **Modify**.

Modify the adaptivity of reinforcement, surface treatment, or edge chamfers in parts

Reinforcement, surface treatment, and edge chamfers react to the modifications of the parent part they are attached to. For example, reinforcement, surface treatment, and edge chamfers automatically adapt to changes in part geometry and size. You can modify the adaptivity settings either for the entire model or for each model object separately. If you modify the adaptivity of individual model objects, these modifications override the default settings that you may have defined for the entire model.

The options are:

- **Default**: the **Default adaptivity** setting of the model is used for the model object

  The selected **Default adaptivity** option may vary between Tekla Structures environments.
- **Off**: adaptivity is not defined

  Changes in the parent part do not affect objects that are attached to the part.
- **Relative**: handles retain their relative distances to the nearest part faces in relation to the part's overall size

- **Fixed**: handles retain their absolute distances to the nearest part faces

Define default adaptivity settings

You can define default adaptivity settings that affect the entire model.

1. On the **File** menu, click **Settings** --> **Options**, and go to the **General** settings.
2. Under **Default adaptivity**, select one of the options.
3. Click **OK** to save the changes.

Modify the adaptivity of an individual model object

You can modify the adaptivity settings for each reinforcement or surface treatment separately. These modifications override the default settings that you may have defined for the entire model.

1. In the model, select the reinforcement (page 591) or surface treatment (page 425) whose adaptivity settings you want to change.
2. Right-click, select **Adaptivity**, and then select one of the options.

   Note that if you select **Default** for a rebar set modifier (page 557), the modifier uses the adaptivity setting of the rebar set (page 494) it belongs to, not the **Default adaptivity** setting of the model from **File** --> **Settings** --> **Options** --> **General**.

Display detailing of a part

In some cases it is useful to see in the model all the objects that are connected to a part, such as components, welds, fittings, reinforcement, and surfaces. You can then examine, for example, whether parts are welded correctly.

1. Select a part.
2. Click  **Display detailing** on the contextual toolbar.

   Alternatively, you can use **Quick Launch**.

Tekla Structures displays all bolts, welds, cuts, fittings, and other details belonging to the part, even if you had defined them as hidden in the display settings (page 676). For concrete parts, Tekla Structures displays also reinforcement, surface treatment, and surfaces.

See also

Add details to parts (page 362)

Adjust how model objects are displayed (page 671)

2.3 Work with assemblies

In Tekla Structures, assembly is a structure that consists of one or several parts or items and possibly other objects, such as bolts, welds and reinforcement. When a part or an item is modeled, Tekla Structures automatically creates an assembly for it. This means that also a loose part or an item has an assembly. When you model more parts in Tekla Structures, you may need to merge several parts into one assembly or join several assemblies together.

Building up assemblies

There are different methods on how to form assemblies in the model and achieve the desired assembly structure. The structure can be flat where the assemblies are on one level, or there can be sub-assemblies on several assembly levels in the assembly structure.

Try out the different ways of composing assembly structures to find out which ones suit your purposes. Note that the different methods to form the assemblies may affect your drawings and reports. We recommend you to test the different outputs early on in the modeling phase to ensure that all the relevant assembly information is included in the needed deliverables, such as production drawings.

If needed, you can later add parts and assemblies to existing assemblies, or otherwise modify the assembly structure (page 450) to produce the desired outcome.

Additionally, consider the direction how you form the assembly: it might be useful to build the assembly from bottom to top, so that you first create sub-assemblies and then form a higher-level assembly (page 441) that includes the sub-assemblies.

NOTE Concrete cast units (page 454) are technically considered as assemblies, and they have similar commands available as steel assemblies. In some cases, the assembly commands need to be used for cast units as well.

Hierarchical assemblies

A sub-assembly is an assembly that is added to another assembly to form a hierarchical nested assembly (page 441). A hierarchical assembly consists of the highest level (main) assembly, and one or more levels of sub-assemblies under it. Hierarchical assemblies support the manufacturing process of large assemblies, by enabling the combining of assemblies from smaller sub-assemblies into a larger one.

Note that the *assembly hierarchy* affects the drawing and report outputs. Report and drawing templates need to be made with similar hierarchy as the assembly hierarchy in the model in order to output the data from the correct assembly levels.

Assembly types and assembly hierarchy

Before you start to build up the assemblies in the model, consider the needed assembly structure, how it needs to be composed, and how is affects the drawings and reports. There are several methods how the assembly structure can be formed, depending on the needed structure.

Method	Assembly type	More information
Bolt or weld parts to an existing assembly as secondary parts.	Basic assembly	Use bolts and welds to create and connect assemblies and sub-assemblies (page 447)
Add parts to an existing assembly as secondary parts.	Basic assembly	Add parts to an existing assembly (page 450)
Bolt or weld assemblies to an existing assembly as sub-assemblies.	Nested assembly	Use bolts and welds to create and connect assemblies and sub-assemblies (page 447)
Add parts or assemblies to an existing assembly as sub-assemblies.	Nested assembly	Create nested assemblies (page 451)
Join existing assemblies together without adding any loose parts.	Nested assembly	Join existing assemblies (page 451)

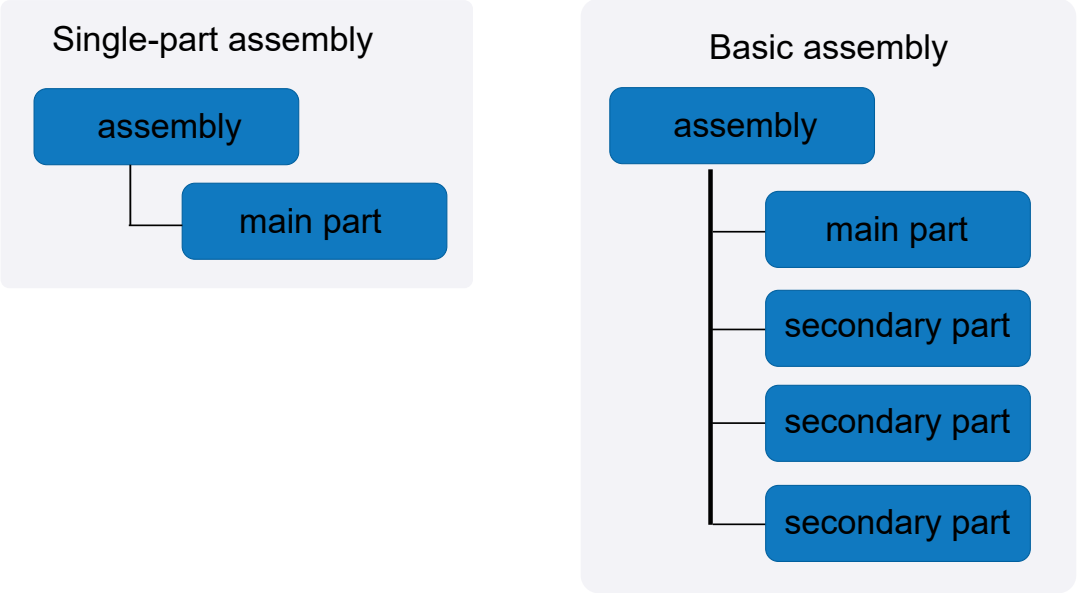
TIP To easily check the structure and hierarchy of an assembly, use the Assembly tree viewer tool from Tekla Warehouse.

Assembly types

Single-part assembly and basic assembly

A *single-part assembly* consists of one single part. When a part is modeled, Tekla Structures automatically creates an assembly for it.

A *basic assembly* consists of a main part with secondary parts, and the assembly is on one assembly level.



Tekla Structures creates basic assemblies when you:

- Use workshop bolts or workshop welds to manually connect parts together.
- Apply a component that automatically creates workshop welds or workshop bolts.
- Use the **Add to assembly** command to add parts to an existing assembly as secondary parts.

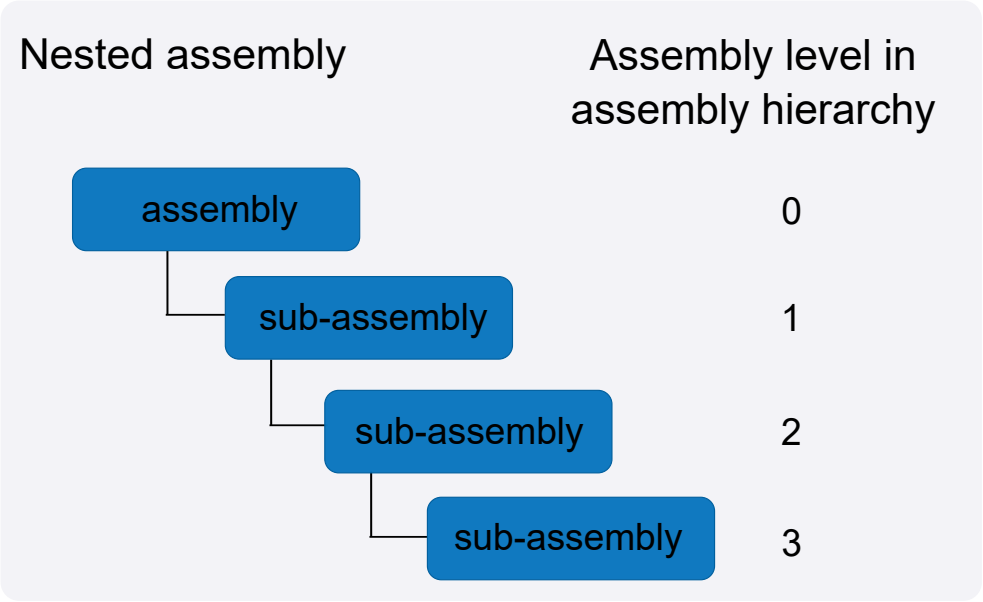
The main part in an assembly has other parts welded or bolted to it, or added with the **Add to assembly** command. By default, the main part is not welded or bolted to any other parts. You can change the main part if needed.

Nested assembly

A *nested assembly* consists of a parent assembly and sub-assemblies which are on the lower assembly levels in the assembly hierarchy.

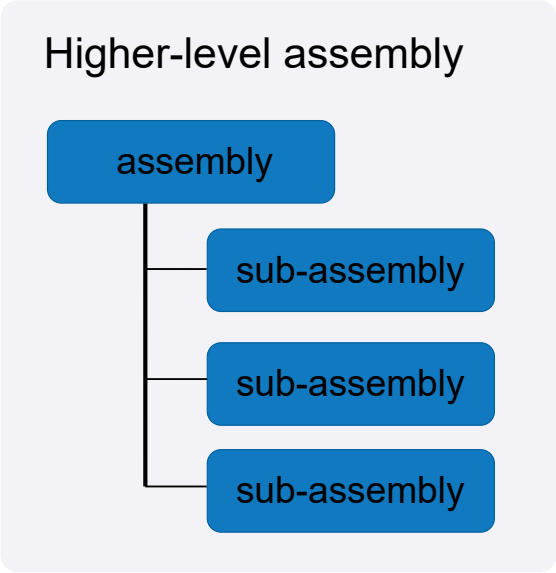
Tekla Structures creates a nested assembly when you:

- Use bolts or welds to connect assemblies to an existing assembly as sub-assemblies.
- Use the **Add as sub-assembly** command to add parts or assemblies to an existing assembly as sub-assemblies.



Another type of a nested assembly is created when you join existing assemblies as sub-assemblies. The sub-assemblies are on the same assembly level.

Use the **Join as sub-assemblies** command to join existing assemblies together. You can add assemblies as sub-assemblies, but not any loose parts (parts or items).



NOTE Sub-assemblies in a nested assembly retain their own assembly information and main part. You can define the properties separately for the sub-assemblies and the parent assembly.

To select assemblies in different assembly hierarchy levels, place the mouse pointer on any part in the assembly, press **Shift** and scroll up or down with the middle mouse button.

How to check the assembly hierarchy in a nested assembly

You can work on any level of a *nested assembly*, from single parts and bolts, through the sub-assemblies, up to the highest level of the nested assembly.

The active selection switch defines on which level you start and toward which direction you move in the assembly hierarchy. The status bar shows the level of the assembly hierarchy you are on.



The assembly on the highest hierarchy level gets the number 0, and the assemblies or objects inside an assembly lower in the hierarchy get the number 1, 2, and so on. You can continue to level 9.

- When the  **Select assemblies** selection switch is active:

  Start from the assemblies on the highest level, move to their sub-assemblies, and finally select single parts, bolts, and so on.

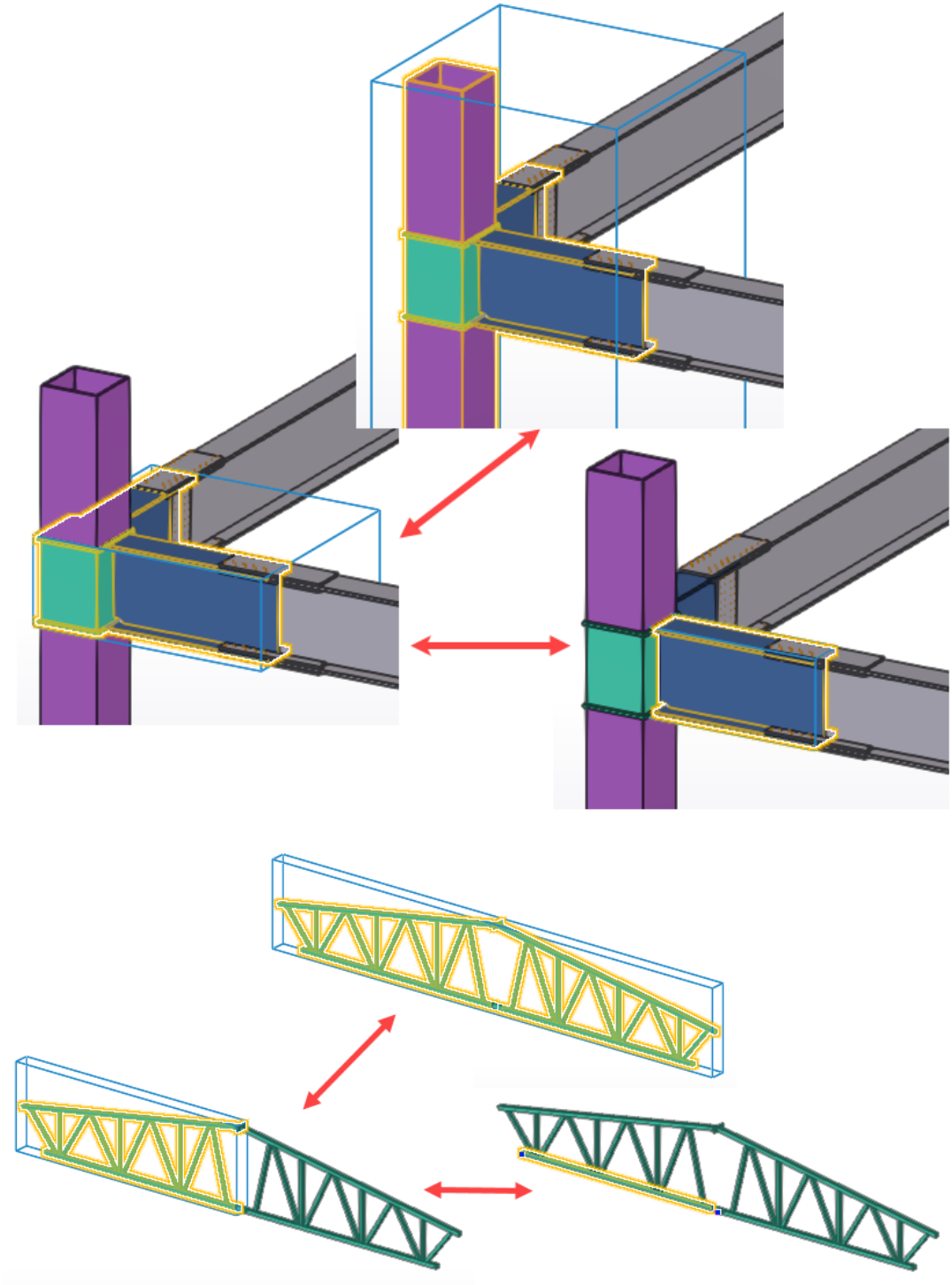
  To shift to a specific level in an assembly, place the mouse pointer on any part in the assembly, press **Shift** and scroll up with the middle mouse button to highlight the objects on the lower levels in assembly hierarchy, level by level.

- When the  **Select objects in assemblies** selection switch is active:

  Start from single objects and move to higher levels of nested assemblies.

  To shift to a specific level in an assembly, place the mouse pointer on any part in the assembly, press **Shift** and scroll up with the middle mouse button to highlight the objects on the higher levels in assembly hierarchy.

The blue highlighting box indicates the assembly that you can select.



The assembly hierarchy in nested assemblies affects drawings and reports. Report and drawing templates need to be made with similar hierarchy as the assembly hierarchy in the model to output the data from the correct assembly levels.

You can create separate drawings and reports of the sub-assemblies and the nested assembly, and still produce dimensions, marks, fabrication information, and so on for all the assembly levels.

For more information how to work with the assembly hierarchy, watch the Working with multi-level assemblies video.

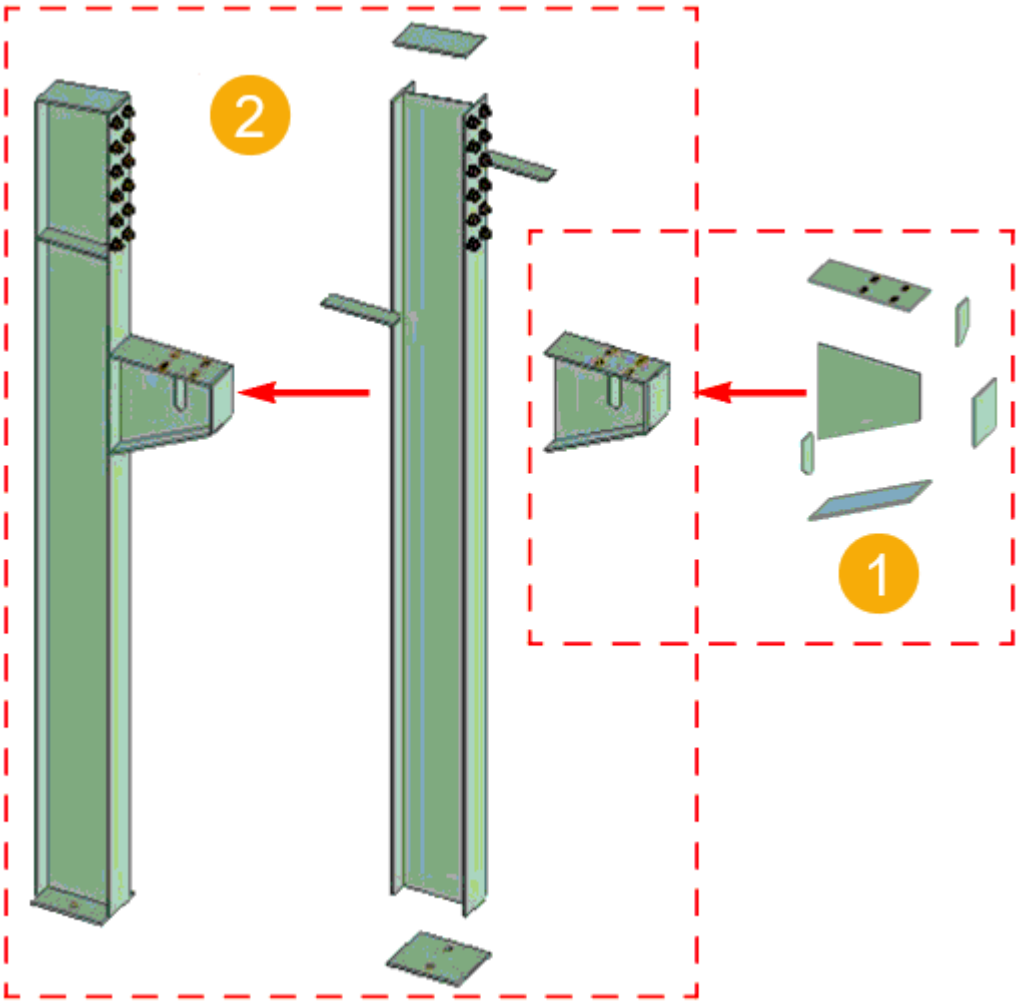
TIP To select an entire assembly or cast unit, instead of using the  **Select assemblies** selection switch, hold down the **Alt** key and select one of the parts in the assembly or cast unit.

The number of objects in the assembly or in the cast unit is shown at the bottom right corner of the status bar.

Assembly examples

Column corbel

A column corbel is fabricated in one workshop, and then attached to the column in another workshop. Model the corbel as a sub-assembly of the column. Then create an assembly drawing for each workshop: one assembly drawing showing how the corbel is welded together, another assembly drawing showing how the corbel and the other parts are welded to the column.



(1) Drawing 1, Workshop 1

(2) Drawing 2, Workshop 2

Complex truss

Model the halves of a complex truss as assemblies. Create assembly drawings for the workshop to fabricate the truss halves. Then create another assembly drawing showing how the halves should be joined on site.

Built-up profile

In a frame of built-up columns and beams, each built-up profile can be a sub-assembly. You can create an assembly drawing showing the entire frame, and separate drawings showing how the columns and beams are constructed.

Use bolts and welds to create and connect assemblies and sub-assemblies

You can use bolt and weld properties in the property pane to create assemblies, or to connect sub-assemblies to an existing assembly.

Use bolts to create and connect assemblies

Assemblies and their main parts are automatically defined when you create workshop bolts. Additionally, you can connect more parts to assemblies by using bolts, or create nested assemblies by connecting sub-assemblies to an existing assembly.

To control how Tekla Structures creates the assembly, use the **Connect as** and **Bolt type** lists in the **Bolt** properties in the property pane

The order in which you select parts when creating the bolt determines the main and secondary parts of the assembly, or the assembly hierarchy.

Assembly type	Connect as	Bolt type	Result
Basic assembly	**As secondary part**	**Workshop**	Basic assembly with the part you are bolting as a secondary part. The first part you select usually becomes the main part in the assembly.
Nested assembly	**As sub-assembly**	**Workshop** or **Site**	Nested assembly with the assembly you are bolting as a sub-assembly. The first part you select determines the assembly to which you are bolting.
	As secondary part	**Site**	No assembly created.

Bolt secondary parts to an assembly

1. Open the **Bolt** properties in the property pane.
2. In the **Connect as** list, select **As secondary part**.
3. Select the main part to which the secondary parts will be bolted.
4. Select the secondary parts.

5. Click the middle mouse button to finish selecting parts.
6. Pick a point to indicate the bolt group origin.
7. Pick a second point to indicate the bolt group x direction.

Bolt sub-assemblies to an assembly

1. Open the **Bolt** properties in the property pane.
2. In the **Connect as** list, select **As sub-assembly**.
3. Select a part in the assembly to bolt to.
4. Select a part in the sub-assembly to be bolted.
5. Click the middle mouse button to finish selecting parts.
6. Pick a point to indicate the bolt group origin.
7. Pick a second point to indicate the bolt group x direction.

Use welds to create and connect assemblies

Assemblies and their main parts are automatically defined when you create workshop welds. Additionally, you can connect more parts to assemblies by using welds, or create nested assemblies by connecting sub-assemblies to an existing assembly.

To control how Tekla Structures creates the assembly, use the **Connect as** and **Workshop/Site** lists in the **Weld** properties in the property pane

The order in which you select parts when creating the weld determines the main and secondary parts of the assembly, or the assembly hierarchy. The first part you select becomes the main part of the assembly. Tekla Structures dimensions secondary parts relative to the main part in assembly drawings.

Assembly type	Connect as	Bolt type	Result
Basic assembly	**As secondary part**	**Workshop**:	Basic assembly with the part you are welding as a secondary part. The first part you select usually becomes the main part in the assembly.
Nested assembly	**As sub-assembly**	**Workshop**: or **Site**: 	Nested assembly with the assembly you are welding

Assembly type	Connect as	Bolt type	Result
			as a sub-assembly. The first part you select determines the assembly to which you are welding.
	As secondary part	**Site**:	No assembly created.

Weld secondary parts to an assembly

1. Open the **Weld** properties in the property pane.
2. In the **Connect as** list, select **As secondary part**.
3. Select the part to weld to.
4. Select the parts to be welded.

Weld sub-assemblies to an assembly

1. Open the **Weld** properties in the property pane.
2. In the **Connect as** list, select **As sub-assembly**.
3. Select a part in the assembly to weld to.
4. Select a part in the sub-assembly to be welded.

Modify the assembly structure

You can modify the assembly structure by adding more parts or assemblies, or by changing the main part of an assembly, for example.

Concrete cast units (page 454) are technically considered as assemblies, and they have similar command available as steel assemblies. In some cases, the assembly commands need to be used for cast units as well.

NOTE The active selection switch controls whether you can select assemblies or parts when you use the different assembly commands.

Add parts to an existing assembly

You can add secondary parts to a basic assembly, or to any level of a nested assembly.

1. Ensure that the **Select objects in assemblies**  or **Select objects in components**  selection switch is active.
2. Select the parts you want to add.
3. Right-click and select **Assembly** --> **Add to assembly**.
4. Select the assembly to which you want to add the parts.

Create nested assemblies

You can add parts and assemblies as sub-assemblies in another assembly on a lower level in the assembly hierarchy.

1. Ensure that the **Select assemblies**  selection switch is active.
2. Select the parts or assemblies you want to add to another assembly.

   They will become sub-assemblies in the nested assembly.
3. Right-click and select **Assembly** --> **Add as sub-assembly**.
4. Select the assembly to which you want to add the parts or assemblies.

Alternatively, you can use the **Assembly** --> **Add as sub-assembly** command on the ribbon.

Join existing assemblies

You can join existing assemblies as sub-assemblies without adding any loose parts. The joined sub-assemblies are on the same assembly level. For example, you can model two halves of a complex truss as assemblies, and then join the halves together to form a higher-level assembly that contains the halves as sub-assemblies. The assembly with the largest volume becomes the main assembly.

1. Ensure that the **Select assemblies**  selection switch is active.
2. Select the assemblies you want to join.
3. Right-click and select **Assembly** --> **Join as sub-assemblies**.

Alternatively, you can use the **Assembly** --> **Join as sub-assemblies** command on the ribbon.

NOTE This command adds a new assembly hierarchy level on top, and the resulting new assembly contains only sub-assemblies.

If needed, you can change the main assembly. Use the **Assembly** --> **Set as main object** command on the ribbon.

Create a sub-assembly of assembly parts

You can create a sub-assembly of parts that are already in an assembly.

1. Ensure that the **Select objects in assemblies**  or **Select objects in components**  selection switch is active.
2. Select the parts you want to include in the sub-assembly.

   Note that you need to select several parts to create the sub-assembly.
3. Right-click and select **Make into sub-assembly**.

Check and highlight objects in an assembly

Use the **Inquire** tool to check which objects belong to a particular assembly, cast unit, or rebar assembly.

1. On the ribbon, click the down arrow next to , and then select **Assembly objects** to inquire the assembly objects.
2. Select an assembly, cast unit, or rebar assembly. Ensure that you are on the intended assembly hierarchy level.

   Tekla Structures highlights the other objects that belong to the same assembly, cast unit, or rebar assembly. The following colors are used:

Object type	Highlight color
Steel part - main part Main reinforcement object in rebar assembly	orange
Steel part - secondary part Secondary reinforcement object in rebar assembly	yellow
Concrete - main part	magenta
Concrete - secondary part	cyan
Reinforcement in cast unit	blue

Alternatively, to quickly show the assembly, right-click an object in the assembly and select **Assembly** --> **Show assembly**. Tekla Structures draws a blue highlighting box around the assembly.

Change the main part in an assembly

The *main part* in an assembly has other parts welded or bolted to it. By default, the main part is not welded or bolted to any other parts. You can change the main part in an assembly.

1. If needed, check what is currently the main part of the assembly.

   Use the **Inquire** tool to check which objects belong to a particular assembly.

   To easily check the structure and hierarchy of an assembly, use the Assembly tree viewer tool from Tekla Warehouse.

2. Ensure that the **Select objects in assemblies**  or **Select objects in components**  selection switch is active.
3. Select the new main part.
4. Right-click and select **Assembly** --> **Set as new main part of assembly**.

   Tekla Structures changes the main part.

Alternatively, you can use the **Assembly** --> **Set as main object** command on the ribbon.

Use this command also to change the main part in a concrete cast unit.

Change the main assembly

If you have joined existing assemblies together, the assembly with the largest volume has become the main assembly. You can change the main assembly within the sub-assemblies.

1. Ensure that the **Select assemblies**  selection switch is active.
2. Place the mouse pointer on any part in the assembly, press **Shift** and scroll with the middle mouse button in the assembly hierarchy to see the sub-assemblies.
3. Select the new main assembly.
4. Right-click and select **Set as new main sub-assembly**.

   Tekla Structures changes the main sub-assembly. When the assembly is inquired, Tekla Structures highlights the main sub-assembly's main part in orange.

Alternatively, you can use the **Assembly** --> **Set as main object** command on the ribbon.

Remove objects from assemblies

If you want to disassemble your assembly, you can remove objects from an assembly, or ungroup the objects belonging to an assembly.

Remove a part or a sub-assembly from an assembly

1. Select the part or sub-assembly you want to remove.
2. Right-click and select **Assembly** --> **Remove from assembly**.
3. To remove another part or sub-assembly, restart the command.

Explode an assembly or a sub-assembly

1. Ensure that the  **Select assemblies** selection switch is active.
2. Select the assembly or sub-assembly you want to explode.
3. Do one of the following:
   - To explode the entire assembly, right-click and select **Assembly** --> **Explode**.
   - To only explode the sub-assembly, right-click and select **Assembly** --> **Explode sub-assembly**.

When you explode a nested assembly, always start from the highest assembly hierarchy level. Tekla Structures breaks the assembly hierarchy level by level and ungroups the objects that belong to the assembly. You need to use the **Explode** command several times to break a nested assembly back to single parts.

You can also use the **Explode sub-assembly** to explode sub-assemblies to single parts without breaking the entire assembly hierarchy.

2.4 Create cast units

In Tekla Structures, each modeled concrete part is considered as its own *cast unit*. When you model concrete parts in Tekla Structures, you need to create cast units when you have several concrete parts which should be cast as one monolithic structure. For example, a single cast unit could consist of a column with corbels. The cast unit can then have the production drawings, quantity reports, and exports showing the content of the cast unit.

Concrete cast units are technically considered as assemblies, and they have similar commands available as steel assemblies (page 440). In some cases, the assembly commands need to be used for cast units as well.

Create a cast unit

You need to specify which parts form the cast unit. Cast units can include reinforcement, as well as concrete parts.

1. On the **Concrete** tab, click **Cast unit** --> **Create cast unit**.

2. Select the objects you want to include in the cast unit.
3. Click the middle mouse button to create the cast unit.

Check and highlight objects in a cast unit

Use the **Inquire** tool to check which objects belong to a particular cast unit or assembly.

1. On the ribbon, click the down arrow next to , and then select **Assembly objects**.
2. Select a part that belongs to a cast unit or an assembly.

   Tekla Structures highlights the other parts that belong to the same cast unit or assembly. The following colors are used:

Object type	Highlight color
Concrete - main part	magenta
Concrete - secondary part	cyan
Reinforcement	blue
Steel part - main part	orange
Steel part - secondary part	yellow

Change the cast unit main part

The *main part* in a concrete cast unit is the one with the largest volume of concrete. You can change the main part in a cast unit.

1. If needed, check what is currently the main part of the cast unit.

   Use the **Inquire** tool to check which objects belong to a particular cast unit.

2. Ensure that the **Select objects in assemblies**  selection switch is active.
3. Select the new main part.
4. Right-click and select **Set as new main part of assembly**.

Define the cast unit type of a part

You must define the cast unit type of concrete parts. Tekla Structures checks the cast unit type of the main part each time you create or modify a cast unit. It is important to use the correct cast unit type, because some functionalities, for example numbering, are based on the cast unit type.

1. Double-click a concrete part to open the part properties in the property pane.
2. Go to the **Cast unit** section.
3. In the **Cast unit** list, select one of the following:
   - **Cast in place**

     Cast units that are constructed fully in their final location.
   - **Precast**

     Cast units that are built at another location and transported to their final location for placement in the full structure.
4. Click **Modify** to save the changes.

Some best practices when you are working with cast units

- Always check that all the concrete parts and related accessories are connected to the cast unit.
- Cast units always have a main part. The main part in a cast unit is usually the part with the largest volume of concrete. You can change the main part. Always ensure that the main part is the intended one as the cast unit gets many of its properties from the main part. Keep the possible sub-assembly hierarchy clear.
- Cast units get their position numbers (page 752) according to the numbering settings. Identical cast units have the same position number and they are included in the same drawing. In addition to the position numbers, you can assign other identifiers to separate the cast units with the same drawing and position number.
- The cast unit drawings contain all the hierarchical content included in the cast unit. Always check that all of the concrete parts, cuts, reinforcement, and surface treatment are connected to the correct parts and that embeds are added as sub-assemblies to the cast unit. This ensures that all the objects are included in the drawings.
- To indicate the casting direction (page 458) of a precast concrete part, you can define which part face you want to face upwards in the casting form. Therefore, this face will be the front face view in the fabrication drawing. Casting direction affects the numbering of parts. By modeling beams and interior walls from left to right, bottom to top, the default front face is towards the viewer, and any fixed part marks are oriented upright. When using wall layout for exterior walls, model in the clockwise direction.
- If you copy fully detailed cast units in a model, use the **Copy special --> Copy all content to another object** command. With this command, all objects in a cast unit are copied to other similar cast units without individually selecting each object to copy. This means that, for example,

secondary parts, sub-assemblies, components, and cuts and fittings are copied with the cast unit.

Additionally, you can

- inquire (page 721) the cast unit's general properties, such as volume and weight which are automatically calculated
- inquire the cast unit's own properties, such as position number
- inquire the cast unit's main part properties
- list all parts, rebars and sub-assemblies of the cast unit.

Add objects to cast units

You can use different methods to add objects to cast units. The available methods depend on the objects' material and on the hierarchy you want to create in the cast unit.

- The default way of working is to use the **Add to cast unit** command. Using this command you can add concrete parts that need to be in the same casting unit. The parts retain their geometry and you can use them as input when modeling components, shown in drawings.
- Use the **Add as sub-assembly** command to add embeds. Embeds are always added as sub-assemblies on a lower level in the cast unit hierarchy. Sub-assemblies can consist of one or multiple parts. In both cases the parts have to be added to a cast unit as sub-assemblies. Sub-assemblies in a cast unit retain their own assembly information and their main part. If in a precast cast unit the precast element consists of numerous shells or layers, the secondary cast units need to be added as sub-assemblies to the main cast unit.
- Use the **Attach to part** command to attach one or more concrete parts to another part in the cast unit. Additionally, you can use the command to merge parts if you need them to be reported or drawn as one part.

To	Do this	Available for
Add an object as a secondary part	1. On the **Concrete** tab, click **Cast unit** --> **Add to cast unit**. 2. Select the object you want to add. 3. Select an object in the cast unit.	Concrete, timber, miscellaneous materials
Add an object as a sub-assembly	1. If you are adding a custom part, ensure that the **Select components (custom objects)**  selection switch is active.	Steel, concrete, timber, miscellaneous materials

To	Do this	Available for
	2. On the **Steel** tab, click **Assembly** --> **Add as sub-assembly**. 3. Select the object you want to add. 4. Select the cast unit to which you want to add the object.	

Remove objects from cast units

If you want to disassemble your cast unit, you can remove objects from a cast unit, or ungroup the objects belonging to a cast unit.

Remove objects from a cast unit

1. On the **Concrete** tab, click **Cast unit** --> **Remove from cast unit**.
2. Select the object you want to remove from the cast unit.

   You need to restart the command to remove another object.

Explode a cast unit

You can ungroup the objects belonging to a cast unit.

1. On the **Concrete** tab, click **Cast unit** --> **Explode**.
2. Select an object in the cast unit you want to explode.

Casting direction

To indicate the casting direction of a concrete part, you can define which part face you want to face upwards in the casting form. The top-in-form face is displayed in the front view of a drawing.

Casting direction for concrete and non-concrete parts

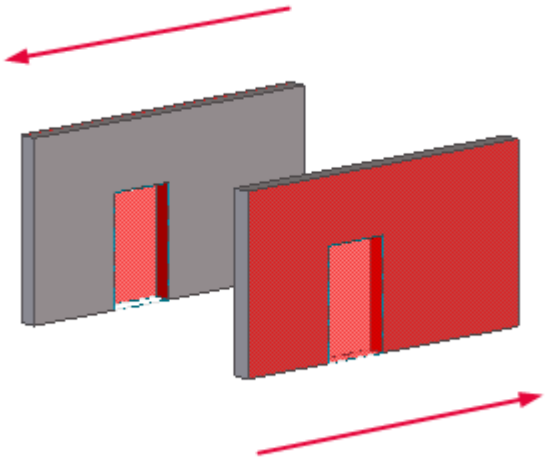
To enable this feature for non-concrete parts as well, and to indicate in the model the part face that is shown in the drawing main (front) view, use the advanced option XS_SET_FIXEDMAINVIEW_UDA_TO_AFFECT_NUMBERING.

The casting direction affects the numbering of parts. If you define the casting direction for parts that differ only by their modeling direction, they get different position numbers. This is because the modeling direction affects the top-in-form face of the parts. By default, the casting direction of the parts is undefined, which means the modeling direction does not affect numbering.

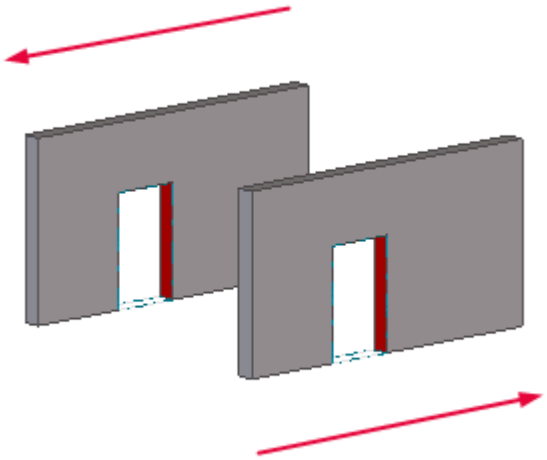
NOTE In drawings, use the **Fixed** coordinate system to show the top-in-form face in the front view.

Example

In the following example, each cast unit gets a **different** position number, because the top-in-form setting and the orientation of the panels is different. The red arrow indicates the modeling direction.



In the following example, the cast units get the **same** position number, because their top-in-form setting has not been defined. The red arrow indicates the modeling direction.



Define the casting direction of a part

You can define the casting direction for concrete parts.

1. Set the rendering of parts to **Rendered** by doing one of the following:
   - On the **View** tab, click **Rendering** --> **Parts rendered**.
   - Press **Ctrl + 4**.
2. On the **Concrete** tab, click **Cast unit** --> **Set top-in-form face**.
3. Move the mouse pointer over the selected concrete part.
4. Select the part face that will face upwards in the form.

TIP Alternatively, you can do this in the user-defined attributes of the part.

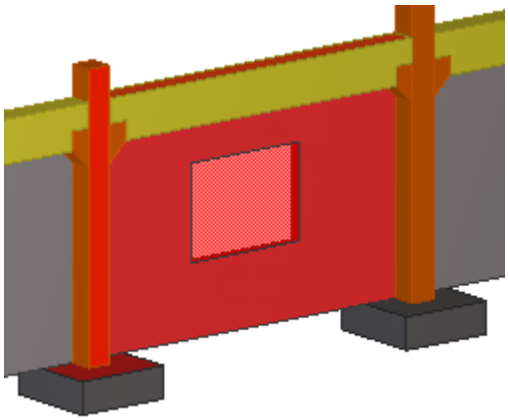
- Concrete parts: Select an option for the **Top in form face** user-defined attribute.
- Non-concrete parts: With XS_SET_FIXEDMAINVIEW_UDA_TO_AFFECT_NUMBERING set to `STEEL`, `TIMBER`, and/or `MISC`, select an option for the **Fixed drawing main view** user-defined attribute.

Show the top-in-form face

You can display the top-in-form face of a concrete part in a model view.

1. On the **Concrete** tab, click **Cast unit** --> **Show top-in-form face**.
2. Click the concrete part whose top-in-form face you want to show.

   Tekla Structures highlights the top-in-form face in red:



TIP To hide the top-in-form face again, right-click the view and select **Update window**.

Note that if you have not specifically set the top-in-form value, the **Show top-in-form face** command highlights the default face, which is the front face of the part.

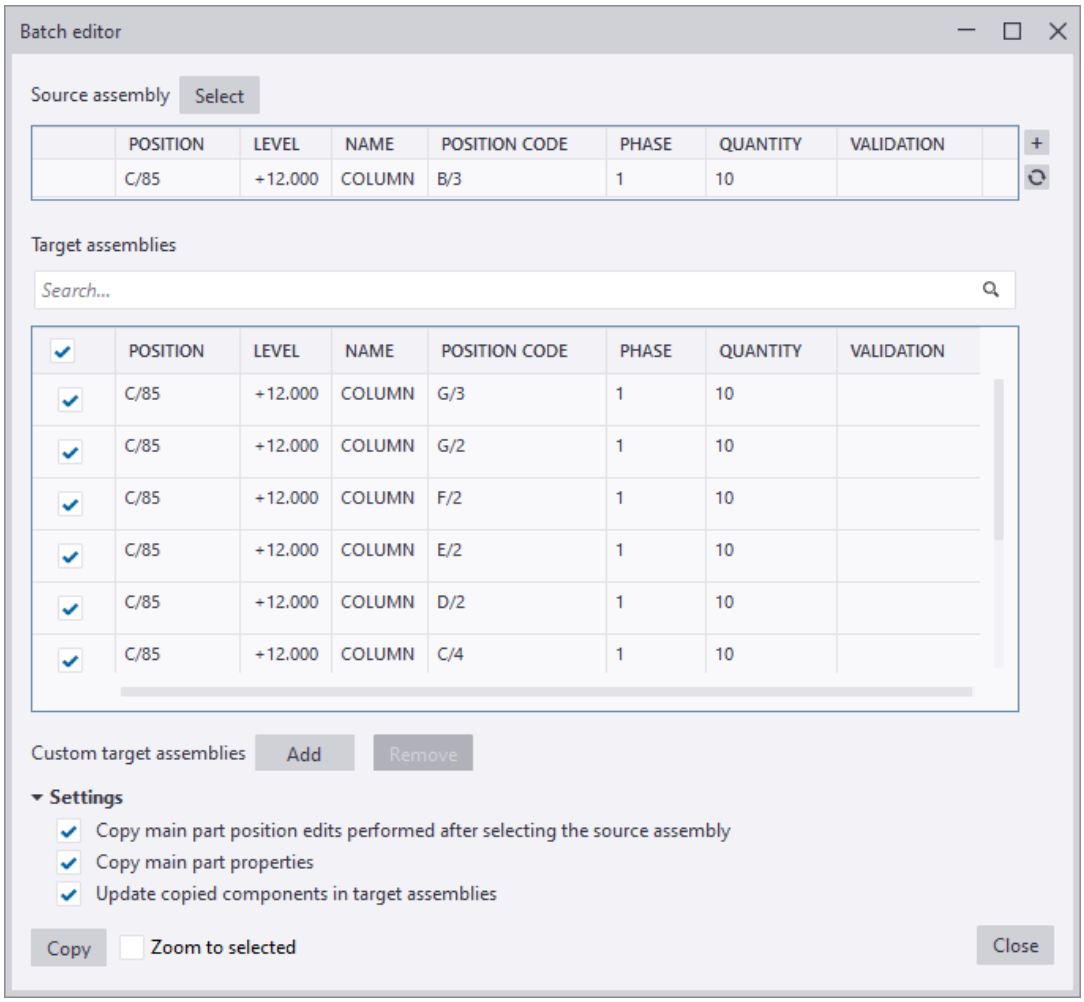
2.5 Batch editing of assemblies or cast units

Using the **Batch editor** tool you can quickly copy modifications done to an assembly or a cast unit to other, identical assemblies or cast units that have the same position number, or to very similar but differently numbered assemblies or cast units.

Batch editor reduces the amount of repetitive work in situations where you need to edit very similar assemblies or cast units by modifying the object geometry or by changing the part properties.

Unlike the **Copy special** --> **To another object** or **Copy special** --> **All content to another object** commands which replace the duplicate objects in the target assemblies or cast units, **Batch editor** detects matching objects in the targets and modifies their geometry and properties without replacing the objects.

This is important especially in the late stages of a project when the detailing of all objects is done and the documentation has been created or issued. For example, replacing of objects and components might damage not only the target but nearby assemblies or cast units which would not require any modifications, and the changes would have a negative impact on the drawings that are ready for issuing.



With **Batch editor** you can

- Select a source assembly or cast unit in the model, and based on the source, generate a list of target assemblies or cast units to which you can copy the edits done in the source. **Batch editor** only detects assemblies or cast units which have the same position number as the source.

  If needed, you can manually add very similar but differently numbered assemblies or cast units to the list of targets.

- Use the list to copy all edits done in the source assembly or cast unit to the specified target assemblies or cast units in one go.

  Batch editor attempts to avoid replacing of modified objects. **Batch editor** detects matching objects in the targets assemblies or cast units and modifies their geometry and properties.

  Both the main and secondary part geometry and property edits, as well as modifications in component properties can be copied. Note that to

copy the main part geometry edits, the **Copy main part position edits performed after selecting the source assembly** option needs to be **on**.

If there are secondary parts in the target assemblies or cast units which do not exist in the source anymore, **Batch editor** deletes the excess secondary parts in the targets assemblies or cast units when you copy the edits.

- Use the list of target assemblies or cast units for
  - viewing the differences between the source and the target assemblies or cast units before copying the edits.
  - browsing the list of target assemblies or cast units and fixing possible errors before running numbering.

  Batch editor uses the **Validation** attribute to report detected differences in the number of objects between the selected source and the listed target assemblies or cast units, and to compare the assembly or cast unit weight and volume. The **Validation** attribute also reports any missing or additional objects in the target assemblies or cast units.

Limitations and recommendations in using Batch editor

- **Batch editor** works only on the assembly or on the cast unit level.

  Assembly or cast unit secondary parts cannot be selected as a source or target. For example, you cannot copy edits from one part to another, or from one sub-assembly to another.
- **Batch editor** does not copy the following:
  - values of unique user-defined attributes
  - sub-assembly properties
  - assembly numbering information
- **Batch editor** does not support the following object types:
  - Pour units and pour objects
  - Load object types in analysis and design
- We recommend that you do not use **Batch editor** if the source assembly or cast unit has been split.
- We recommend that you do not use **Batch editor** with mirrored assemblies or cast units.

  Rebar groups and components are not always adapted correctly to the main part. This applies especially to cases where new objects are added to the mirrored target assemblies or cast units.

- We recommend that you do not use **Batch editor** with assemblies or cast units that look similar but are modeled using different methods.

  This applies to, for example, assemblies or cast units that have different orientation of secondary object, or assemblies or cast units that have been modeled using different object types as the main part.
- Rebar groups and some macros do not adapt to the geometry of the main part with different dimensions or different shape.
- The main part geometry edits that are copied to rotated or mirrored assemblies or cast units may not work as expected.
- **Batch editor** overrides the overall dimensions of the parts that the **Wall layout** tool creates. Therefore, we recommend to use **Batch editor** for copying edits to precast walls that have identical dimensions.

Edit similar assemblies or cast units with Batch editor

1. On the **Edit** tab, click **Batch editor**.

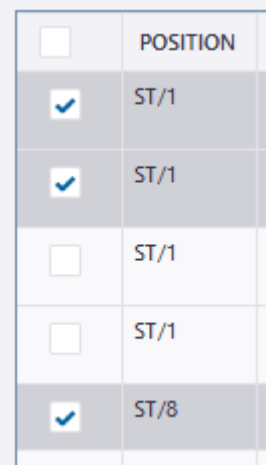
   The **Batch editor** dialog opens.
2. Click the **Select** button and select a source assembly or a cast unit in the model.

   Batch editor finds all assemblies or cast units with the same position number. The assemblies or cast units are listed in the **Target assemblies** list.
3. To add target assemblies or cast units manually, select them in the model and click the **Add** button.

   To remove target assemblies or cast units from the list, select them in the list and click the **Remove** button.

   Note that if you delete the source in the model, the source list becomes empty. Similarly, if you delete the targets in the model, the list of targets becomes empty.
4. In the **Target assemblies** list, use the checkboxes to select the assemblies or cast units to which you want to copy the modifications from the source assembly or cast unit.

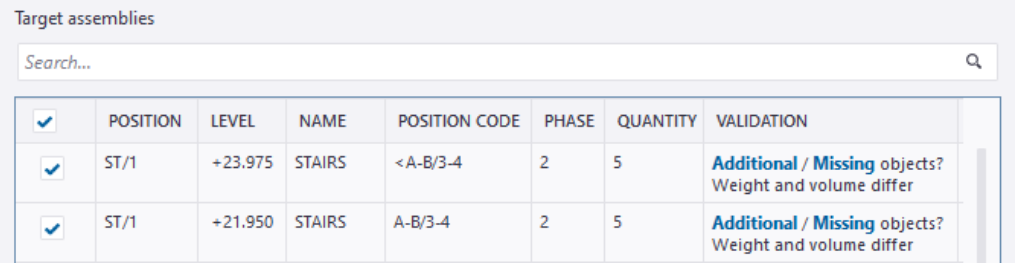
To select several check boxes in one go, select the targets, and hold down the **Shift** key and click the left mouse button.



5. To view the differences between the source and the target assemblies or cast units, or to fix errors, use the **Validation** attribute in the **Target assemblies** list.

   Batch editor uses the **Validation** attribute to report detected differences in the number of objects between the selected source and the listed target assemblies or cast units, and to compare the assembly or cast unit weight and volume. The **Validation** attribute also reports any missing or additional objects in the target assemblies or cast units.

   You can check the missing and additional objects in the model. Click the **Additional** or **Missing** objects link to check the highlighted the objects in the model.



6. To copy the modification from the source assembly or the cast unit to the targets, click the **Copy** button.

   Batch editor analyzes the detected differences between the selected source and the targets, and makes all the selected target assemblies or cast units identical to the source.

7. To review the copying results in the model, select the target objects in the list.

   Select the **Zoom to selected** checkbox to automatically zoom to the object selected in the list.

8. Number the modified assemblies or cast units to validate that all modifications have been copied properly.

Settings in Batch editor

Use the **Settings** to define how the edits are copied.

- **Copy main part position edits performed after selecting the source assembly**

  When the option is **on**, **Batch editor** copies the geometrical modifications done to the assembly or cast unit main part, such as dragging of end point handles of the main part, or changing the main part location with the **Move** or **Move special** commands.

  NOTE **Batch editor** recognizes only the edits that you have done after selecting the source assembly or cast unit, and switching the option on. If you first modify the main part geometry and only then switch the option on, the main part edits are not copied.

  If the option is **off**, only the secondary part and component edits are copied.

- **Copy main part properties**

  When the option is **on**, **Batch editor** copies the assembly or cast unit main part property modifications to the selected targets. Use this option, for example, when you want to copy the edits to targets with different main part profile dimensions.

  Note that user-defined attributes are copied but unique user-defined attributes are not copied.

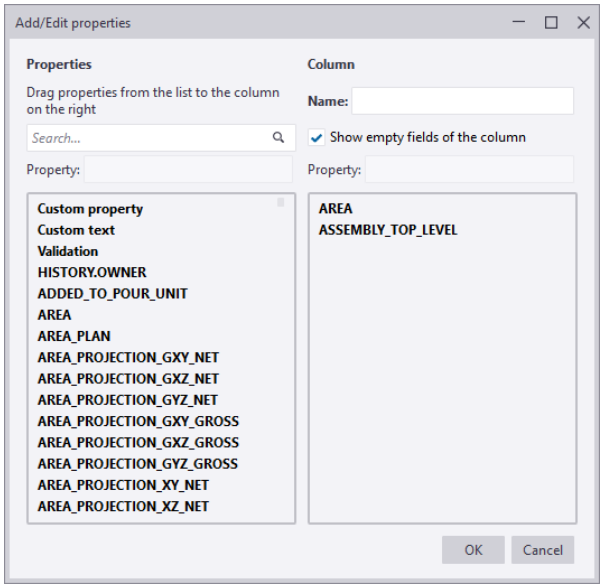
- **Update copied components in target assemblies**

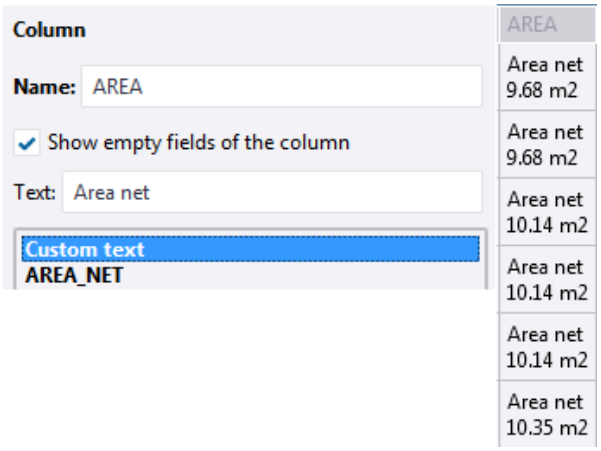
  When the option is **on**, **Batch editor** modifies the components in the target assemblies or cast units according to the properties set in the component properties dialog. All manual changes, such as modifying the geometry of component objects, are lost.

  If the option is **off**, manual changes in the component, for example, modifying the geometry of component objects or the component object properties, are copied to the target assemblies or cast units.

Work with property columns in Batch editor

You can organize the list of target assemblies or cast units and the property columns in **Batch editor**. Add, edit, or remove property columns to show the needed properties about the target assemblies or cast units.

To	Do this
Add more property columns in the **Target assemblies** list	1. Click the + button in the upper right corner of the **Batch editor**. Alternatively, you can right-click the property column header and select **Edit**. 2. In the **Add/Edit properties** dialog, do the following: • Select the required property from the list on the left and drag it to the list on the right. Use the **Search** box for searching properties. You can add several properties to the same column.  • If you want to add a custom text in a column cell, select **Custom text** and type the required text in the displayed **Text** box.

To	Do this
	Then press **Enter** to add the custom text in the list on the right.  • To show the property cells even though there is no value, select the **Show empty fields of the column** option. 3. Enter the name for the property column and click **OK**. The new property columns are added to the **Target assemblies** view.
Change the order of the property columns	Drag the property column header to a new location in the list of targets. 
Change the sort order of a property column	Click the column header. The arrow symbol next to the column header indicates if the sort order is ascending or descending .
Resize a property column	Drag the edge between this and the following column header.
Refresh the **Target assemblies** list	Click in the upper right corner of the **Batch editor**.

2.6 Manage pours

With the pour management functionality of Tekla Structures, you can view the geometry of cast-in-place concrete structures, show them as parts or as pour objects, plan pours and pour breaks, and report pour information, such as

concrete volumes and formwork areas. You can define pours, pour units, pour objects, and pour breaks for concrete parts whose cast unit type is **Cast in place**.

In Tekla Structures, a *pour object* is a building object that consists of one or more cast-in-place concrete parts, or parts of them. The cast-in-place concrete parts are merged into one pour object if they have the same material grade and they touch each other. They also need to be in the same *pour phase* to be merged. Pour objects are visible in *pour views*.

A *pour unit* is an entity for cast-in-place concrete and it consists of a pour object and all related reinforcement, embeds, and other objects that need to be in place before concrete can be poured on the building site.

A *pour* is a group of pour objects that is poured at one go.

With a *pour break* you can split a pour object into smaller pour objects.

NOTE Pour management is mainly targeted at contractors for quantity take-off, planning, and on-site activities. By default, pour management is disabled in new models in most roles. You can enable pour management (page 469) in the current model using the advanced option XS_ENABLE_POUR_MANAGEMENT.

See also

Enable pour management (page 469)

View cast-in-place concrete structures (page 471)

Define the pour phase of a part (page 474)

Pour objects (page 475)

Pour units (page 477)

Pour breaks (page 482)

Troubleshoot pours (page 488)

Example: Create concrete geometry and work with pours (page 491)

Enable pour management

By default, pour management is disabled in new models in most roles. You can enable pour management in the current model in the **Advanced Options** dialog.

WARNING If pour management is enabled in the model, do not disable it using `XS_ENABLE_POUR_MANAGEMENT`, especially in the middle of the project. This may cause problems if you have drawings containing pour objects, and if you are sharing your model. The

pour objects and pour breaks in the model and in the drawings may get invalid, and you may lose all pour-related modeling work.

1. On the **File** menu, click **Settings** --> **Advanced options** to open the **Advanced options** dialog.
2. Under **Concrete detailing**, set XS_ENABLE_POUR_MANAGEMENT to `TRUE`.
3. Click **OK**.
4. Save and re-open the model for the change to take effect.

   The commands that show and create pour objects and pour breaks are now available in the model and in drawings.

See also

Disable pour management temporarily (page 470)

Disable pour management temporarily

You can temporarily disable pour management. This may be needed if pour management seems to significantly slow down your model, for example, when the pours and pour objects are very large and require splitting into smaller ones.

When pour management is temporarily disabled, the existing pour objects and pour breaks are still present in the model, but any modifications to the model geometry that would normally update pour objects and pour breaks automatically will not do that. Any information related to pours will be outdated and inaccurate, for example, in reports, and the pour breaks will not be adaptive. They will be updated automatically when you re-enable pour management.

To disable or re-enable pour management:

1. Go to **Quick Launch**, start typing `pours and pour breaks`, and select the **Toggle pours and pour breaks** command from the list that appears.
2. Click **Yes** in the confirmation dialog.

NOTE If you are working in a Tekla Model Sharing model, remember to re-enable pour management before you write out. Similarly, if you are working in the multi-user mode, re-enable pour management before you save the model. In this way, information related to pours stays up to date for all users of the model.

TIP If you have problems opening a large model with pour objects that contain many parts, you may need to disable pour management before you open the model.

You can do this by modifying the `xs_user.[user name]` file located in the model folder. Set `PAPB` to `0` to disable pours, and then save the file.

When needed, remember to re-enable pour management.

See also

Enable pour management (page 469)

View cast-in-place concrete structures

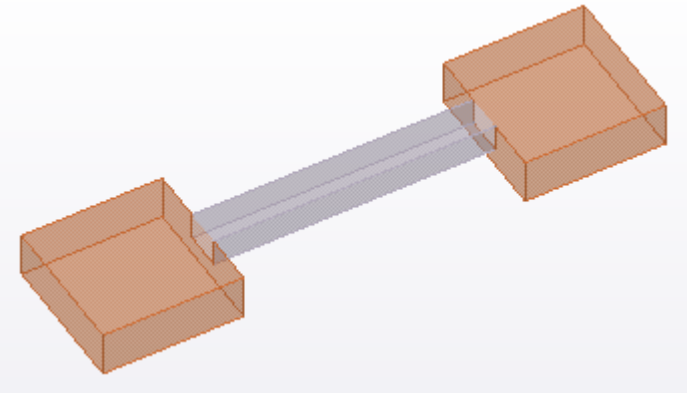
When pour management is enabled, you can view cast-in-place concrete structures in model views either as parts or as pour objects.

Depending on your needs, you can switch between the different representation options for cast-in-place concrete structures. For example, working in a part view is useful when you want to reinforce individual parts or change their geometry. A pour view is useful when you want to find out the volume of concrete to be poured or check which objects belong to a pour unit, or when you want to reinforce continuous structures that span multiple parts.

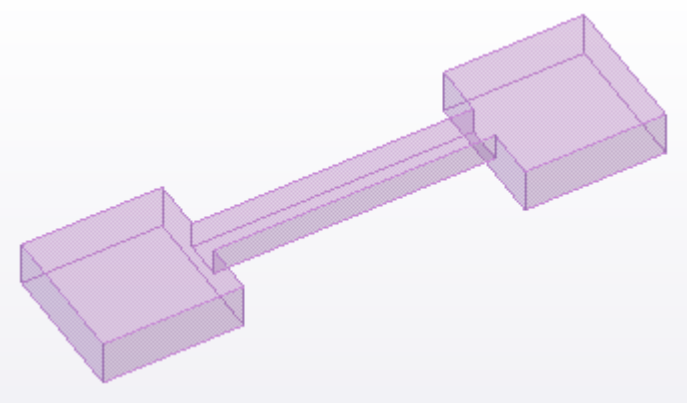
Set the appearance of cast-in-place concrete structures

You can define how cast-in-place concrete structures appear in a model view.

1. Ensure that pour management is enabled (page 469).
2. Double-click the view to open the **View properties** dialog.
3. Click **Display** to open the **Display** dialog.
4. Ensure that the **Parts** checkbox is selected.
5. In the **Cast in place** list, select either:
   - **Parts**



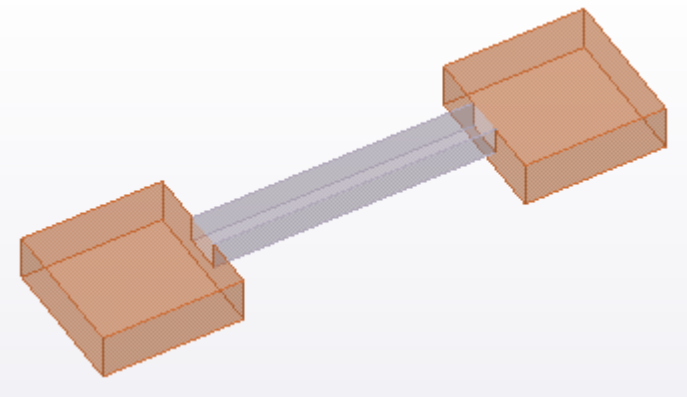
   - **Pours**



6. If you selected **Parts** for the cast-in-place concrete structures, in the **Cast in place parts** list, select either:

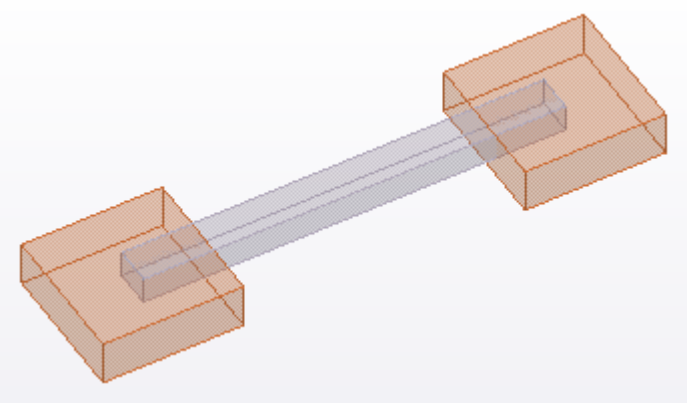
   - **Merged**

     Tekla Structures shows concrete parts as merged in the model if their cast unit type is **Cast in place**, if they have the same material grade and pour phase (page 474) number, and if they touch or overlap one another. When these criteria are met, Tekla Structures removes the outlines of the individual parts within each continuous concrete structure.



   - **Separated**

     Tekla Structures shows concrete parts as individual parts and separated by their outlines.



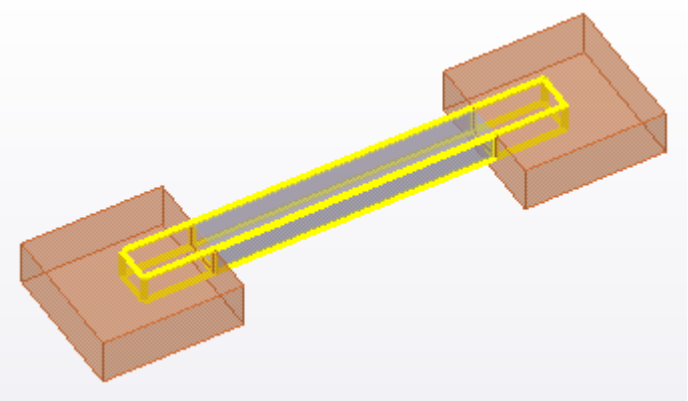
7. Ensure that the view is selected.
8. Click **Modify** to save the changes.

TIP To quickly change the representation of the active view from **Parts** to **Pours**, and the other way round, click  **Pour view** on the **Concrete** tab.

You can also create two views, a pour view and a part view, and keep them both open side by side on your screen.

Part view vs. pour view

Continuous concrete structures cannot be selected or highlighted in part views. When you hold the mouse pointer over a concrete structure in a part view, Tekla Structures highlights the original parts belonging to it. You can select a part and modify it if needed:



Duplicates and overlapping parts are counted only once in the volume calculations for pour objects. Note that single part and cast-unit volumes are still calculated the same way as before, which means that the sum of single

part and cast-unit volumes may be higher than the volume of pour objects that are defined from exactly the same part geometry.

When you reinforce a concrete structure, you need to reinforce the individual concrete parts within it in part views, or you can reinforce pour objects by using **Reinforcing Bar Shape Catalog** or rebar sets in pour views. Therefore, you can reinforce a part of a continuous concrete structure independently from the whole continuous concrete structure. All reinforcement are visible both in part views and in pour views.

Define the pour phase of a part

Use the pour phase property to separate pour objects from one another. By defining pour phases, you can prevent cast-in-place concrete parts from merging even if they have the same material grade and they touch or overlap one another.

NOTE Pay attention to pour phases when you create cast-in-place concrete parts. For example, use pour phase 0 for horizontal structures, like beams and slabs, and pour phase 1 for vertical structures, like columns and walls, to separate them to different pour objects. In this way you can make sure that the number of parts included in each single pour object is reasonable, and that your models do not slow down due to too large pour objects.

To modify the pour phase of a part:

1. Double-click a concrete part to open the part properties in the property pane.
2. Under **Cast unit**:
   a. In the **Cast unit type** list, ensure that the cast unit type is set to **Cast in place**.
   b. In the **Pour phase** box, enter a pour phase.

   By default, the pour phase is 0. If you are unable to change the value, the cast unit type that you set in step 2a is incorrect.
3. Click **Modify**.

NOTE When you define pour phases, make sure that parts in different pour phases do not overlap. If you use parts (not pour objects) to report geometric information, overlapping volumes of different pour phases are not merged but counted twice in calculations, and you may end up with incorrect volume, area, or weight information.

See also

View cast-in-place concrete structures (page 471)

Pour objects

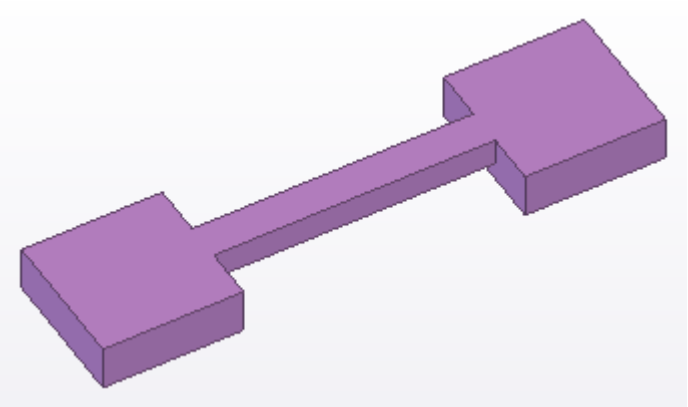
When pour management is enabled (page 469), each concrete part whose cast unit type is **Cast in place** automatically forms a pour object.

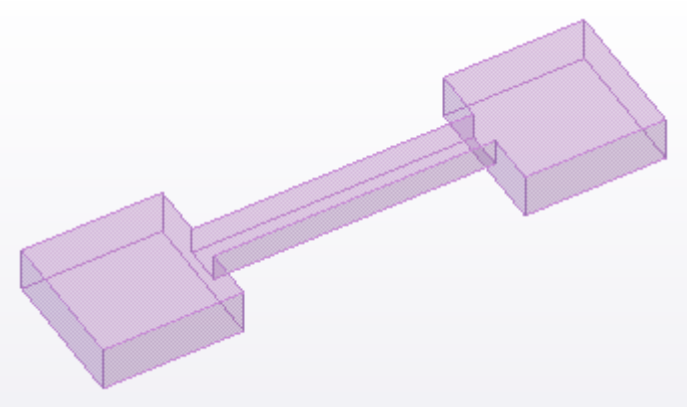
Tekla Structures automatically merges multiple cast-in-place concrete parts to a pour object if they have the same material grade and pour phase (page 474) number, and if they touch or overlap one another.

By creating pour breaks (page 482) you can split pour objects into smaller pour objects.

NOTE Make sure that the number of parts included in each single pour object is reasonable. Too large number of parts and part surfaces in a pour object slows down the model.

Pour objects are visible in pour views (page 471). All pour objects are displayed using the same color, regardless of the individual parts' color within a concrete structure. You can change the default color by using the advanced option XS_POUR_OBJECT_COLOR in **File** --> **Settings** --> **Advanced options** --> **Concrete detailing** .





You can also use different color and transparency settings (page 476) to show pour object groups, for example, by pour number.

TIP You can group pour objects using **Organizer** or **Task manager**.

Limitations

The following commands are not available for pour objects: **Copy**, **Move**, **Delete**, **Split**, and **Combine**. This is because pour object geometry is defined by parts. If you want to change the geometry of pour objects, you have to modify the parts instead of pour objects, or you can create pour breaks.

See also

Modify the properties of a pour object (page 477)

Reinforce pour objects using Rebar shape catalog (page 528)

Create a rebar set (page 494)

Change the color and transparency of pour objects

By default, all pour objects are displayed using the same color in pour views, regardless of the individual parts' color. You can customize the pour object color and transparency in model views by defining object groups and then selecting specific color and transparency settings for each group.

TIP To change the default color of pour objects, use the advanced option XS_POUR_OBJECT_COLOR in **File** --> **Settings** --> **Advanced options** --> **Concrete detailing** .

1. On the **View** tab, click **Representation**.

   The **Object Representation** dialog is displayed.

2. Create a new object group for the pour objects whose color and transparency you wish to change.

   a. In the **Object Representation** dialog, click **Object group...**.

   b. In the **Object group - Representation** dialog, click **Add row**.

   c. To direct the settings to pour objects instead of parts, select the following options for the row:

      - **Category** = **Object**
      - **Property** = **Object type**
      - **Condition** = **Equals**
      - **Value** = **Pour object**

   d. If needed, add any additional filtering criteria.

      For example, to filter pour objects by a certain user-defined attribute, add a row with **Pour object** as **Category**, and define the **Property**, **Condition**, and **Value** as desired.

e. Enter a unique name in the box next to the **Save as** button.

f. Click **Save as** to save the object group.

g. Click **Close**.

3. Repeat step 2 if you wish to create more object groups.
4. In the **Object Representation** dialog, select an object group from the **Object group** list.
5. In the **Color** list, choose a color for the object group.
6. In the **Transparency** list, set the transparency of the object group.
7. Click **Modify**.

   The object group's color and transparency changes in the model.

See also

Change the color and transparency of model objects by using object representation (page 686)

Modify the properties of a pour object

Pour objects have properties and user-defined attributes which you can view, define, and modify.

For example, you can enter a **Pour number** that you can use to define the sequence of pours, and a **Pour type** that you can use to describe each pour object.

1. Ensure that you are using a pour view. If not, click **Pour view** on the **Concrete** tab to show the pour objects.
2. Ensure that the  **Select objects in assemblies** selection switch is active.
3. Double-click the pour object whose properties you want to modify.
4. In the property pane, enter or modify the pour object properties.
5. Click **Modify**.

See also

Pour objects (page 475)

Pour units

When pour management is enabled, you can create pour units that combine together pour objects and other objects. A *pour unit* is an entity for cast-in-place concrete and it consists of a pour object and all related reinforcement,

embeds, and other objects that need to be in place before concrete can be poured on the building site.

For each pour object (page 475) in the model, there is a corresponding pour unit to which the pour object belongs. You can automatically add other objects to pour units by using the **Calculate pour units** command. You can also modify the pour units manually.

The following model objects can be added to pour units:

- Reinforcement, such as single reinforcing bars, bar groups, reinforcement meshes and strands
- Assemblies (for example, rebar assemblies and embeds)
- Sub-assemblies (for example, embeds in cast-in-place cast units)
- Bolts (for example, anchor bolts and shear studs)
- Surfaces added to the pour object
- Precast cast units

Precast cast units can only be added manually, not by using the **Calculate pour units** command.

Note that some model objects such as parts and welds cannot be directly added to a pour unit. Instead, these objects are indirectly linked to the pour unit through the assemblies and cast units they belong to.

One model object can be included in only one pour unit at a time.

Calculate pour units

You can have Tekla Structures detect which objects form pour units and automatically add the objects to pour units.

1. Ensure that pour management is enabled (page 469).
2. On the **Concrete** tab, click **Calculate pour units**.

   Tekla Structures adds the objects (page 481) to pour units.

You can check the pour units in a pour view, or using the **Inquire** tool, **Organizer**, or reports.

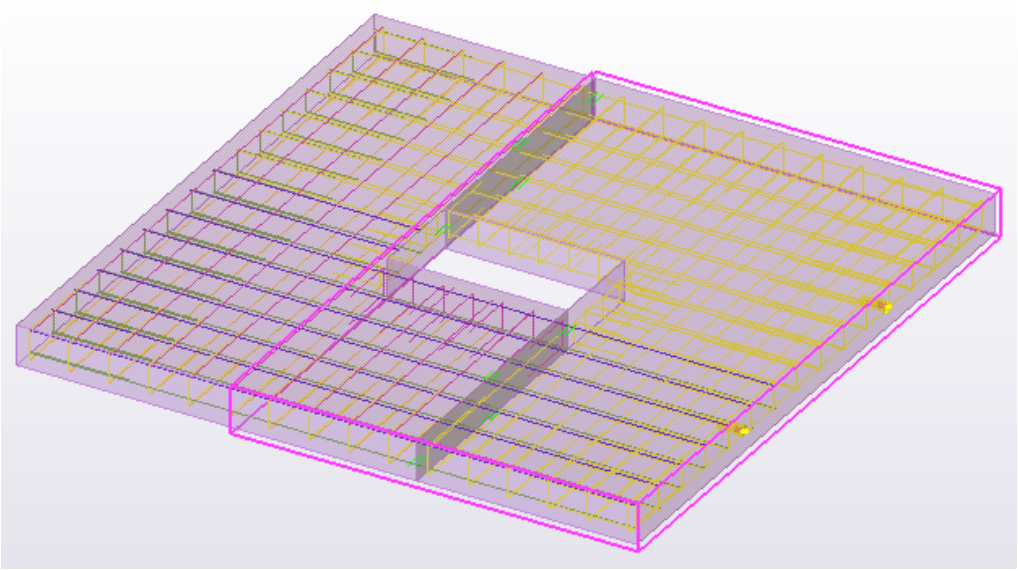
If you want to modify the pour units, you can manually add and remove objects. The manual additions will be kept even if you use the **Calculate pour units** command again, but the objects you manually remove from pour units will be added again.

Check and inquire objects in a pour unit

You can visually check which objects are included in a pour unit. You can also use the **Inquire** tool to get information about the pour unit and the objects in it.

1. Ensure that you are using a pour view (page 471). If not, on the **Concrete** tab, click **Pour view** to show the pour objects.
2. Ensure that the  **Select assemblies** selection switch is active.
3. Click a pour object to select its pour unit.

   A magenta box indicates the pour unit.



4. Right-click and select **Inquire** .

   Tekla Structures lists the objects in the pour unit and shows their properties in the **Inquire object** dialog.

Add objects to a pour unit

In addition to using the **Calculate pour units** command, you can manually add objects to pour units.

1. Ensure that you are using a pour view (page 471). If not, on the **Concrete** tab, click **Pour view** to show the pour objects.
2. Select the objects that you want to add to a pour unit.

   You can add reinforcement, assemblies, precast cast units, and bolts.

   If you have other objects selected, they will not be added.
3. Right-click and select **Pour** --> **Add to pour unit**.

   Alternatively, you can go to **Quick Launch** and use the **Add selected objects to pour unit** command. You can also assign a keyboard shortcut to this command.
4. Click a pour object to add the objects to the corresponding pour unit.

Tekla Structures adds all the objects that can be added to the pour unit. The unallowed objects are not added.

TIP If you do not have objects selected, you can first start the **Add to pour unit** command by using **Quick Launch** or a customized keyboard shortcut, and then select the object that you want to add to the pour unit.

Remove objects from a pour unit

After using the **Calculate pour units** command, you can manually remove objects from pour units.

1. Select the object that you want to remove from a pour unit.
2. Right-click and select **Pour** --> **Remove from pour unit**.

   Alternatively, you can go to **Quick Launch** and use the **Remove selected objects from pour unit** command. You can also assign a keyboard shortcut to this command.

The removed object can then be added to another pour unit either manually with the **Pour** --> **Add to pour unit** command, or automatically with the **Calculate pour units** command.

TIP If you do not have objects selected, you can first start the **Remove from pour unit** command by using **Quick Launch** or a customized keyboard shortcut, and then select the object that you want to remove from the pour unit.

Reset pour unit relations

In some cases you may need to reset all or some of the pour unit contents and relations defined using the **Calculate pour units** command and/or the **Add to pour unit** command.

To do this:

1. Go to **Quick Launch**.
2. Search for and select the appropriate command of the following:
   - **Reset all pour unit relations**
   - **Reset all manually assigned pour unit relations**
   - **Reset all pour unit relations except manual assignments**
3. In the confirmation dialog, click **Yes** to reset pour unit relations.

   Note that if you use the **Reset all pour unit relations** command, and then click **No** in the confirmation dialog to cancel the resetting of the manual assignments, the automatic relations are still reset. If you want to recreate the automatic relations, use the **Calculate pour units** command again.

Modify the properties of a pour unit

You can modify the pour unit properties in the same way as pour object properties, but using a different selection switch.

1. Ensure that you are using a pour view (page 471). If not, click **Pour view** on the **Concrete** tab to show the pour objects.
2. Ensure that the  **Select assemblies** selection switch is active.
3. Double-click the pour unit whose properties you want to modify.
4. In the property pane, enter or modify the pour unit properties.

   For example, you can define the pour unit name and user-defined attributes.
5. Click **Modify**.

How Tekla Structures automatically adds objects to pour units

When you use the **Calculate pour units** command, Tekla Structures automatically adds objects to pour units.

Each object that collides with a pour object, meaning that the object at least partly overlaps the pour object, is added to the same pour unit to which the pour object belongs.

If any object in an assembly collides with a pour object, the entire assembly is added to the pour unit.

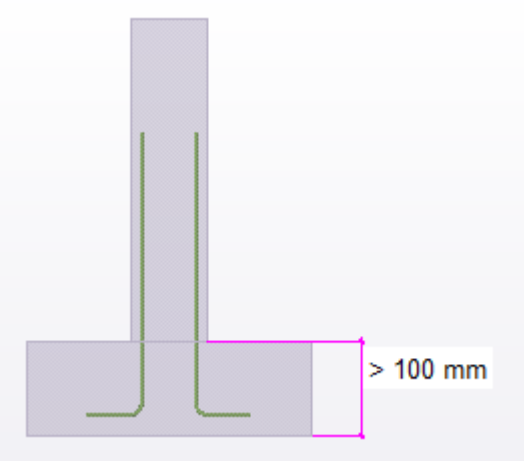
Only reinforcement objects that belong to cast-in-place parts are added to pour units.

If any object in a reinforcing bar group or strand group collides with a pour object, the entire group is added to the pour unit. On the other hand, individual reinforcing bars in a rebar set can be added to different pour units.

Objects colliding with more than one pour object

If an object collides with more than one pour object, the object will be associated with the pour object whose bounding box has the lowest global z coordinate.

For example, the reinforcing bars colliding with a footing pour object and a column pour object are associated with the footing pour object because its bottom face has a lower global z coordinate than the column pour object.



If the lowest global z coordinates of the pour object bounding boxes are the same or differ less than 100 mm, the object will be associated with one of the pour objects according to these rules:

1. If the object's center of gravity is within only one of the colliding pour object bounding boxes, the object will be associated with that pour object.
2. If the object's center of gravity is within more than one pour object bounding box, or completely outside any pour object bounding box, the object will be associated with the pour object whose center of gravity is closest to the object's center of gravity.

If there are changes in pour units

Whenever there are changes in a pour object or pour unit, all associations to that pour unit are reset. Similarly, if there are changes in an object associated with the pour unit, this association is reset. The next time you use the **Calculate pour units** command, only the unresolved associations are calculated.

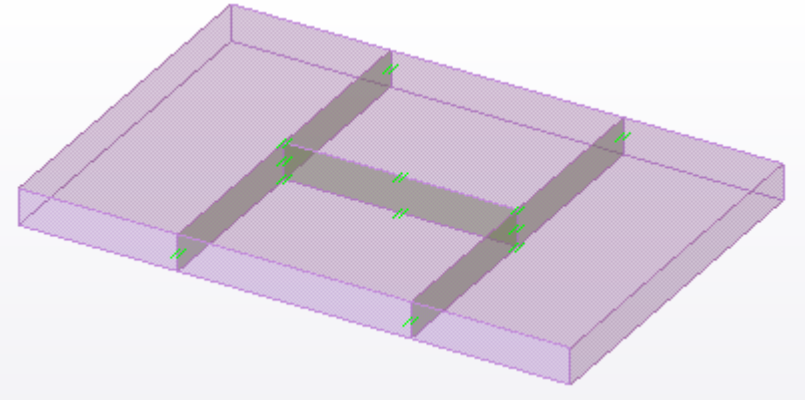
See also

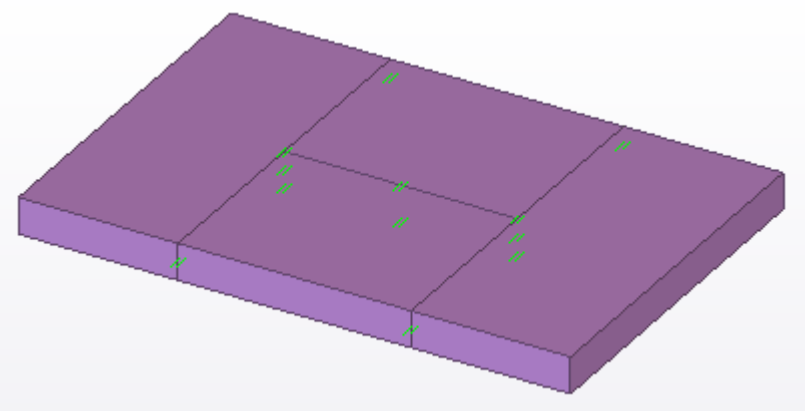
Pour units (page 477)

Pour breaks

When pour management is enabled, you can use pour breaks to split pour objects into smaller pour objects.

Pour breaks are visible and you can work with them both in pour views and in part views (page 471). Pour breaks are displayed as a thin plane or line, depending on the part rendering option (page 671) you use.





WARNING If you move or copy a part, pour breaks do not follow the part. Pour breaks remain at their original locations, and adapt to any cast-in-place concrete parts (page 483) they still touch.

If a pour break does not split a pour object completely into two, the pour break is displayed in red by default. This means that it is invalid and needs to be remodeled.

See also

Set the visibility of pour breaks (page 484)

Create a pour break (page 485)

Modify a pour break (page 487)

Pour break adaptivity

Pour breaks are adaptive to changes in cast-in-place concrete parts and pour objects. This means that if you change the geometry or location of a cast-in-place concrete part or pour object, its pour breaks change accordingly.

If you delete a cast-in-place concrete part, its pour breaks disappear as well.

If you modify a cast-in-place concrete structure in any of the following ways, its pour breaks adapt:

- Change the profile or dimensions of a part
- Add or remove cuts or fittings
- Change chamfer shape or dimensions
- Add or remove parts of the cast-in-place concrete structure by:
  - Changing the cast unit type of a part from **Precast** to **Cast in place** or vice versa
  - Changing the pour phase of a part
  - Changing the concrete grade of a part
  - Moving, copying, or deleting parts

If you move a cast-in-place concrete part outside its pour breaks, the pour breaks disappear. If you move a part so that it still hits one or more pour breaks, the pour breaks that are inside the part stay in their original locations and adapt to the part in the new location.

If you copy or move a pour break and it hits a cast-in-place concrete part in the destination location, the pour break adapts to the part. Also the pour breaks that you copy from another model adapt to the parts in the model they are copied to.

If a pour break is dependent on another pour break that is split or deleted, the dependent pour break is deleted as well. If a pour break is dependent on another pour break that is moved, the dependent pour break adapts inside the pour object, as long as the pour break plane can touch the moved pour break.

If a pour break is split so that it becomes partial, the pour break is deleted. A partial pour break can only split a cast-in-place part or pour object in conjunction with other pour breaks.

Set the visibility of pour breaks

You can show pour breaks in model views.

Before you start, ensure that pour management is enabled (page 469).

1. Double-click a model view to open the **View properties** dialog.
2. Click **Display...** to open the **Display** dialog.
3. Select the **Pour break** checkbox.
4. Click **Modify**.

See also

Pour breaks (page 482)

Create a pour break

You can add pour breaks to pour objects or concrete parts whose cast unit type is **Cast in place**.

You can create pour breaks by picking one, two, or more points in the model.

When you create a pour break that traverses more than two points, the pour break will be limited to the pour object it splits and perpendicular to the current work plane. If you need to create an inclined or horizontal pour break using multiple points, shift the work plane (page 56) first.

TIP Use the  **Snap to nearest points (points on line)** snap switch (page 89) to start or end pour breaks on part or pour object edges.

Use the  **Snap to any position** snap switch (page 89) to pick intermediate points for pour breaks.

To create a pour break, do any of the following:

To	Do this
Create a pour break, perpendicular to a part face, using one point	1. On the **Concrete** tab, click **Pour break** --> **Single point** . 2. Pick the location for the pour break.
Create a pour break that splits all cast-in-place concrete parts and pour objects located between two points	1. On the **Concrete** tab, click **Pour break** --> **Two points**. 2. Pick two points to define the location of the pour break.
Create a pour break using multiple points	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points**. 3. Pick the points you want the pour break to go through.
Create a pour break defined by the opposite corners of a rectangle	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points**.

To	Do this
	3. Hover over , and then click  on the toolbar that appears. 4. Pick two opposite corner points of the pour break.
Create a pour break defined by the center and one corner of a rectangle	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points**. 3. Hover over , and then click  on the toolbar that appears. 4. Pick the center point of the pour break. 5. Pick one corner point of the pour break.
Create a pour break defined by three corners of a rectangle	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points**. 3. Hover over , and then click  on the toolbar that appears. 4. Pick three corner points of the pour break.
Create a pour break defined by one side midpoint and two corners of a rectangle	1. If needed, shift the work plane. 2. On the **Concrete** tab, click **Pour break** --> **Multiple points**. 3. Hover over , and then click  on the toolbar that appears. 4. Pick one side midpoint of the pour break. 5. Pick two corner points of the pour break.

If the pour break you are creating does not split a pour object or cast-in-place part completely into two, Tekla Structures does not add the pour break to the model. You may need to use another **Pour break** command to create a valid pour break, for example **Multiple points** instead of **Single point**.

See also

Modify a pour break (page 487)

Pour breaks (page 482)

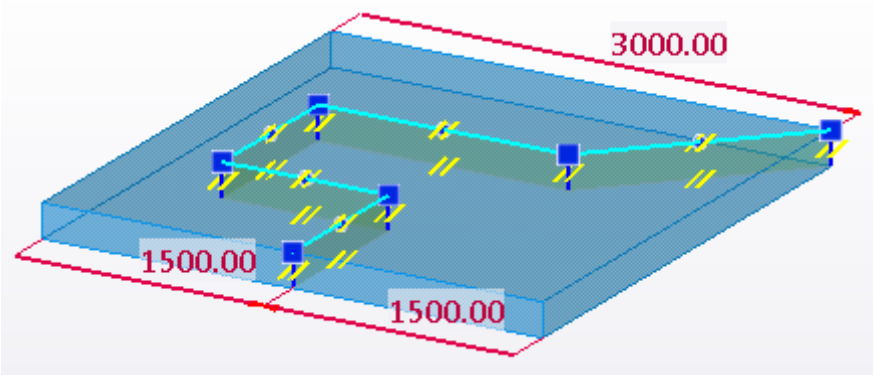
Modify a pour break

You can modify, copy, move, and remove existing pour breaks.

Before you start:

- Ensure that the  **Direct modification** switch is active.
- Ensure that the  **Select pour breaks** selection switch is active.
- Select the pour break.

  Tekla Structures displays the handles and dimensions that you can use to modify the pour break.



To modify a pour break:

To	Do this
Change the shape or location of the pour break	Drag a corner point or an end point to a new location.
Change a location dimension	Drag a dimension arrowhead to a new location, or: 1. Select the dimension arrowhead which you want to move. 2. Using the keyboard, enter the value with which you want the dimension to change.

To	Do this
	To start with the negative sign (-), use the numeric keypad. To enter an absolute value for the dimension, first enter $, then the value. 3. Press **Enter**, or click **OK** in the **Enter a Numeric Location** dialog.
Add an intermediate point to the pour break	Drag a midpoint handle  to a new location.
Remove an intermediate point from the pour break	1. Select an intermediate corner point. 2. Press **Delete**.
Modify the pour break properties	1. Double-click the pour break to open the property pane. 2. Modify the properties. 3. Click **Modify**.
Copy the pour break	Copy (page 137) the pour break like any other object in Tekla Structures. For example, right-click and select **Copy**.
Move the pour break	Move (page 155) the pour break like any other object in Tekla Structures. For example, right-click and select **Move**. You may need to move the pour break if you have moved the part, because the pour break does not follow the part.
Remove the pour break	Press **Delete**.

See also

Pour breaks (page 482)

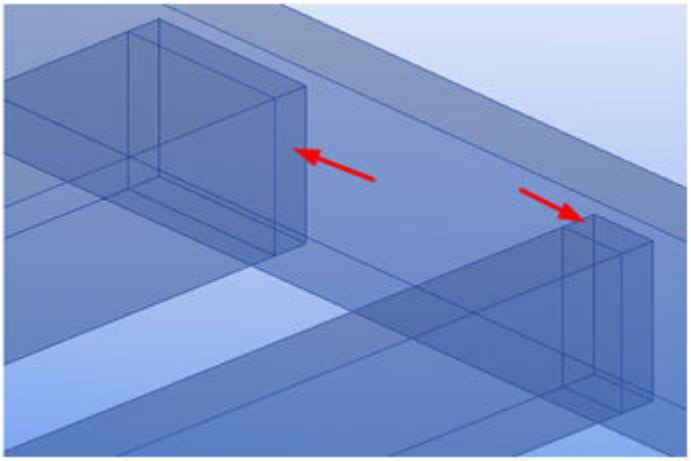
Resize and reshape model objects (page 119)

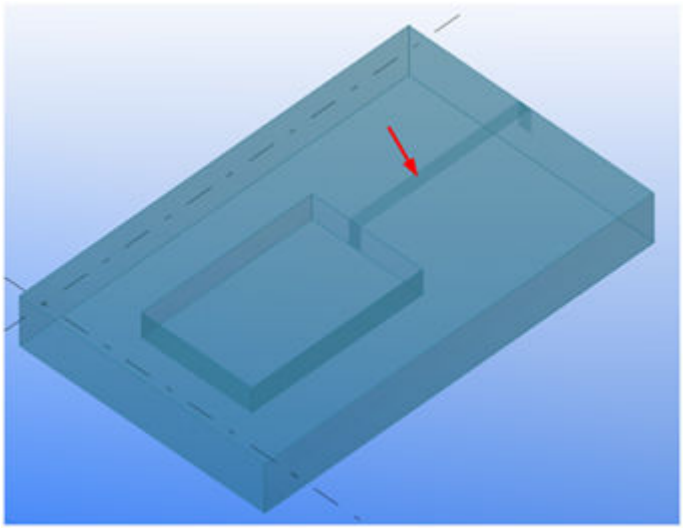
Troubleshoot pours

When you work with cast-in-place concrete parts, it is important that you regularly check the resulting pour objects, and try to get rid of the errors related to them, before you start detailing, or creating drawings and reports. The errors in solid pour objects may lead to inaccuracies in volume and other quantity calculations, and to incorrect representation and hatching in drawings.

While you model, use the following methods to check the model for pour-related errors:

- Check if there are `Solid error` rows in the session history log file.
- Ensure that the cast-in-place concrete parts and pour objects look continuous in model views. They should not have part outlines or shadow lines inside them, like in the following images:





If you notice errors or overlapping volumes or faces, try remodeling some of the parts.

You can also try out the following tips to avoid pour-related errors:

- Make sure that the number of parts included in a single pour object is reasonable.
- Sometimes modeling the parts in a different order may fix errors in pour objects.

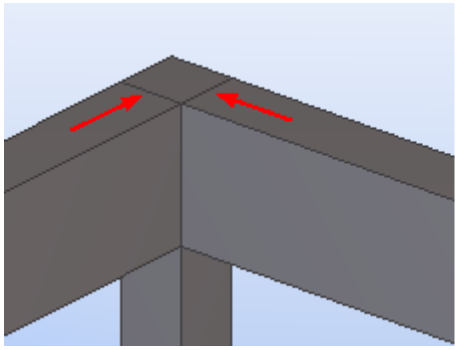
- To control which lines are visible in drawings, use the advanced options XS_DRAW_CAST_PHASE_INTERNAL_LINES and XS_DRAW_CAST_UNIT_INTERNAL_LINES.

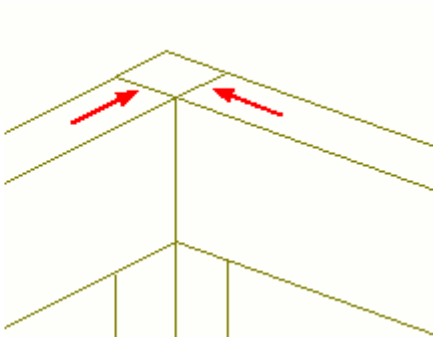
  This may help because the cast-in-place concrete parts that have errors are treated in the same way in drawings as the precast concrete parts.

If remodeling the parts does not fix errors in solid pour objects, then overlap the parts as little as possible to ensure that the volume and quantity calculations are close to the correct values.

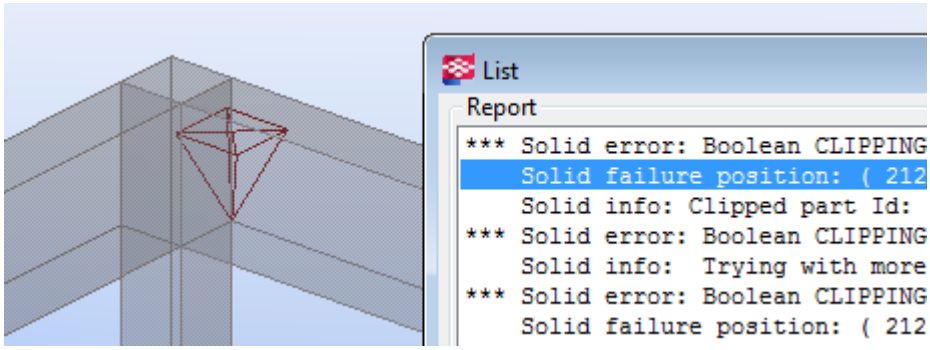
Example: Identify and fix a pour error

This is how an error related to a solid pour object can be indicated in a model view and in a drawing. The pour object is not shown as continuous, and there are extra lines between the parts in the pour object:

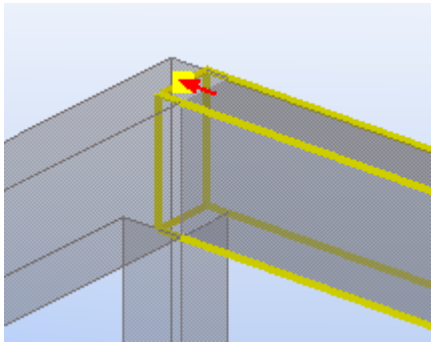




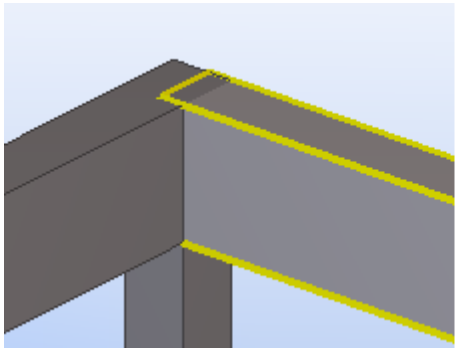
Checking the session history log file and clicking a `Solid failure position` row helps you to locate the error in the model (press **Ctrl+2** to see through the parts):

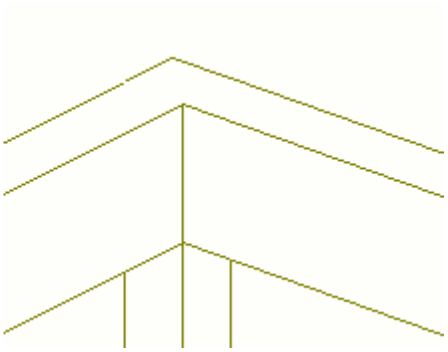


Try moving a beam end so that it is no longer on the same surface as the column side:



This is how the model and the drawing look like after you have fixed the model:





Also the volume of the pour object is now correct, for example, in reports. The overlapping volume of the beam and the column is only counted once.

See also

Manage pours (page 468)

Example: Create concrete geometry and work with pours

The guidelines in this example help you to efficiently model cast-in-place concrete geometries, and to define, visualize, sequence, and report pours and pour breaks.

Before you start, ensure that you have pour management enabled. See Enable pour management (page 469).

1. If possible, use an existing engineering or architectural model or drawing as a basis when you create concrete structures in Tekla Structures.

   Import the existing model or drawing as a reference model to your Tekla Structures model.

   See Insert a reference model and Reference models and compatible formats.

2. If you are using an IFC model as a reference model:

a. Convert the concrete structures you need from the IFC model to native Tekla Structures objects.

   See Convert IFC objects into native Tekla Structures objects and Example: Convert IFC objects into Tekla Structures objects at one go.

b. Check the conversion results.

c. If needed, modify the converted objects.

   For example, you may need to change the profile, material, or cast unit type of the converted objects.

   TIP Use **Organizer** for checking and selecting objects.

3. If you are using a different reference model type, or if there are structures that cannot be converted from an IFC model, model the needed concrete structures as cast-in-place concrete parts in Tekla Structures.

   You can model by tracing over the reference model.

   See Create parts and modify part properties (page 217).

4. For each cast-in-place concrete part, define a pour phase number to divide your Tekla Structures model into pour objects.

   For example, use the default pour phase 0 for horizontal structures, like beams and slabs, and the default pour phase 1 for vertical structures, like columns and walls, to separate them to different pour objects.

   See Define the pour phase of a part (page 474).

   TIP Use selection filters or **Organizer** to efficiently select multiple parts and to modify them all at the same time.

5. View and check the pour objects in a pour view.

   See View cast-in-place concrete structures (page 471) and Pour objects (page 475).

6. If needed, modify the pour phases or create pour breaks to fine-tune the pour objects.

   For example, create pour breaks to split large slabs into smaller pour objects.

   See Create a pour break (page 485) and Pour breaks (page 482).

7. Once you are ready with the concrete geometry and pour objects, you can define pour sequences by entering pour numbers for pour objects, or by using the **Organizer** categories.

   See Modify the properties of a pour object (page 477) and Categories in Organizer.

8. Calculate the pour units, and modify them by adding and removing objects if needed.

   See Pour units (page 477).

9. You can also define other properties for pour objects and pour units, for example, concrete mixtures, or dates or status of workflow.

   See Modify the properties of a pour unit (page 481) and Categories in Organizer.

10. Use **Organizer** to categorize pours. Then you can select them by their sequence and report pour-specific information, such as pour volumes and formwork areas.

    See View object properties in Organizer and Example: Organize the model into location and custom categories, and view quantities.

11. If you wish, use **Task manager** to include pour objects and pour units in tasks and to schedule pours. You can then visualize pour status information based on planned and actual dates by using **Project status visualization**.

    See Create a task in Task manager and Project status visualization.

12. Create general arrangement drawings for pour units.

    Select a pour unit using the  **Select assemblies** switch, create a 3D view of the pour unit, and then create a GA drawing using the 3D view.

    This way you can automatically include in the drawing all reinforcement, embeds, and other objects that need to be shown with the pour object.

    See Pours in drawings.

2.7 Create reinforcement

Once you have created a model of concrete parts, you will need to reinforce the parts to gain higher strength for the parts.

In Tekla Structures, you can use different methods to create reinforcement. In many cases, you may need to use a combination of several reinforcement tools to get the desired results.

The most automated method is to use the various reinforcement components Tekla Structures contains. We recommend that you use reinforcement components to create reinforcement whenever possible. The components are adaptive, attached to a concrete part, and updated automatically if the dimensions of the reinforced part change.

Rebar sets (page 494) are another flexible and versatile method to create reinforcement. Also the rebar sets are adaptive to concrete geometry, and easy to modify using direct modification.

In addition to these methods, you can manually create:

- single reinforcing bars (page 520)
- reinforcing bar groups (page 520)

  For more automated reinforcing bar group creation you can use Rebar shape catalog (page 522), which contains predefined reinforcement shapes.
- reinforcement meshes (page 536)
- prestressed strands (page 541)
- reinforcement splices (page 543)

When needed, you can also combine several reinforcement objects into rebar assemblies (page 643), for example, to create reinforcement cages.

Create a rebar set

Rebar sets consist of reinforcing bars that are automatically grouped according to bar geometry and that you can modify by using direct modification and the rebar set guidelines, leg faces, and local modifiers. You can create rebar sets when you want to flexibly reinforce various areas and geometries in concrete parts or pour objects.

You have several options for creating rebar sets. Longitudinal and crossing rebar sets, and rebar sets created according to concrete part or pour object faces, are attached and adaptive to a concrete part or pour object. Using the **Point input** command you can create rebar sets even outside concrete objects. You can also use Rebar shape placing tool (page 507) to create rebar sets.

NOTE When you work with rebar sets, ensure that the  **Direct modification** switch is active.

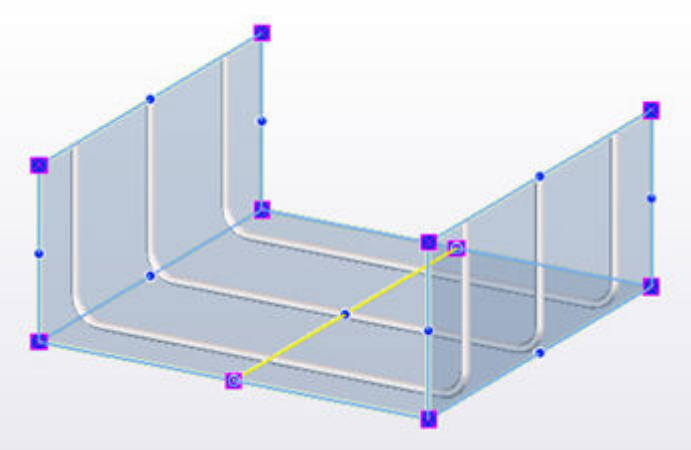
Basic concepts related to rebar sets

The leg faces (page 549) of a rebar set are planes that define where the reinforcing bar legs are created. Tekla Structures creates leg faces at the reinforced faces of concrete parts or pour objects, or according to the points you pick when you create rebar sets.

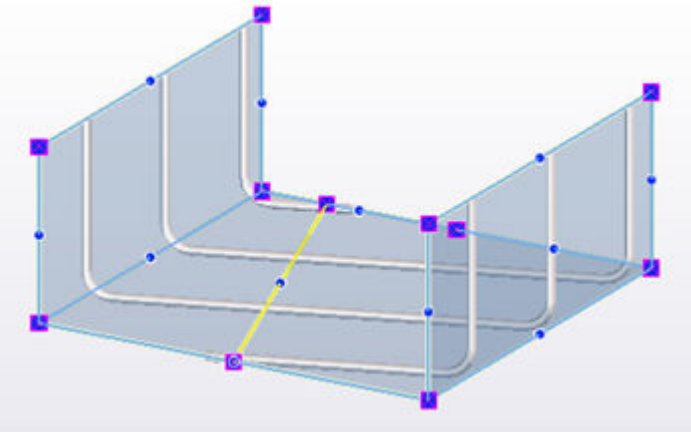
The rebar set *leg surfaces* are similar to leg faces, but they can be curved. Tekla Structures creates leg surfaces when you create rebar sets using the **By guidelines** command.

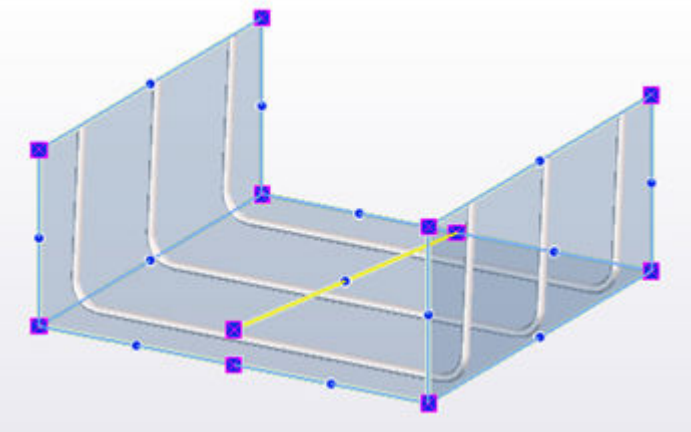
Each rebar set has at least one *guideline* that defines the distribution direction of the bars. The spacing of the bars is also measured along the guideline. The guideline can be a line, or a polyline that may have corner chamfers.

In the example below, the leg faces are shown in gray, and the guideline is highlighted in yellow:



The position of the guideline affects the bar creation. If you move or lift an end of the guideline, the bars turn respectively. For example:



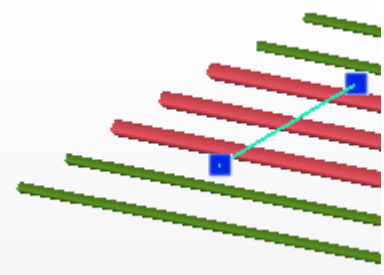


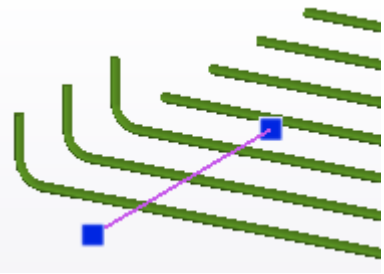
If needed, you can create an unlimited number of *secondary guidelines* for a rebar set that has leg faces. For a rebar set that has leg surfaces, you can only create one secondary guideline.

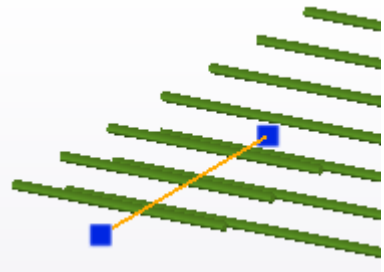
You can use guidelines to define different spacings along the rebar set. Each guideline has its own offset and spacing settings, and secondary guidelines have an option to inherit both settings from the primary guideline.

When you create rebar sets using the **Longitudinal** command, Tekla Structures automatically creates the needed number of guidelines in curved (page 513) beams, polybeams, strip footings, and multi-segment wall panels. In flat polybeams, one guideline is created for each sharp corner, and three guidelines for each curved corner.

If you need to modify a rebar set only at certain locations, you can create local (page 557) *property modifiers*, *end detail modifiers*, and *splitters*.







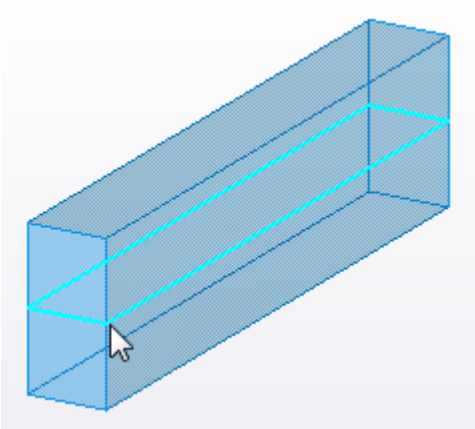
Create longitudinal rebars

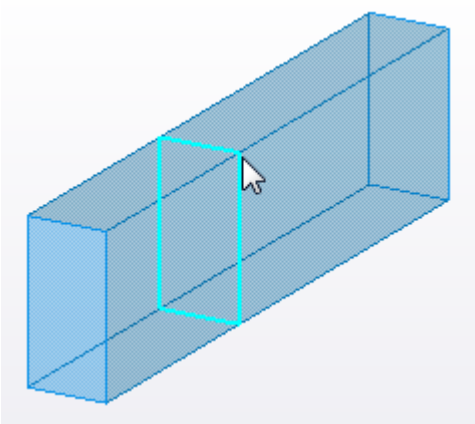
You can create rebar sets that consist of longitudinal reinforcing bars in a concrete part or pour object.

1. Depending on the concrete object you want to reinforce, use a part view or a pour view (page 471).

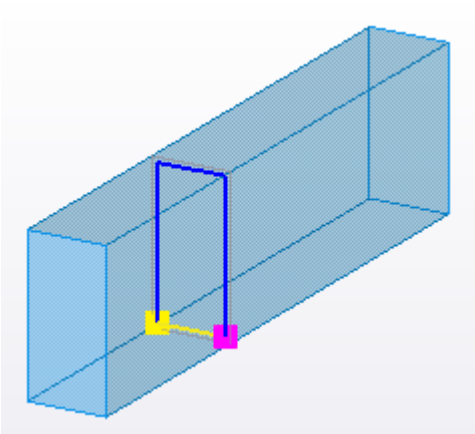
2. On the **Rebar** tab, click **Longitudinal**.
3. Move the mouse pointer over the edges of a concrete part or pour object.

   Tekla Structures highlights the cross sections that you can select.





4. Select the cross section that you want to reinforce.



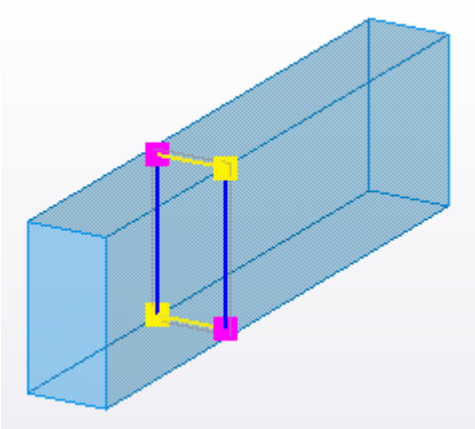
5. If needed, modify the cross section size or shape for the bars.

   To do this, click on the contextual toolbar, and then drag the cross section handles.

6. In the selected cross section, select the faces that you want to reinforce.

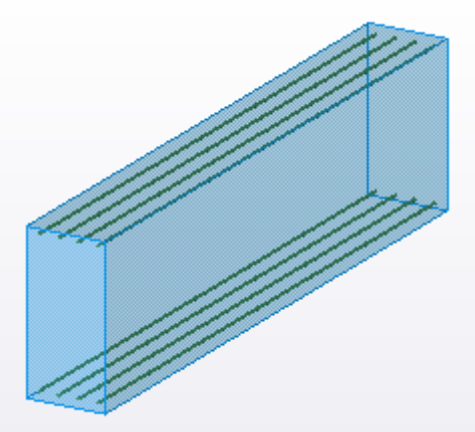
   By default, only one of the faces is selected. To select multiple faces, hold down **Shift** or **Ctrl**.

   Tekla Structures highlights the selected faces in yellow.



7. To extend or shorten individual faces, click on the contextual toolbar. Then drag the yellow and magenta end handles.
8. To finish, click the middle mouse button, or click **Create the rebar set** on the contextual toolbar.

Tekla Structures creates a rebar set at each selected face, and the bars perpendicular to the selected cross section.



Create crossing rebars

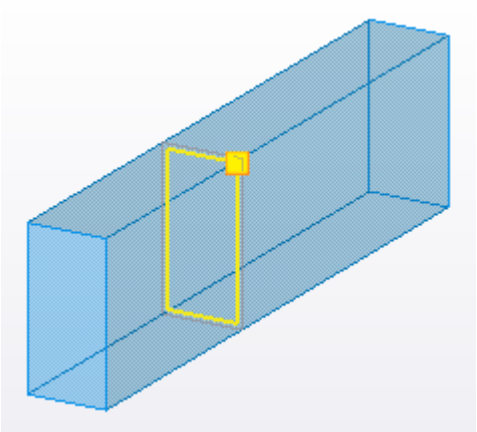
You can create a rebar set that consists of crossing reinforcing bars in a concrete part or pour object.

1. Depending on the concrete object you want to reinforce, use a part view or a pour view (page 471).
2. On the **Rebar** tab, click **Crossing**.

3. Move the mouse pointer over the edges of a concrete part or pour object.

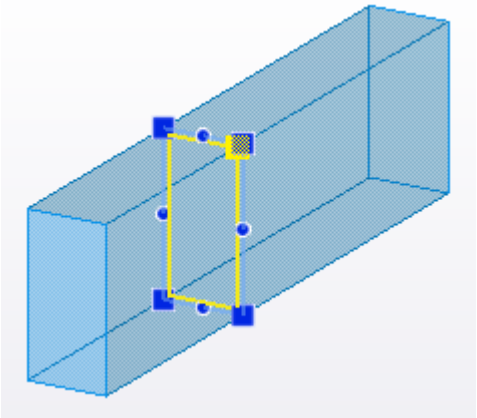
   Tekla Structures highlights the cross sections that you can select.

4. Select the cross section that you want to reinforce.



5. If needed, modify the shape of the bars.

   To change the cross section size for the bars, click on the contextual toolbar and then drag the cross section handles.

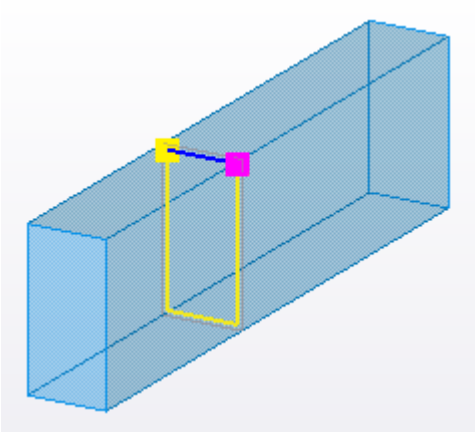


6. With active on the contextual toolbar, in the selected cross section, select the bar legs that you want to create.

   By default, all legs are selected and Tekla Structures creates a leg for each object face.

   - To unselect a selected leg, hold down **Ctrl** and click the leg.
   - To unselect all legs, click on the contextual toolbar.
   - To select more than one leg, select the first leg, and then hold down **Ctrl** or **Shift** when selecting the rest of the legs.
   - To select all legs, click on the contextual toolbar.

Tekla Structures highlights the selected legs in yellow, and creates a continuous bar shape of the legs.

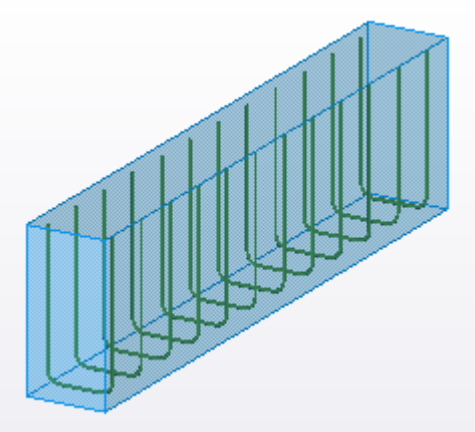


7. If you want to rotate the bar shape, for example, to move stirrup hooks to another corner, press **Tab** for counterclockwise or **Shift+Tab** for clockwise direction.
8. If you need to extend or shorten individual legs, click on the contextual toolbar and then drag the bar end handles.

   This way you can also create overlapping bar shapes, or extend bar ends outside the concrete object.
9. If you want to modify the distribution length of the bars, adjust the length of the rebar set guideline.

   Click on the contextual toolbar and then drag the guideline end handles .
10. To finish, click the middle mouse button, or click **Create the rebar set** on the contextual toolbar.

Tekla Structures creates the bars parallel to the selected cross section, and distributes the bars along the length of the guideline.



Create rebars by face

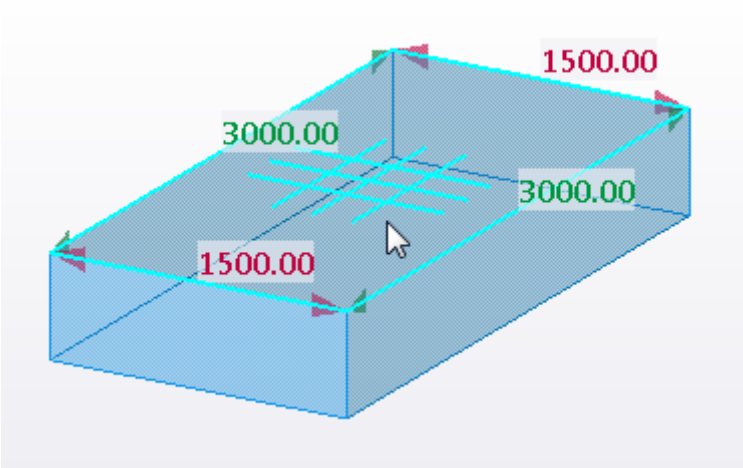
You can create rebar sets according to a face of a concrete part or pour object.

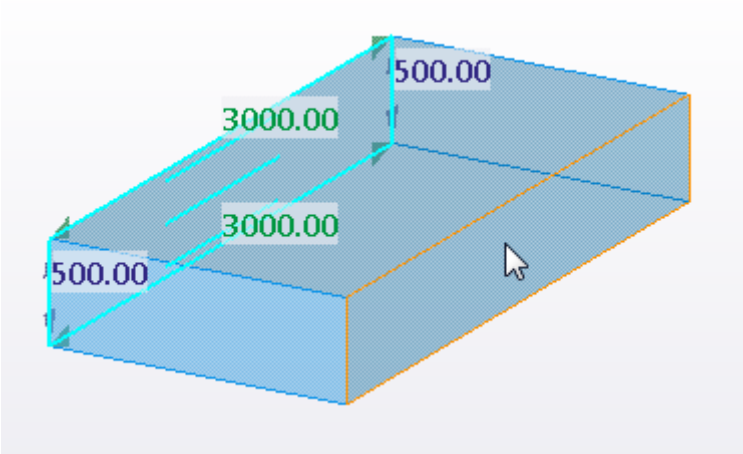
1. Depending on the concrete object you want to reinforce, use a part view or a pour view (page 471).
2. On the **Rebar** tab, click **By face**.
3. Define the faces and areas of the concrete object you want to reinforce, and the direction of the bars using the following options on the contextual toolbar:

Click this button	To do this
	Create bars at the near face of the concrete object.
	Create bars at the far face of the concrete object.
	Create bars parallel to the longest object face edge. With the  option, bars are created so that they are parallel to the edge that is closest to the mouse pointer.
	Create bars perpendicular to the longest object face edge. With the  option, bars are created so that they are perpendicular to the edge that is closest to the mouse pointer.
	Create bars in two directions: one set of bars parallel to the longest object face edge, and the other set of bars perpendicular to the edge. With the  option, one set of bars is parallel and the other set of bars perpendicular to the edge that is closest to the mouse pointer.
	Create bars for an entire object face.
	Create bars for a rectangular area on an object face.
	Create bars for a polygonal area on an object face.

4. Depending on the area you selected to reinforce, do one of the following:
   - To reinforce an entire object face:
     a. Move the mouse pointer over the faces of a concrete part or pour object.

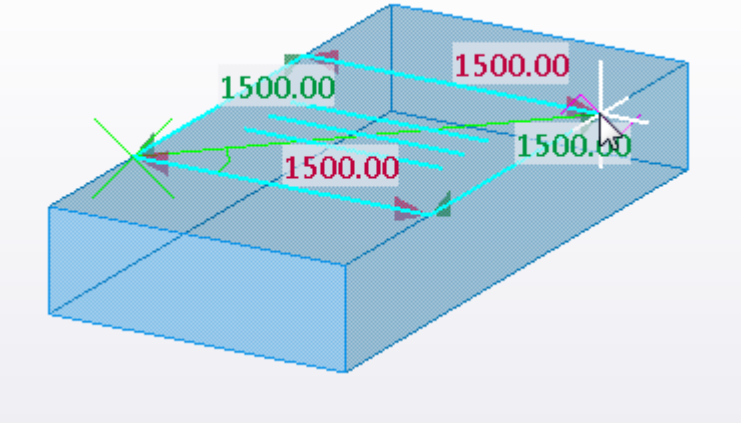
Tekla Structures shows the object face dimensions, and a symbol that indicates the direction of the bars.



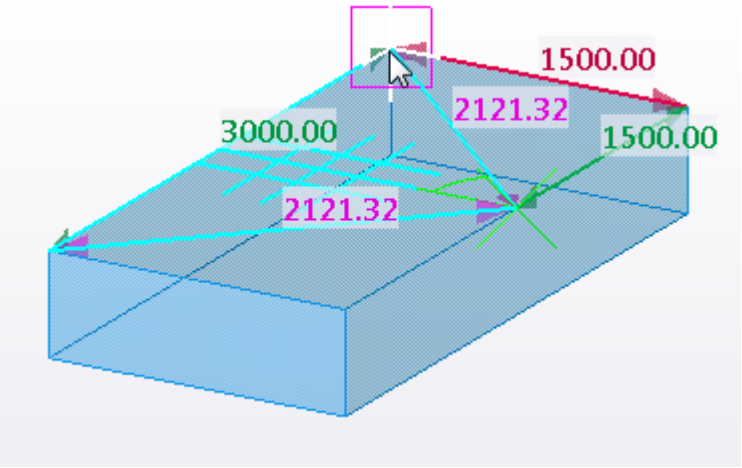


b. Select the object face.

- To reinforce a rectangular area, pick two opposite corners of the area.

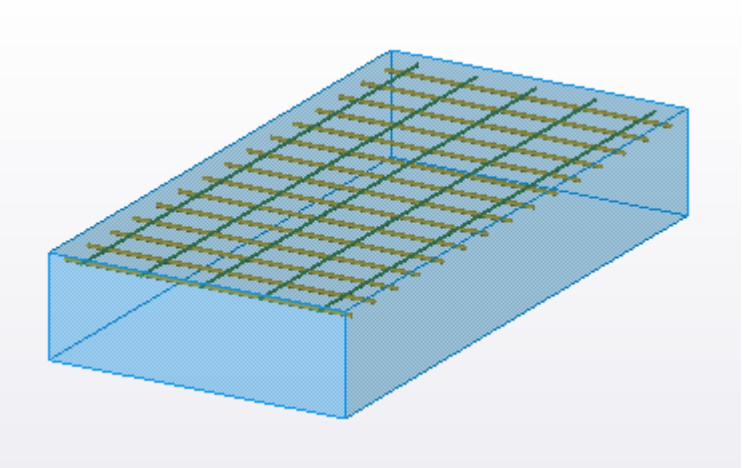


- To reinforce a polygonal area, pick the polygon corners.



Then click the middle mouse button.

Tekla Structures creates the bars according to the options you selected. If you selected to create bars in two directions, Tekla Structures creates two rebar sets: one with bars parallel to the longest object face edge, and one with bars perpendicular to it.

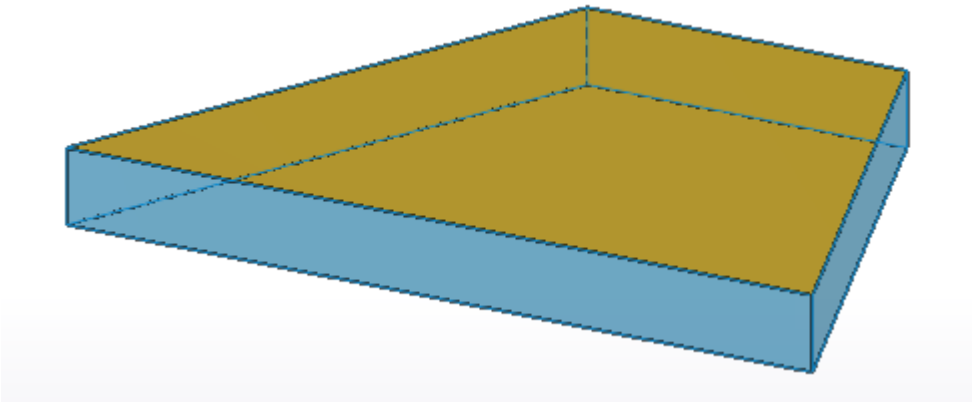


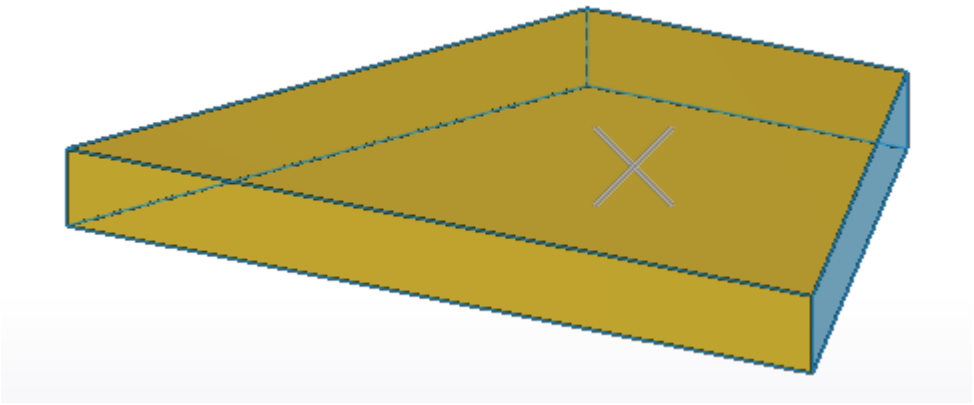
Create rebars by guidelines

You can create a rebar set at one or more faces of a concrete part, according to one or two guidelines you define by picking points.

1. On the **Rebar** tab, click **By guidelines**.
2. Select the part faces that you want to reinforce.

Tekla Structures highlights the selected faces in yellow.

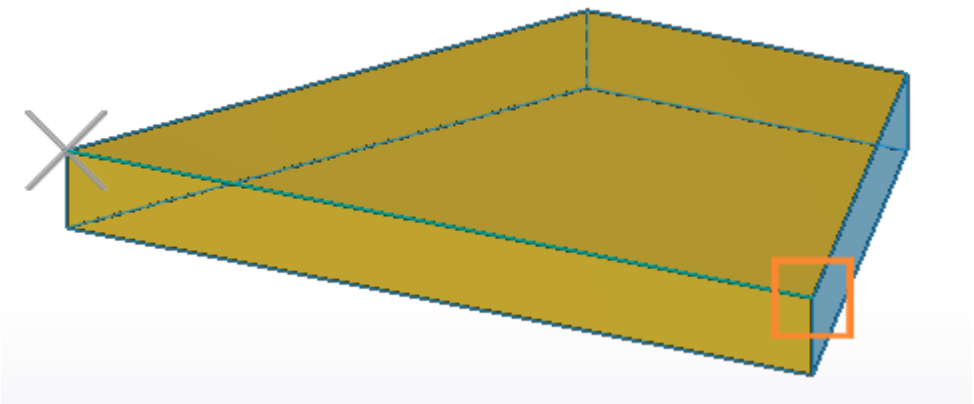




You can select as many faces as you like.

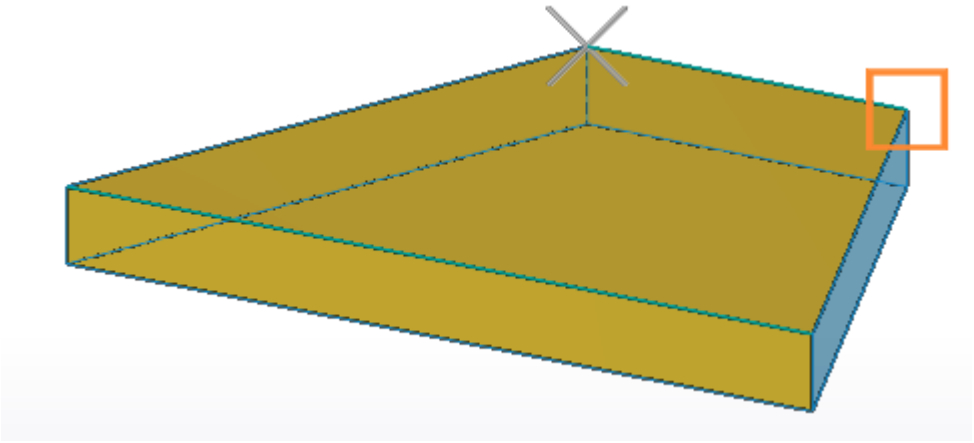
If you need to deselect a selected face, click the face again.

3. Click the middle mouse button to finish selecting faces.
4. Pick points to define the location for the primary guideline, and then click the middle mouse button.



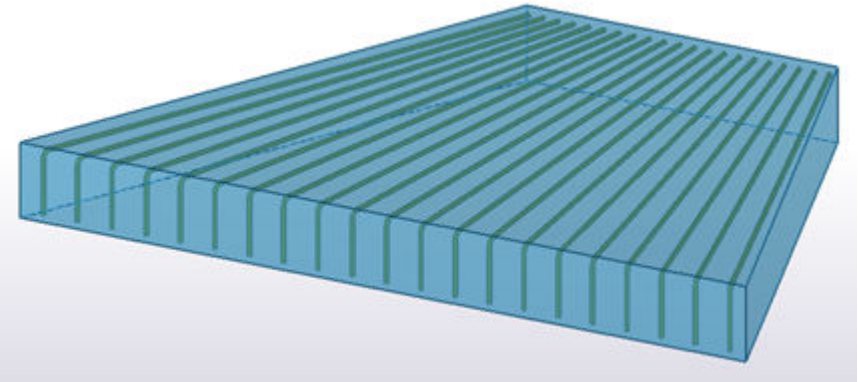
= start point, = end point

5. If you want to create a secondary guideline, pick points to define its location.



6. Click the middle mouse button to create the rebar set and the guidelines.

Tekla Structures creates a rebar set that extends over the faces you selected, and distributes the bars along the guidelines.



Create rebars by point input

You can create a set of reinforcing bars so that you define the shape of the bars by picking points in the model.

1. On the **Rebar** tab, click **More** --> **Point input**.
2. On the contextual toolbar, select an option to define the rebar set type and the number of cross sections in the rebar set.

   The options are:

   - **Normal**
   - **Tapered**

- **Tapered ridge**
- **Tapered curved**
- **Tapered N**

If you select the **Tapered N** option, enter the number of cross sections.



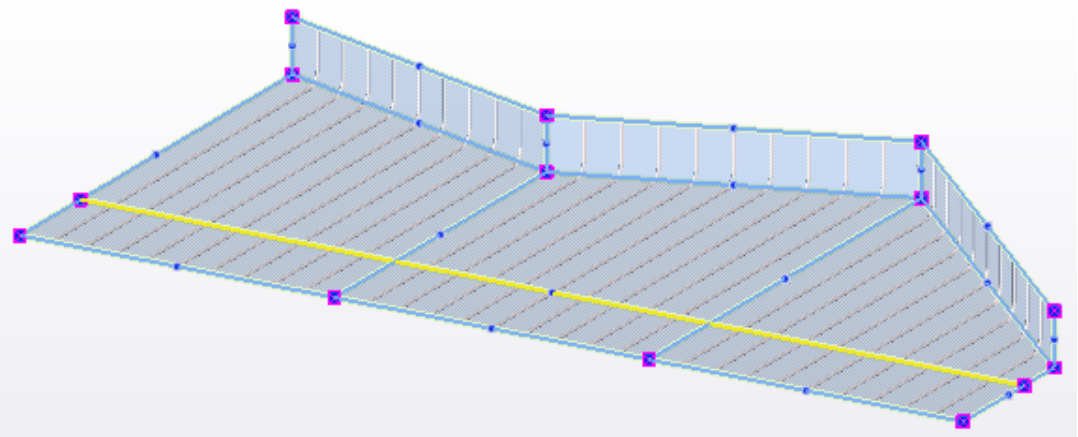
3. Pick points to define the shape of the bar at the first cross section.

   You can use different snapping (page 88) methods, such as **Ortho** and temporary reference points.

4. Click the middle mouse button to finish picking.
5. Depending on the rebar set type and the number of cross sections, do one of the following:

   - For a  **Normal** rebar set, pick the start point and the end point of the rebar set.
   - At the second and subsequent cross sections for other types of rebar set, pick points to define the bar shape. Then click the middle mouse button to finish picking at each cross section.

Tekla Structures creates a rebar set with leg faces between each cross section.



Rebar set properties

Use the contextual toolbar or the property pane to view and modify the properties of rebar sets. The file name extension of the property file is `.rst`.

See also Rebar set properties (page 1068) and Modify a rebar set (page 545).

Limitations

- Roundings in bent bar corners are not taken into account in automatic clash avoidance when Tekla Structures creates rebar sets and arranges them to layers.
- You cannot create rebar sets in deformed parts.

Create a rebar set using Rebar shape placing tool

You can create a rebar set by selecting a predefined bar shape from **Rebar shape placing tool**. The predefined shapes in **Rebar shape placing tool** are based on the shapes that have been defined in **Rebar shape manager** and saved in the `RebarShapeRules.xml` file.

Use **Rebar shape placing tool** to reinforce parts and pour objects. The rebar sets may extend across single or multiple objects.

Rebar shape placing tool does not work with round, spiral, or 3D bar shapes, or in tapered variable cross sections.

Create rebar sets

1. On the **Rebar** tab, click **More** --> **Rebar placing**.

   The **Rebar shape placing tool** dialog opens.
2. If you want to create bars that extend across several parts or pour objects, for example dowel bars, select **Multiple objects** from the list at the bottom of the dialog.
3. If you want to create several rebar sets in the same cross section, select the **Keep cross section** checkbox.
4. Select one of the predefined bar shapes from the tree view on the left.

   If the shape that you need is not available, or if you want to remove the shapes that you do not need, you can reorganize the tree view (page 512).
5. Define the bar dimensions.

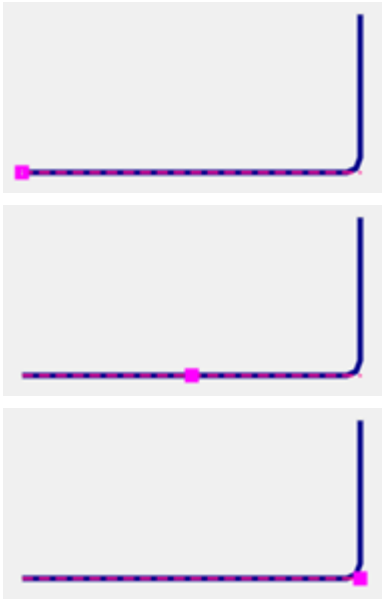
   The dimensions you can define vary depending on the selected bar shape.

   The hook properties are visible only if you have set the advanced option XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION to `FALSE` in **File menu** --> **Settings** --> **Advanced Options** --> **Concrete Detailing**.

   - To set a **Length of leg** value, click a leg in the preview of the shape.

     If you do not enter a **Length of leg** value, the leg length is calculated automatically according to the dimensions of the concrete structure.
   - To set a **Bending angle** value for a bend that is not 90 degrees, click one of the legs next to the bend.

6. Set the rebar set reference point to start, middle, or end by double-clicking the different legs or hooks in the preview of the shape.



When you are placing the rebar set in the model, you can move the rebar set preview to a new location by dragging the reference point.

7. Modify the other bar properties as needed.

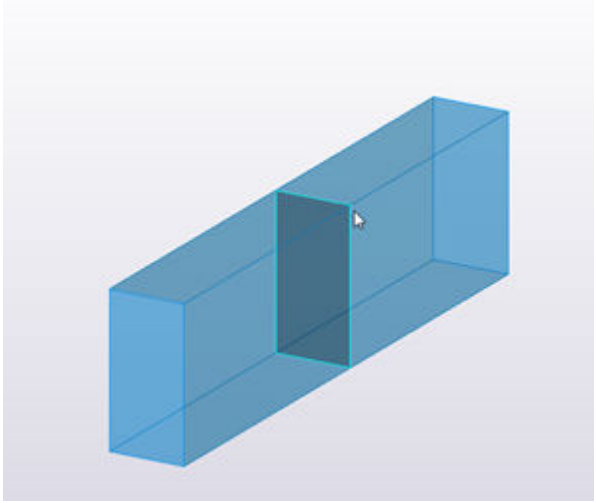
   For example, you can use **Layer order number** to arrange bars to layers when two or more rebar sets overlap.

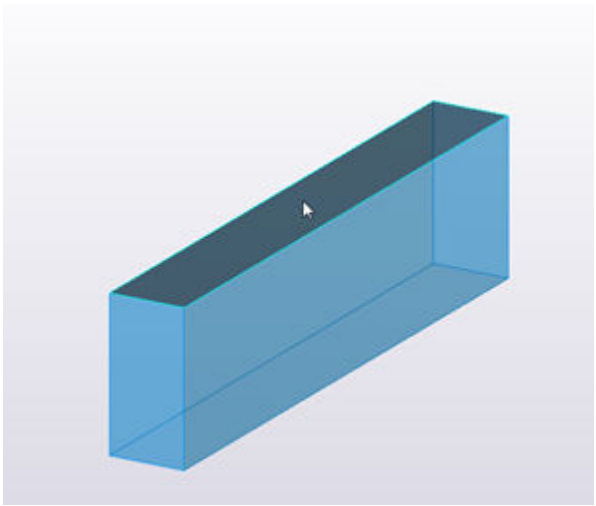
8. On the **Spacing** tab, define the spacing properties of the rebar set.

9. To place the rebar set in the model, move the mouse pointer over the edges and faces of a concrete structure.

   Depending on the concrete structure you want to reinforce, use a part view or a pour view (page 471).

   Tekla Structures highlights the cross sections and faces that you can select. For example:

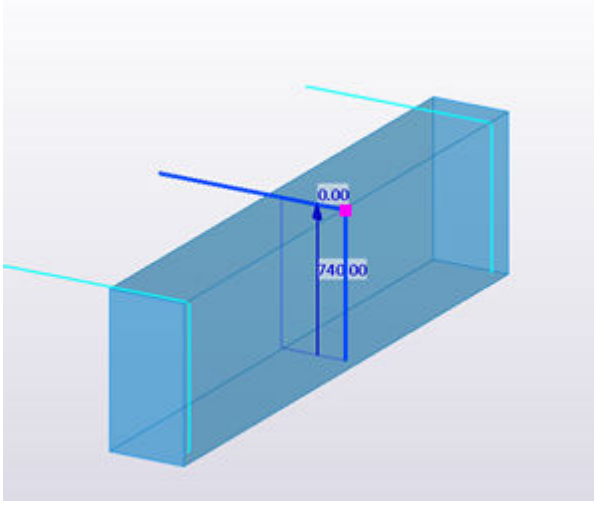




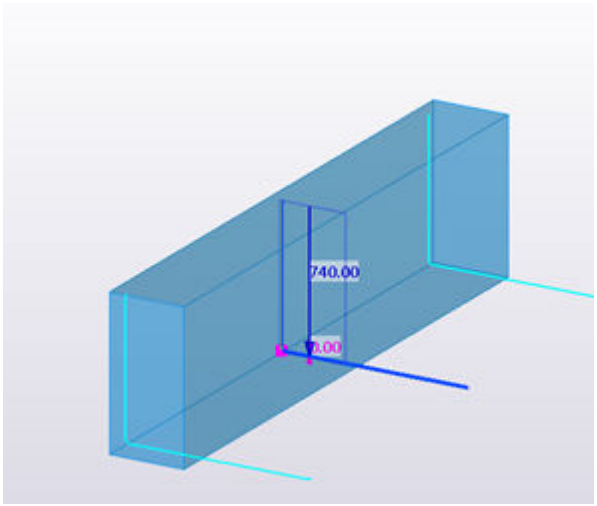
10. Select the cross section or face that you want to reinforce.

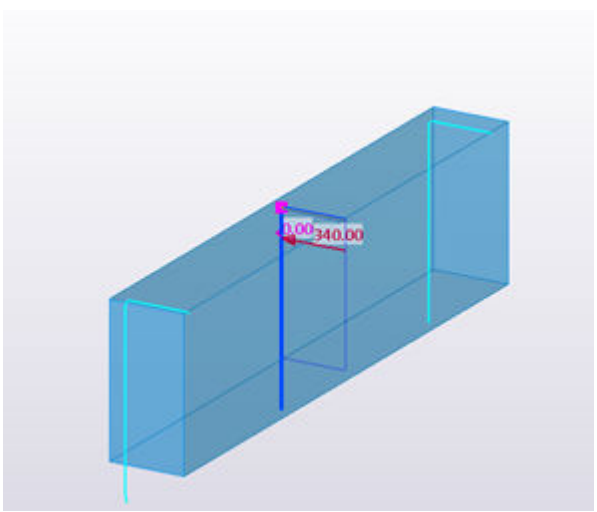
    If you have **Multiple objects** selected, click each cross section or face to select them. Click the middle mouse button to finish selecting.

    Tekla Structures shows a preview of the bar shape in the model, and the first and last bars in the rebar set in cyan.



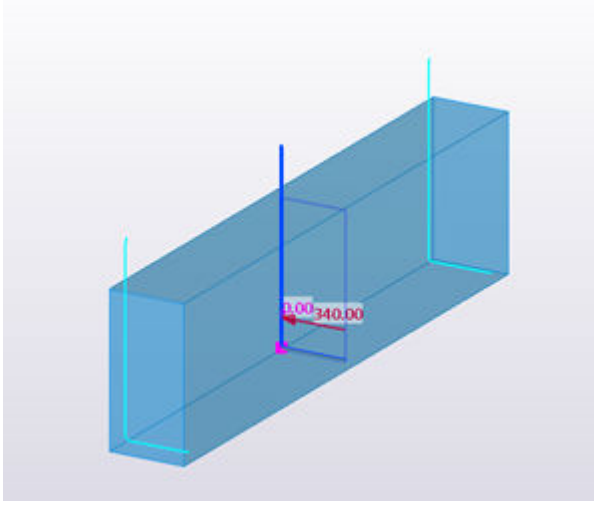
11. To move the rebar set to the desired location in the selected cross section or face, do any of the following:
    - Click a blue line segment to place the rebar set reference point on that line segment. For example:



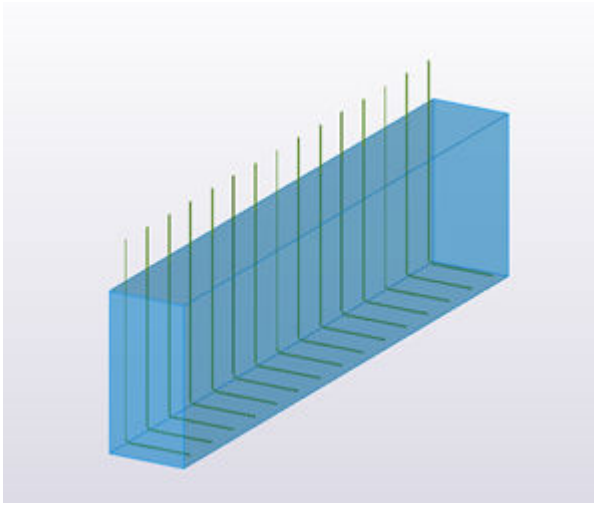


If needed, you can still change the reference point location in the **Rebar shape placing tool** dialog by double-clicking the desired location in the preview.

- Drag the magenta reference point handle  to a new location on the blue line.
- To turn the bar shape, click  on the contextual toolbar.



12. Click the middle mouse button to create the rebar set.



13. If you have the **Keep cross section** checkbox selected, repeat steps 4–12 to create more rebar sets in the same cross section.

TIP If the **Rebar shape placing tool** dialog is already open but the command is not active, click the **Select cross section** button to start creating rebar sets again.

Add and remove rebar shapes

You can modify the tree view in **Rebar shape placing tool** by adding frequently used bar shapes to the tree, or removing the bar shapes that you do not need.

1. On the **Rebar** tab, click **More** --> **Rebar placing**.

   The **Rebar shape placing tool** dialog opens.

2. Click **Organize catalog**.

3. To create a new category folder, click .
4. Drag and drop the selected shapes to the folder.

   If multiple shapes have the same shape code and you drag them to the categories, the shape codes get a suffix **(1)**, **(2)**, and so on. You can rename the shapes by clicking the name twice and entering a new name or suffix, for example, **(a)**, **(b)**.

   When the shapes are listed in a report, they all get the same shape code.
5. Change the name of the folder in a similar way, if needed.
6. To remove a shape from a category, select the shape and click .
7. Click **OK**.

Examples: Rebar sets in curved structures

You can reinforce curved concrete structures using rebar sets.

The curved concrete structures can include curved beams (page 284), polybeams (page 287) that have **Arc point** chamfers, and flat spiral beams (page 291) whose total rise is zero. You can also reinforce strip footings and wall panels the same way as you reinforce beams and polybeams.

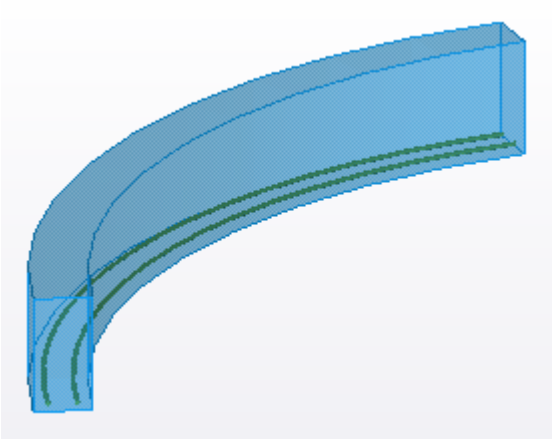
For more information about creating rebar sets, see also Create a rebar set (page 494).

Create longitudinal bars for a curved beam

In this example, we will create longitudinal bottom bars for a curved concrete beam.

1. Create a curved concrete beam.
   a. On the **Concrete** tab, click **Beam**.
   b. Pick two points.
   c. Double-click the beam to modify its properties.
   d. Set the radius and number of segments, and then click **Modify**.
2. Create longitudinal bars at the bottom face of the beam.
   a. On the **Rebar** tab, click **Longitudinal**.
   b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
   c. Click the middle mouse button to create the rebar set.

Tekla Structures creates the curved longitudinal bars according to the beam geometry. For example:



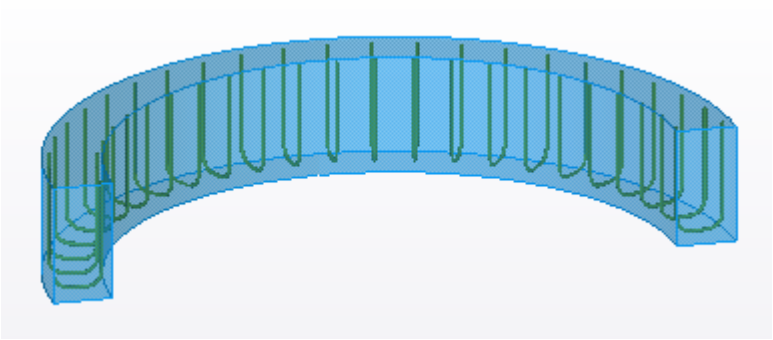
The rebar set has three guidelines: one at each end of the beam and one in the midpoint of the beam.

Create crossing bars for a curved polybeam

In this example, we will create crossing U-shaped stirrups for a curved concrete polybeam.

1. Create a concrete polybeam with curved segments.
   a. On the **Concrete** tab, click **Beam** --> **Polybeam**.
   b. Pick at least three points you want the beam to go through, and then click the middle mouse button.
   c. Select the polybeam.
   d. Select the handle in a polybeam corner, and then select the **Arc point** chamfer type on the contextual toolbar.
2. Create crossing bars that follow the bottom and side faces of the beam.
   a. On the **Rebar** tab, click **Crossing**.
   b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
   c. In the selected cross section, hold down **Ctrl** and click the top bar leg to unselect it.
   d. Click the middle mouse button to create the rebar set.

Tekla Structures creates the crossing bars radially according to the beam geometry. For example:



The rebar set guideline is a polyline with three points, and the midpoint has the **Arc point** chamfer.

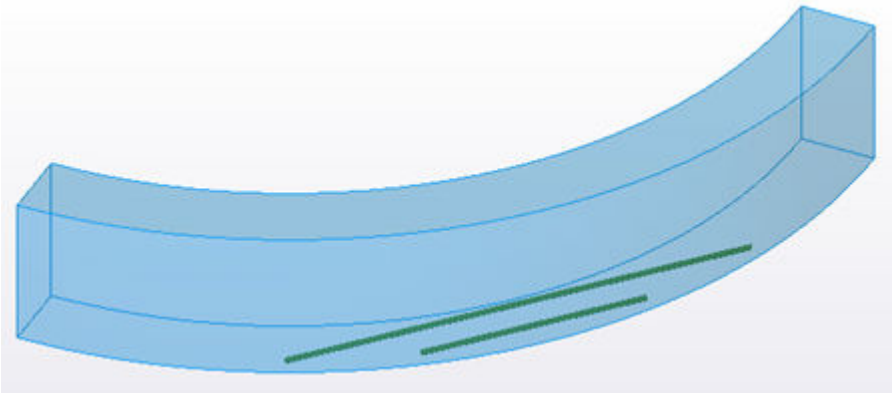
Create reinforcing bars for a spiral beam

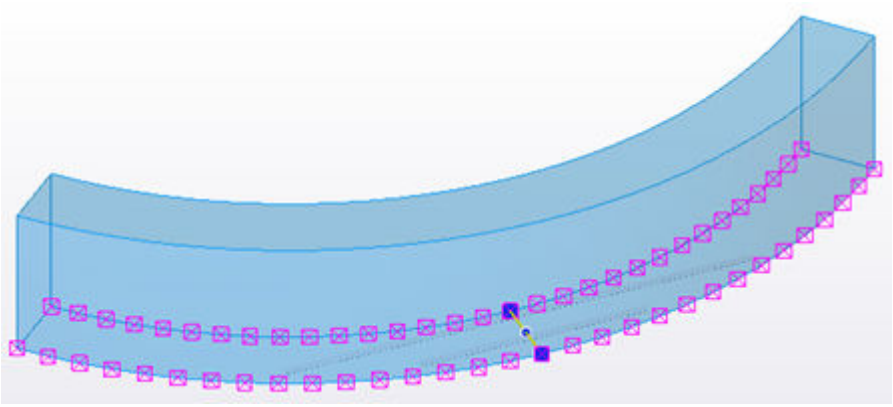
In this example, we will reinforce a spiral beam whose total rise is zero. We will create longitudinal bottom bars and crossing U-shaped stirrups.

You can use the following manual method also for more complex concrete objects that have been imported and that cannot be reinforced automatically.

1. Create a flat spiral concrete beam.
   a. On the **Concrete** tab, click **Beam** --> **Spiral beam**.
   b. Pick the start point of the beam.
   c. Pick a point to indicate the beam's center of curvature.
   d. Click the middle mouse button.
   e. Ensure that **Total rise** is 0.
2. Create longitudinal bars at the bottom face of the beam.
   a. On the **Rebar** tab, click **Longitudinal**.
   b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
   c. Click the middle mouse button to create the rebar set.

Tekla Structures creates a longitudinal rebar set with one guideline.

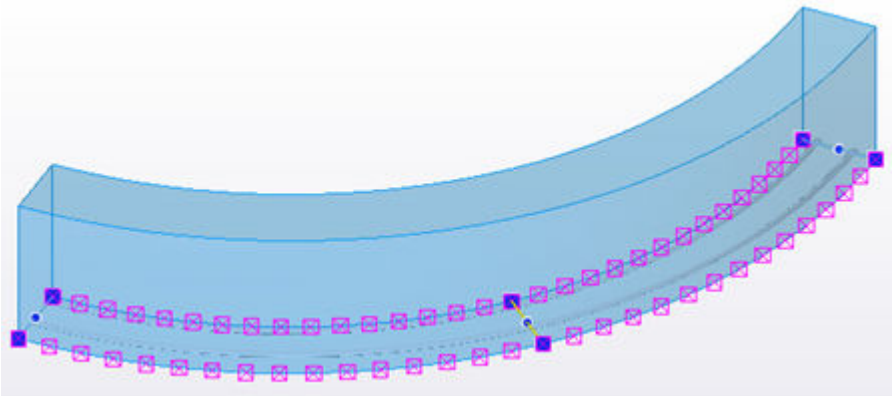


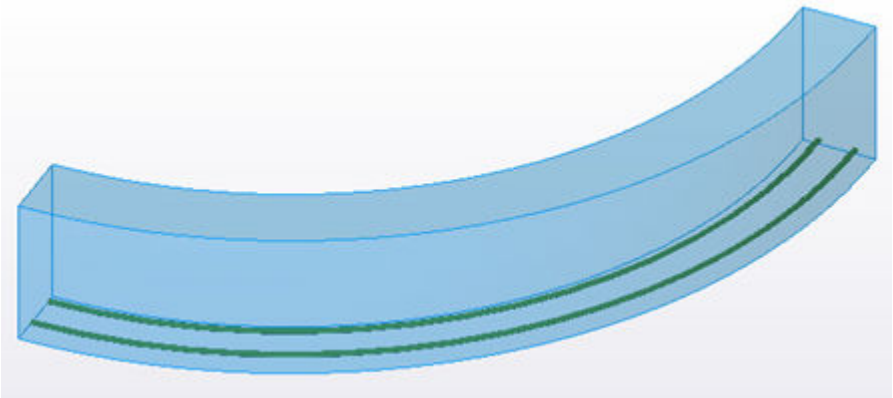


d. Press **Esc** to interrupt the command.

3. Modify the longitudinal rebar set by creating more guidelines.
   a. Select the rebar set.
   b. On the contextual toolbar, click **Add secondary guideline**.
   c. Ensure that you are in the single point picking mode (shown on the contextual toolbar).
   d. Pick the start point for a secondary guideline.
   e. Pick the start point for another secondary guideline.
   f. Press **Esc** to finish creating secondary guidelines.
   g. If needed, move the guidelines to desired locations by dragging them or their end point handles.

For example, you can move the primary guideline to the start of the beam, one of the secondary guidelines to the midpoint of the beam, and the other secondary guideline to the end of the beam.



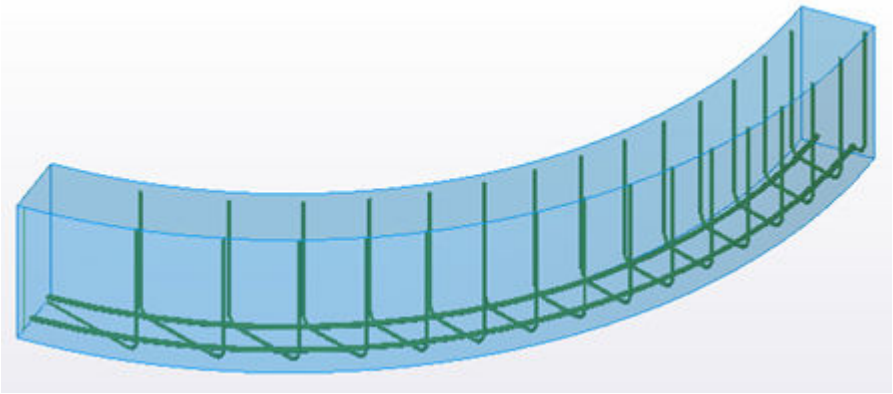


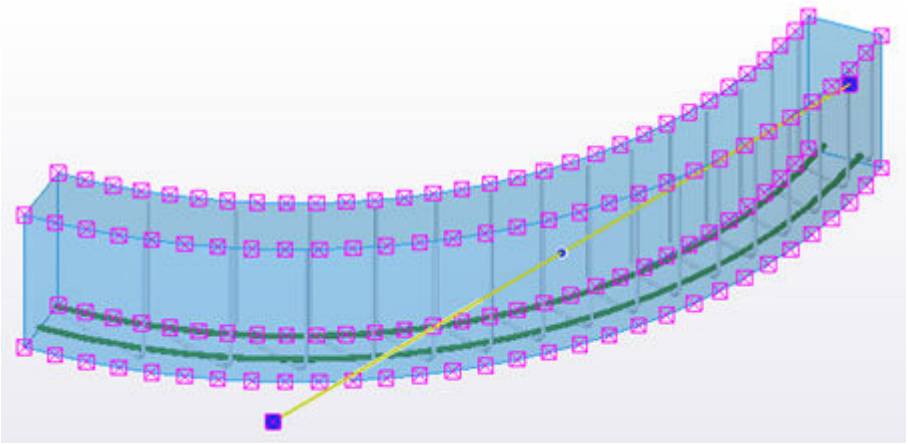
h. Modify the geometry (page 547) and properties (page 1073) of the guidelines as needed.

   The bars are constructed according to the locations and spacing settings of these three guidelines.

4. Create crossing bars that follow the bottom and side faces of the beam.
   a. On the **Rebar** tab, click **Crossing**.
   b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
   c. In the selected cross section, hold down **Ctrl** and click the top bar leg to unselect it.
   d. Click the middle mouse button to create the rebar set.

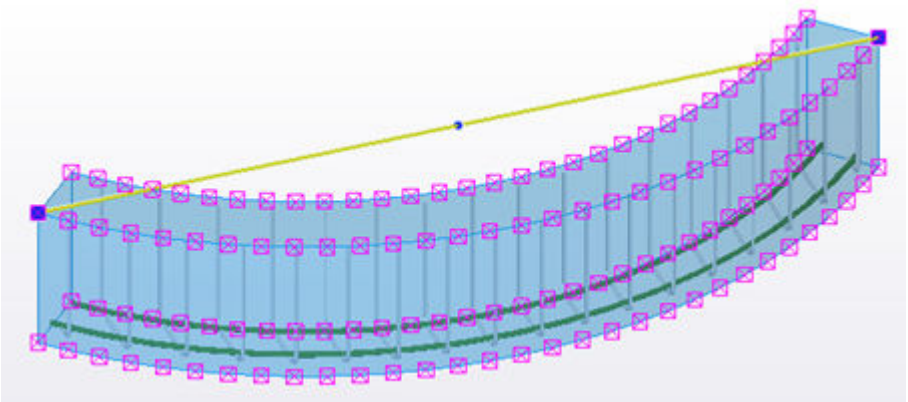
Tekla Structures creates a crossing rebar set with one guideline.



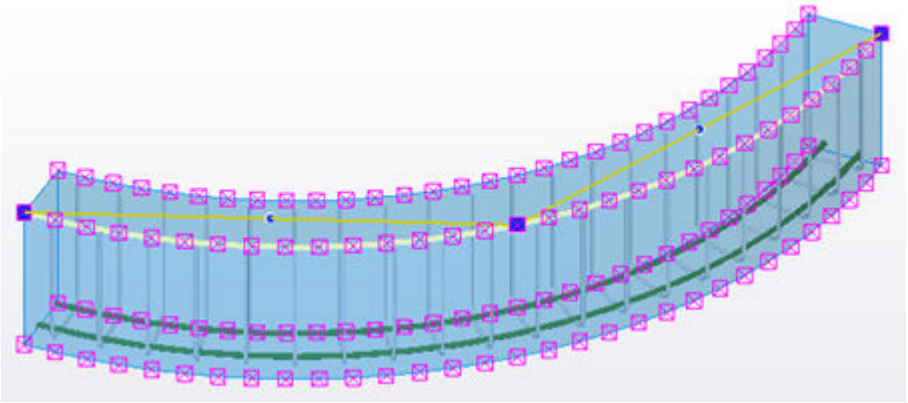


e. Press **Esc** to interrupt the command.

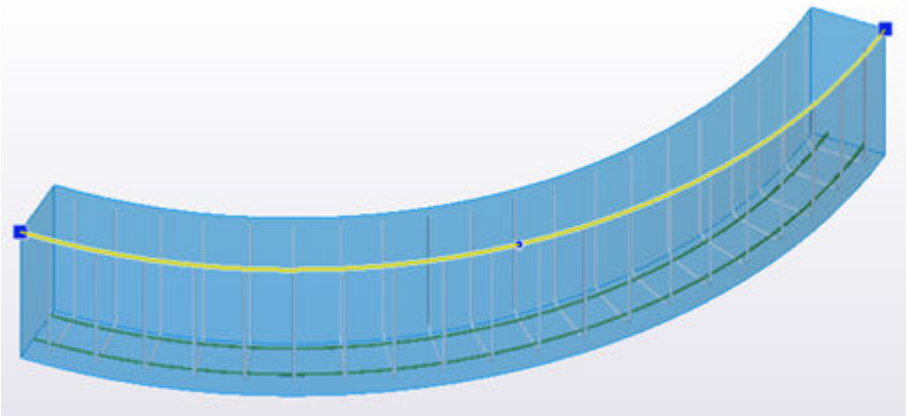
5. Modify the crossing rebar set by modifying the guideline.
   a. Select the rebar set to highlight the guideline.
   b. Drag the guideline end points to the beam ends.



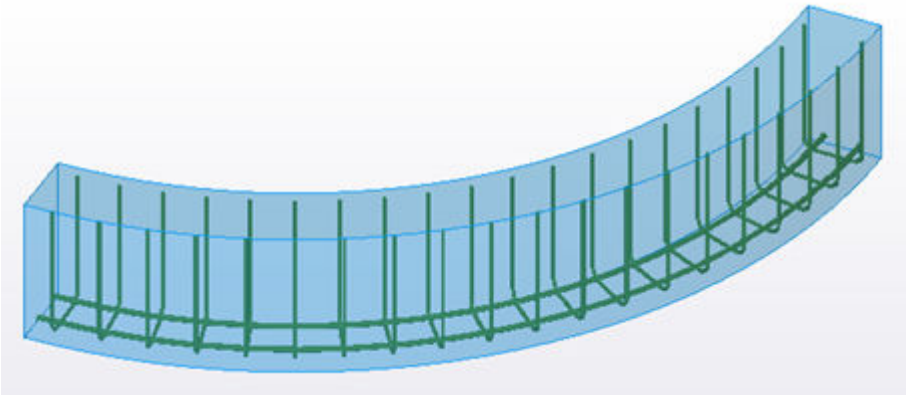
c. Drag the guideline midpoint to the beam midpoint. Ensure that the new guideline corner has the **Arc point** chamfer.



Alternatively, you can click on the contextual toolbar to make the guideline follow the leg face edges that are located between the guideline end points.



Tekla Structures arranges the crossing bars radially along the beam.



Limitations

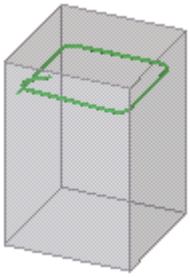
- If curved longitudinal bars have too small start and/or end offset values, the bars closest to leg face edges may be divided into small bar segments. To avoid this, increase the offset values.

Create a single reinforcing bar

1. On the **Rebar** tab, click  **Single bar**.

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Single bar** command to open the **Single rebar** properties.
2. Select the part to reinforce.
3. Pick the bar start point.
4. Pick the other bar reference points to set the bar shape.
5. Click the middle mouse button to finish picking.

   Tekla Structures attaches the bar to this part.



6. If you want to modify the reinforcement, do one of the following:

   - Use direct modification (page 572). Ensure that the  **Direct modification** switch is active.
   - Double-click the reinforcement to open the **Single rebar** properties and modify the properties (page 1061).

See also

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a reinforcing bar group (page 520)

Create a reinforcing bar group

A reinforcing bar group includes several identical, or very similar, reinforcing bars. Tekla Structures always treats these bars as a group, modifies them in the same way, deletes them all at the same time, and so on. You first define the shape of a single bar, then the direction in which Tekla Structures distributes the bars.

NOTE If you do not want to manually define the bar shape, use Rebar shape catalog (page 522) and its predefined reinforcement shapes instead.

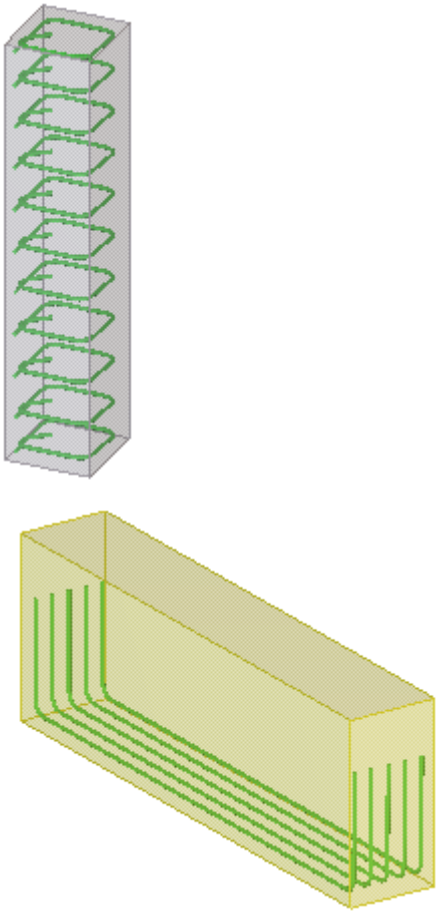
1. On the **Rebar** tab, click .

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the command button to open the **Rebar group** properties.
2. Select the part to reinforce.

   Tekla Structures attaches the bar group to this part.
3. Pick the bar start point.
4. Pick the other bar reference points.

   These points define the plane of the first bar and the shape of a single bar in the group.
5. Click the middle mouse button to finish picking.
6. Pick the start point of the bar group.
7. Pick the end point of the bar group.

The start and end points indicate the distribution length and direction of the bars. Usually the distribution length of the bars is perpendicular to the plane so that the cover thickness on the sides can be defined.



8. If you want to modify the reinforcement, do one of the following:

   - Use direct modification (page 572). Ensure that the  **Direct modification** switch is active.
   - Double-click the reinforcement to open the **Rebar group** properties and modify the properties (page 1061).

See also

Create a curved reinforcing bar group (page 529)

Create a circular reinforcing bar group (page 531)

Create a tapered or spiral reinforcing bar group (page 533)

Create a reinforcing bar group using Rebar shape catalog

A reinforcing bar group includes several identical, or very similar, reinforcing bars. You can create a reinforcing bar group by selecting a predefined reinforcement shape from **Rebar shape catalog**. The predefined shapes in **Rebar shape catalog** are based on the shapes that have been defined in **Rebar shape manager** and saved in the `RebarShapeRules.xml` file.

Rebar shape catalog does not work with tapered reinforcing bar groups (page 533) or with 3D bar shapes.

NOTE If you do not want to use the predefined shapes but want to manually define the bar shape, use the Bar group (page 520) command instead.

---

1. On the **Rebar** tab, click **Bar group** --> **Rebar shape catalog**.

   The **Rebar Shape Catalog** dialog opens.

2. Select one of the predefined shapes from the tree view on the left.

   You can add frequently used shapes to the tree view (page 525), or delete the shapes that you do not need.

   If you select an existing reinforcement in the model and click the **Get** button, the properties of that reinforcement are displayed in the **Rebar Shape Catalog** dialog.

3. If needed, modify the bar properties.

   - To set a **Length of leg** value, click a leg in the preview of the shape.

     If you do not enter a **Length of leg** value, the leg length is calculated automatically according to the dimensions of the concrete part.

   - To set a **Bending angle** value for a bend that is not 90 degrees, click one of the legs next to the bend.

   - For circular, polygonal and spiral reinforcement, you can enter **Circle diameter** and **Overlap distance** values.

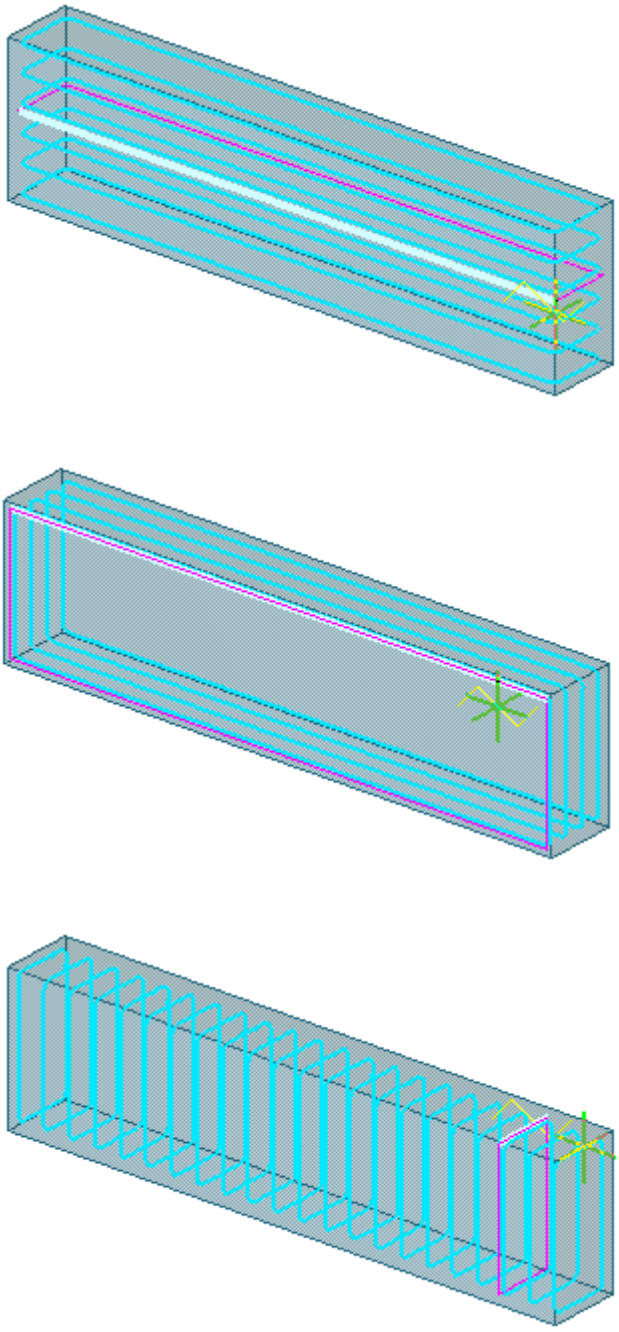
   The hook properties are visible only if you have set the advanced option XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION to `FALSE` in **File menu** --> **Settings** --> **Advanced Options** --> **Concrete Detailing**.

4. If needed, set the reference point of the reinforcement (page 525) to start, middle, or end by double-clicking the different legs or hooks in the preview of the shape.

5. Click **OK**.

6. In the model, place the mouse pointer over a part face or edge.

A preview showing the placing and dimensions of the reinforcement is displayed.



7. Based on the preview, select a placing for the reinforcing bar group and click the left mouse button.

   Tekla Structures creates the reinforcement.

8. If you want to modify the reinforcement, do one of the following:

- Use direct modification (page 572). Ensure that the  **Direct modification** switch is active.
- Double-click the reinforcement to open the reinforcing bar group properties, and modify the properties (page 1061).

See also

Create a reinforcing bar group (page 520)

Create a rebar set using Rebar shape placing tool (page 507)

Add more reinforcement shapes to the tree view in Rebar shape catalog

You can modify the tree view in **Rebar shape catalog** by adding frequently used shapes to the tree, or deleting the shapes that you do not need.

1. On the **Rebar** tab, click **Bar group** --> **Rebar shape catalog**.

   The **Rebar Shape Catalog** dialog opens.
2. Click **Organize catalog**.
3. Create a new category folder by clicking .
4. Drag and drop the selected shapes to the folder.

   If multiple shapes have the same shape code and you drag them to the categories, the shape codes get a suffix **(1)**, **(2)**, and so on. You can rename the shapes as you wish by clicking the name twice and entering a new name or suffix, for example, **(a)**, **(b)**.

   When the shapes are listed in a report, they all get the same shape code.
5. Change the name of the folder in a similar way, if needed.
6. To remove a shape from a category, select the shape and click .
7. Click **OK**.

See also

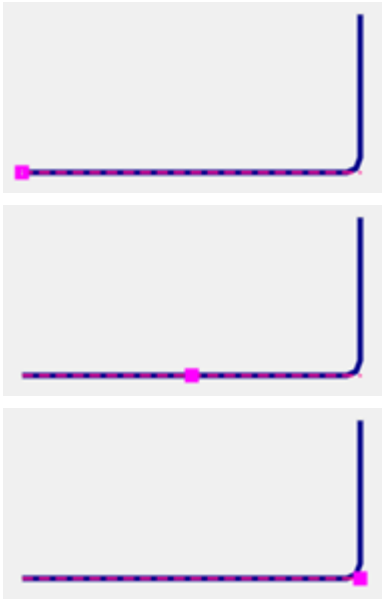
Create a reinforcing bar group using Rebar shape catalog (page 522)

Set the reinforcement reference point in Rebar shape catalog

When you use **Rebar shape catalog** and select a shape, you can set the reference point to the start, middle or end of the reinforcing bar leg. When you create the reinforcement in the model, you can move the reinforcement to a new location by dragging the reference point. This is useful, for example, when the reinforcing bar legs are of certain length and you want to aim the reference point, for example, to the middle of a part edge. You can also move the reference point of circular reinforcement shapes.

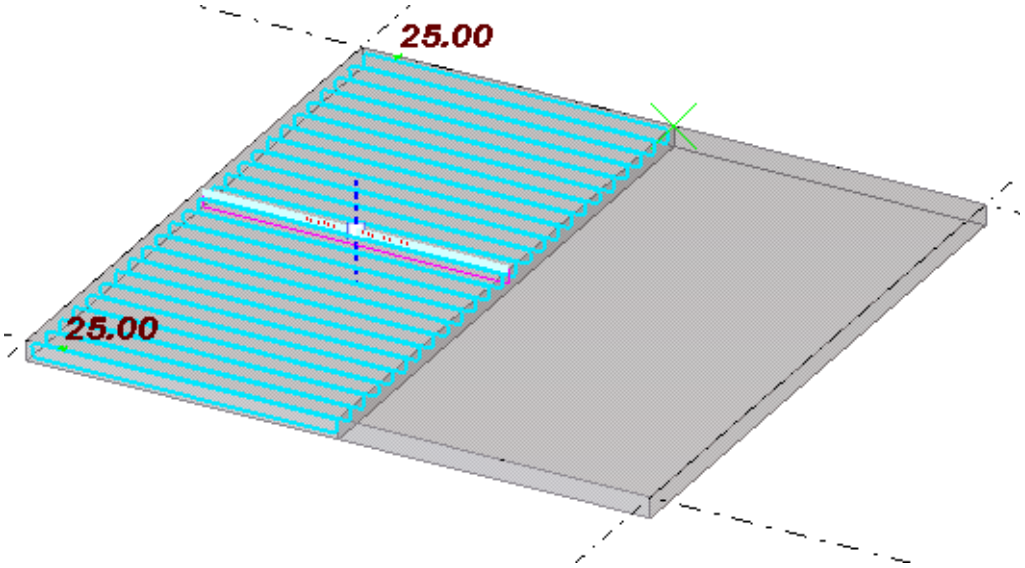
1. On the **Rebar** tab, click **Bar group** --> **Rebar shape catalog**.

   The **Rebar Shape Catalog** dialog opens.
2. Select a reinforcement shape.
3. Set the reference point to the desired location (start, middle, end) by double-clicking the position in the preview of the shape.

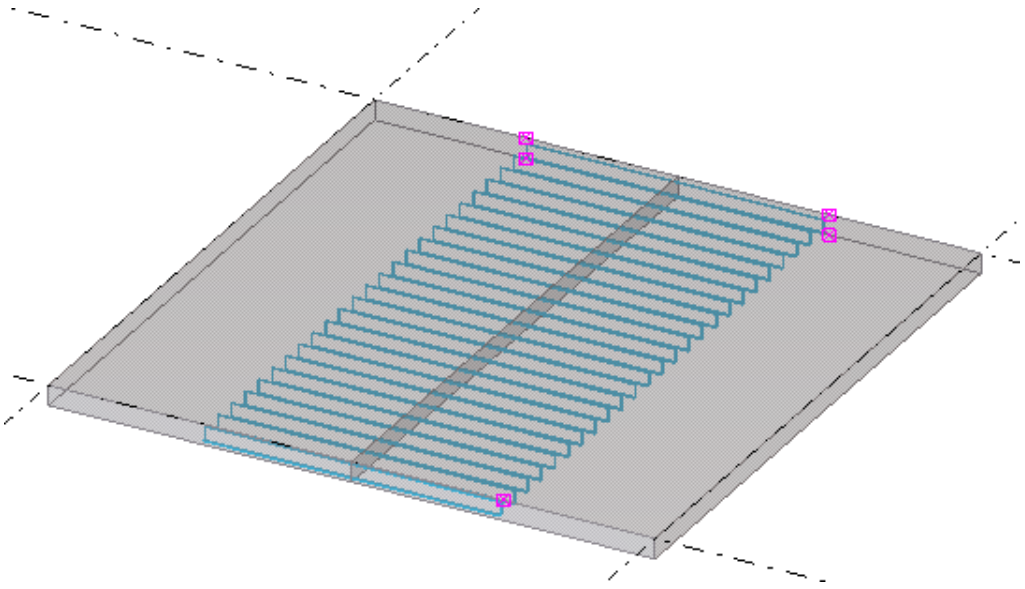


4. If needed, modify the bar properties.
5. Click **Apply** or **OK**.
6. In the model, place the mouse pointer over a part face or edge.
7. Based on the preview, select the desired placing and hold down the **Alt** key and click the left mouse button.

The reference point is displayed.



8. Move the reinforcement to a new location by dragging the reference point.
9. Click the middle mouse button to create the reinforcement.



NOTE For circular reinforcement you can set the reference point to the center line as follows:

a. Place the mouse pointer over a column edge to have the reinforcement oriented correctly.

b. Hold down the **Alt** key and click the left mouse button.

c. Drag the reference point and hold down **Shift** key to snap to the center of column.

d. Click the middle mouse button to create the reinforcement.

See also

Create a reinforcing bar group using Rebar shape catalog (page 522)

Reinforce pour objects using Rebar shape catalog

You can reinforce pour objects in pour views using **Rebar shape catalog**.

NOTE Rebar sets (page 494) and **Rebar shape catalog** are the methods to reinforce pour objects in pour views. If you want to use other reinforcement commands, such as Bar group (page 520), or reinforcement components, you need to reinforce single parts in part views. All reinforcement are visible both in part views and in pour views.

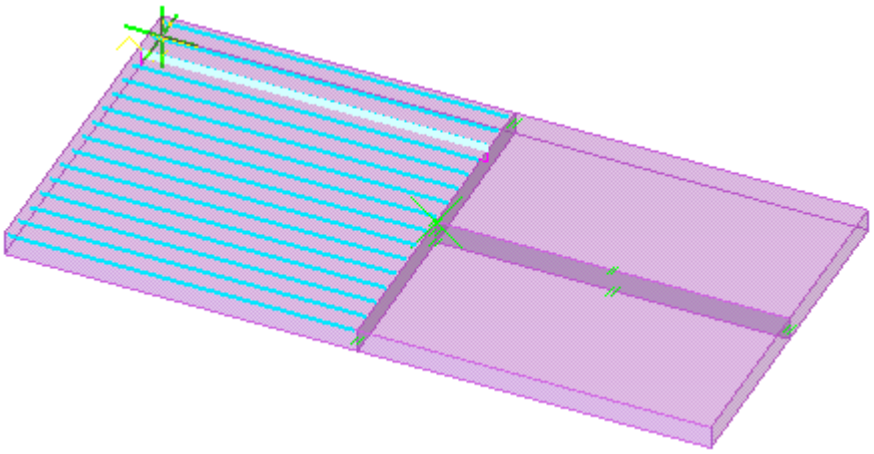
When you reinforce pour objects using **Rebar shape catalog**:

- The reinforcement is attached to the reinforced part, not to the pour object.
- The reinforcement geometry is defined in accordance with the pour object geometry even though the reinforcement is attached to a part. For example, pour breaks can limit the length of reinforcing bars.
- In reports the reinforcement information is listed according to the part, not to the pour object.

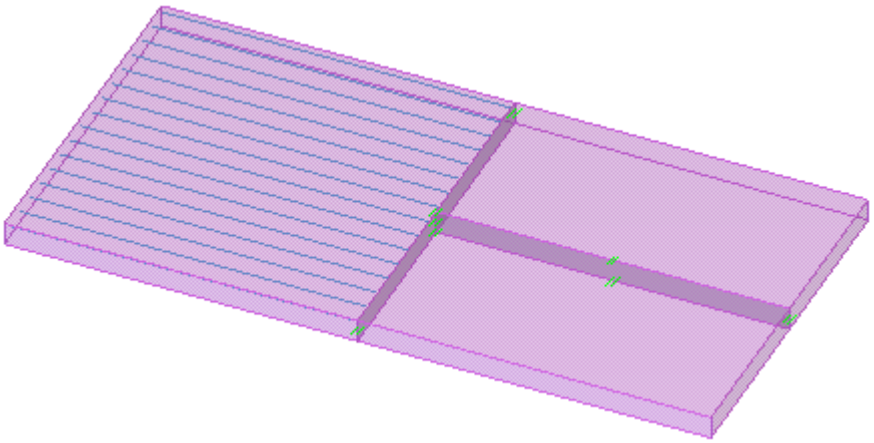
Before you start, create concrete parts whose cast unit type is **Cast in place**. Tekla Structures automatically forms pour objects of them.

1. Ensure that you are using a pour view. If not, click **Pour view** on the **Concrete** tab.
2. If needed, create pour breaks by selecting any of the **Pour break** commands on the **Concrete** tab:
   - **Single point**
   - **Two points**
   - **Multiple points**
3. To insert a reinforcement to a pour object, on the **Rebar** tab, click **Bar group** --> **Rebar shape catalog**.

   The **Rebar Shape Catalog** dialog opens.
4. Select a shape from the tree view on the left and modify the properties, if needed.
5. Click **OK**.
6. In the model, place the mouse pointer over a face or an edge of a pour object.



7. Based on the preview, select a placing for the reinforcement and click the left mouse button to create the reinforcement.



See also

Create a reinforcing bar group using Rebar shape catalog (page 522)

Manage pours (page 468)

Create a curved reinforcing bar group

You can reinforce curved segments in a concrete beam or a curved wall.

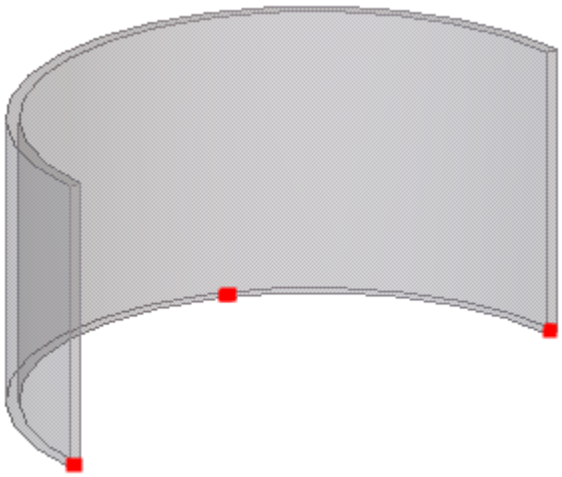
1. On the **Rebar** tab, click **Bar group** --> **Curved group**.

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Curved group** command to open the **Curved bar** properties.

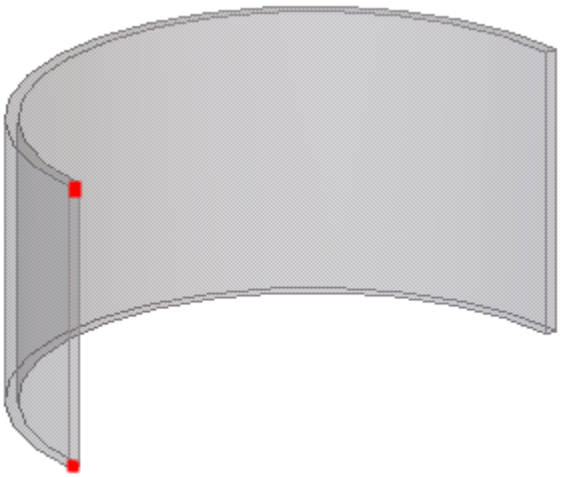
2. Select the part to reinforce.

   Tekla Structures attaches the bar group to this part.

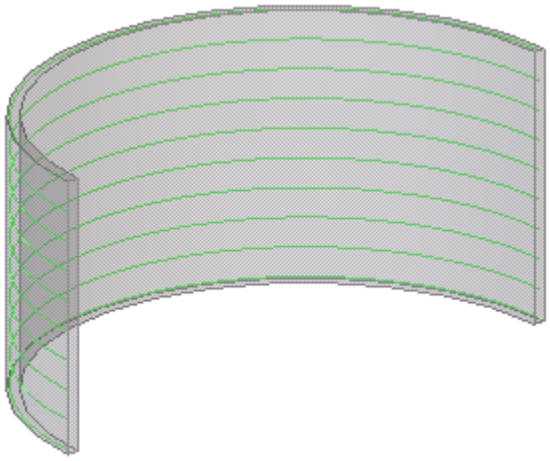
3. Pick three points on an arc to define the curve.



4. Pick two points to indicate the distribution direction of the bars.



Tekla Structures creates a group of curved reinforcing bars.



5. If you want to change the curved reinforcing bar group properties:
   a. Double-click the curved reinforcing bar group to open the **Curved bar** properties.
   b. Modify the properties (page 1061).
   c. Click **Modify**.

See also

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a reinforcing bar group (page 520)

Create a circular reinforcing bar group (page 531)

Create a tapered or spiral reinforcing bar group (page 533)

Modify reinforcement (page 544)

Create a circular reinforcing bar group

You can reinforce round circular columns.

1. On the **Rebar** tab, click **Bar group** --> **Circular group**.

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Circular group** command to open the **Circular rebar** properties. For example, to create a circular spiral bar, set **Rebar group type** to **Spiral**.
2. Select the part to reinforce.

   Tekla Structures attaches the bar group to this part.
3. Pick three points on the outer contour of the concrete part to define the circular bars.

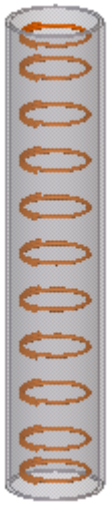
The radius is automatically calculated from these three points.



4. Pick two points to indicate the distribution direction of the bars.



Tekla Structures creates a group of circular reinforcing bars.



NOTE If you want to modify the splice length of the round stirrups, enter negative values in the **Start** and **End** boxes in the **Circular rebar** properties.

5. If you want to change the circular reinforcing bar group properties:
   a. Double-click the circular reinforcing bar group to open the **Circular rebar** properties.
   b. Modify the properties (page 1061).
   c. Click **Modify**.

See also

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a reinforcing bar group (page 520)

Create a curved reinforcing bar group (page 529)

Create a tapered or spiral reinforcing bar group (page 533)

Modify reinforcement (page 544)

Create a tapered or spiral reinforcing bar group

For example for rectangular concrete parts with variable cross sections, you can create tapered reinforcing bar groups by specifying the bar shape at each cross section. For concrete parts with a uniform cross section, you can also create spiral reinforcing bar groups, and it is sufficient to pick two points to define the distribution area of the bar group. If the part shape is not rectangular, an alternative bar shape can be defined.

Use the **Rebar group type** list in the reinforcing bar group properties to select or change the bar group type.

To create a circular spiral bar group (page 531), use the **Bar group** --> **Circular group** ribbon command and the **Circular rebar** properties instead.

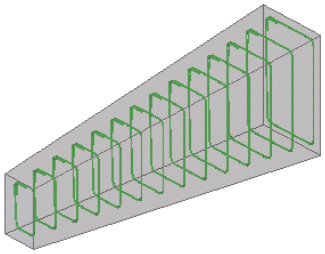
1. On the **Rebar** tab, hold down **Shift** and click .

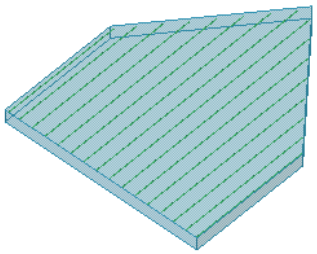
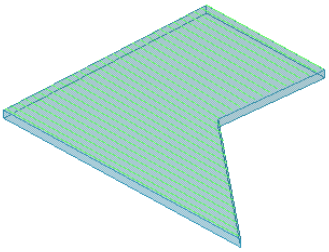
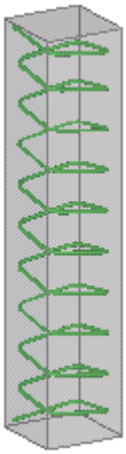
   The **Rebar group** properties open.
2. In the **General** section, select a tapered or spiral option from the **Rebar group type** list.
3. If needed, enter or modify the other bar properties.
4. Select the part to reinforce.

   Tekla Structures attaches the bar group to the part.
5. Pick points to define the shape of the bar at the first cross section.
6. Click the middle mouse button to finish picking.
7. Depending on the bar group type and the number of cross sections, do one of the following:
   - At the second and each subsequent cross section for a tapered bar group, pick points to define the bar shape. Then click the middle mouse button to finish picking at the cross section.
   - For a spiral bar group, pick two points to define the distribution area of the bar group.

Tekla Structures creates the reinforcement.

Reinforcing bar group types

Option	Description	Example
 Normal	Not tapered. Pick two points to define the distribution area of the bar group.	
 Tapered	One bar dimension changes linearly in the group.	

Option	Description	Example
 Tapered ridge	One bar dimension changes linearly in the group. The dimension is longest in the middle of the group.	
 Tapered curved	One bar dimension changes along a curve. The dimension is longest in the middle of the group.	
 Tapered N	One bar dimension changes linearly between N cross sections. Enter the number of cross sections in the **Number of cross sections** box.	
 Spiral	The reinforcing bars rise in a polygonal or circular shape along the longitudinal axis of the part.	

See also

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a reinforcing bar group (page 520)

Reinforcing bar and bar group properties (page 1061)

Modify a single reinforcing bar, bar group, or mesh (page 572)

Create a reinforcement mesh

You can create a reinforcement mesh that consists of two perpendicular bar groups. Tekla Structures treats mesh bars as one unit but distinguishes the main and crossing bars.

The reinforcement mesh can be rectangular, polygonal, or bent. You can also create a customized reinforcement mesh.

NOTE You cannot change the mesh type once the mesh has been created.

Create a rectangular reinforcement mesh

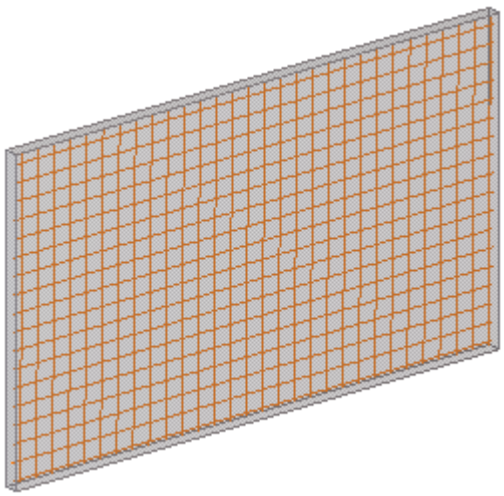
1. On the **Rebar** tab, hold down **Shift** and click **Mesh**.

   The **Rebar mesh** properties open in the property pane.
2. In the **Mesh type** list, select **Rectangle**.

   NOTE You cannot change the mesh type once the mesh has been created.
3. Select the part to reinforce.

   Tekla Structures attaches the mesh to this part.
4. Pick the start point of the mesh.
5. Pick a point to indicate the direction of the longitudinal bars.
6. Click the middle mouse button to finish picking.

   Tekla Structures creates the mesh parallel to the work plane, to the left of the points you picked.



7. If you want to modify the reinforcement mesh, do one of the following:

   - Use direct modification (page 572). Ensure that the  **Direct modification** switch is active.
   - Double-click the reinforcement to open the **Rebar mesh** properties and modify the properties (page 1064).

Create a polygonal reinforcement mesh

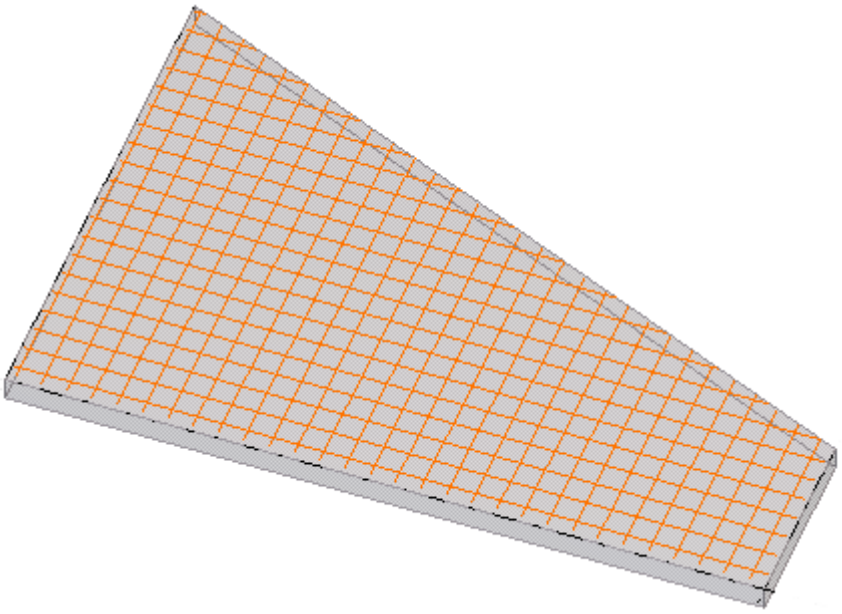
1. On the **Rebar** tab, hold down **Shift** and click  **Mesh**.

   The **Rebar mesh** properties open in the property pane.
2. In the **Mesh type** list, select **Polygon**.

   NOTE You cannot change the mesh type once the mesh has been created.
3. Select the part to reinforce.

   Tekla Structures attaches the mesh to this part.
4. Pick the start point of the mesh.
5. Pick the corner points of the mesh.
6. Click the middle mouse button to finish picking.
7. Pick a point to indicate the direction of the longitudinal bars.

   Tekla Structures creates the mesh.



8. If you want to modify the reinforcement, do one of the following:

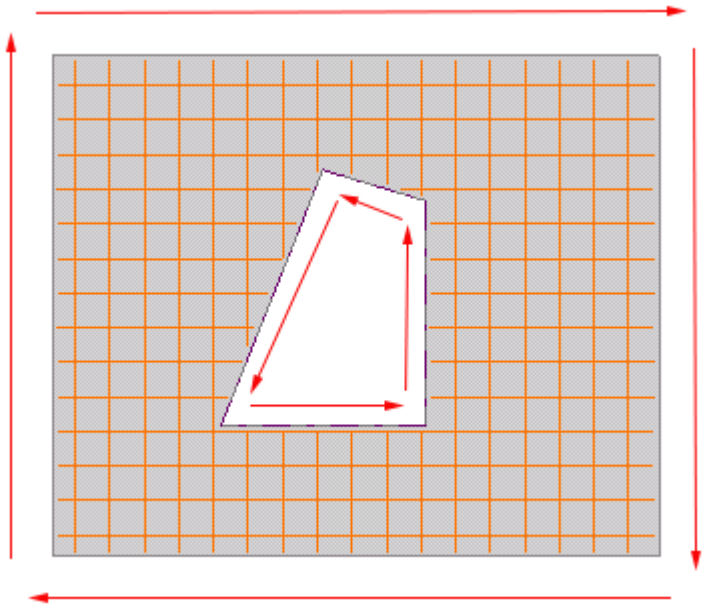
   - Use direct modification (page 572). Ensure that the  **Direct modification** switch is active.
   - Double-click the reinforcement to open the **Rebar mesh** properties and modify the properties (page 1064).

Reinforcement mesh with holes

If you want to reinforce a part that has holes, you need to pick the corner points of holes when you create the reinforcement.

1. Select the part to reinforce.
2. Pick the start point of the mesh.
3. Pick the corner points of the mesh.
4. Pick the corner points of the hole.

   Note that you need to pick the corner points of the hole in the opposite direction than the corner points in the mesh.



5. Click the middle mouse button to finish the picking.
6. Pick a point to indicate the direction of the longitudinal bars.

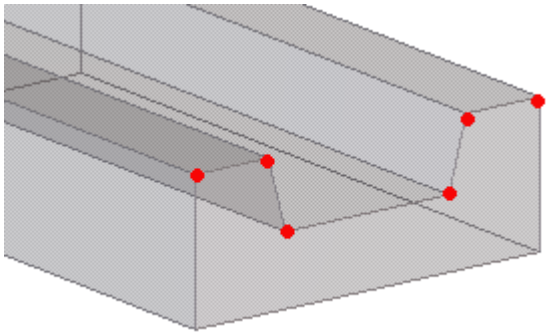
Create a bent reinforcement mesh

1. On the **Rebar** tab, hold down **Shift** and click  **Mesh**.

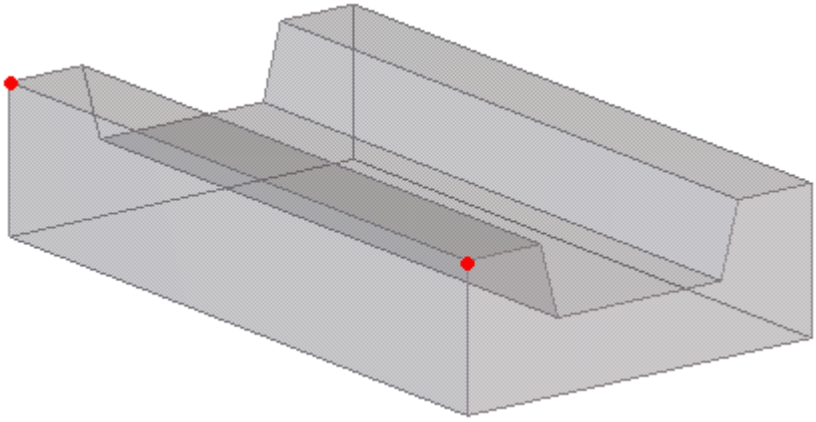
   The **Rebar mesh** properties open in the property pane.
2. In the **Mesh type** list, select **Bent**.

   NOTE You cannot change the mesh type once the mesh has been created.
3. Enter the bending radius.
4. Select the part to reinforce.

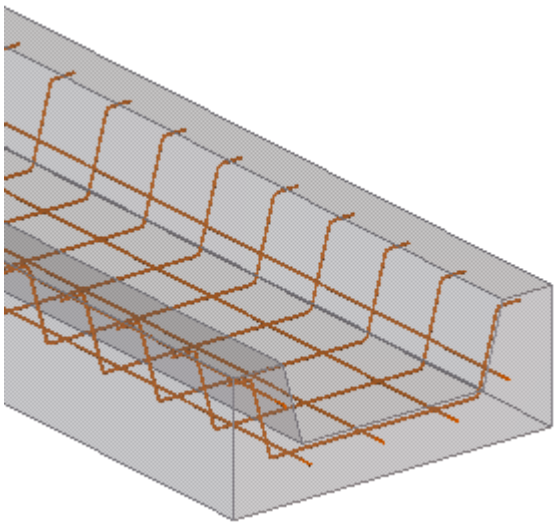
   Tekla Structures attaches the mesh to this part.
5. Pick three or more points to indicate the bending shape of the crossing bars.



6. Click the middle mouse button to finish picking.
7. Pick two points to indicate the length and direction of the longitudinal bars.



Tekla Structures creates the mesh.



8. If you want to modify the reinforcement mesh, do one of the following:

   - Use direct modification (page 572). Ensure that the  **Direct modification** switch is active.
   - Double-click the reinforcement to open the **Rebar mesh** properties and modify the properties (page 1064).

Create a customized reinforcement mesh

You can create a customized reinforcement mesh that consists of two perpendicular bar groups.

1. On the **Rebar** tab, hold down **Shift** and click  **Mesh**.

   The **Rebar mesh** properties open in the property pane.
2. In the **Layout** section, select the **Custom Mesh** option.
3. Enter an identifier for the mesh in the **Mesh** box.

   The default identifier is **Custom Mesh**.
4. Modify the other mesh properties (page 1065) as needed.
5. Select the part to reinforce.

   Tekla Structures attaches the mesh to this part.
6. Pick two points to indicate the direction of the longitudinal bars.
7. If you want to define the mesh plane, pick one more point.
8. Click the middle mouse button to finish picking.

9. If needed, you can save customized properties as property files, and load these properties later on when you create new meshes.

Create a reinforcement strand pattern

You can create prestressed straight or deflected strands for concrete parts.

NOTE To allow positioning of the strands, first create points to the part you are creating the strands for. On the **Edit** tab, click **Points** and select **On plane** to open the **Point Array** dialog. Define the point coordinates.

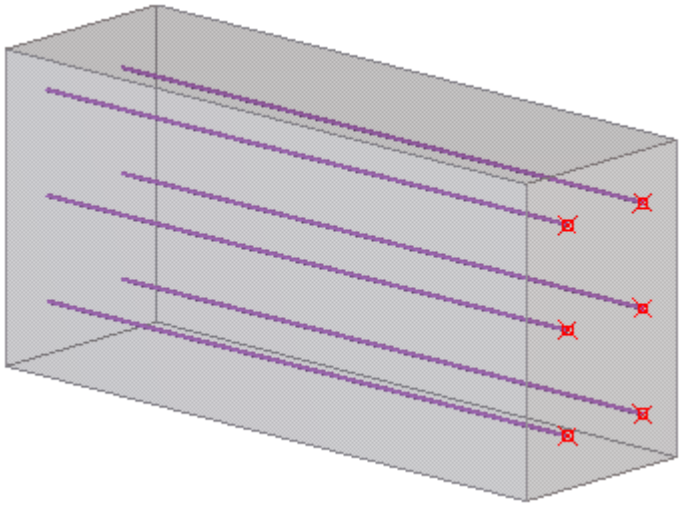
---

1. On the **Rebar** tab, click  **Strand**.

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Strand** command to open the **Strand pattern** properties.
2. Select the part you are creating strands for.
3. Pick each of the points that you are using to position the strands (for example, at the end of a part).

   The points you pick define the first cross section.
4. Click the middle mouse button to finish picking.
5. Pick points to position the strands.
   - If you create a single cross section, pick two points to define the length of the strands.
   - If you create two or more cross sections, for each cross section, pick points to indicate the strand positions. Pick the strand positions in the same order as for the first cross section.
6. Click the middle mouse button to finish picking.

Tekla Structures creates the strands.



7. If you want to change the strand properties:
   a. Double-click the strand pattern to open the **Strand pattern** properties.
   b. Modify the properties (page 1088).
   c. Click **Modify**.

See also

Debond reinforcement strands (page 542)

Debond reinforcement strands

1. Double-click the strand pattern in which you want to debond strands.

   The **Strand pattern** properties opens.
2. Click the **Debonding** button to open the debonding properties.
3. On the **Debonding** tab, click the **Add** button to create a new row in the table.
4. Enter the strand numbers in the **Debonded strands** field.

   The strand number is the selection order number of the strand.

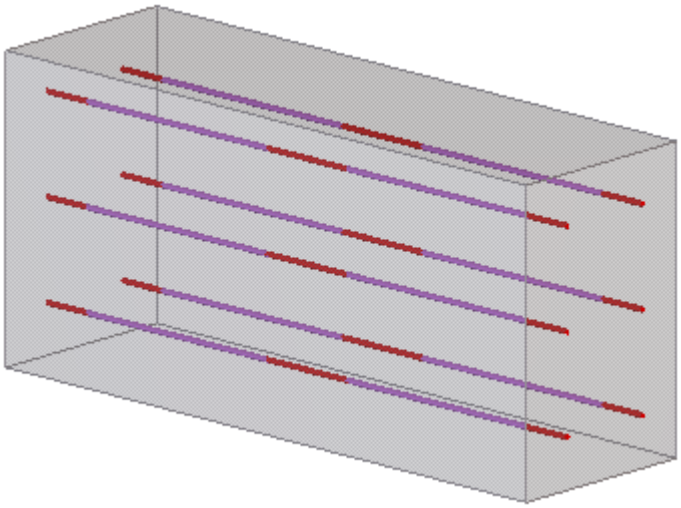
   - To set the same values for all the strands, enter all the strand numbers, separated by a space. For example, `1 2 3 4`.
   - To set separate values for each strand, click **Add** to add a new row, then enter the strand number in the **Debonded strands** field.

5. Define the debonded lengths.

   To set symmetrical lengths, select the **End lengths = start lengths** checkbox and only enter values in the **From start** or **Middle to start** fields.

6. Click **Modify**.

   Tekla Structures displays the debonded section of the strand in red.



See also

Create a reinforcement strand pattern (page 541)

Reinforcement strand properties (page 1088)

Create a reinforcement splice

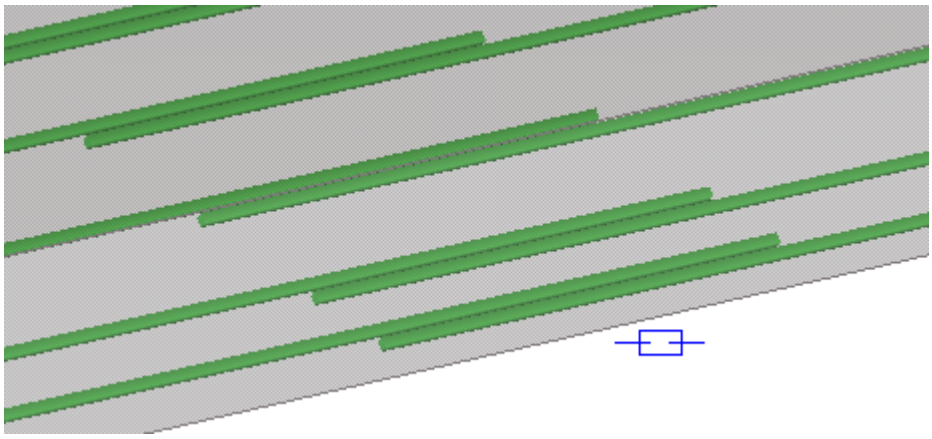
You can join reinforcing bars or reinforcing bar groups together with reinforcement splices. There can be a gap between the bars or groups.

1. On the **Rebar** tab, click **Edit** --> **Rebar splice**.

   If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Rebar splice** command to open the **Rebar splice** properties.

2. Select the first reinforcing bar or bar group.
3. Select the second reinforcing bar or bar group.

   Tekla Structures creates the splice. The splices have blue splice symbols  in the model.



4. If you want to change the splice properties:
   a. Double-click the splice to open the **Rebar splice** properties.
   b. Modify the properties.
   c. Click **Modify**.

Splice properties

Use the **Rebar splice** properties to view and modify the properties of splices. The file name extension of a saved splice property file is `.rsp`.

Option	Description
Joint type	Splice type. **Lap left** creates the lap to the direction of the first reinforcing bar or bar group selected, **Lap right** to the direction of the second. **Lap both** centers the lap between the bars or bar groups.
Lap length	Length of the lap joint.
Offset	Offset of the splice center point from the point where the bars originally met.
Bar positions	Select whether the lapping bars are on top of each other or parallel to each other.

See also

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a reinforcing bar group (page 520)

Split and splice a reinforcement (page 582)

2.8 Modify reinforcement

Once you have added reinforcement to your model, you can modify the shape of the reinforcement, for example. Tekla Structures includes several methods for the modification.

Rebar sets

When you modify rebar sets, you can use direct modification on the rebar set guidelines (page 545), leg faces (page 549), and modifiers (page 557). You can also split rebar sets (page 568) and bars in rebar sets (page 569). To prevent unwanted automatic changes in rebar sets, you can freeze rebar sets (page 571).

Single reinforcing bars, bar groups, and meshes

When you modify single reinforcing bars, bar groups, or meshes, you can use:

- direct modification (page 572)
- handles (page 584)
- grouping (page 580)
- combining (page 581)
- splitting (page 581)

See also

Use adaptivity to modify a reinforcement (page 591)

Attach a reinforcement object to a concrete part (page 593)

Split and splice a reinforcement (page 582)

Classify reinforcement to layers (page 595)

How to calculate the reinforcing bar length (page 599)

How to calculate the reinforcing bar leg length (page 602)

Modify a rebar set

You can modify rebar sets by changing the rebar set properties, by using the rebar set guidelines or leg faces, or by creating local rebar set modifiers. The guidelines, leg faces, and modifiers all have direct modification handles.

NOTE When you work with rebar sets, ensure that the  **Direct modification** switch is active.

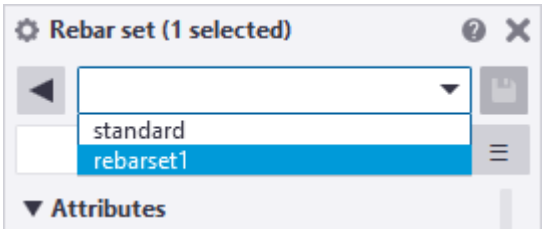
In case you open an existing model using a new version of Tekla Structures, always update the existing rebar sets first: on the **Rebar** tab, click **More** --> **Regenerate**.

See also Modify a rebar set using leg faces and leg surfaces (page 549) and Modify a rebar set locally using modifiers (page 557).

Modify the properties of a rebar set

You can change the properties of a rebar set in the property pane or on the contextual toolbar.

1. Double-click the rebar set that you want to modify.
2. If you want to use previously saved properties from a file, select the property file from the topmost list in the property pane:



3. Modify the rebar set properties (page 1068) in the property pane.
4. Click **Modify** to save the changes.
5. To save the properties for later use, enter a name for the property file in the topmost box in the property pane, and then click .

TIP Alternatively, you can modify the rebar set properties on the contextual toolbar.

Change the layer order of a rebar set

You can adjust the order of the bar layers when two or more rebar sets overlap.

By default, the layer order is based on the creation order of the rebar sets. Tekla Structures automatically places the bars that are created first closest to the concrete surface, and the bars created last will be the furthest.

TIP To check which rebar sets affect the bar layer placement of each other, click the down arrow next to  on the ribbon, and use the **Rebar set layers** command.

---

1. Select a rebar set.
2. On the contextual toolbar, check and adjust the layer order by using the following buttons:
   - Click  to move the bars to the outermost layer.
   - Click  to move the bars one layer outwards.
   - Click  to move the bars one layer inwards.
   - Click  to move the bars to the innermost layer.
   - Click  to check which rebar sets affect the selected rebar set, and which rebar sets are affected by the selected rebar set.

   Alternatively, you can enter a number in the **Layer number** box in the property pane, and then click **Modify** to save the changes.

   The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers.

   If you set the same layer number for several rebar sets, the bars will be placed on the same layer, and the bars may collide.
3. If needed, fine-tune the layer order at any individual leg face or leg surface (page 549) separately.

   These modifications override the default settings and the layer order settings of the entire rebar set.

Modify a rebar set using guidelines

The guidelines of a rebar set define the distribution direction of the bars. The spacing of the bars is also measured along the guidelines. You can modify the rebar set guidelines by using direct modification.

See also Resize and reshape model objects (page 119), Distribute bars in a rebar set (page 565), and Create a secondary guideline (page 562).

To show or hide (page 564) the guidelines when you select rebar sets in the model, go to the **Rebar** tab and click **Visibility** --> **Guidelines**. Alternatively, you can use the keyboard shortcut **Alt+2** or the advanced option XS_REBARSET_SHOW_GUIDELINES.

To modify a guideline, select a rebar set and do any of the following:

- To move a guideline, drag the line handle.
- To move a guideline point, drag the point handle .
- To add a new point to the start or end of a guideline:
  1. Select the start or end point of the guideline .
  2. Click  **Add new point** on the contextual toolbar.
  3. Pick a location for the new start or end point.
- To add an intermediate point to a guideline, drag a midpoint handle .
- To remove a point from a guideline, select the point and press **Delete**.
- To modify chamfers at intermediate corner points of a guideline:
  1. Select a corner point.
  2. Define the chamfer type and dimensions (page 420) on the contextual toolbar.
- To change the direction (page 563) of a guideline, select the guideline and click  **Swap ends** on the contextual toolbar.
- To make a guideline follow the leg face edges that are located between the guideline end points:
  1. Move the guideline end points to the leg face edges.
  2. If there are cuts at the edges, add intermediate points to the guideline, and drag the handles to the corners of the cuts.
  3. Double-click the guideline to open its properties in the property pane. Select **Yes** in the **Follow edges** list, and then click **Modify**.

     Alternatively, select the guideline and click  on the contextual toolbar.
- To adjust how rebar set bars follow a sharp or curved part corner:
  1. Select the guideline at the part corner.
  2. Click the chamfer type button on the contextual toolbar, and then select  **None** for a sharp corner or  **Arc point** for a curved corner to switch between the chamfer types.

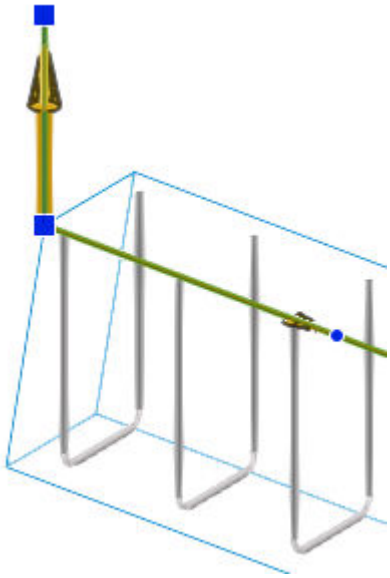
  Alternatively, select one or more part corner guidelines, go to the property pane, and select **None** or **Arc point** in the **Chamfer type** list.

Change the bar plane orientation of a rebar set

By default, Tekla Structures creates rebar set bars at planes that are perpendicular to the guidelines. In slanted or complex concrete structures, you may want to align the bar planes with the global z direction of the model, or some other way.

1. Select a rebar set.
2. On the contextual toolbar, click **Bar plane orientation**.

   Tekla Structures shows a yellow arrow with two blue direct modification handles, at the start point of the primary guideline. The bars become aligned with the global z direction.



3. To adjust the bar plane orientation some other way, drag the handles.

   For example, you can let the top handle snap to a certain location, such as a part corner, in the model.
4. If you want to revert to the original bar plane orientation, and to hide the yellow arrow, click again on the contextual toolbar.

Modify a rebar set using leg faces and leg surfaces

In addition to modifying an entire rebar set, you can make changes to any individual leg face or leg surface. Leg surfaces are especially useful when you reinforce curved or complex concrete structures.

NOTE If a leg face or leg surface is shown in red in the model, it means that the geometry of some bars in the rebar set is not valid. Check and fix the properties of the rebar set (page 1068) (for example, the bending radius) or the leg face (page 1075) or leg surface (page 1076).

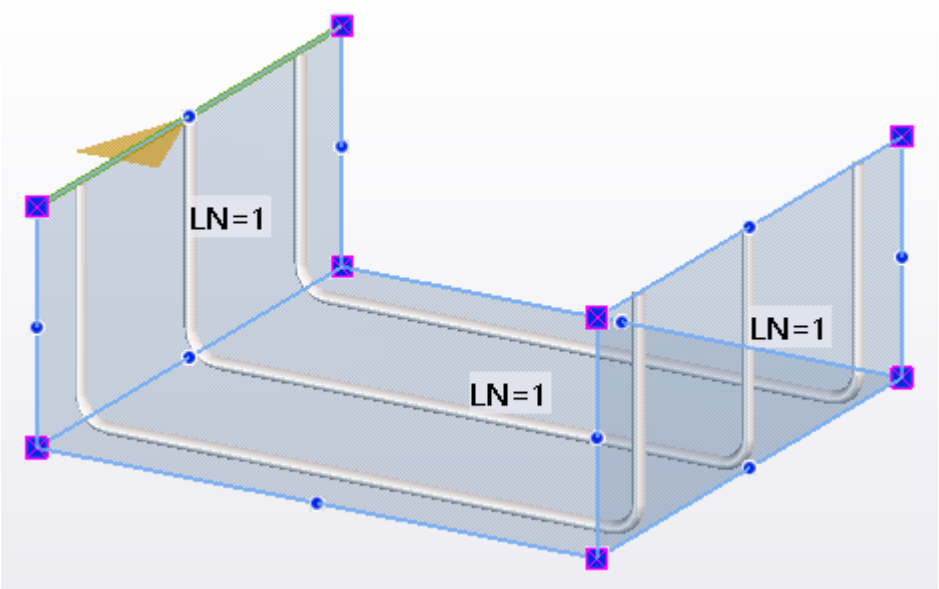
Show the leg faces and leg surfaces
To modify rebar sets by using leg faces, you first need to make the leg faces visible.

Tekla Structures also shows the leg surfaces. They automatically adapt to the geometry of the parts they are associated with. You can only modify leg surfaces by using their properties (page 1076) or contextual toolbar, not by using direct modification.

1. Ensure that the  **Direct modification** switch is active.
2. On the **Rebar** tab, click **Visibility** --> **Leg faces**.

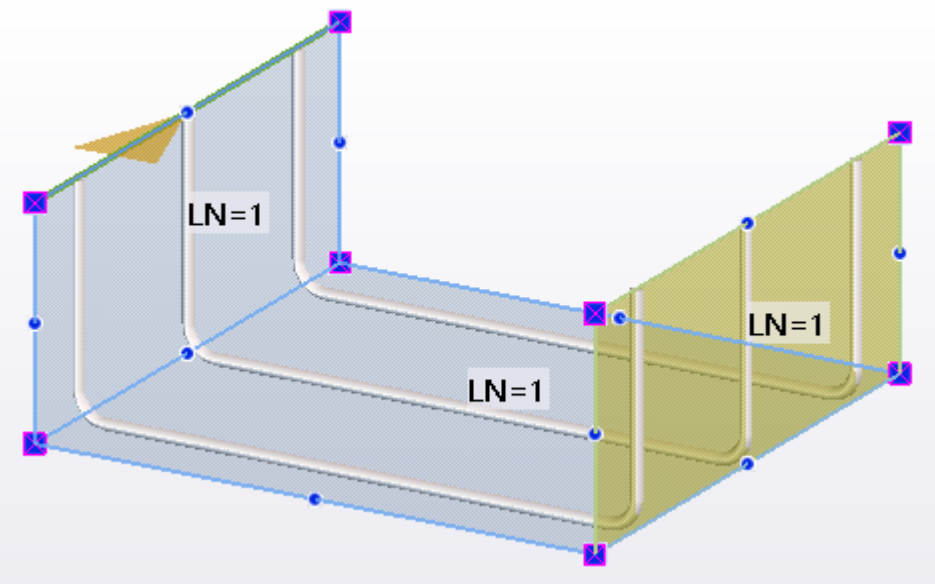
   Alternatively, you can use the keyboard shortcut **Alt+1**.
3. Select a rebar set.

   Tekla Structures shows the leg faces and leg surfaces. Tekla Structures also shows the bar layer numbers of the rebar set on each leg face or leg surface, for example LN=1.



4. Move the mouse pointer over a leg face or leg surface and click to select it.

Tekla Structures highlights the leg face or leg surface in yellow.



Modify the leg faces

You can use any of the following methods when you modify rebar set leg faces.

1. To move a leg face, drag it to a new location.

   The connected leg faces are extended or shortened, but the angle remains unchanged.

   If you hold down **Alt** when dragging, also the angle of the connected leg faces may change, depending on the dragging direction. The size of the dragged leg face remains the same.

   If you want to detach the leg face from the connected leg faces, hold down **Shift** when dragging.

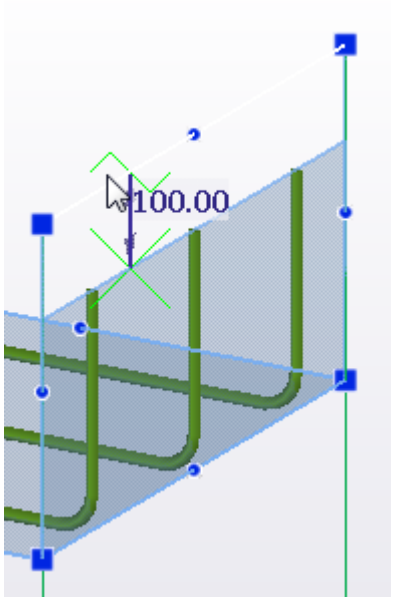
2. To move an edge of a leg face, drag the edge to a new location.

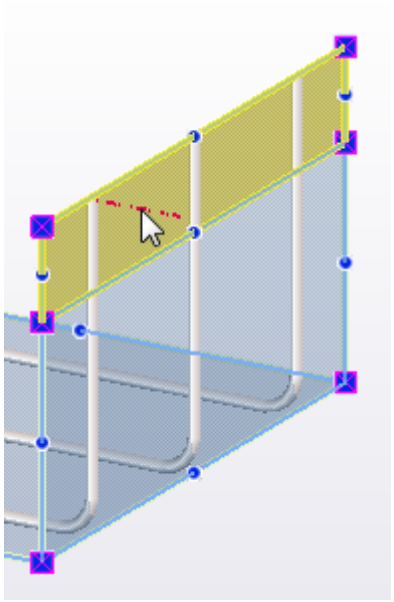
   The connected leg faces follow, if possible.

   If you hold down **Alt** when dragging, the edge is separated from the connected leg face.

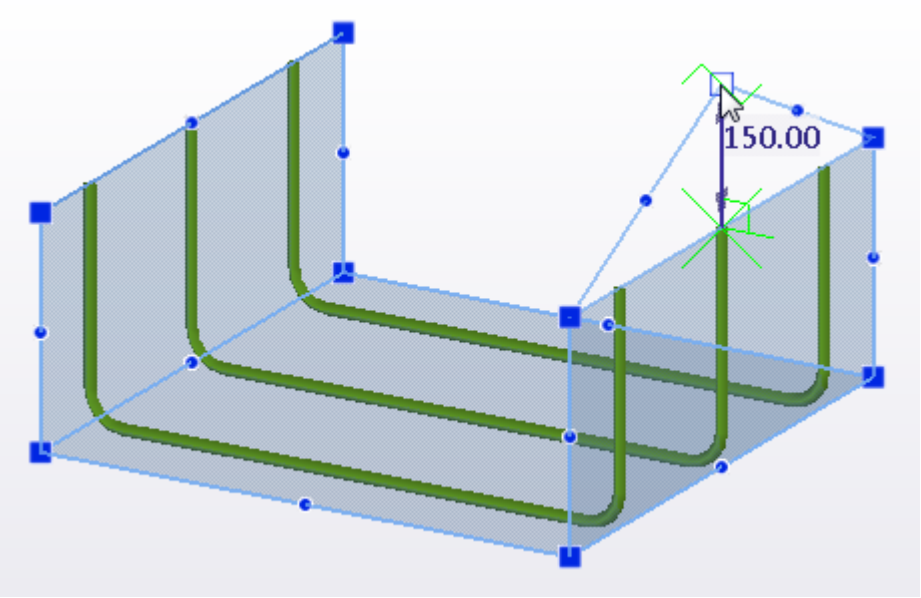
3. To create a parallel copy of a leg face, hold down **Ctrl** and drag the leg face.

4. To create a new, connected leg face, hold down **Ctrl** and drag an edge of the leg face.

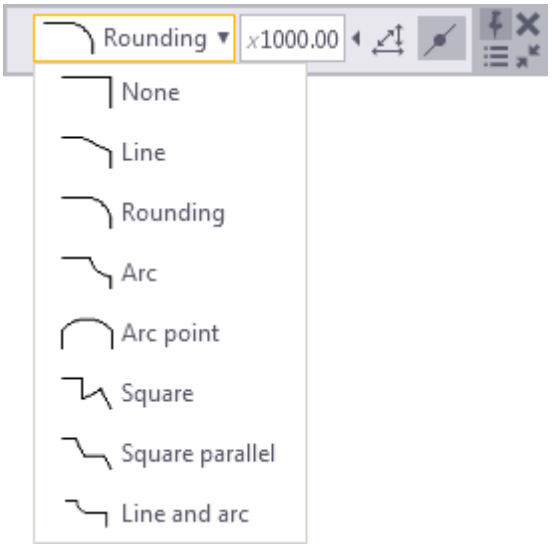


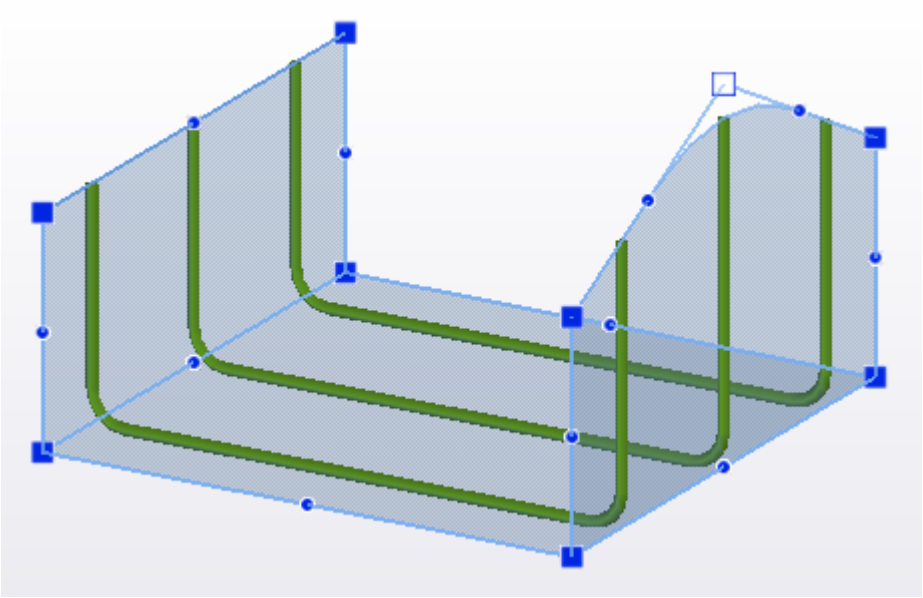


5. To add a new leg face at a part face or pour object face, ensure that the rebar set is selected, click **Add leg face** on the contextual tab on the ribbon, and then select the part face or pour object face.
6. To create a new leg face on the basis of its corner points, switch **Picking mode** to , click **Add leg face** on the contextual tab on the ribbon, and then pick points to indicate the leg face corners. Click the middle mouse button to finish picking.
7. To add a new corner point to a leg face, drag a midpoint handle.



8. To remove a corner point from a leg face, select the point and press **Delete**.
9. To modify a corner chamfer of a leg face, select the corner point, and then select the chamfer type (page 420) and enter the chamfer dimensions on the contextual toolbar.

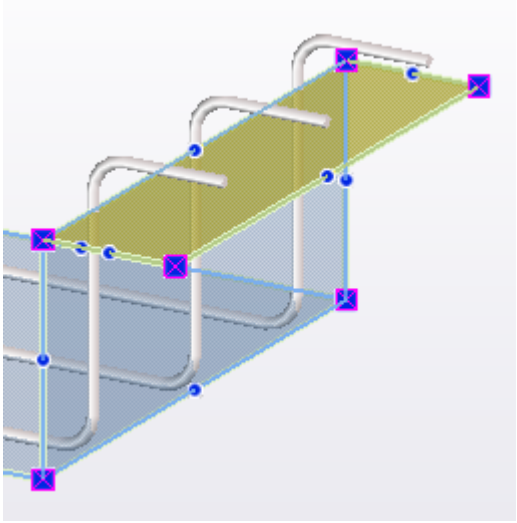


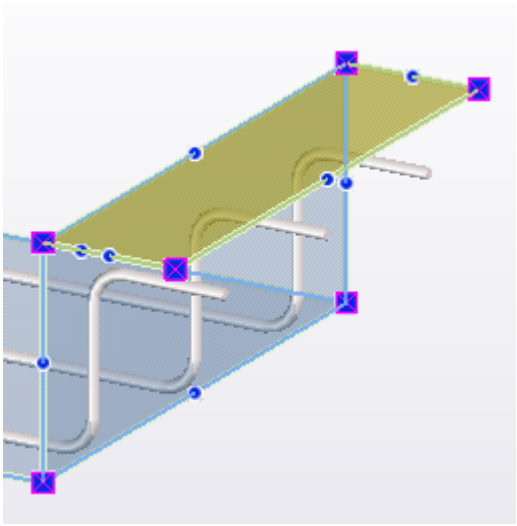


10. To remove a leg face, select it and press **Delete**.
11. To define an additional offset between a leg face and the bars, select the leg face and enter a value for **Additional offset** on the contextual toolbar, for example, 

.

    A negative value moves the bars outside the concrete.
12. To flip bars over to the other side of a leg face, select the leg face and click  **Flip bar side** on the contextual toolbar.





Note that after flipping, Tekla Structures searches for concrete on the other side of the leg face to create a concrete cover and apply the concrete cover settings. If there is no concrete, the concrete cover thickness will be zero.

13. To check and change the order of the bar layers at an individual leg face, select the leg face and adjust the layer order by using the following buttons on the contextual toolbar:

    - Click  to move the bars to the outermost layer.
    - Click  to move the bars one layer outwards.
    - Click  to move the bars one layer inwards.
    - Click  to move the bars to the innermost layer.

- Click  to check which rebar sets affect the selected rebar set, and which rebar sets are affected by the selected rebar set.

Alternatively, you can enter a number in the **Layer number** box in the property pane, and then click **Modify** to save the changes.

The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers.

These modifications override the layer order settings of the entire rebar set (page 545).

TIP You can modify leg face properties (page 1075) also in the property pane.

Modify the leg surfaces

Use these methods to modify rebar set leg surfaces.

1. To add a new leg surface at a part face, ensure that the rebar set is selected, click  **Add leg surface** on the contextual tab on the ribbon, and then select the part face.

   If you try to add a leg surface to a part face that already has a leg surface that belongs to the same rebar set, Tekla Structures does not add the new, duplicate leg surface.
2. To remove a leg surface, select it and press **Delete**.
3. To check and change the order of the bar layers at an individual leg surface, select the leg surface and adjust the layer order by using the following buttons on the contextual toolbar:

   - Click  to move the bars to the outermost layer.
   - Click  to move the bars one layer outwards.
   - Click  to move the bars one layer inwards.
   - Click  to move the bars to the innermost layer.
   - Click  to check which rebar sets affect the selected rebar set, and which rebar sets are affected by the selected rebar set.

   Alternatively, you can enter a number in the **Layer number** box.

   The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers.

   These modifications override the layer order settings of the entire rebar set (page 545).

4. To define an additional offset between a leg surface and the bars, select the leg surface and enter a value for **Additional offset** on the contextual toolbar.

   A negative value moves the bars outside the concrete.

5. To define whether the bars at a leg surface are cut by the openings in the concrete, select the leg surface and click the  **Create holes** button on the contextual toolbar.

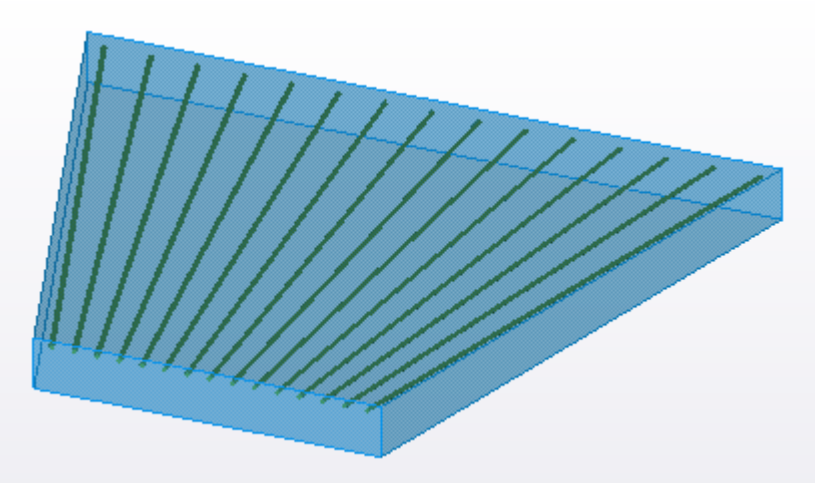
TIP You can modify leg surface properties (page 1076) also in the property pane. After modifying the properties of the selected leg surfaces, click **Modify** to save the changes.

Modify a rebar set locally using modifiers

You can use *modifiers* to modify a rebar set at certain locations only.

For example, you can create a local *property modifier* to only change the properties of certain bars in a rebar set, or you can create hooks or threading by adding an *end detail modifier*, or you can split rebar set bars into stock lengths by using *splitters*.

You can also create secondary guidelines for the rebar set. With a secondary guideline you can define a different spacing at the start and end of the rebar set bars, for example.

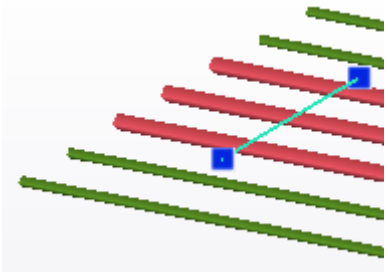
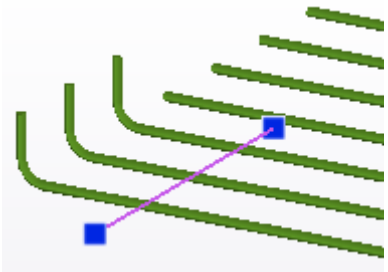
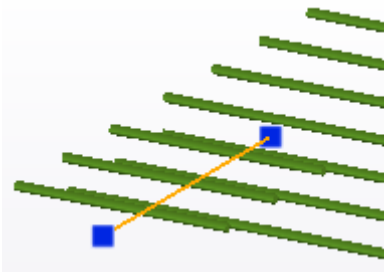


Modifiers are lines, or polylines that may have corner chamfers. Modifiers are projected to rebar set leg faces. Each modifier then only affects the rebar set bars that its projection touches.

NOTE When you work with rebar sets, ensure that the **Direct modification** switch is active.

Also, ensure that the required modifiers are visible (page 564).

To select entire rebar sets, or bar groups or individual bars within rebar sets in the model, you can use the three reinforcement selection switches . Tekla Structures then shows the existing modifiers that affect the selected rebar set bars, and the direct modification handles of the modifiers. Different modifiers have different colors as follows:

Modifier	Color	Example
Property modifier	Light green	
End detail modifier	Magenta	
Splitter	Orange	
Secondary guideline	Light blue	

An arrowhead symbol  close to the midpoint of each modifier indicates the direction (page 563) of the modifier, pointing from the start towards the end of the modifier.

When you select a modifier, Tekla Structures indicates the rebar set bars that are affected by the modifier, and shows the other, unaffected bars as semi-transparent.

You can modify the modifiers by using direct modification, or by changing their properties in the property pane or on the contextual toolbar. When you change the modifier properties, the properties of the rebar set bars change at the location defined by the modifier.

When you delete a modifier, the rebar set reverts to the state it had without the modifier.

Create a property modifier

Property modifiers are shown in light green.

1. Using the reinforcement selection switches , select the rebar set bars for which you want to create the modifier.
2. On the **Rebar set** contextual tab on the ribbon, click  **Property modifier**.
3. Define how you want to place the modifier in the model.

   Click the **Picking mode** button on the contextual tab to cycle through the picking modes and to select the picking mode.

   The  button indicates that you can pick one point, and the  button indicates that you can pick multiple points. The  button indicates that the modifier will be created for the selected bars only.
4. Depending on the selected picking mode, do one of the following:
   - Pick one point to create a single line modifier for the selected rebar set, bar group, or bars.
   - Pick two points to define the end points of a single line modifier. Then click the middle mouse button.
   - Pick multiple points to create a polyline modifier. Then click the middle mouse button to finish picking.
5. To end the command, press **Esc**.
6. To apply local modifications to the rebar set bars at the modifier location:
   a. Select the modifier.

b. If you need to change the modifier geometry, drag the direct modification (page 119) handles.

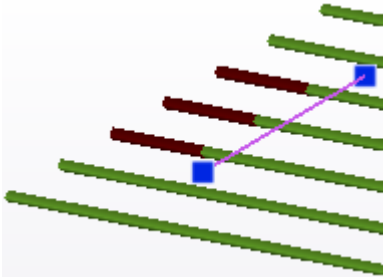
c. Change the property modifier properties (page 1078) on the contextual toolbar or in the property pane.

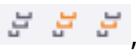
d. If you used the property pane, click **Modify** to save the changes.

Create an end detail modifier

End detail modifiers affect the nearest end of the bar, so by moving the modifier you can change the bar end that it affects. End detail modifiers are shown in magenta.

Tekla Structures displays the threaded bar ends in dark red.



1. Using the reinforcement selection switches , select the rebar set bars for which you want to create the modifier.
2. On the **Rebar set** contextual tab on the ribbon, click  **End detail**.
3. Define how you want to place the modifier in the model.

   Click the **Picking mode** button on the contextual tab to cycle through the picking modes and to select the picking mode.

   The  button indicates that you can pick one point, and the  button indicates that you can pick multiple points. The  button indicates that the modifier will be created for the selected bars only.
4. Depending on the selected picking mode, do one of the following:
   - Pick one point to create a single line modifier for the selected rebar set, bar group, or bars.
   - Pick two points to define the end points of a single line modifier. Then click the middle mouse button.
   - Pick multiple points to create a polyline modifier. Then click the middle mouse button to finish picking.
5. To end the command, press **Esc**.
6. To apply local modifications to the rebar set bars at the modifier location:

a. Select the modifier.

b. If you need to change the modifier geometry, drag the direct modification (page 119) handles.

c. Change the end detail modifier properties (page 1081) on the contextual toolbar or in the property pane.

d. If you used the property pane, click **Modify** to save the changes.

Create a splitter

Splitters split bars and create lap or crank splices. Splitters are shown in orange.

1. Using the reinforcement selection switches , select the rebar set bars for which you want to create the modifier.
2. On the **Rebar set** contextual tab on the ribbon, click  **Splitter**.
3. Define how you want to place the splitter in the model.

   Click the **Picking mode** button on the contextual tab to cycle through the picking modes and to select the picking mode.

   The  button indicates that you can pick one point, and the  button indicates that you can pick multiple points. The  button indicates that the splitter will be created for the selected bars only.
4. Depending on the selected picking mode, do one of the following:
   - Pick one point to create a single line splitter for the selected rebar set, bar group, or bars.
   - Pick two points to define the end points of a single line splitter. Then click the middle mouse button.
   - Pick multiple points to create a polyline splitter. Then click the middle mouse button to finish picking.

   TIP If you want to create the splitter at a rounded distance from a bar end, and the dimension shown is measured from the other bar end, hold down **Shift** when you are placing the splitter in the model to switch the measuring point to the other bar end.
5. To end the command, press **Esc**.
6. To apply local modifications to the rebar set bars at the splitter location:

   a. Select the splitter.

b. If you need to change the splitter geometry, drag the direct modification (page 119) handles.

c. Change the splitter properties (page 1085) on the contextual toolbar or in the property pane.

d. If you used the property pane, click **Modify** to save the changes.

TIP To automatically split long bars in rebar sets so that the bars do not exceed the stock length, you can use the **Auto splitter** detailing component (page 569).

Create a secondary guideline

You can create secondary guidelines for a rebar set. Secondary guidelines are shown in light blue.

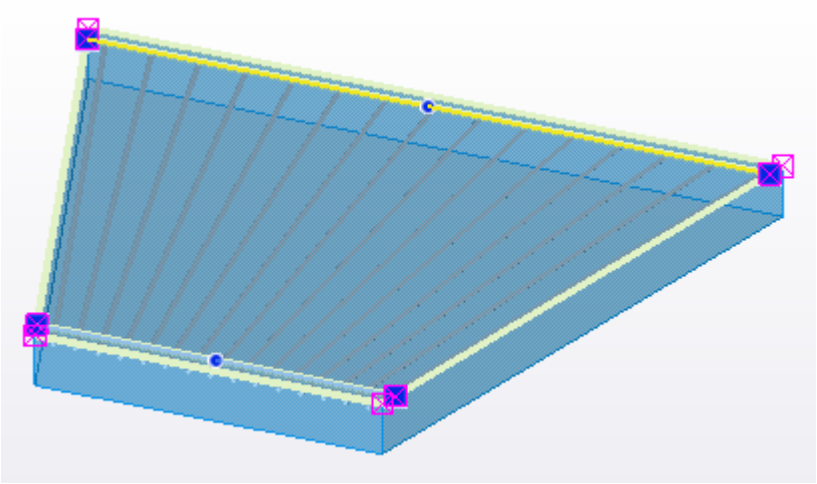
Note that if a rebar set was created using the **By guidelines** command, or if it otherwise has leg surfaces, you can only create one secondary guideline.

1. Select a rebar set.
2. On the **Rebar set** contextual tab on the ribbon, click  **Secondary guideline**.
3. Define how you want to place the guideline in the model.

   The  button on the contextual tab indicates that you can pick one point, and the  button indicates that you can pick multiple points. Click the button to change the picking mode.
4. Depending on the picking mode, do one of the following:
   - Pick one point to create a single line guideline.
   - Pick two points to define the end points of a single line guideline. Then click the middle mouse button.
   - Pick multiple points to create a polyline guideline. Then click the middle mouse button to finish picking.
5. To create another secondary guideline, repeat steps 3 and 4.

   If there are already two guidelines in a rebar set that has leg surfaces, the tooltip of the  button changes to **Maximum number of secondary guidelines reached** and you cannot create more guidelines.
6. Press **Esc** to stop creating secondary guidelines.
7. If needed, select a secondary guideline and modify its geometry (page 119) and properties (page 1073).

For example, you can adjust the secondary guideline's length or spacing values.



See also Distribute bars in a rebar set (page 565) and Modify a rebar set using guidelines (page 547).

8. To set a secondary guideline as the primary guideline, select the secondary guideline and click **Set as primary** on the contextual toolbar.

TIP Alternatively, you can create secondary guidelines the same way as you copy other modifiers (page 563): hold down **Ctrl** and drag the primary guideline.

Create a modifier by copying

You can copy rebar set modifiers.

1. Select a rebar set, bar group, or bar to show its modifiers.

   If you cannot see modifiers, ensure that the required modifier types are set visible (page 564).

2. Select the modifier that you want to copy.
3. Hold down **Ctrl** and drag the modifier to the desired location.

   Tekla Structures creates a new modifier when you release the mouse button.

4. Select the modifier to modify its geometry (page 119) and properties as needed.

Change the direction of a modifier

You can change the direction of rebar set modifiers, splitters, and guidelines.

1. Select a rebar set, bar group, or bar to show its modifiers.

   If you cannot see modifiers, ensure that the required modifier types are set visible (page 564).

2. Select the modifier whose direction you want to change.

3. On the contextual toolbar, click **Swap ends**.

   The arrowhead symbol close to the midpoint of the modifier changes direction, indicating the changed modifier direction.

Make a modifier follow edges

You can define that a rebar set modifier, splitter, or guideline attempts to follow the leg face edges that are located between the modifier end points. This is useful when you reinforce and detail curved concrete structures, for example.

1. Move the modifier end points to the leg face edges.
2. If there are cuts at the edges, add intermediate points to the modifier, and drag the handles to the corners of the cuts.
3. Double-click the modifier to open its properties in the property pane. Select **Yes** in the **Follow edges** list, and then click **Modify**.

   Alternatively, select the modifier and click on the contextual toolbar.

Show or hide rebar set modifiers

When you modify rebar sets using modifiers, ensure that the modifiers you need are visible. You can show and hide modifiers according to their type. If you have many rebar set modifiers in your model, it may be useful to only show some of them when you select rebar set bars, and hide the ones that are not currently needed.

For example, you can show the end detail modifiers only, and hide all property modifiers and splitters.

You can also show or hide primary and secondary guidelines.

1. Go to the **Rebar** tab and click **Visibility**.
2. Do any of the following:
   - Click **Guidelines** to switch the guidelines on or off.
   - Click **Property modifiers** to switch the property modifiers on or off.
   - Click **Splitters** to switch the splitters on or off.
   - Click **End detail modifiers** to switch the end detail modifiers on or off.

Alternatively, you can use the following keyboard shortcuts:

- **Alt+2** for guidelines
- **Alt+3** for property modifiers
- **Alt+4** for splitters

- **Alt+5** for end detail modifiers

To show or hide the rebar set modifiers that have
been created by using components, use the advanced option XS_REBARSET_SHOW_MODIFIERS_CREATED_BY_COMPONENTS. By default, this advanced option is set to `TRUE` and these modifiers are shown when you select rebar set bars.

How to cut rebar sets

You can cut rebar sets automatically by the existing cuts in the concrete parts or manually by using the cutting commands on the **Edit** tab. You can modify cuts in rebar sets the same way as you modify cuts in parts in the model, by using direct modification.

You can use the following commands to create cuts (page 412):

- **Line cut**
- **Polygon cut**
- **Part cut**

Concrete cover settings are also applied to cuts, even on cut edges that are parallel to reinforcing bars.

Cut a rebar set using a cut in a concrete part

When you create rebar sets for concrete parts using the **Longitudinal**, **Crossing**, **By face**, and **By guidelines** commands, Tekla Structures automatically cuts the new rebar sets using the existing cuts in the concrete parts. If you add a new cut to a concrete part with a rebar set, the rebar set is not automatically cut. If you want to cut the rebar set as well, use the **Part cut** command, and use the new cut as the cutting part.

1. On the **Edit** tab, click **Part cut**.
2. Select the rebar set you want to cut.
3. Select the cut in the concrete part.

   Tekla Structures cuts the rebar set.

Modify a cut in a rebar set

You can modify cuts in rebar sets using direct modification. For example, you can make a cut in a rebar set of a different size or shape from a cut in a concrete part.

1. Ensure that the  **Direct modification** switch is active.
2. Select the cut in the rebar set.
3. Modify the cut using direct modification (page 119).

Distribute bars in a rebar set

Rebar sets may have different spacing values between the bars. The spacing of the bars is measured along the rebar set guidelines. You can modify the spacing values in model views or in the property pane, or by using property modifiers. You can also add, move, and delete single bars.

NOTE When you work with rebar sets, ensure that the **Direct modification** switch is active.

Modify the spacing properties

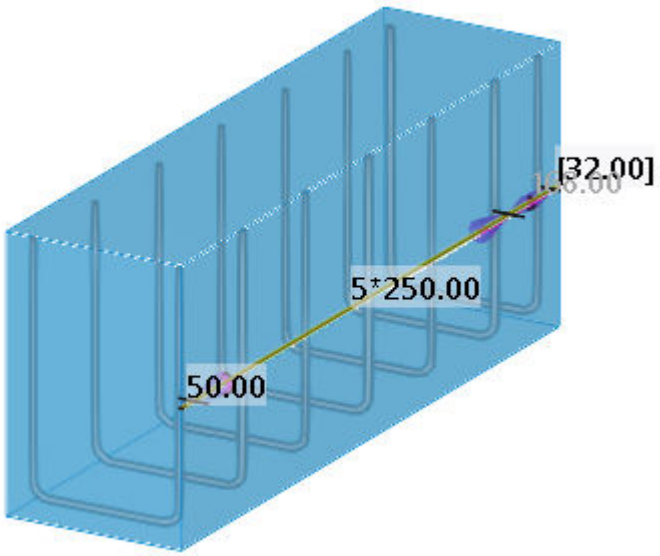
You can modify the spacing properties in model views and using the contextual toolbar, or in the property pane.

1. Select a rebar set.
2. Ensure that the guidelines and property modifiers are visible (page 564).
3. If you want to modify spacing properties using a secondary guideline or property modifier, select the secondary guideline or property modifier.

   To modify secondary guideline spacings independently from the primary guideline, click on the contextual toolbar, or set **Inherit from primary** to **No** in the secondary guideline properties (page 1073) and click **Modify**.

   For the property modifier (page 1078), set **Modify distribution** to **Yes** and click **Modify**.
4. On the contextual toolbar, click **Modify the spacings**.

   Tekla Structures shows the spacing values and the start and end offsets in the model.



For rebar sets and property modifiers, the values are shown on the primary guideline, for secondary guidelines on the secondary guideline.

When spacing modification is enabled, you cannot modify the geometry of the guidelines.

5. To specify how the bars are spaced, select an option for **Creation method** on the contextual toolbar.
6. To modify a spacing value, an offset, or the number of bars, click the value in a model view, enter a new value, and then press **Enter**.

   When **Creation method** is set to **By exact spacings**, you can list multiple spacing values (separated with space characters) and/or use multiplication to repeat the same spacing, for example, `100 200 5*300`.

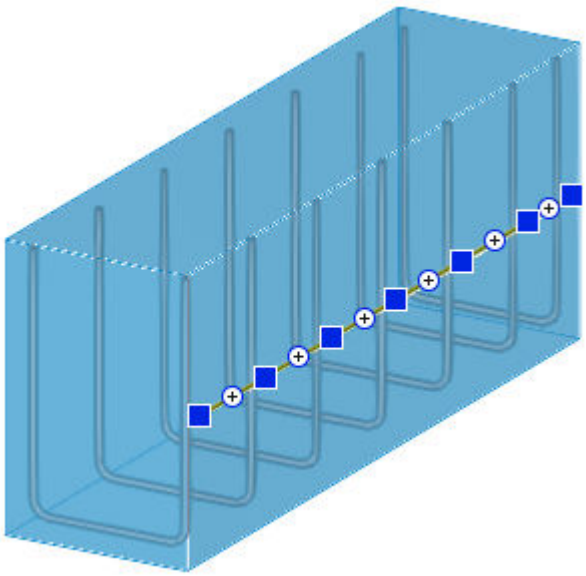
Alternatively, you can modify the spacing properties in the property pane.

Add, move, and delete single bars

1. Select a rebar set.
2. Ensure that the guidelines are visible (page 564).
3. If you want to move bars along a secondary guideline, select the secondary guideline.

   Click on the contextual toolbar, or set **Inherit from primary** to **No** in the secondary guideline properties (page 1073) and click **Modify**.
4. On the contextual toolbar, click **Move, add, delete reinforcement**.

   Tekla Structures displays a handle for each bar on the guideline, and symbols between the bars.



5. Do any of the following:

   - To add a bar between two existing bars, click .
   - To move a bar, select the  handle of the bar and drag it to a new location.

     You can also use the keyboard to enter a numeric location (page 103). Then press **Enter** to confirm.
   - To delete a bar, select the  handle of the bar and press **Delete**.

You can also use the **Exclude** list in the rebar set (page 1068), secondary guideline (page 1073), or property modifier (page 1078) properties to leave out the first and/or last rebar set bars.

Split a rebar set

You can split rebar sets into two rebar sets. The rebar sets to split can have leg faces or leg surfaces.

You may need to split a rebar set when you want to add only a portion of it into a rebar assembly, for example.

Rebar sets with leg faces can be split in any direction.

Limitations:

- Splitting only works with rebar sets that have one or two straight guidelines. If you try to split a rebar set that has more than two guidelines or a curved guideline, Tekla Structures shows a message saying "The object cannot be split in this position".
- Rebar set leg surfaces cannot be cut. If a rebar set has leg surfaces, splitting only works perpendicular to the guideline (=parallel to the bars).

1. On the **Edit** tab, click **Split**.
2. Select the rebar set that you want to split.
3. Pick the first point for the splitting line.
4. Pick the second point for the splitting line.

When a rebar set is split into two:

- The two input points are projected onto the nearest leg face to define a plane, which is then used for splitting the rebar set.
- When needed, the rebar set modifiers and cuts are copied to the new rebar set, which is created as a result of the split.
- The leg faces of the existing rebar set are cut at the split plane, and new leg faces are created for the new rebar set.

Split bars in a rebar set

Use the **Auto splitter** detailing component to generate splitters that split long bars in rebar sets so that the bars do not exceed the stock length.

You can define the portion of the reinforcement to be split and spliced in the same cross section, and the location, symmetry, and length of the splices.

NOTE **Auto splitter** works only with rebar sets and it automatically updates if the split rebar set is modified.

To split reinforcing bar groups and single bars, use the Automatic splicing tool (page 582) or the **Edit** --> **Split** command (page 581) instead.

If the reinforcement that is going to be split belongs to a rebar assembly (page 643), the newly split bars are also added to the rebar assembly.

Limitation: When **Auto splitter** creates a splitter at a non-orthogonal angle, there may be a few millimeter inaccuracies in the bar cut lengths. This should be handled using the rounding settings of the rebar set.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for **Auto splitter**.
3. Double-click **Auto splitter** to start the component.
4. In the **Auto splitter** dialog, adjust the settings as needed:
   a. Select the manufacturer of the reinforcement.

      The maximum bar lengths and lap lengths are then listed by the grade and size of the bar.

      If needed, you can define the length information in the `AutomaticSplicingTool_Manufacturers.dat` file. You can copy the default file from `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system`, edit it, and save it to your project or firm folder. The same file is also used by the **Automatic splicing tool**.
   b. For the bar grades and sizes that are not listed in the `AutomaticSplicingTool_Manufacturers.dat` file, use the **Unspecified stock and lap length** settings to define the maximum bar length after which the bars are split and spliced, and to define the lap length as a distance or in relation to the nominal bar diameter.
   c. Define which proportion of the reinforcement can be spliced in the same cross section.
   d. Define the symmetry that is applied when the reinforcing bars are spliced.

For example, to split and splice bars so that the resulting bars have equal length, select the following **Splicing symmetry** option:



The bar length will be as close to the maximum allowed length as possible.

e. Define the offset of the splice center point.

f. Define the minimum longitudinal distance between two parallel bar splices.

g. Define whether the lapping bars are on top of each other or parallel to each other.

h. Define which rebar set bars will be split and spliced.

- To split and splice bars in one rebar set only, select the rebar set in the model.
- To split and splice several specific rebar sets, select the rebar sets in the model, and then click **Create on selected objects**.

  You can also use a selection filter that selects the required objects. Select the filter in the **Filter on selected object** list.

i. To save the **Auto splitter** settings for later use, enter a file name in the box at the top of the dialog, and then click **Save**.

The file is saved in the `\attributes` folder under the current model folder with the file name extension `.AutoSplitter.AutoSplitterWindow.xml`.

j. Click **OK** to split and splice the rebar set bars.

To modify the group of splitters created using **Auto splitter**, double-click the blue **M** symbol in the model to open the component properties. After modifying the properties, click **Modify** at the bottom of the **Auto splitter** dialog.

If you modify rebar sets that have splitters created using **Auto splitter**, the splitter properties and locations adapt automatically, and all manual modifications are overridden. For example, if you change the bar size, the maximum bar length and the lap length adjust accordingly.

If you have to modify individual splitters, you can either:

- First select the blue **M** symbol in the model, then use the  **Objects in components** switch and select an individual splitter and modify it. These manual modifications will be overridden if you modify the rebar set or the component properties.
- Explode the component, but then the component no longer updates automatically. Select the blue **M** symbol in the model, right-click and select **Explode**.

Freeze or unfreeze a rebar set

You can freeze rebar sets in the model to prevent automatic adjustments and unwanted changes in them when surrounding objects, such as neighboring rebar sets or the parent concrete part, are modified. Freezing rebar sets is useful, for example, when bars have already been detailed, fabricated, or put in place on site.

When you freeze a rebar set, its automatic, intelligent adjustment behavior is disabled, and it starts to behave similarly to a basic reinforcing bar group with **Adaptivity** set to **Off**. Freezing affects the **Bar placement** properties of the rebar set (page 1068) as follows:

- **Automatic layering** is set to **Off**.
- **Layer number** becomes unavailable.
- The **Offsets** settings become available.
- **Adaptivity** is set to **Off**.

For each leg face or leg surface in a frozen rebar set, **Automatic layering** is set to **Off**, **Layer number** becomes unavailable, and the current bar leg distance from the leg face is saved as a fixed offset (**From leg face to bar edge** or **From leg face to bar center**).

A frozen rebar set does not automatically react to modifications in the parent concrete part's geometry, concrete cover values, or other rebar sets. However, you can still manually modify the rebar set properties, or the geometry of guidelines, modifiers, and leg faces. Modifying a leg face of a frozen rebar set, for example, by adding a new corner point, unfreezes that specific leg face. If you add new leg faces to a frozen rebar set, they are also unfrozen. Note that the bars in a frozen rebar set that are at leg surfaces still react to changes in the parent concrete part, because leg surfaces always follow the geometry of the parent part.

If you move, copy, or delete the parent concrete part to which a frozen rebar set is attached, the frozen rebar set will follow.

In model views, frozen rebar sets, leg faces, and leg surfaces are indicated with a snowflake symbol  next to the layer number when you select rebar sets and when the leg faces are visible.

When you unfreeze a frozen rebar set, its automatic, intelligent adjustment behavior is enabled again, and the rebar set starts to react again to changes in the surrounding objects.

Freeze a rebar set

1. On the **Rebar** tab, click **More** --> **Freeze rebar sets**.
2. Select the rebar set that you want to freeze.

   To select multiple rebar sets, use area selection.

Alternatively, and to freeze multiple rebar sets at a time, first select one or more rebar sets in the model, and then start the command in one of the following ways:

- Click **Rebar** --> **More** --> **Freeze rebar sets** on the ribbon.
- Right-click, and then select **Freeze/Unfreeze** --> **Freeze** from the context menu.

Unfreeze a rebar set

1. On the **Rebar** tab, click **More** --> **Unfreeze rebar sets**.
2. Select the rebar set that you want to unfreeze.

   To select multiple rebar sets, use area selection.

Alternatively, and to unfreeze multiple rebar sets at a time, first select one or more rebar sets in the model, and then start the command in one of the following ways:

- Click **Rebar** --> **More** --> **Unfreeze rebar sets** on the ribbon.
- Right-click, and then select **Freeze/Unfreeze** --> **Unfreeze** from the context menu.

Unfreezing affects the **Bar placement** properties of the rebar set as follows:

- **Automatic layering** is set to **On**.
- **Layer number** is set to **Auto**.
- The **Offsets** settings become unavailable. The offset values revert to **Auto** if they were set to **Auto** when the rebar set was frozen and have not been manually adjusted since freezing. If manual values were used before or after freezing, they are preserved.
- **Adaptivity** is set to **Default** (=the model's **Default adaptivity** option selected in the **Options** dialog).

For each leg face or leg surface in the unfrozen rebar set, the previously stored distance values are deleted. The location of the bar legs is then recalculated based on the current concrete cover values and layering information.

Modify a single reinforcing bar, bar group, or mesh

You can modify a reinforcement by using direct modification. You can either modify the reinforcement simply by dragging handles, or select a command from the contextual toolbar.

NOTE Direct modification does not work for the following reinforcement types:

- Circular (page 531), curved (page 529), or 3D reinforcing bars

- Reinforcement strand patterns (page 541)

If you have created the reinforcement using a component, you need to explode the component before using direct modification.

Before you start:

- Ensure that the  **Direct modification** switch is active.
- Select the reinforcement.

  Tekla Structures displays the handles that you can use to modify the reinforcement, and a toolbar icon . Click the icon to open the toolbar and to select the appropriate command. The available commands depend on the type of the reinforcement you are modifying.



To modify single reinforcing bars, reinforcing bar groups, or reinforcement meshes:

To	Do this	Command available for
Change the cover thickness of a reinforcing bar	Drag a line handle to the desired location. 	Reinforcing bars, reinforcing bar groups, reinforcement meshes

To	Do this	Command available for
Add polygon points to a reinforcing bar	Drag a midpoint handle to the desired location.	Reinforcing bars, reinforcing bar groups, polygonal and bent reinforcement meshes
Add points to the start or end of a reinforcing bar	1. Click the start or end reference point of the reinforcing bar . 2. Click the **Add new point** button on the toolbar. 3. Pick a location for the new start or end point.	Reinforcing bars, reinforcing bar groups
Remove points from a reinforcing bar	1. Select one or more reference points. 2. Press **Delete**.	Reinforcing bars, reinforcing bar groups, polygonal and bent reinforcement meshes
Add hooks	1. Click the start or end point of the reinforcing bar . A toolbar for hook properties appears. 2. Select the desired shape for the hook. 3. If you selected **Custom hook**, enter the angle, radius, and length for the hook and click .	Reinforcing bars, reinforcing bar groups
Change the bending radius of a reinforcing bar	1. Click the **Change bending radius** button on the toolbar. 2. Enter a value in the box next to the **Change bending radius** button and press **Enter**.	Reinforcing bars, reinforcing bar groups
Change the diameter of a reinforcing bar	1. Click the **Change diameter** button on the toolbar.	Reinforcing bars, reinforcing bar groups,

To	Do this	Command available for
	2. Select a value from the list next to the **Change diameter** button.	reinforcement meshes
Modify the spacings by adjusting the range	1. Click the **Modify spacings** button on the toolbar. 2. Drag a handle  to the desired location.	Reinforcing bar groups, reinforcement meshes
Modify the spacings by splitting the range in two	1. Click the **Modify spacings** button on the toolbar. 2. Drag a midpoint handle to the desired location and release the handle. Tekla Structures creates a new reinforcing bar and the range is split in two. The spacing in the two new ranges is as close as possible to the original spacing. 3. If needed, change the number of spaces or the spacing value. Click the midpoint handle and enter the desired values in the boxes on the toolbar and press **Enter**.	Reinforcing bar groups, reinforcement meshes
Move, add, or remove reinforcement	1. Click the **Move, add, delete reinforcement** button on the toolbar. Tekla Structures displays the line handles for each reinforcing bar. 2. Do one of the following: • To move a reinforcing bar, highlight it and drag it to the desired location. • To add a reinforcing bar between two reinforcing bars, click .	Reinforcing bar groups, reinforcement meshes

To	Do this	Command available for
	• To delete reinforcing bars, select them and press **Delete**.	

See also

Use handles to modify a reinforcement (page 584)

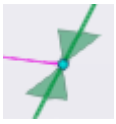
Use adaptivity to modify a reinforcement (page 591)

Check the validity of reinforcement geometry (page 594)

Distribute bars in a reinforcing bar group

You can select how reinforcing bars are distributed in a bar group by modifying the bar spacings.

To modify the spacings of bars in a reinforcing bar group, do one of the following:

To	Do this
Modify the spacings using direct modification (page 572)	1. Ensure that the  **Direct modification** switch is active. 2. Select a reinforcing bar group. 3. On the contextual toolbar, click the **Modify the spacings** button . 4. Drag a handle  to the desired location.
Modify the spacings using the **Rebar group** properties	1. Select a reinforcing bar group. 2. Double-click the reinforcement to open the **Rebar group** properties. 3. In the **Distribution** section, select a spacing option from the **Creation method** list. 4. Enter the required values. 5. Click **Modify**.

Spacing options in the **Creation method** list:

Option	Description	Example
Equal distribution by number of reinforcing bars	Enter the number of reinforcing bars. Tekla Structures divides the available distance by the number of bars. Enter the number of bars in the **Number of reinforcing bars** box.	
Equal distribution by target spacing value	Enter a spacing value. Tekla Structures aims the spacing value as close as possible to the value in the **Target spacing value** box.	
By exact spacing value with flexible first	Enter the spacing value in the **Exact spacing value** box. Creates fixed, regular spaces between the bars. The first space adjusts to even out the bar distribution. If the first space is less than 10% of the exact spacing value, Tekla Structures removes one bar.	
By exact spacing value with flexible last space	Enter the spacing value in the **Exact spacing value** box. Creates fixed, regular spaces between the bars. The last space adjusts to even out bar distribution.	
By exact spacing value with flexible middle space	Enter the spacing value in the **Exact spacing value** box. Creates fixed, regular spaces between the bars. The middle space adjusts to even out bar distribution. If there are an odd number of bars (two middle spaces), the other middle space adjusts to even out bar distribution.	
By exact spacing value with flexible first and last space	Enter the spacing value in the **Exact spacing value** box. Creates fixed, regular spaces between the bars. Both the first and last spaces adjust to even out bar distribution.	

Option	Description	Example
By exact spacings	Enter the spacing values manually in the **Exact spacing values** box. Use multiplication to repeat spacings, for example, `5*200` to create five spaces of `200`.	

See also

Create a reinforcing bar group (page 520)

Create a reinforcing bar group using Rebar shape catalog (page 522)

Modify a single reinforcing bar, bar group, or mesh (page 572)

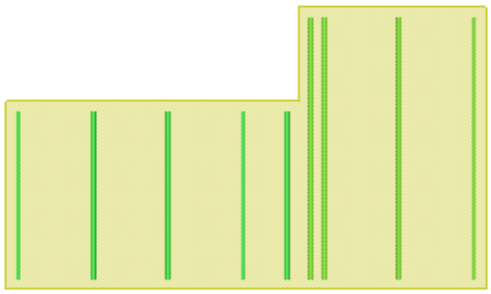
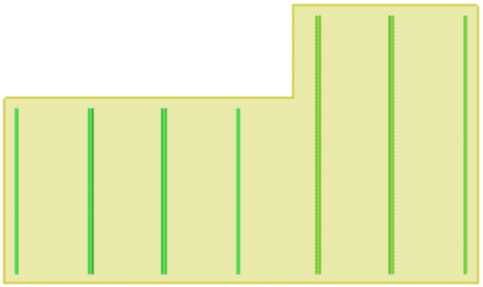
Delete bars from a reinforcing bar group

You may occasionally need to delete or exclude specific reinforcing bars. For example, when several reinforced areas intersect, causing reinforcing bars to overlap, or when you want to start bar distribution at a specific distance from the end of a part.

To delete reinforcing bars from a group, do one of the following:

To	Do this
Delete bars using direct modification (page 572)	1. Ensure that the  **Direct modification** switch is active. 2. Select a reinforcing bar group. 3. On the contextual toolbar, click the **Move, add, delete reinforcement** button . 4. Select the bars to be deleted and press **Delete**.
Delete bars using the **Rebar group** properties	1. Select a reinforcing bar group. 2. Double-click the reinforcement to open the **Rebar group** properties. 3. In the **Distribution** section, select an option from the **Exclude** list. 4. Click **Modify**.

Examples when you have used the **Exclude** options:

Before excluding bars	After excluding bars
Two reinforcing bar groups have been added to a concrete beam: • one bar group with flexible last space • one bar group with flexible first space 	Two reinforcing bar groups without the excluded bars: • one bar group with the last bar excluded • one bar group with the first bar excluded 

See also

Create a reinforcing bar group (page 520)

Create a reinforcing bar group using Rebar shape catalog (page 522)

Modify a single reinforcing bar, bar group, or mesh (page 572)

Ungroup a reinforcement

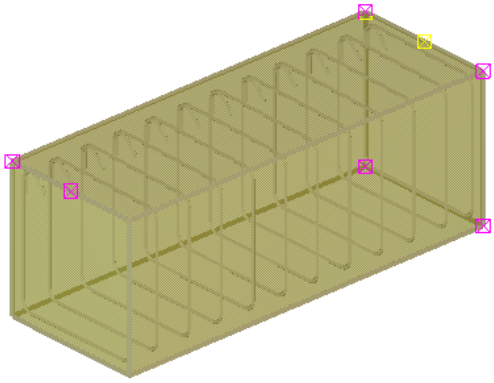
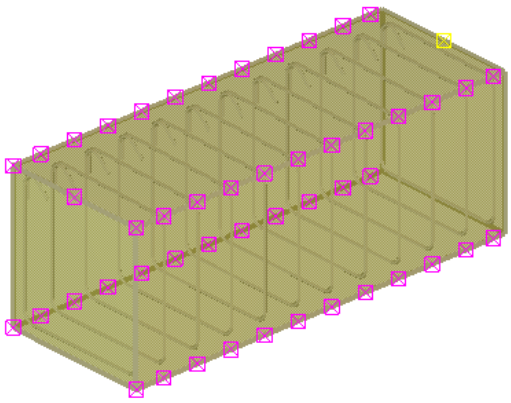
You can ungroup reinforcing bar groups and reinforcement meshes. Only reinforcement where each reinforcing bar is in one plane can be ungrouped.

NOTE You cannot ungroup circular (page 531) or curved (page 529) reinforcing bar groups.

1. On the **Rebar** tab, click **Edit** --> **Group**.
2. Select one of the reinforcing bars in a reinforcing bar group or in a reinforcement mesh.

   The reinforcing bar group is replaced with single reinforcing bars. The single bars get the same properties and offsets as the group.

   If you ungroup a reinforcement mesh, the offsets for single bars are zero.

Before ungrouping	After ungrouping
	

See also

Modify reinforcement (page 544)

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a reinforcing bar group (page 520)

Create a reinforcement mesh (page 536)

Group reinforcement

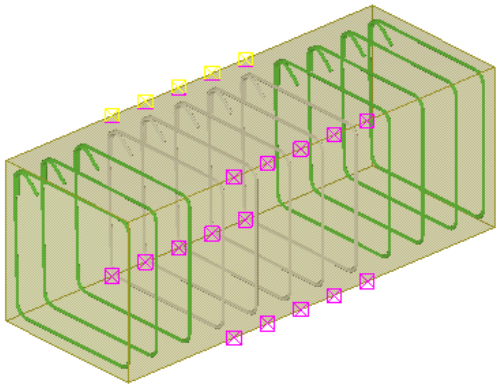
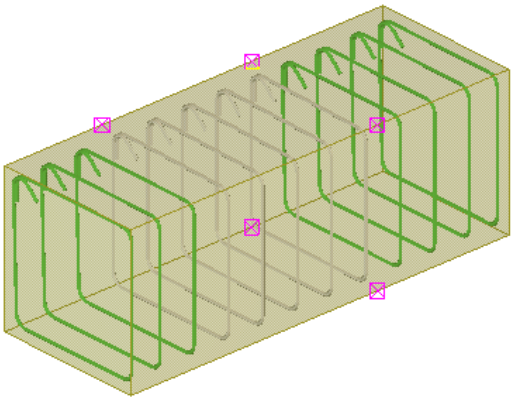
You can group single reinforcing bars and reinforcing bar groups. Only reinforcement where each reinforcing bar is in one plane can be grouped. All groups are created with exact spacings. Single reinforcing bars need to have the same bending shape.

NOTE You cannot create circular (page 531) or curved (page 529) reinforcing bar groups by grouping.

1. On the **Rebar** tab, click **Edit** --> **Group**.
2. Select all the reinforcing bars or reinforcing bar groups that you want to group.
3. Click the middle mouse button.
4. Select one reinforcing bar or reinforcing bar group to copy the properties from.

   The new group gets the same properties as the selected reinforcing bar.

   NOTE The reinforcing bar or reinforcing bar group that you copy the properties from is also added to the group. This means, for example, that you cannot copy properties from a separate reinforcing bar group which you do not want to include in your new reinforcing bar group.

Before grouping	After grouping
	

See also

Modify reinforcement (page 544)

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a reinforcing bar group (page 520)

Create a single reinforcing bar (page 520)

Combine two reinforcing bars or reinforcing bar groups into one

You can combine two single reinforcing bars or reinforcing bar groups into one. Reinforcing bars can be combined if their end points are connected, or the bars are parallel and close to each other. However, in certain cases it is possible to combine bars or groups that are not connected nor parallel. The combined reinforcement gets the same properties as the bar selected first.

NOTE You cannot combine **Tapered N** reinforcing bar groups.

1. On the **Edit** tab, click **Combine**.
2. Select the first single bar or bar group to be combined.
3. Select the second single bar or bar group to be combined.

   Tekla Structures combines the reinforcing bar groups or bars into one.

See also

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a reinforcing bar group (page 520)

Create a single reinforcing bar (page 520)

Modify reinforcement (page 544)

Split a reinforcing bar group

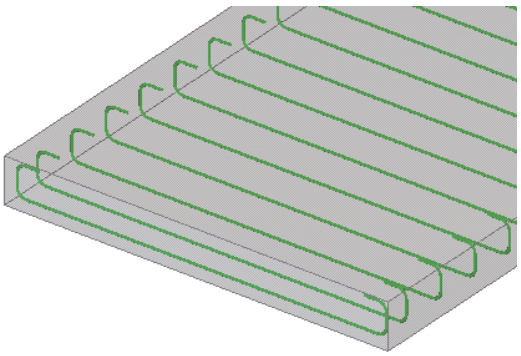
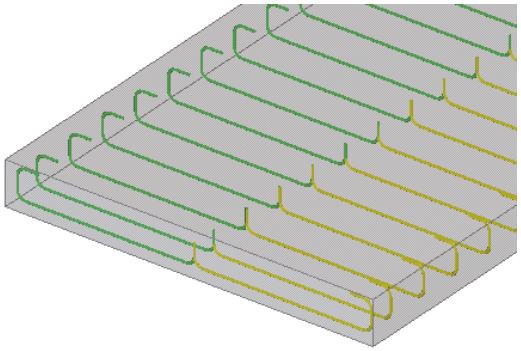
You can split normal and tapered reinforcing bar groups into two groups. You can also split single reinforcing bars into two.

1. On the **Edit** tab, click **Split**.
2. Select a reinforcing bar group.
3. Pick two points to indicate where to split the group.

   Tekla Structures splits the reinforcing bar group.

NOTE You cannot split reinforcing bar groups diagonally.

Once split, each new reinforcing bar group retains the properties of the original group. For example, if the bars in the original group had hooks at both ends, bars in the new groups also have hooks at both ends. Modify the properties of the new groups, if needed.

Before splitting	After splitting
	

See also

Split and splice a reinforcement (page 582)

Modify a single reinforcing bar, bar group, or mesh (page 572)

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a reinforcing bar group (page 520)

Create a single reinforcing bar (page 520)

Split and splice a reinforcement

You can split long reinforcing bars and bar groups that exceed the stock length, and create splices in split locations.

Use the **Automatic splicing tool** macro to split and splice reinforcement that exceeds the stock length. You can first check the length of the reinforcing bars in the model according to the manufacturer. You can then define the portion

of the reinforcement to be split and spliced in the same cross section, and the location, symmetry, type, and length of the splices.

NOTE **Automatic splicing tool** does not work with rebar sets (page 494). To split bars in rebar sets, use Auto splitter (page 569) and splitters instead.

If the reinforcement that is going to be split belongs to a rebar assembly (page 643), the newly split bars are also added to the rebar assembly.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **Automatic splicing tool** to start the macro.
4. In the **Automatic Splicing Tool** dialog box:
   a. Select the manufacturer of the reinforcement.

      The maximum bar lengths and lap lengths are then listed by the grade and size of the bar.

      If needed, you can define the length information in the `AutomaticSplicingTool_Manufacturers.dat` file. You can copy the default file from `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system`, edit it, and save it to your project or firm folder.

   b. For the bar grades and sizes that are not listed in the `AutomaticSplicingTool_Manufacturers.dat` file, use the **Maximum length for unspecified stock** box to define the maximum reinforcing bar length after which the bars are split and spliced.

   c. To check if the length of the reinforcing bars exceeds the maximum length, use one of the buttons next to **Perform check to**:

      - To check all reinforcement in the model, click **All**.
      - To check specific reinforcement, first select the reinforcement or cast units in the model, and then click **Selected**.

      Tekla Structures lists the reinforcing bars that are longer than the maximum length under **Longer rebars** on the right side of the dialog box.

      When you select a row in the **Longer rebars** list, Tekla Structures highlights the corresponding reinforcement in the model.

   d. Define which proportion of the reinforcement can be spliced in the same cross section.

   e. Define the symmetry that is applied when the reinforcing bars are spliced.

f. Define the offset of the splice center point.

g. Define the minimum longitudinal distance between two parallel bar splices.

h. Select the splice type.

   You can create lap splices, coupler splices, or welded splices.

i. For lap splices, define the default lap length as a distance or in relation to the nominal bar diameter.

   This value will be used if there is no lap length defined for a bar grade and size in the `AutomaticSplicingTool_Manufacturers.dat` file.

j. For lap slices, define whether the lapping bars are on top of each other or parallel to each other.

k. To split and splice reinforcement, use one of the buttons next to **Perform split and splice to**:

   - To splice all reinforcement in the model, click **All**.
   - To splice specific reinforcement, first select the reinforcement in the **Longer rebars** list, or in the model using the **Select objects in components** switch , and then click **Selected**.

See also

Create a reinforcement splice (page 543)

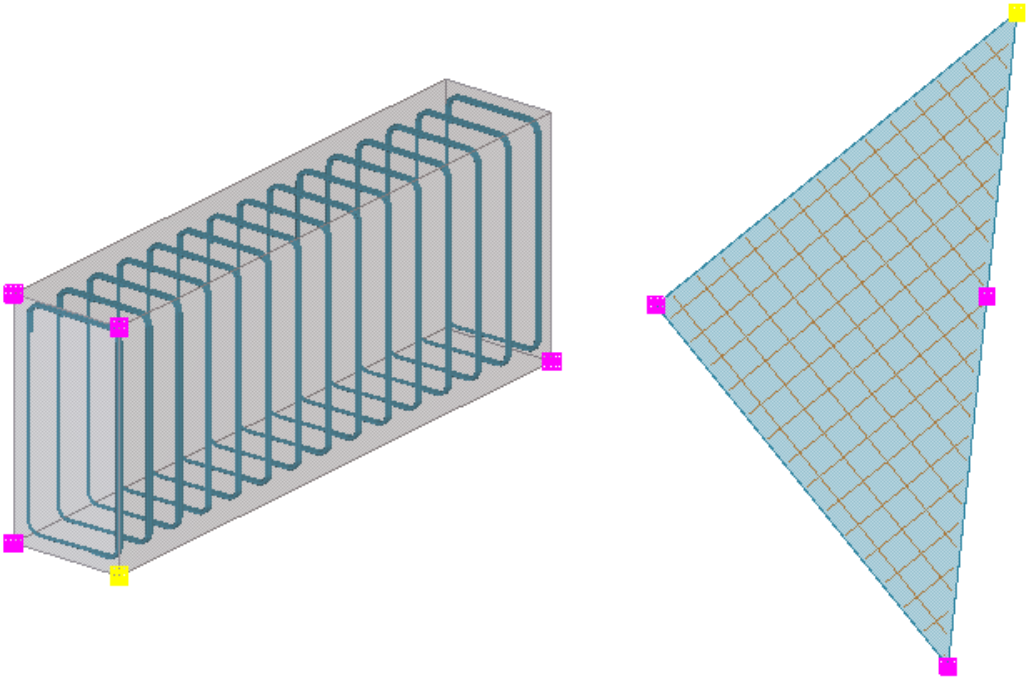
Use handles to modify a reinforcement

If you do not want to use direct modification to modify a reinforcement, you can use, for example, the reinforcement handles to modify the reinforcement.

Tekla Structures uses handles to indicate:

- The ends and corners of a reinforcing bar.
- The distribution length of a bar group.
- The corners and main bar direction of a mesh.

When you select a reinforcement, Tekla Structures highlights the handles. The handle of the first end point is yellow, the rest are magenta.



1. Select the reinforcement.

   Tekla Structures highlights the handles.
2. Click one of the handles to select it.
3. Move the handle like any other object in Tekla Structures.

   For example, if **Drag and drop** is active, just drag the handle to a new position.

NOTE If you want to use the reinforcement handles, ensure that the **Direct modification** switch  is not active. If the switch is active and direct modification (page 572) is on, Tekla Structures displays direct modification handles for the reference points, ends, legs, and leg midpoints of the selected reinforcement. These handles are blue.

See also

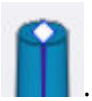
Check the validity of reinforcement geometry (page 594)

Add hooks to reinforcing bars

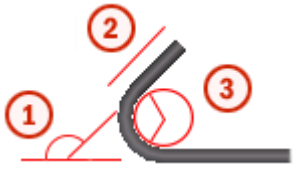
You can add hooks to the ends of reinforcing bars for anchoring purposes.

NOTE Hooks are intended to be used only for anchoring purposes. Do not use hooks as a method to model other reinforcing bar geometry, because it may cause problems with visibility in drawings, in adaptivity, and in bar bending shape recognition.

To add hooks to reinforcing bars, do one of the following:

To	Do this
Add hooks using direct modification (page 572)	1. Ensure that the  **Direct modification** switch is active. 2. Select a single reinforcing bar or a reinforcing bar group. 3. Click the start or end point of the reinforcing bar . A toolbar for hook properties appears. 4. Select the desired shape for the hook. 5. If you select **Custom hook**, enter the angle, radius, and length for the hook. Click .
Add hooks using the **Single rebar** or **Rebar group** properties	1. Select a single reinforcing bar or a reinforcing bar group. 2. Double-click the reinforcement to open its properties. 3. In the **Hooks** section, select a hook type for the bar start and/or end from the **Hook type** list. 4. If you select **Custom hook**, enter the angle, radius and length for the hook. 5. Click **Modify**.
Add hooks to rebar sets using end detail modifiers	See Modify a rebar set locally using modifiers (page 557).

For custom hooks you need to enter the hook information:

Option	Description	
Angle	Enter a value between -180 and +180 degrees.	 1. Angle 2. Length 3. Radius
Radius	Enter the internal bending radius of the hook. Use the same radius for the hook and for the reinforcing bar. If the hook and the reinforcing bar have different radiuses, Tekla Structures does not recognize the shape of the bar.	
Length	Enter the length of the straight part. If the length is set to zero, no hooks are created.	

Hook examples



	Description
1	Standard 90-degree hook
2	Standard 135-degree hook
3	Standard 180-degree hook
4	Custom hook

If you select a standard hook, the **Angle**, **Radius**, and **Length** use predefined dimensions.

The `rebar_database.inp` file contains the predefined minimum bending radius and minimum hook length for all standard hooks.

See also

Create a reinforcing bar group (page 520)

Create a reinforcing bar group using Rebar shape catalog (page 522)

Modify a single reinforcing bar, bar group, or mesh (page 572)

Define the reinforcement cover thickness

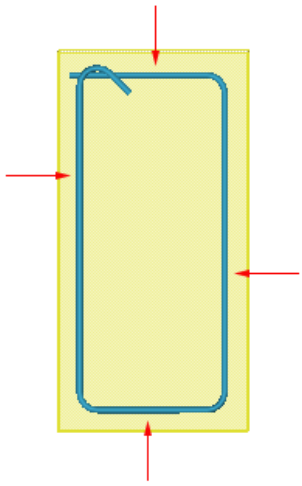
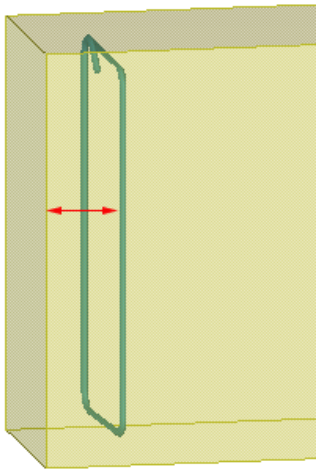
Reinforcing bars need a concrete cover to protect them against harmful elements, such as the weather and fire. When you create single bars, Tekla Structures uses the thickness of concrete cover to determine the position of the bar.

To define the reinforcement cover thickness, do one of the following:

To	Do this
Change the cover thickness using direct modification (page 572)	1. Ensure that the  **Direct modification** switch is active. 2. Select a single reinforcing bar, a reinforcing bar group, or a mesh. 3. Drag a line handle to the desired location. 
Change the cover thickness using the **Single rebar**, **Rebar group**, or **Rebar mesh** properties	1. Select a single reinforcing bar, a reinforcing bar group, or a mesh. 2. Double-click the reinforcement to open its properties. 3. Define the reinforcing bar cover thickness in the **Cover thickness** section. The cover thickness can be defined in three directions: • On plane, that is, the distance from part's bottom, top, and side faces to the bar. You can enter several values. Enter the values in the order you pick the points to create the bar. If you enter less values than there are bar legs, Tekla Structures uses the last value for the remaining legs. • From plane, that is, the distance from the end face of the part to the bar. If the reinforcing bar is outside the part, enter a negative value in the **On plane** and/or the **From plane** boxes.

To	Do this
	• In the longitudinal direction of the bar, that is, start and end. To define the length of an ultimate leg of a bar, use the **Leg length** option and the **Snap to nearest points** switch. Then pick anywhere on a part edge or line to indicate the direction for the bar leg. 4. Click **Modify**.
Change the default cover thickness of the rebar sets in a model	1. On the **File** menu, click **Settings** --> **Options** to open the **Options** dialog box. 2. Go to the **Rebar set** settings and to the **Covers and locations** tab. 3. Modify the settings and click **OK**. You can define the default cover thickness values in the global and/or parts' local coordinate systems and at different part faces. In each concrete part's properties, you can then select whether to use the global or local cover thickness values. 4. To apply the changes to all or selected existing rebar sets in the model, go to the **Rebar** tab on the ribbon and click **More** --> **Regenerate**.
Change the cover thickness of the rebar sets in an individual concrete part or for a part type	1. Double-click a concrete part to access its properties. 2. Go to the **Concrete covers for rebar sets** section. Alternatively, you can click **User-defined attributes** and go to the **Rebar set** tab. 3. Select the coordinate system: **Global** or part's **Local**. If you select the empty option, Tekla Structures uses the global cover thickness values from the **Options** dialog box. 4. To override the default values from the **Options** dialog, define the cover thickness at each required part face. • With the **Global** coordinate system, you can enter the top, bottom, and side face values. • With the **Local** coordinate system, you can enter the top, bottom, front, back, start, and end face values.

To	Do this
	5. Click **Modify** to update the selected part's properties. 6. To save the cover thickness values for later use for this part type, update the standard file or create a property file.
Change the concrete cover thickness of the rebar sets at a concrete part face or pour object face	1. Add a surface (page 436) to the object face at which you want to change the concrete cover. 2. Double-click the surface to modify its properties in the property pane. 3. In the **Type** list, select **Concrete cover**. 4. In the **Rebar set** section, enter the concrete cover thickness in the **Concrete cover** box. 5. Click **Modify** to apply the changes. **NOTE:** If you add surfaces to pour object faces, update the rebar sets each time you reopen the model. On the **Rebar** tab on the ribbon, click **More** --> **Regenerate**.

On plane	From plane	Start and end
		

See also

Create a reinforcing bar group (page 520)

Create a reinforcing bar group using Rebar shape catalog (page 522)

Create a rebar set (page 494)

Modify reinforcement (page 544)

Select the definition for a reinforcement

When you create (page 493) or modify (page 544) reinforcement, such as a reinforcing bar group, rebar set, or reinforcement component, you can select a definition for the bars from the rebar catalog. Selecting the definition automatically sets some basic reinforcement properties, such as grade, size, and bending radius.

1. Open the properties of the reinforcement.
2. In the property pane or component dialog, click the **...** button next to the **Size** box.

   The **Select rebar** dialog appears.
3. If needed, organize the rebar catalog view.

   For example, you can filter rebar definitions, or group and sort them differently.
4. Select a rebar definition from the list.
5. Click **OK** to close the **Select rebar** dialog.
6. Click **Modify** in the reinforcement properties to apply the changes.

Use adaptivity to modify a reinforcement

Reinforcement follows the shape of the part when the reinforcement handles are located on the face or edge of the part.

The following types of adaptivity are available:

- Default adaptivity: the **Default adaptivity** setting of the model is used for the reinforcement.

  Note that if you select the **Default** option for a rebar set modifier (page 557), the modifier uses the adaptivity setting of the rebar set (page 494) it belongs to, not the **Default adaptivity** setting of the model.
- Adaptivity is off: changes in the parent part do not affect objects that are attached to the part.
- Fixed adaptivity: handles retain their absolute distances to the nearest part faces.
- Relative adaptivity: handles retain their relative distances to the nearest part faces in relation to the part's overall size.

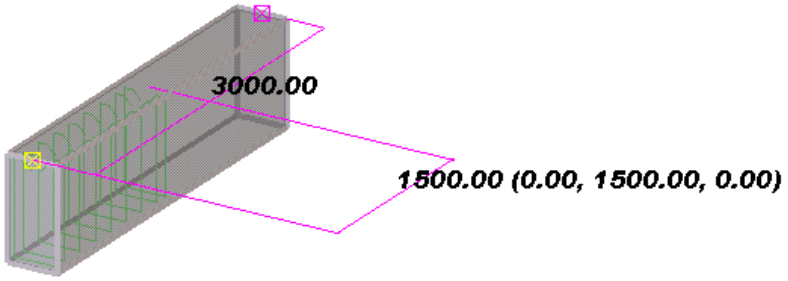
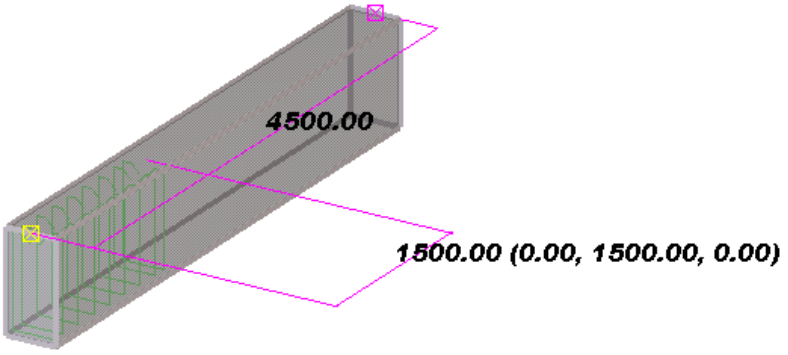
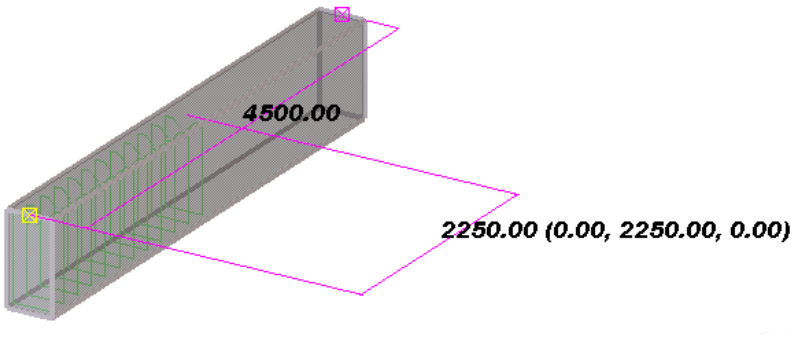
TIP To modify the general adaptivity settings, click **File** --> **Settings** --> **Options** --> **General**.

You can also modify the adaptivity settings for each reinforcement separately. These modifications override the general settings in the **Options** dialog box.

1. Select a reinforcement.
2. Right-click, select **Adaptivity**, and then select one of the adaptivity options.

   When a part is modified, Tekla Structures handles the reinforcement according to the adaptivity selection.

Adaptivity examples

Reinforcing bars in their original position	
Fixed adaptivity	
Relative adaptivity	

See also

Check the validity of reinforcement geometry (page 594)

Modify a single reinforcing bar, bar group, or mesh (page 572)

Modify the adaptivity of reinforcement, surface treatment, or edge chamfers in parts (page 438)

Attach a reinforcement object to a concrete part

When you create a reinforcement object, such as a bar group, rebar set, or mesh, Tekla Structures attaches it to the concrete part that you have selected to reinforce. Later in some cases, you might need to attach the reinforcement object to another concrete part manually. The attached reinforcement objects follow the part if the part is moved, copied, or deleted.

NOTE Only attached reinforcement objects are shown in cast unit drawings and reports with the part.

1. If you want to manually attach only certain single bars in a rebar set to a part, create a property modifier (page 559) for the bars that you want to attach.
2. Select the reinforcement object that you want to attach.

   To attach single rebar set bars, first select the rebar set, then the property modifier that affects the bars that you want to attach.
3. Right-click and select **Attach to part**.
4. Select the part to which you want to attach the reinforcement object.

   The reinforcement object is attached to the part.

Note that if you have manually attached single bars in rebar sets to parts in Tekla Structures 2022 SP2 or older versions, using property modifiers for manual attachment does not work with these bars. To revert all manually attached bars in rebar sets to automatic attachment (page 595), select all the rebar sets in the model and use the **Detach from part** command.

Detach a reinforcement object from a concrete part

If needed, you can detach a reinforcement object from a concrete part.

1. Select the reinforcement object that you want to detach.

   To detach single rebar set bars, first select the rebar set, then the property modifier that affects the bars that you want to detach.
2. Right-click and select **Detach from part**.

   The reinforcement object is detached from the part.

Note that if you detach a rebar set or a property modifier, the bars in the rebar set are reattached automatically if they are inside a concrete part.

TIP Alternatively, you can use the ribbon when you manually attach or detach reinforcement objects.

1. On the **Rebar** tab, click **More** --> **Attach to part** or **More** --> **Detach from part**.
2. Select the reinforcement object that you want to attach or detach.
3. If you are attaching the reinforcement object, select the part to which you want to attach the reinforcement object.

See also

How Tekla Structures automatically attaches rebar set bars to concrete parts (page 595)

Modify reinforcement (page 544)

Check the validity of reinforcement geometry

Reinforcement creation or modification can result in invalid reinforcement geometry. For example, too big bending radius can cause invalid reinforcement geometry. If a model contains a reinforcement with invalid geometry, the reinforcement is not shown in the drawings. The reinforcement becomes visible and the drawings are updated when the geometry is corrected.

If a rebar set leg face or leg surface (page 549) is shown in red in the model, it means that the geometry of some bars in the rebar set is not valid. Check and fix the properties of the rebar set (page 1068) (for example, the bending radius) or the leg face (page 1075) or leg surface (page 1076).

NOTE Reinforcement geometry validity check does not work with circular (page 531) or curved (page 529) reinforcing bar groups.

1. On the **File** menu, click **Diagnose & Repair** and in the **Model** area, click **Diagnose model**.
2. Check the results.

   If there are inconsistencies in the geometry, Tekla Structures displays a warning message, and draws a thin line between the reinforcement handles to show the invalid geometry.

   You can correct the reinforcement geometry by selecting the line and modifying the reinforcement properties.

See also

Modify a single reinforcing bar, bar group, or mesh (page 572)

Modify a rebar set (page 545)

Classify reinforcement to layers

To be able to show in drawings what is the order of different reinforcement layers near a surface of a concrete part, you need to classify reinforcement in the model. You can do this by using the **Rebar classification** macro.

Rebar classification classifies the reinforcing bars and reinforcement meshes by their order of depth in concrete slabs and panels. The reinforcing bars and meshes get an attribute indicating the layer where they are placed inside the concrete part.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **Rebar classification** to start the macro.
4. In the **Rebar classification** dialog:
   a. Enter the prefixes that you want to use for the reinforcement layers near the top, bottom, front, and back surfaces of the concrete parts.
   b. Select whether you want to classify **All objects** or **Selected objects**.

      If you select **Selected objects**, select the reinforcement or the concrete parts containing the reinforcement that you want to classify.
   c. Click **Preview** to view the properties of the reinforcement in each layer.

      The layers are named using the relevant surface prefix and numbered starting from the surface.
   d. If you do not want to classify a reinforcement, select it from the list and click **Delete item**.
   e. To save the classification attributes of the reinforcement, do one of the following:
      - Click **Modify** to also keep the **Rebar classification** dialog open.
      - Click **OK** to also close the **Rebar classification** dialog.
5. In a drawing, run the **Rebar layering marker** macro to create layer-specific markers for reinforcement.

How Tekla Structures automatically attaches rebar set bars to concrete parts

When you create or modify a rebar set, Tekla Structures automatically attaches each bar in the rebar set to a concrete part. This concrete part is the *parent part* of the rebar set bar.

Depending on the cast unit type, bottom level, and volume of the concrete parts, and the bar length, location, and orientation, and the number of bars in the rebar set, Tekla Structures searches for and chooses the parent part for each of the bars as follows:

- Each bar that is at least partly inside only one concrete part is attached to that part.
- Each bar that is inside both precast and cast-in-place concrete parts is attached to a precast part.

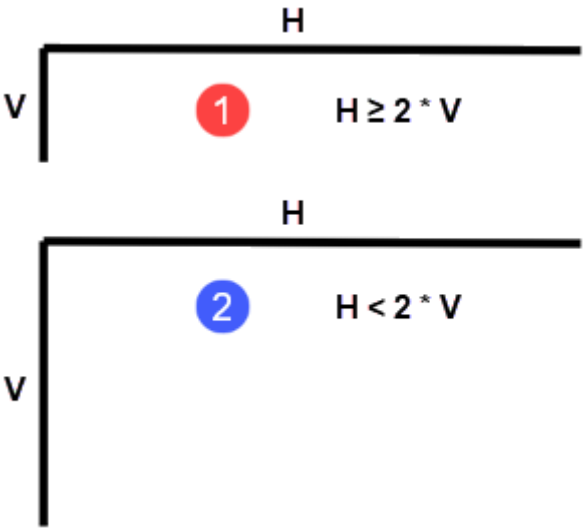


(1) = precast part, **(2)** = cast-in-place part

In this and the following images, the parent part is shown in green.

Then, for each bar that is inside two or more concrete parts, either precast parts or cast-in-place parts, but not both:

- A bar is considered horizontal if its overall horizontal extent **H** is longer than or equal to its overall vertical extent **V** multiplied by the default factor of 2. Otherwise the bar is considered vertical.

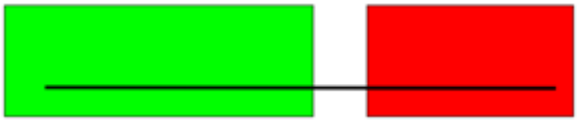


(1) = horizontal bar, **(2)** = vertical bar

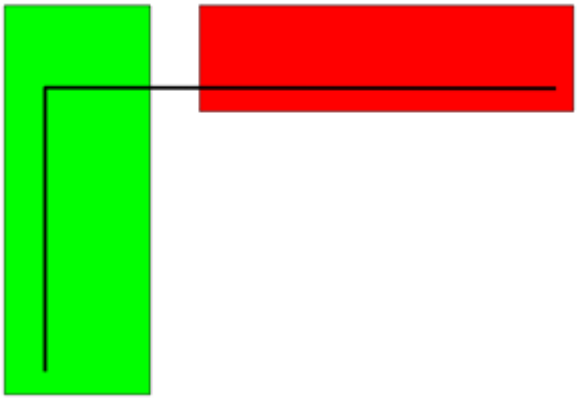
In general, a bar in a rebar set is considered horizontal if:

H ≥ XS_REBAR_VERTICAL_FACTOR * **V**

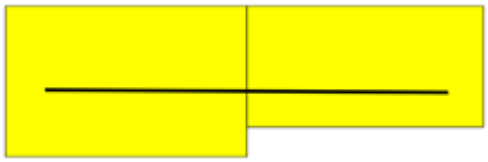
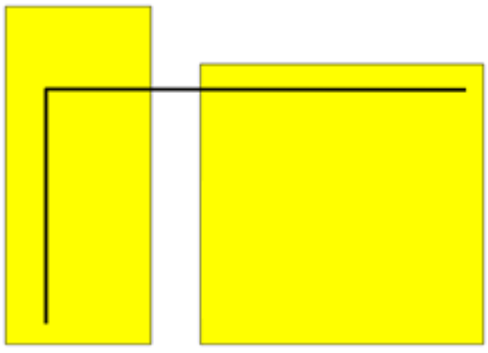
- If the bar is considered horizontal, it is attached to the part that contains the longest bar length.



- If the bar is considered vertical and if the bar is fully inside one part and partially inside other parts, it is attached to the part that contains the whole bar.
- If the bar is considered vertical and placed partly inside several parts, it is attached to the part whose bottom face has the lowest global z coordinate.



Sometimes several parts might match one of the above criteria and could be the parent part of a rebar set bar. For example:

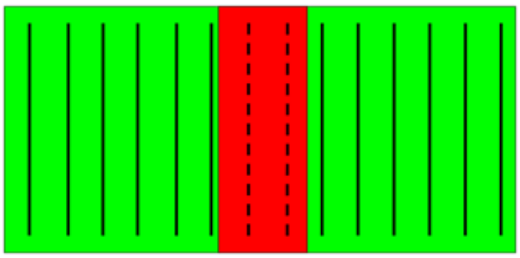
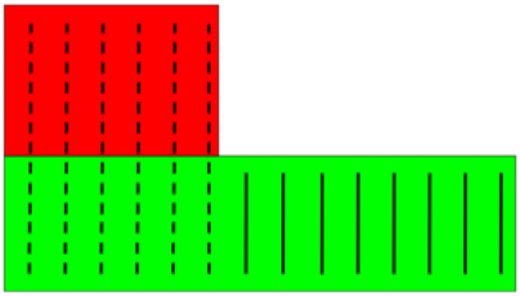
	A horizontal bar is equally in two parts.
	Two parts contain some of a vertical bar and their bottom levels are at equal height.

	Two parts contain all of a vertical bar but are at different heights.

In cases like these, Tekla Structures chooses the parent part for each bar from among the matching parts as follows:

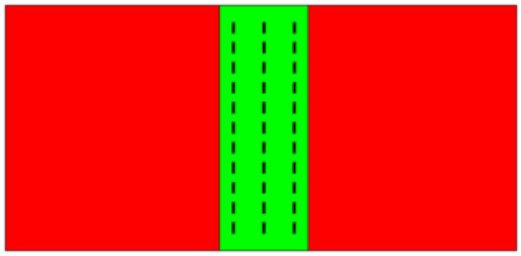
- If one of the parts is the more common parent part in the rebar set than the other parts, Tekla Structures attaches the bar to that more common parent part.

  For example:

	In this plan or elevation view, two parts overlap and the dashed bars are entirely in both parts. The dashed bars are attached to the green part because it contains more bars of the rebar set than the red part.
	In this plan view, the dashed bars are in two parts. The bar planes are vertical and the parts are at equal height. The dashed bars are attached to the green part because the most bars of the rebar set are attached to it.

- If there is no more common parent part for a rebar set, Tekla Structures attaches the bar to the part that has the smallest volume.

  For example:

	In this plan view, two parts overlap and a rebar set with the dashed bars is entirely in both parts. The bars are attached to the green part because it has the smaller volume.

If the volumes of the parts are equal for several parts, Tekla Structures chooses the part with the smallest ID.

If you need to override the automatic attachment of rebar set bars to concrete parts, you can manually attach (page 593) rebar sets and bars in them.

How to calculate the reinforcing bar length

You have three options how to calculate the reinforcing bar length in Tekla Structures:

- Along center line, which is the default method
- As a sum of leg lengths
- Using a formula

Along center line

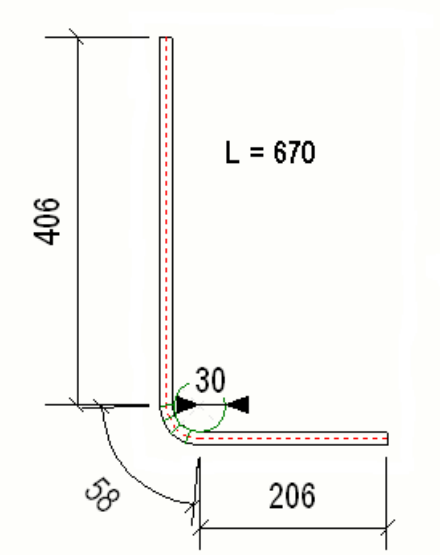
The center line length calculation is used by default when XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT is set to `FALSE` in **File menu** --> **Settings** --> **Advanced options**.

Center line length calculation uses the actual reinforcing bar diameter by default.

In the example below, the center line length is calculated as follows: `450 - (30 + 14) + 2*3.14*(30+14/2)*1/4 + 250 - (30 + 14) = 670.1`

where

- `30` = bending radius
- `14` = actual diameter (12 is nominal)

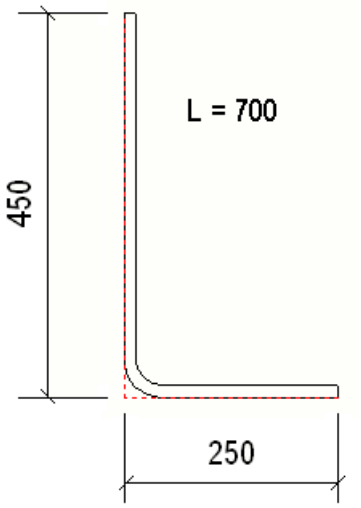


Sum of leg lengths (SLL)

The sum of leg length calculation is based on the dimensions of the straight legs and it does not take the bending radius into account.

This calculation is used when XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT and XS_USE_USER_DEFINED_REBARSHAPERULES are set to `TRUE` in **File menu** --> **Settings** --> **Advanced options**.

In the example below, the reinforcing bar length is `450 + 250 = 700`



If the length value is shown as zero in reports and inquiries, you need to define the length in **Rebar Shape Manager** for each shape.

To define the length in **Rebar Shape Manager**:

1. In **Bending schedule fields**, right-click in the **L** cell and select **SLL (Sum of leg lengths)** from the pop-up menu.

2. Click **Update**.
3. Click **Save**.

Using a formula

You can also use a formula in **Rebar Shape Manager** to calculate the reinforcing bar total length.

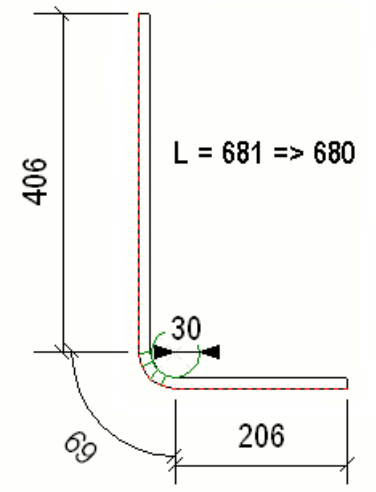
You need to set XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT and XS_USE_USER_DEFINED_REBARSHAPERULES to `TRUE` **File menu** --> **Settings** --> **Advanced options**.

For example, to take the bending radius into account and to calculate the length along the reinforcing bar outer surface, do the following:

1. In **Bending schedule fields**, right-click in the **L** cell and select **(formula)** from the pop-up menu.
2. Enter the following formula for the length calculation: `S1 + S2 + 2*3.14*(RS + DIA)*1/4`

where

- `S1` = straight leg length 1 (`406`)
- `S2` = straight leg length 2 (`206`)
- `RS` = rounding radius (`30`)
- `DIA` = actual diameter (`14`)



Accuracy

The accuracy of reinforcing bar length is defined in the `rebar_config.inp` file. The values can vary in each environment.

For example, the values shown below are from a `rebar_config.inp` file. In the default environment the file is located in `..\ProgramData\Trimble\Tekla Structures\<version>\Environments\default\system\` folder.

The following settings define the accuracy and rounding for the leg lengths:

- `ScheduleDimensionRoundingAccuracy=1.0`
- `ScheduleDimensionRoundingDirection="DOWN"`

The following settings define the accuracy and rounding for the total reinforcing bar length:

- `ScheduleTotalLengthRoundingAccuracy=10.0`
- `ScheduleTotalLengthRoundingDirection="DOWN"`

Note that also XS_USE_ONLY_NOMINAL_REBAR_DIAMETER affects the reinforcing bar length calculation.

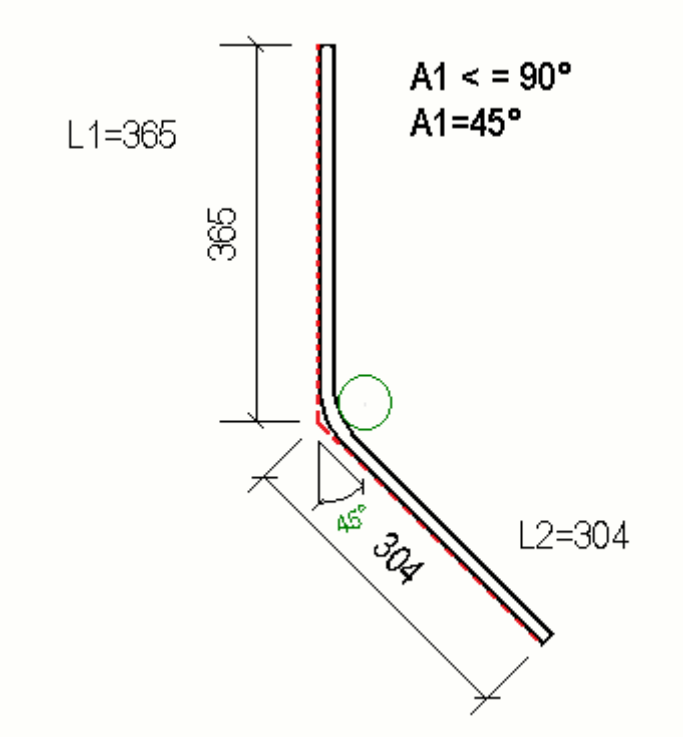
See also

Rebar shape manager in reinforcement shape recognition (page 604)

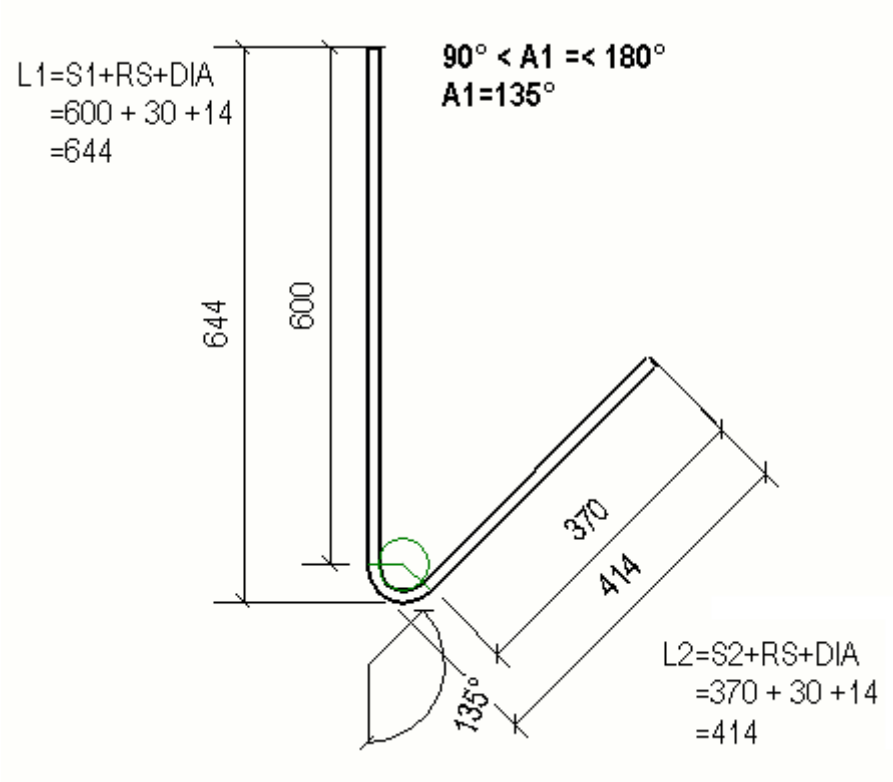
How to calculate the reinforcing bar leg length

How the reinforcing bar leg length is calculated depends on the angle between the bar legs.

- When the angle is <= 90°, the length is measured to the extension of a leg along the outer edge



- When the angle is > 90 ° and <= 180 °, the tangential length is used



The leg lengths are calculated using **Rebar Shape Manager**, where

- `S1` = straight portion of a bar for the first segment
- `S2` = straight portion of a bar for the second segment
- `A1` = bending angle measured between the extension of the first leg and the second leg. The angle is 0° if the second segment continues to the same direction as the first segment (the bar is straight)
- `L1` = leg length for the first reinforcing bar segment
- `L2` = leg length for the second reinforcing bar segment
- `RS` = bending radius
- `DIA` = actual diameter of the reinforcing bar

See also

Rebar shape manager in reinforcement shape recognition (page 604)

Reinforcing bar and bar group properties (page 1061)

Reinforcement shape recognition

Tekla Structures recognizes different reinforcing bar bending shapes and assigns shape codes to them. Tekla Structures then uses the shape and

dimension information in bending schedules, pull-out pictures, templates and reports.

Tekla Structures includes two methods for shape recognition.

User-defined bending shape definitions.	These definitions are created with Rebar shape manager (page 604) and saved in the `RebarShapeRules.xml` file. The file is located in the environment folder under `..\ProgramData\Trimble\Tekla Structures\<version>\Environments\`. The exact file location may vary depending on the folder structure of your environment files.
Tekla Structures internal, hard-coded bending type definitions. Only used if the advanced option XS_USE_USER_DEFINED_REBARSHAPE RULES is set to `FALSE`.	These internal bending types (page 617) of reinforcing bars are mapped to area-specific reinforcing bar bending type codes in the `rebar_schedule_config.inp` file. The file is located in `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system` folder.

See also

Reinforcement in templates (page 641)

Rebar shape manager in reinforcement shape recognition

You can define your own bar bending shapes and assign shape codes with **Rebar shape manager**, and thus increase the amount of recognized bar shapes. User-defined bending shapes are useful when Tekla Structures does not recognize the bending shape and assigns the `UNKNOWN` bending type to the shape.

Rebar shape manager is meant for users who need to customize the bending shapes according to company or project requirements.

With **Rebar shape manager** you can:

- Customize the existing bending shapes and create new bending shapes (page 605).
- Establish your own rules (page 608) for defining the bending shapes.

- Compare selected bars (page 614) in the model to the existing bending shapes.
- Customize your own dimension mappings that are used in templates and reports (page 615).
- Import and export user-defined bending shapes.
- Use user-defined bending shapes in bending schedules and pull-out pictures.

NOTE **Rebar shape manager** is a tool for recognizing reinforcing bar shapes. You cannot control the reinforcing bar creation properties, such as cover thickness, reinforcing bar grade, or size, with this tool.

See also

Tips for reinforcement shape recognition in Rebar shape manager (page 616)

Define reinforcing bar bending shapes in Rebar shape manager

With **Rebar shape manager** you can establish your own rules for defining the bending shapes. When you define your own reinforcing bar bending shapes and shape codes, an `.xml` file called `RebarShapeRules.xml` is created in the current model folder.

In addition, Tekla Structures installation contains by default another `.xml` file called `RebarShapeRules.xml`. This file contains the most typical bending shapes in your environment, and it is located in the environment folder under `..\ProgramData\Trimble\Tekla Structures\<version>\Environments\`. The exact file location may vary depending on the folder structure of your environment files.

When you define new shapes, the shapes in the default `RebarShapeRules.xml` rule file can be appended to your own shapes. Tekla Structures reads valid `RebarShapeRules.xml` rule files in the model, project, firm, and system folders in that order. When the shape codes and report field values are applied, Tekla Structures uses the first matching shape in a `RebarShapeRules.xml` file which is found first based on the search order. If a shape is unknown according to the definitions in the first `RebarShapeRules.xml` file, only then the next rule file (or files) will be used. All the found bending shapes are displayed in **Rebar shape manager**.

1. Select reinforcing bars in the model.
2. On the **File** menu, click **Editors** --> **Rebar shape manager**.

   Rebar shape manager opens and lists the selected reinforcing bars in the **Model rebars** list.

   Alternatively, you can first open **Rebar shape manager** and then select reinforcing bars in the model. Click **Get selected** to add the reinforcing bars to the **Model rebars** list.

- The **Model rebars** list shows the ID and the shape code of the selected reinforcing bars.
- The **Shape catalog** list shows the shapes that exist in the default `RebarShapeRules.xml` rule file.
- The **Tolerances** tab shows the tolerances used when the bending shape rules are compared.

3. Select one unknown shape from the **Model rebars** list.

   Tekla Structures shows a preview of the shape. The blue numbers in the preview are related to the straight bar legs in the shape, and the green numbers to the arc legs.

   Under the preview, Tekla Structures shows a description of the recognized bar geometry, and the number of bendings and arcs. If you move the mouse pointer over the text, a tooltip shows which tolerance values and other properties are in use for the selected bar. If the bar geometry has been simplified in the recognition process, **(Simplified)** is shown next to the description text, and the tooltip shows the simplification result.

4. If needed, you can compare selected bars (page 614) in the model to the existing bending shapes in the rule files.

5. To define the needed information for a bending shape, do the following:

To define	Do this
Tolerances	Enter tolerance values for the following measurements: • **Dimension** (leg lengths and other distances) • **Angle** (bending and twist angles) • **Radius** (bending radiuses) Note that the tolerance values are stored along with the rule file `RebarShapeRules.xml`, and thus the tolerances are specific to each rule file. The following tolerances are also used in rebar shape recognition, and they can be adjusted using the `XS_REBAR_RECOGNITION_...` advanced options, the rebar catalog (`rebar_database.inp`), and the **Shape recognition** user-defined attributes of individual reinforcement objects: • Extra point shortening • Curve tolerance • Max curve radius requiring bending Additionally, you can use the **Recognize as straight bar** user-defined attribute to have Tekla Structures recognize individual reinforcement objects as straight bars even though they are bent in the model.

To define	Do this
Shape code	Enter a shape code for an unknown shape. Note that multiple reinforcement shapes that are variants of the same shape can have the same **Shape code** but different **Bending shape rules**.
Bending shape rules	If the bending shape rules that **Rebar shape manager** defines automatically are not sufficient to distinguish certain bending shapes, you can manually add (page 608) new bending shape rules. Add or delete a bending shape rule by clicking the **Add** and **Delete** buttons on the right. Use the **Reset** button to restore the original values.
Check hooks	Select the checkbox if you want to define different shape codes or bending schedule fields for two bars that otherwise have exactly the same geometry, but one bar has hooks and the other one does not. If you select the checkbox, hooks are considered as hooks. If you clear the checkbox, hooks are considered as normal legs. Note that the **Check hooks** option works independently from the advanced option , and it allows bars that have different hooks to have different shape code or schedule fields regardless of the value of the advanced option.
Update	Update the existing shape code definition of the selected reinforcing bar. You can update the definition if you have modified the shape code, bending shape rules or the contents of bending schedule fields.
Bending schedule fields	Define the content (page 615) for a bending schedule. Right-click a field to select a bending shape property or to enter a formula. The names of the **Bending schedule fields** (**A**, **B**, and so on) are used in templates and reports. To make sure that old reports also work correctly, we recommend that you use the same `DIM_XX` fields as in the `rebar_schedule_config.inp` file.
Schedule fields...	Click the **Schedule fields** button to add, remove or change the order of the available schedule fields. If needed, you can reset the schedule fields to original defaults. If you change the set of available schedule fields and update an existing shape, the old schedule fields which do not exist anymore will be cleared. Therefore we recommend that you do not to remove any of the default schedule fields unless

To define	Do this
	you are sure they have not been used in any of the existing shapes. You can change the names of the existing schedule fields or, if you add new fields, give names to them. To use the fields in templates and reports, use the `DIM_XX` or `ANG_xx` fields, and replace `xx` with the schedule field name.

6. When you have finished defining the new shape, click **Add** to add the bending shape definition to the `RebarShapeRules.xml` file.

   To enable the **Add** button you need to change the bending shape rule, enter a shape code or select the **Check hooks** checkbox.

7. Click **Save** to save the `RebarShapeRules.xml` file.

   By default, the file is located in the current model folder.

   When you create, for example, a bending schedule, Tekla Structures uses the updated bending shape information, recognizes the added bending shape and assigns a correct shape code to it.

NOTE **Rebar shape manager** is able to recognize bending shapes regardless of the modeling direction of the bars. This means that the modeling direction has no effect on the shape definition and the shape code.

When defining the bending shapes, the start or the end of modeling direction is always sorted based first on bending angles, then on twist angles, and finally on leg lengths. However, bending radius is not taken into account in the sorting. This means that radius 1 may not always be less than radius 2 or vice versa.

After the sorting, the bars that are not recognized with their original input point order are automatically examined also in the reversed input point order. This eliminates the need to create duplicate definitions for certain bar shapes.

See also

Rebar shape manager in reinforcement shape recognition (page 604)

Add new bending shape rules manually in Rebar shape manager

In some cases, the bending shape rules that are defined in **Rebar shape manager** are not sufficient to distinguish certain bending shapes. If needed, you can manually add new bending shape rules for reinforcing bars in **Rebar shape manager**.

1. In **Rebar shape manager**, click **Add** next to the **Bending shape rules** list.

2. In the **New bending rule** dialog, select the options from the lists to define the new rule.

   The content of the lists depends on the shape and the bending of the reinforcing bar.

3. Click **OK** to add the new rule to the **Bending shape rules** list.

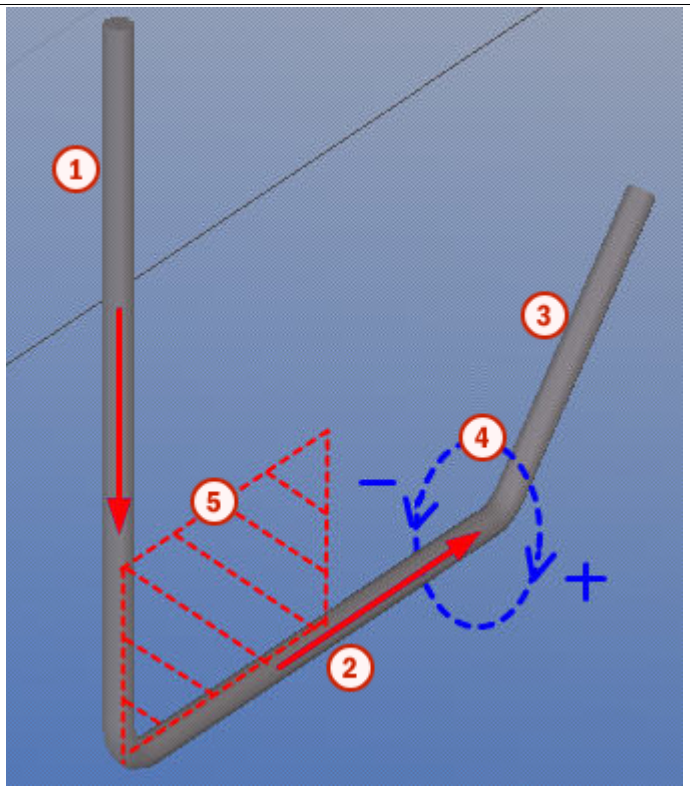
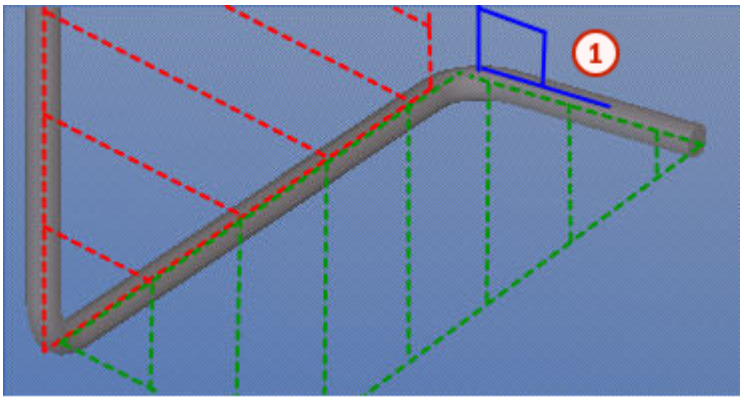
   The **OK** button is enabled only when the rule is valid.

Bending shape rule settings

All the rule options are available in the **New bending rule** dialog, even though only certain selections are valid, depending on the type of the conditions used. The left and right condition of a rule need to be of the same type. The values in the parentheses are the values that were used to create the bar shape.

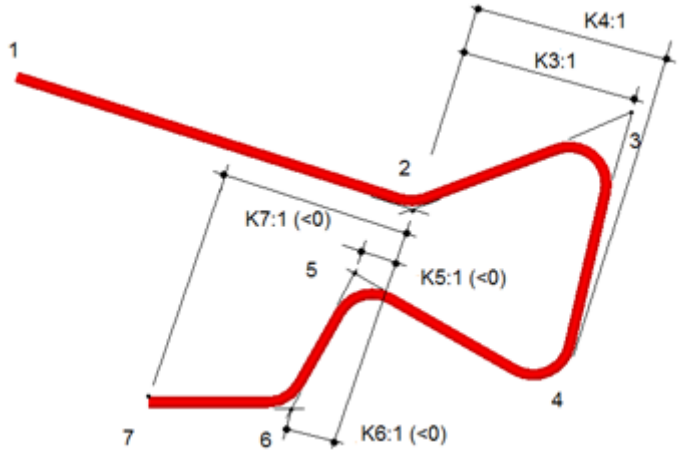
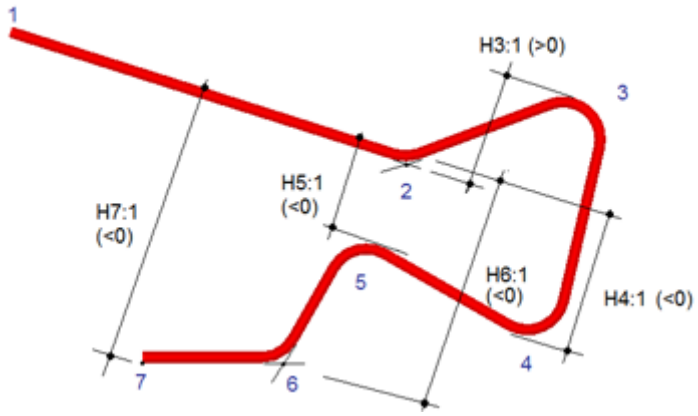
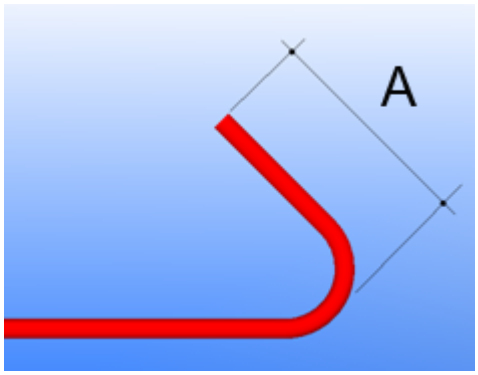
Use the **New bending rule** dialog in **Rebar shape manager** to manually define rules for reinforcing bar bending shapes.

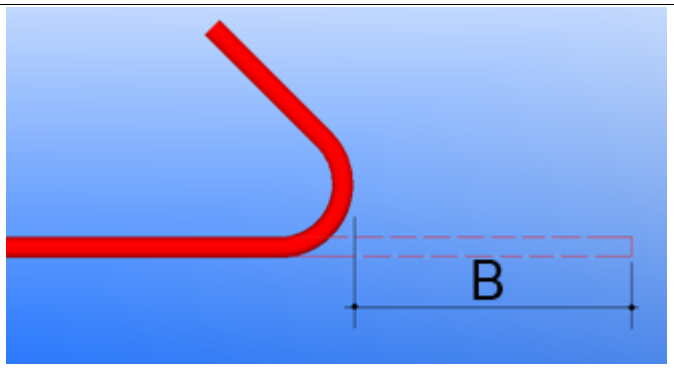
Option	Description
Angle (A)	Bending angle between the legs. Bending angle is always between 0 and +180 degrees. The angle cannot be negative.
Twist angle (T)	Rotation angle of a plane that has been created by two bars of consecutive lengths. The plane is rotated around the axis of the last bar creating the plane. For bars where all the legs lie in the same plane, the twist angle is either 0 degrees or +180 degrees. If the bar twists out of the plane, i.e. the bar is in 3D, the twist angle is between -180 and +180 degrees.

Option	Description
	 1. Leg 1 2. Leg 2 3. Leg 3 4. Twist angle direction 5. Plane created by legs 1 and 2
Twist angle example	The twist angle between two planes is +90 degrees. The planes are created by legs 1 and 2, and legs 2 and 3.  1. Twist angle: +90 degrees

Option	Description
Radius (R, RX)	Bending radius of the bending. `(RX) Radius *` is the value of the bending radius when all the bendings have equal radius. Otherwise the value is zero (0). `Radius * = Radius 1` ensures that all the bendings have been created using the same radius.
Bending length (BL)	Center line length of the bending.
Straight length (S)	Straight length between the start and the end of adjacent bendings. The rule is generated only when there is no straight part, for example, `Straight length 2 = 0`.
Leg length (L)	Length of the leg.
Leg (V)	Leg direction as a vector value.
Bar diameter (DIA, DIAX)	Diameter of the reinforcing bar.
Nominal diameter (NDIA, NDIAX)	Nominal diameter of the reinforcing bar.
Center line length (CLL)	Leg length according to the center line.
Sum of leg lengths (SLL)	Sum of all leg lengths.
Reversed	Reversed reinforcing bar. You can use **Reversed** to have additional bending shape rules and/or formulas for the schedule fields. When used in a rule, you can have separate definitions in shape code and/or schedule fields for reinforcing bars that have different modeling order of the points. When used as a part of a formula, you can eliminate the automatic normalization of the modeling order of the points. For example, a formula `if (REVERSED) then L2 else L3 endif` forces the content of the field to show the desired leg length depending on the order of the points or legs.
Arc inner radius (RI)	Inner radius of the arc.
Arc outer radius (RO)	Outer radius of the arc.
Arc angle (AA)	Angle of the arc.
Arc length (AL)	Length of the arc.

Option	Description
Arc twist angle (AT)	Rotation angle of the arc.
Arc width (AW)	Width of the arc.
Arc height (AH)	Height of the arc.
Curve width (CW)	Extreme width of the curved bar.
Curved height (CH)	Extreme height of the curved bar.
RFACTOR	Relative radius.
LFACTOR	Relative length.
Spiral rounds (SR)	Rounds of the spiral bar.
Spiral pitch (SP)	Pitch of the spiral bar.
Spiral length (SL)	Distance between the reference points of the spiral bar.
Spiral total length (STL)	Total length of the spiral bar when the bar is installed on site.
Standard radius (RS)	Standard minimum bending radius. The bending radius depends on the size and the grade of the bar.
Weight per length (WPL)	Weight per leg length.
Leg distance from leg (D)	Similar to **Point/arc distance off from leg (H)**. The difference is that **Point/arc distance off from leg (H)** considers the bending radius, whereas **Leg distance from leg (D)** is measured from the sharp corner. When the legs are parallel, both **Leg distance from leg (D)** and **Point/arc distance off from leg (H)** give the same result.
Point/arc distance along leg (K)	Distance parallel to a leg from outer edge to outer edge, or tangential to the bending. The distances are positive or negative depending on the leg direction. Example:

Option	Description
	
Point/arc distance off from leg (H)	Distance perpendicular to a leg from outer edge to outer edge, or tangential to the bending. The distances are positive or negative depending on the leg direction. Example: 
Start hook (SH) **End hook (EH)** **Hook angle (SHA, EHA)** **Hook radius (SHR, EHR)** **Hook straight length (SHS, EHS)**	Start and end hook properties. Use method A or B for the hook length calculation: 

Option	Description
Hook length, method A (SHLA, EHLA) **Hook length, method B (SHLB, EHLB)**	
Constant angle	Constant value of the angle. Enter the value in the rightmost box.
Constant radius	Constant value of the radius. Enter the value in the rightmost box.
MAXCURVE	Maximum radius of an arc that requires bending. When an arc's bending radius exceeds **MAXCURVE**, the arc is treated as a straight leg.
Custom properties, template attributes, user-defined attributes	The custom properties, template attributes, and user-defined attributes defined in the `RebarShapeManager.CustomProperties.dat` file appear at the end of the list and can be used like any other option.

See also

Define content for templates and reports in Rebar shape manager (page 615)

Rebar shape manager in reinforcement shape recognition (page 604)

Compare reinforcing bars to bending shapes in Rebar shape manager

In **Rebar shape manager**, you can compare selected reinforcing bars in the model to the existing bending shapes in the `RebarShapeRules.xml` files.

Comparing is useful, for example, when you want to check if a bar with a special shape complies with any of the bending shapes or shape rules, or when you want to check why a bar is not recognized as you would expect. You can then modify either the bar shape in the model or the bending shape rules as needed.

1. Select reinforcing bars in the model.
2. On the **File** menu, click **Editors** --> **Rebar shape manager**.

Rebar shape manager opens and lists the selected reinforcing bars in the **Model rebars** list.

Alternatively, you can first open **Rebar shape manager** and then select reinforcing bars in the model. Click **Get selected** to add the reinforcing bars to the **Model rebars** list.

3. In the **Model rebars** list, select the bar shape that you want to compare.
4. In the **Shape catalog** list, select an existing bending shape.
5. Under the **Shape catalog** list, click **Compare to selected**.

In the **Bending shape rules** list, you can see the passed rules in green and the failed rules in red. An additional warning dialog is shown if there are other reasons for mismatches between the shapes.

See also

Rebar shape manager in reinforcement shape recognition (page 604)

Define reinforcing bar bending shapes in Rebar shape manager (page 605)

Add new bending shape rules manually in Rebar shape manager (page 608)

Define content for templates and reports in Rebar shape manager

Use the **Bending schedule fields** in **Rebar shape manager** to define the content for templates and reports. Each of the **Bending schedule fields** cells can contain a shape property or a formula.

When you right-click in a **Bending schedule fields** cell, you can:

- Select a shape property from the list. The content of the list depends on the reinforcing bar geometry.
- Select the **(empty)** option to clear the content of the current cell.
- Select the **(formula)** option to enter a formula. The variables in the formula can either be the shape properties visible in the pop-up menu, or direct references to other non-empty bending schedule field cells.

  You can use the following functions in the formulas, in the same way as in custom components:

  - Mathematical functions
  - Statistical functions
  - Trigonometric functions

    When you map angles and trigonometric functions in the **Schedule field formula** dialog, enter the functions (sin, cos, tan) in small letters,

for example `sin(A1)`. Capital letters are not recognized, and a blank space will appear in reports.

If you have angles in a formula, the formula needs to be in radians. For example, if you want to subtract 180 degrees from angle A1, enter `A1-PI` in capital letters. If you enter `A1-180` or `A1-pi`, the formula does not work.

Bending schedule fields cell shows the result of a valid formula. If the formula is not valid, a question mark and text describing the error is shown.

NOTE Use the fields **S**, **T**, **U** or **V** to report angles. If you do not use these fields, you need to override the default unit settings in the **Template Editor**.

Example

Formula is `L1+L3+L5-2*DIA`

- `L1`, `L3` and `L5` are the leg lengths measured from outer edge to outer edge
- `H1` is the total width
- to achieve `H1`: `L1+L3+L5 minus 2*bar diameter`

See also

Add new bending shape rules manually in Rebar shape manager (page 608)

Rebar shape manager in reinforcement shape recognition (page 604)

Tips for reinforcement shape recognition in Rebar shape manager

The reinforcing bar shape recognition is based on each shape's bending shape rules. The shapes and their rules are listed in the `RebarShapeRules.xml` file, located in the environment folder under `..\ProgramData\Trimble\Tekla Structures\<version>\Environments\` by default. (The exact file location may vary depending on the folder structure of your environment files.) Sometimes a shape matches the rules of two shapes, and Tekla Structures does not recognize the reinforcing bar shape as expected.

NOTE The most convenient way to ensure that a shape is recognized correctly is to modify the shape definition by adding more rules (page 608) to the shape in **Rebar shape manager**.

However, if necessary, you can manually modify the `RebarShapeRules.xml` file and thus affect the shape recognition. When Tekla Structures recognizes the shape, the order of the shapes in the `RebarShapeRules.xml` is important:

- The first shape that matches the rules is the one that Tekla Structures recognizes as the shape. If you need to change the order of the shapes to change how Tekla Structures recognizes the shape, you can do it by manually modifying the `RebarShapeRules.xml` file. If you modify the file

and change the order of the shapes, make sure that the structure of the file stays valid.

- You can divide the shape definitions in several `RebarShapeRules.xml` files and different folders. Tekla Structures searches the `RebarShapeRules.xml` file in the model, project, firm, and system folders in that order. Tekla Structures uses the first matching shape in the first `RebarShapeRules.xml` file which is found based on the search order.

See also

Rebar shape manager in reinforcement shape recognition (page 604)

Hard-coded bending type identifiers in reinforcement shape recognition

Tekla Structures recognizes different reinforcing bar bending shapes and assigns bending type identifiers to them.

NOTE This bending shape recognition method and the following bending types are only used if the advanced option XS_USE_USER_DEFINED_REBARSHAPERULES is set to `FALSE`.

We recommend having `XS_USE_USER_DEFINED_REBARSHAPERULES` set to `TRUE`, which is the default value, and using Rebar shape manager (page 604) instead in bending shape recognition.

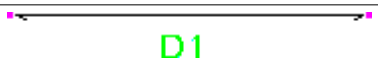
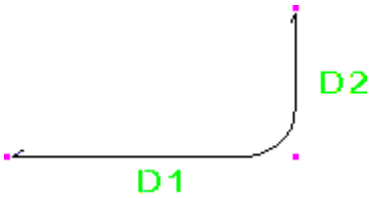
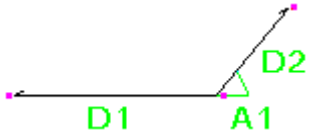
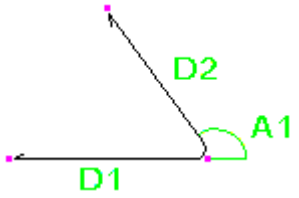
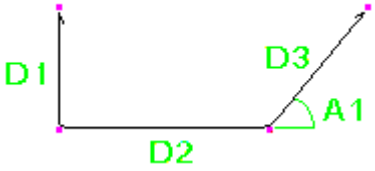
The bending type identifiers in the table below are internal, hard-coded types of Tekla Structures. The leg dimensions (`D1`, `D2`, and so on) and bending angles (`A1`, `A2`, and so on) of reinforcing bars are Tekla Structures internal dimensions and angles. You can map the Tekla Structures internal bending types, for example, to country- or project-specific bending types, and the Tekla Structures internal dimensions and angles to specific template attributes. You can do this in the `rebar_schedule_config.inp` file.

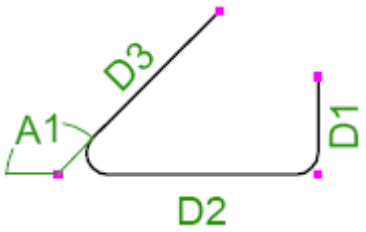
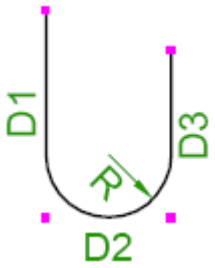
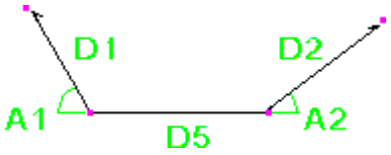
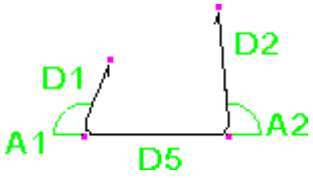
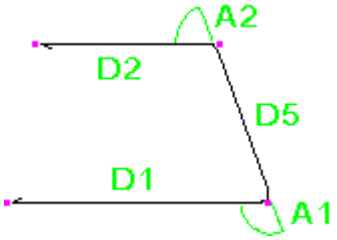
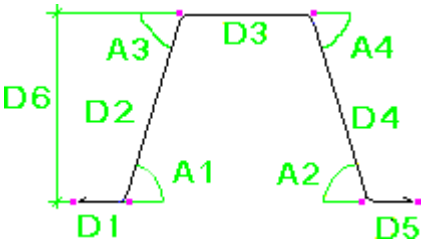
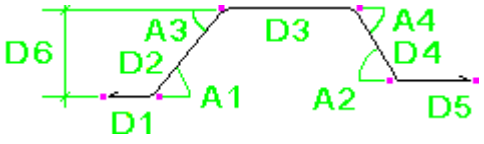
Reinforcing bar bending dimensions are calculated so that the leg dimensions (`D1`, `D2`, and so on) follow the outer edge, or the edge extension, of the reinforcing bar. The total length is calculated according to the center line of the reinforcing bar.

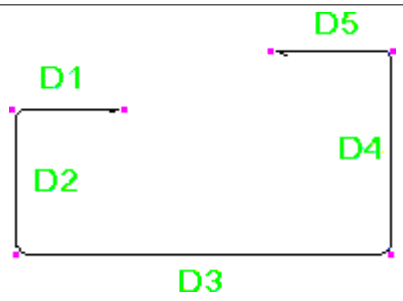
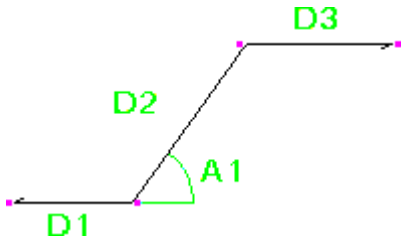
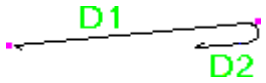
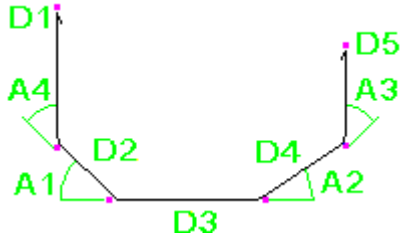
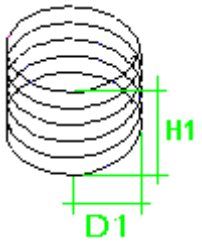
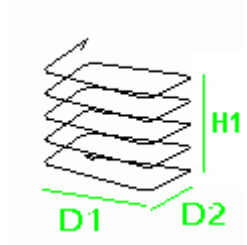
If Tekla Structures does not recognize the shape of a reinforcing bar, it assigns the `UNKNOWN` bending type to it.

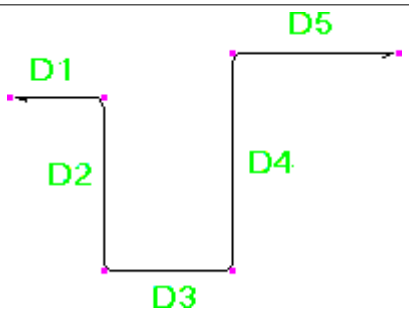
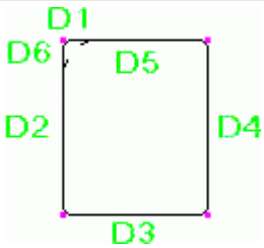
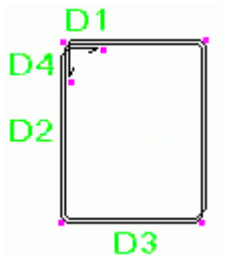
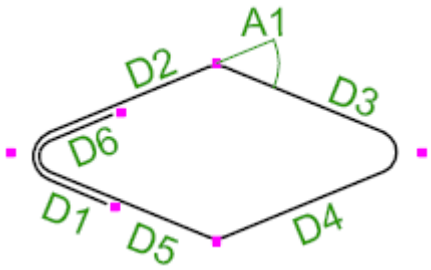
The magenta points in the images in the table below represent the points you pick in the model when you create reinforcing bars.

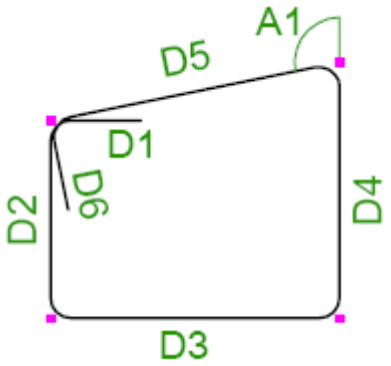
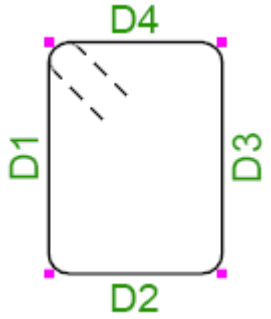
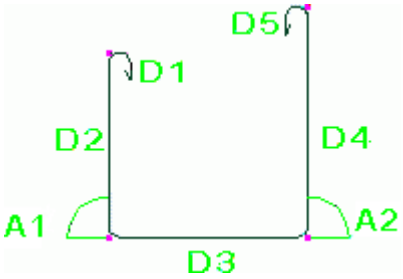
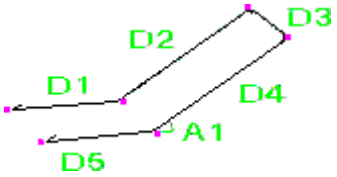
NOTE If you want to customize the hard-coded bending shapes or define new bending shapes, use Rebar shape manager (page 605).

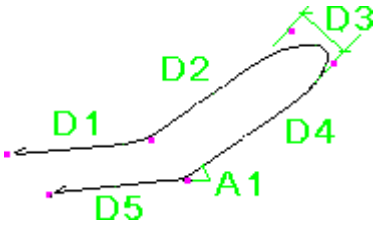
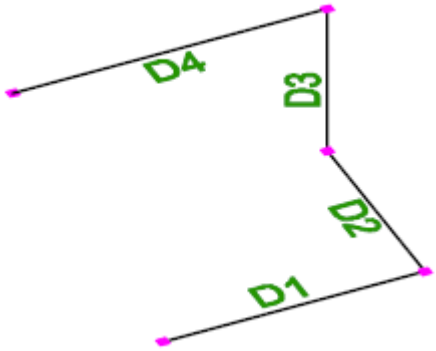
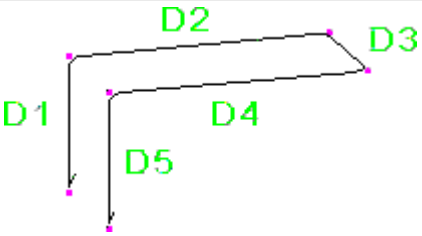
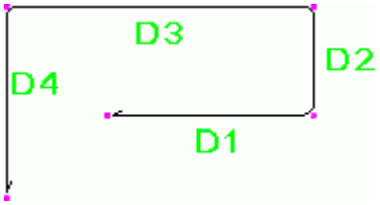
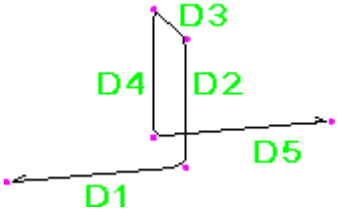
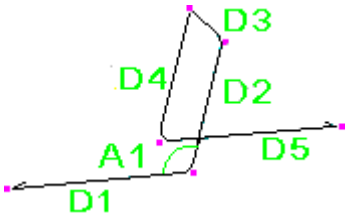
Bending type identifier	Bending shape
1	
2_1	 Requires standard bending radius.
2_2	 Non-standard bending radius.
3_1	
3_2	
4	
4_2	

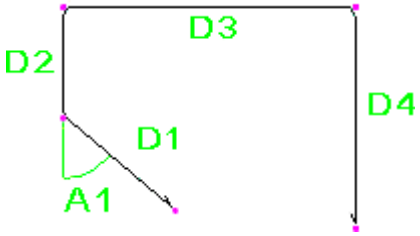
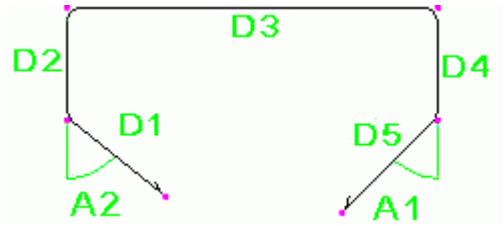
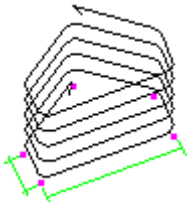
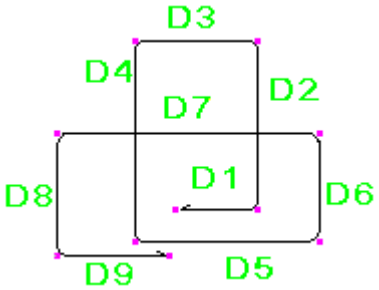
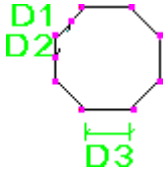
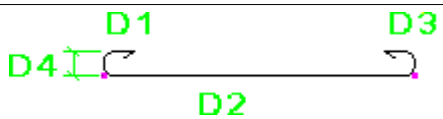
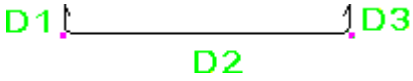
Bending type identifier	Bending shape
4_3	
4_4	
5_1	
5_2	
5_3	
6_1	
6_2	

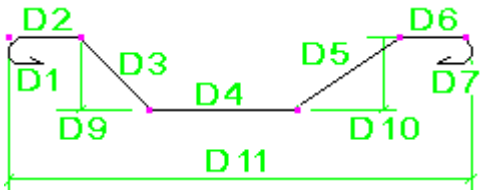
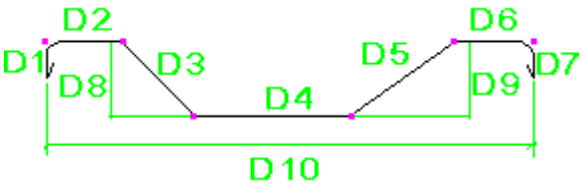
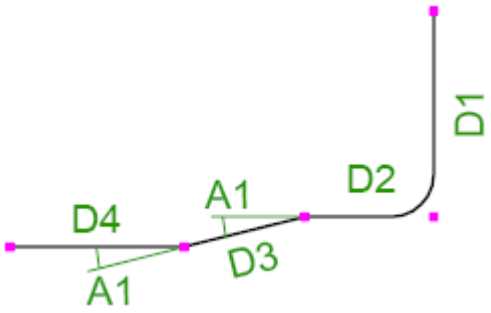
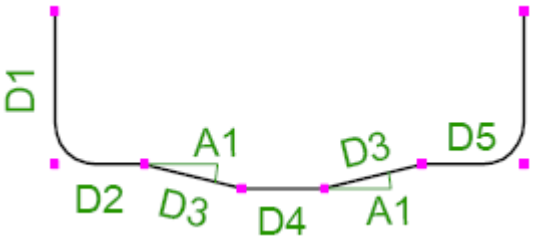
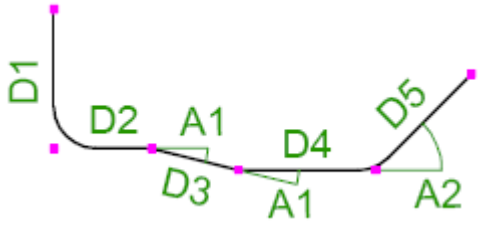
Bending type identifier	Bending shape
7	
8	
9	 Requires 180 degree hook.
10	
11	 D1 = Radius from center of circle to the center line of reinforcing bar.
12	

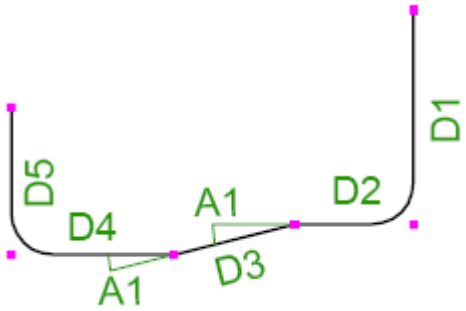
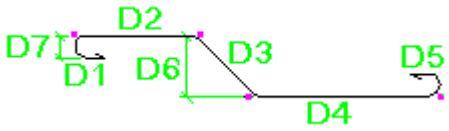
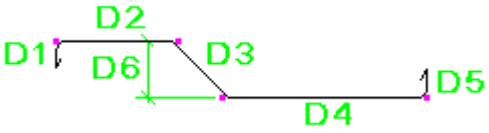
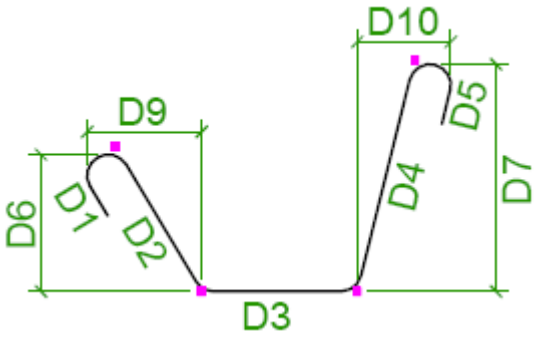
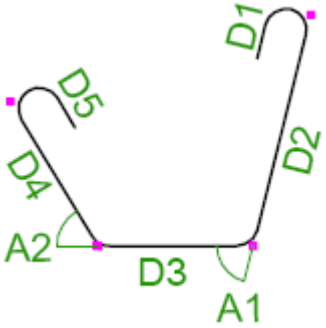
Bending type identifier	Bending shape
13	 Can also be modeled using hooks at both ends (i.e. model D1 and D5 using 90 degree hooks).
14	 Requires 90 degree hooks at both ends.
14_2	
14_3	

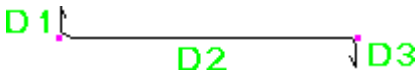
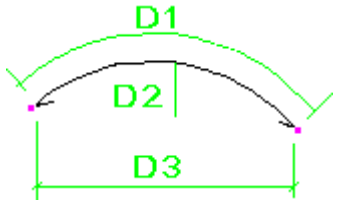
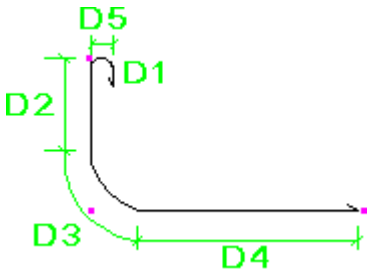
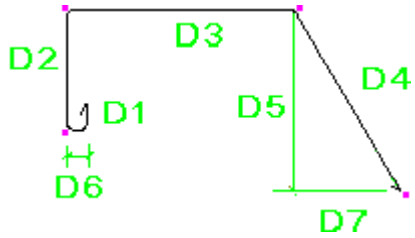
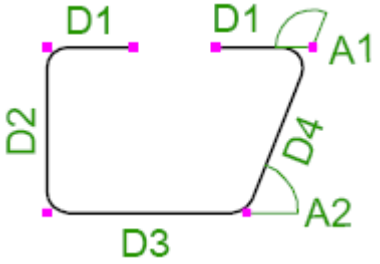
Bending type identifier	Bending shape
14_4	 Requires 90 degree hooks at both ends.
14_5	 Recognized when the start point and end point are in the same location and no hooks are used. If `XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION` is set to `FALSE`, reinforcing bars with hooks (types 14 and 48) are recognized as 14_5.
15	 Requires hooks at both ends.
16_1	

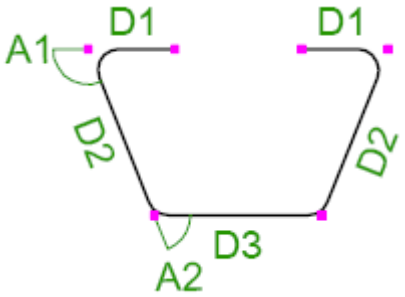
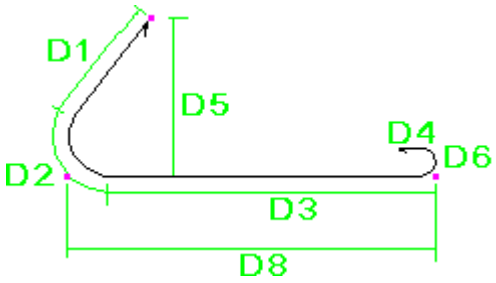
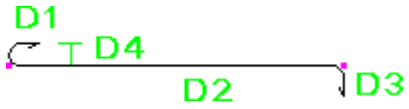
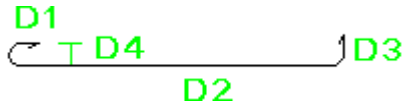
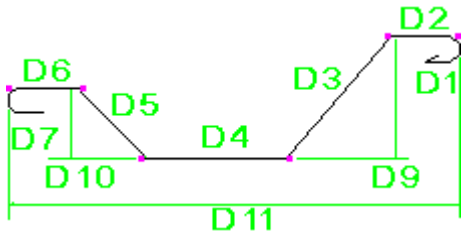
Bending type identifier	Bending shape
16_2	
17	
18	
19	
20_1	
20_2	

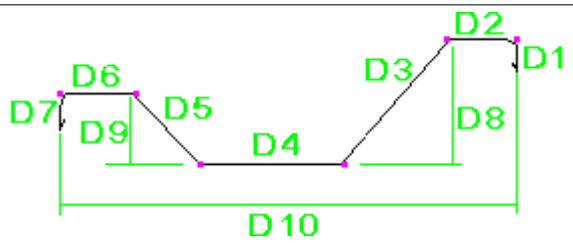
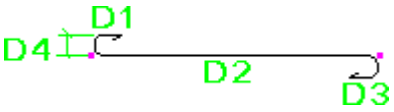
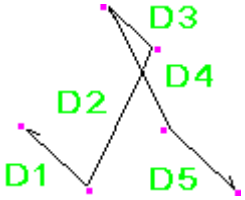
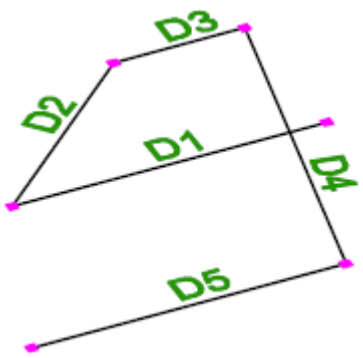
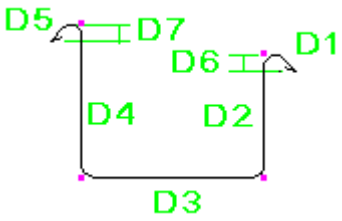
Bending type identifier	Bending shape
21	
22	
23	
24	
25	
26	 Requires 180 degree hooks at both ends.
27	 Requires 90 degree hooks at both ends.

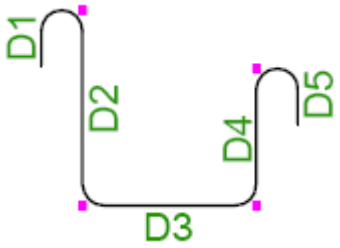
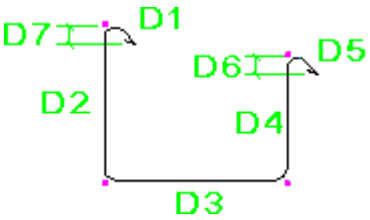
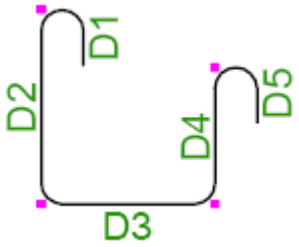
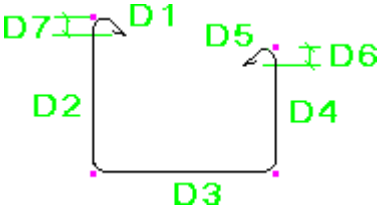
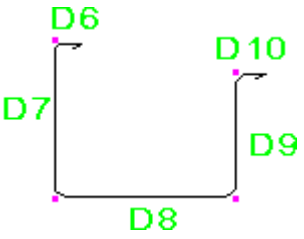
Bending type identifier	Bending shape
28	 Requires 180 degree hooks at both ends.
29	 Requires 90 degree hooks at both ends.
29_2	
29_3	
29_4	

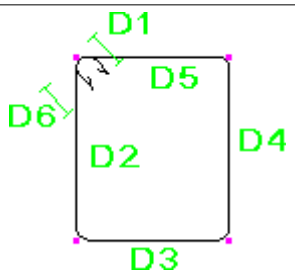
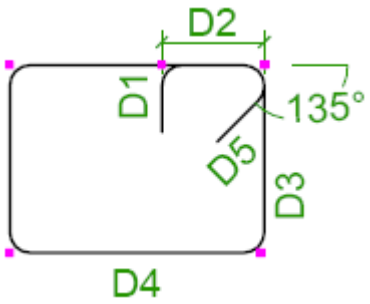
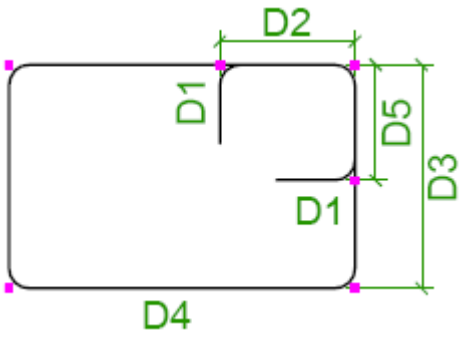
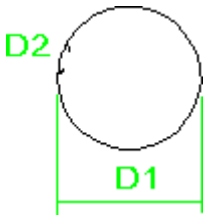
Bending type identifier	Bending shape
29_5	
30	 Requires 180 degree hooks at both ends.
31	 Requires 90 degree hooks at both ends.
32	 Requires 180 degree hooks at both ends.
32_2	

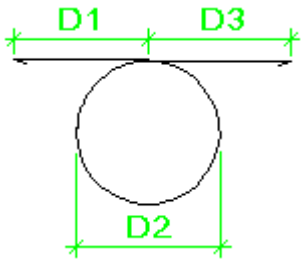
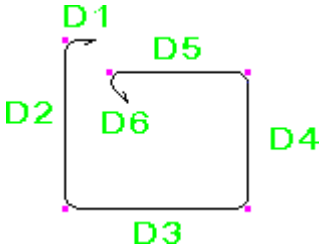
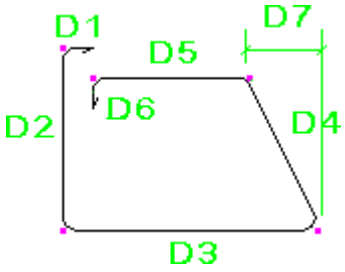
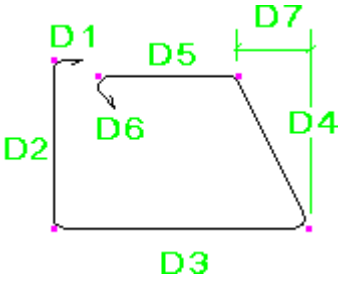
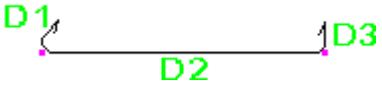
Bending type identifier	Bending shape
33	 Requires 90 degree hooks at both ends.
34	
35	 Requires 180 degree hook.
36	 Requires 180 degree hook.
36_2	 Can also be modeled using hooks at both ends.

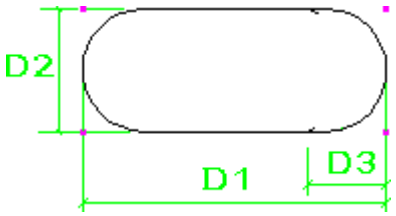
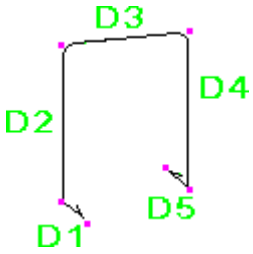
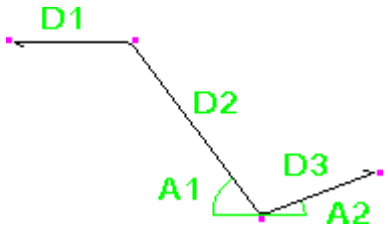
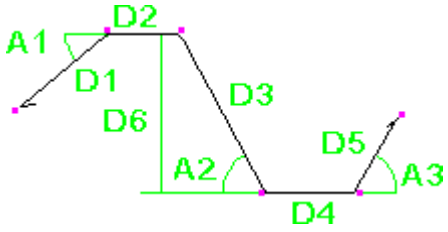
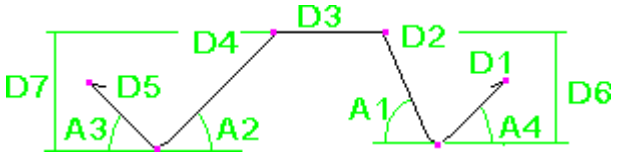
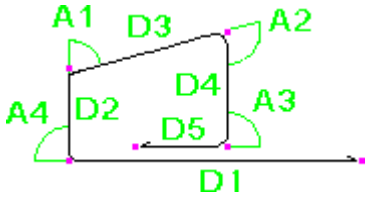
Bending type identifier	Bending shape
36_3	 Can also be modeled using hooks at both ends.
37	 Requires 180 degree hook.
38	 Requires 180 degree hook at one end and 90 degree hook at the other end.
38_2	
39	
40	 Requires 180 degree hooks at both ends.

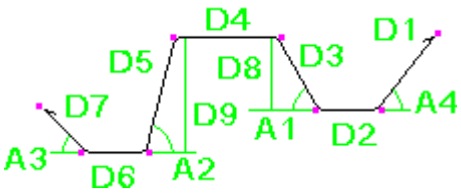
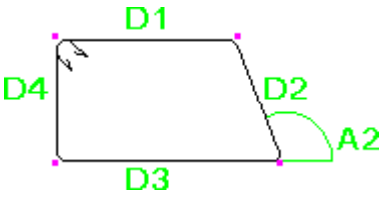
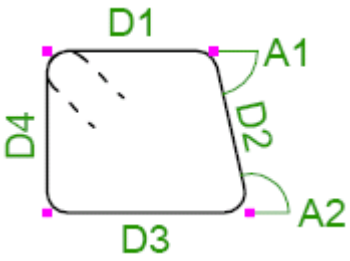
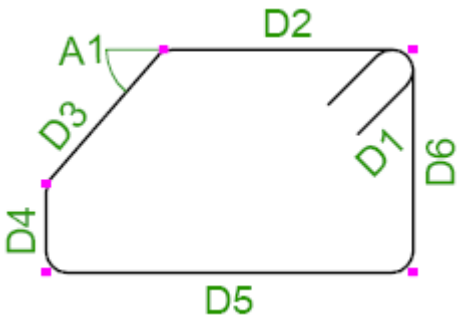
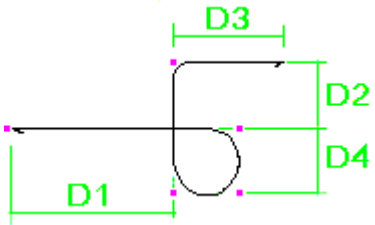
Bending type identifier	Bending shape
41	 Requires 90 degree hooks at both ends.
42	 Requires 180 degree hooks at both ends.
43	
43_2	
44	 Requires hooks at both ends.

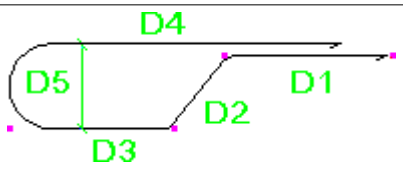
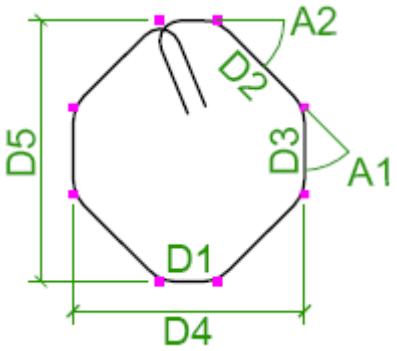
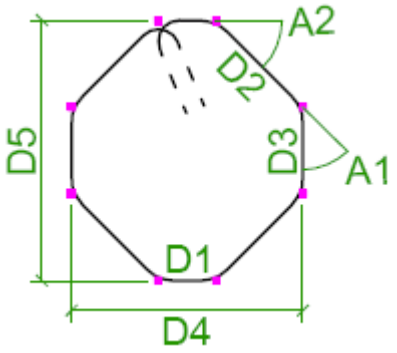
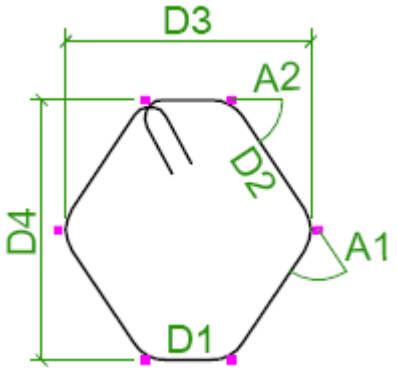
Bending type identifier	Bending shape
44_2	 Requires 180 degree hooks at both ends.
45	 Requires hooks at both ends.
45_2	 Requires 180 hooks at both ends.
46	 Requires hooks at both ends.
47	 Requires 90 degree hooks at both ends.

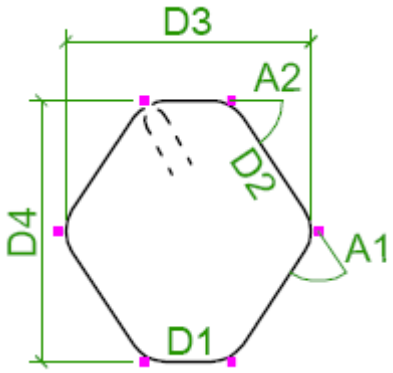
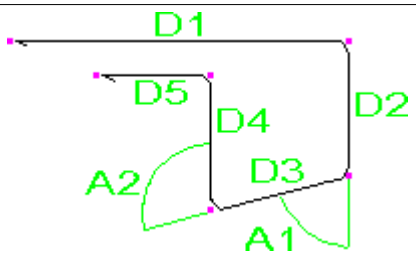
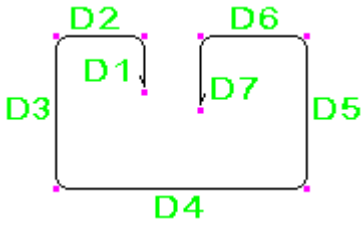
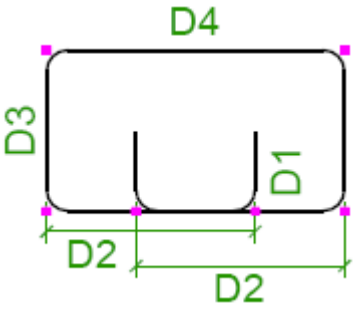
Bending type identifier	Bending shape
48	 Requires hooks at both ends.
48_2	 Requires hooks at both ends.
48_3	
49	 D1 = Reinforcing bar center line diameter.

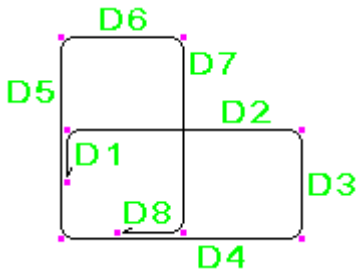
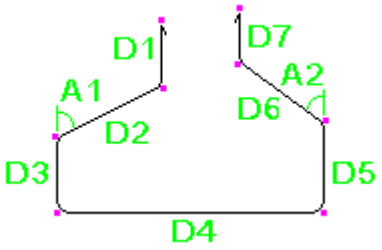
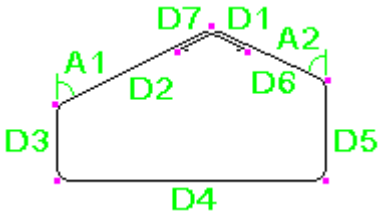
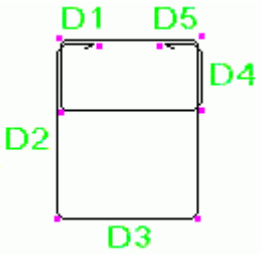
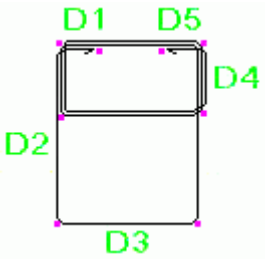
Bending type identifier	Bending shape
49_2	
50	 Requires hooks at both ends.
51	 Requires 90 degree hooks at both ends.
52	 Requires hooks at both ends.
53	 Requires hooks at both ends.

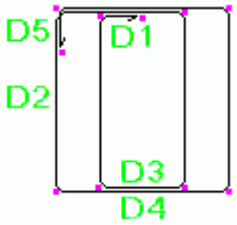
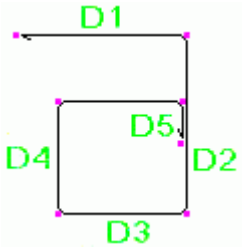
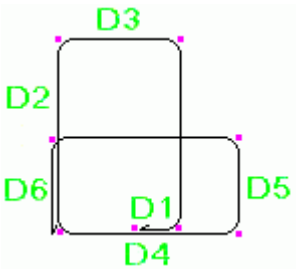
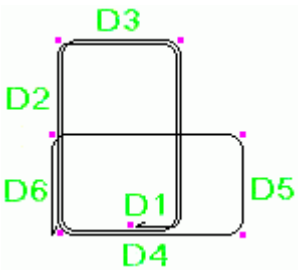
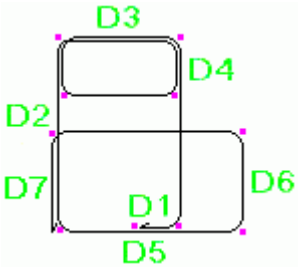
Bending type identifier	Bending shape
54	 Requires hooks at both ends.
55	
56	
57	
58	
59	

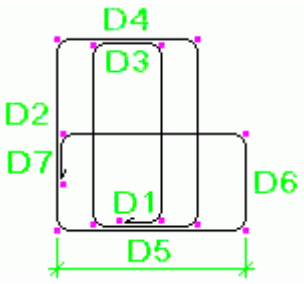
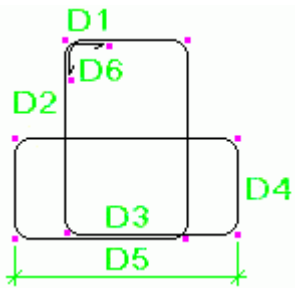
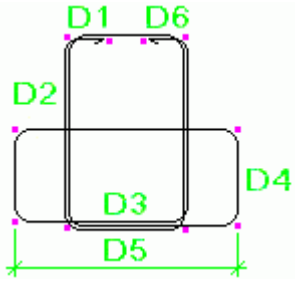
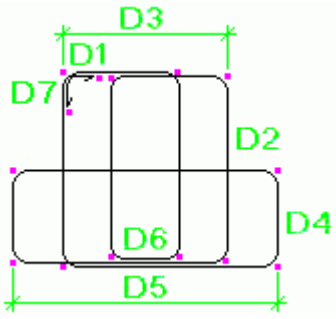
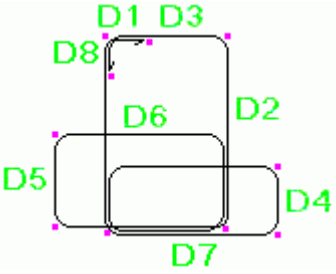
Bending type identifier	Bending shape
60	
61	 Requires hooks at both ends.
61_2	 Recognized if `XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION` is set to `FALSE`.
61_3	 Requires hooks at both ends.
62	 Requires hook.

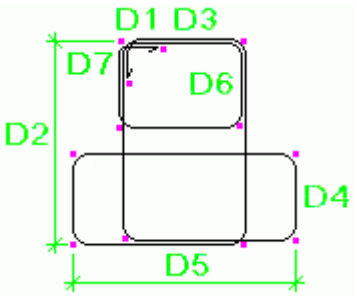
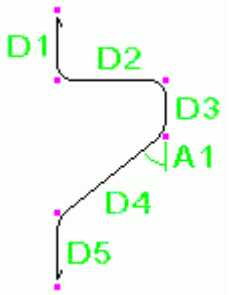
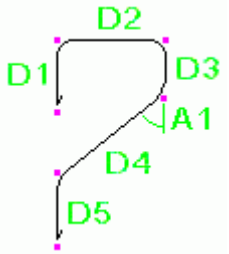
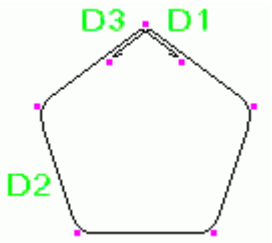
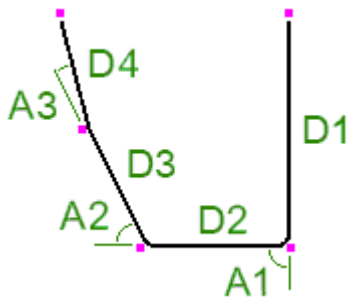
Bending type identifier	Bending shape
63	 Requires hook.
64	 Requires hooks at both ends.
64_2	 Recognized if `XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION` is set to `FALSE`.
65	 Requires hooks at both ends.

Bending type identifier	Bending shape
65_2	 Recognized if `XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION` is set to `FALSE`.
66	
67	
67_2	

Bending type identifier	Bending shape
68	
69_1	
69_2	
70_1	
70_2	

Bending type identifier	Bending shape
71	
72	
73_1	
73_2	
73_3	

Bending type identifier	Bending shape
74	
75_1	
75_2	
76	
77	

Bending type identifier	Bending shape
78	
79_1	
79_2	
80	
UNKNOWN	For example: 

See also

Reinforcement in templates (page 641)

Reinforcement shape recognition (page 603)

Reinforcement in templates

Sometimes you need to localize reinforcing bar bending types or to create templates for reinforcing bar bending schedules.

NOTE If you want to customize the hard-coded bending shapes or define new bending shapes, use **Rebar shape manager**. See Define reinforcing bar bending shapes in Rebar shape manager (page 605).

Reinforcement templates

You can show dimensions, bending angles, and bending types of reinforcing bars in drawings and reports by including reinforcement-specific attributes, such as `DIM_A`, `ANG_S`, `SHAPE`, and `SHAPE_INTERNAL`, in template fields. For more information on creating templates, see Template Editor User's Guide.

Mapping dimensions

Use the `rebar_schedule_config.inp` file in the `..\ProgramData\Trimble\Tekla Structures\<version>\environments\<environment>\system` folder to map

- Tekla Structures internal reinforcing bar dimensions and angles with specific template attributes
- Tekla Structures internal reinforcing bar bending types with specific bending types

These mappings are environment-specific by default. You can modify them to suit your company or project needs.

You can use equations, functions, and `if` statements to calculate the dimensions and angles you need to show.

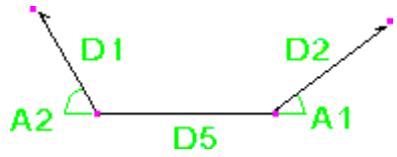
Use any standard text editor (for example, Microsoft Notepad) to edit the `rebar_schedule_config.inp` file.

Examples

The following example of the `rebar_schedule_config.inp` file maps the internal bending type `5_1` to the bending type identifier `E`, and the leg dimensions and bending angles to specific template attributes.

rebar_schedule_config.inp

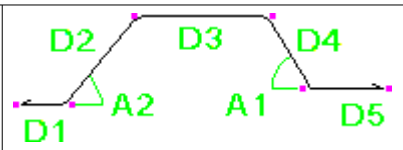
```
BEND_TYPE_5_1[1]="E"
BEND_TYPE_5_1[2]="DIM_A=D1"
BEND_TYPE_5_1[3]="DIM_B=D5"
BEND_TYPE_5_1[4]="DIM_C=D2"
BEND_TYPE_5_1[5]="DIM_TD=TD"
BEND_TYPE_5_1[6]="ANG_U=A1"
BEND_TYPE_5_1[7]="ANG_V=A2"
```



With this mapping, the internal bending type `6_2` becomes `XY`, and the template attributes `DIM_B` and `DIM_C` will show the horizontal and vertical dimensions of the second leg `D2`, and `DIM_E` and `DIM_F` the horizontal and vertical dimensions of the fourth leg `D4`.

rebar_schedule_config.inp

```
BEND_TYPE_6_2[1]="XY"
BEND_TYPE_6_2[2]="DIM_A=D1"
BEND_TYPE_6_2[3]="DIM_B=D2*COS(A2*PI/180)"
BEND_TYPE_6_2[4]="DIM_C=D2*SIN(A2*PI/180)"
BEND_TYPE_6_2[5]="DIM_D=D3"
BEND_TYPE_6_2[6]="DIM_E=D4*COS(A1*PI/180)"
BEND_TYPE_6_2[7]="DIM_F=D4*SIN(A1*PI/180)"
BEND_TYPE_6_2[8]="DIM_G=D5"
BEND_TYPE_6_2[9]="DIM_TD=TD"
```



The following example maps the internal bending type `4` to the bending type identifier A **if** the dimensions `D1` and `D3` are the same. Otherwise it maps `4` to `B`.

rebar_schedule_config.inp

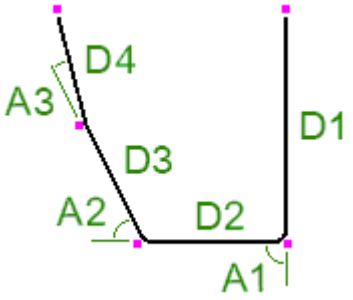
```
BEND_TYPE_4[1]=if (D1==D3) then ("A") else ("B") endif
BEND_TYPE_4[2]="DIM_A=D1"
BEND_TYPE_4[3]="DIM_B=D2"
BEND_TYPE_4[4]="DIM_C=D3"
BEND_TYPE_4[5]="DIM_TD=TD"
```



If Tekla Structures does not recognize a reinforcing bar bending shape, it uses the internal bending type `UNKNOWN` for it. In the `rebar_schedule_config.inp` file you can also define how unknown bending types appear in drawings and reports. For example, you may just want to use the bending type identifier `???`, and list all leg dimensions and bending angles.

rebar_schedule_config.inp

```
BEND_TYPE_UNKNOWN[1]="???"
BEND_TYPE_UNKNOWN[2]="DIM_A=D1"
BEND_TYPE_UNKNOWN[3]="DIM_B=D2"
BEND_TYPE_UNKNOWN[4]="DIM_C=D3"
BEND_TYPE_UNKNOWN[5]="DIM_D=D4"
BEND_TYPE_UNKNOWN[6]="DIM_E=D5"
BEND_TYPE_UNKNOWN[7]="DIM_F=D6"
BEND_TYPE_UNKNOWN[8]="ANG_S=A1"
BEND_TYPE_UNKNOWN[9]="ANG_T=A2"
BEND_TYPE_UNKNOWN[10]="ANG_U=A3"
BEND_TYPE_UNKNOWN[11]="ANG_V=A4"
BEND_TYPE_UNKNOWN[12]="DIM_TD=TD"
```



No.	Grade	Size	Mark	Length	Type	A	B	C	D	E	F	S	T	U	V	TD
1	A615-40	#4	R/5	1930	???	740	420	430	380			90	65	15		76

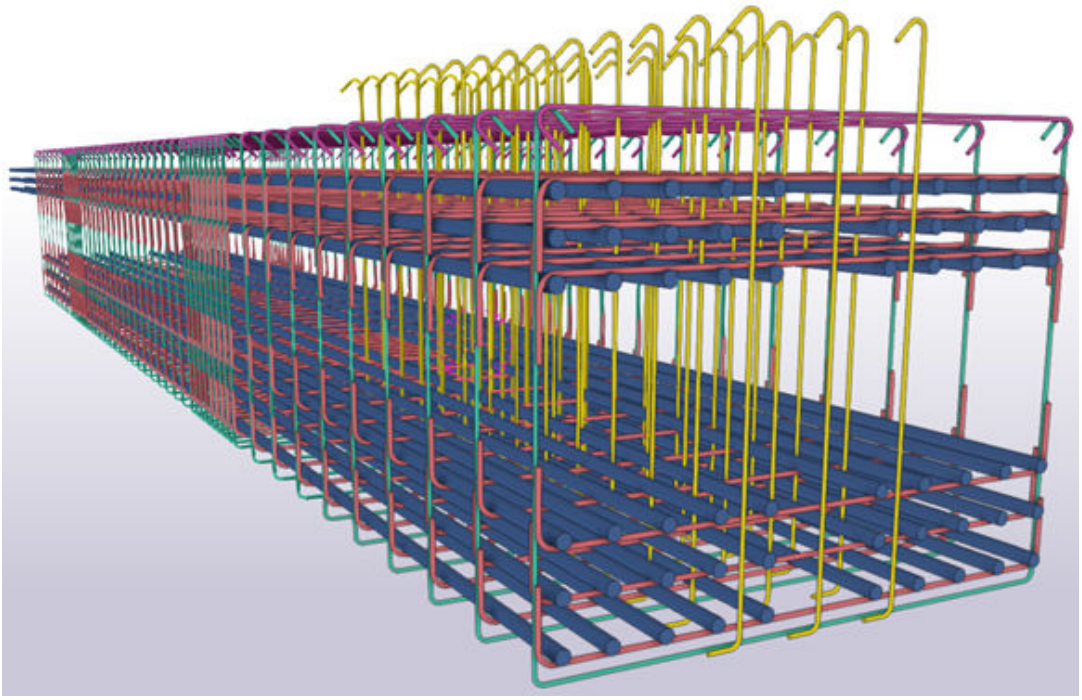
See also

Hard-coded bending type identifiers in reinforcement shape recognition (page 617)

Reinforcement shape recognition (page 603)

2.9 Work with rebar assemblies

A *rebar assembly* is a prefabricated reinforcement entity or cage that usually consists of several reinforcement objects, such as single reinforcing bars, different bar groups and rebar sets, reinforcement meshes, and possibly strands. Rebar assemblies can also contain assemblies and parts as sub-assemblies.



The structure of a rebar assembly can be flat where all objects are on one assembly hierarchy level, or there can be sub-assemblies on several assembly levels in the assembly structure.

Note that the assembly hierarchy affects the drawing and report outputs. Report and drawing templates need to be made with similar hierarchy as the assembly hierarchy in the model in order to output the data from correct assembly levels. We recommend you to test the different outputs early on in the modeling phase to ensure that all the relevant assembly information is included in the needed deliverables, such as production drawings.

When you work with rebar assemblies, use the same selection switches and the same methods to move in the assembly hierarchy as with other nested assemblies (page 444).

NOTE For the IFC export of rebar assemblies, use the **File** --> **Export** --> **IFC4** command. Rebar assemblies do not work in the IFC2x3 export.

Create a rebar assembly

You can create basic, one-level rebar assemblies or nested, hierarchical rebar assemblies. Start by creating basic rebar assemblies. If needed, you can later add more objects or sub-assemblies, or form hierarchies by adding these basic rebar assemblies as sub-assemblies to higher-level rebar assemblies.

A reinforcement object can only have one relation, either to a concrete part or to a rebar assembly. When you add a reinforcement object to a rebar

assembly, it will be detached from the part. If a reinforcement object already belongs to a rebar assembly, it cannot be attached to a part.

Create a basic rebar assembly

You can create a basic rebar assembly with various reinforcement objects on the same assembly hierarchy level.

The reinforcement object with the highest weight will become the main object of the rebar assembly.

1. Create the reinforcement objects that you want to include in the rebar assembly.
2. On the **Rebar** tab, click **Assembly** --> **Create rebar assembly**.
3. Select the objects that you want to include in the rebar assembly.

   You can select reinforcing bars, bar groups, rebar sets, reinforcement meshes, and strands.

   Note that individual rebar set bars or rebar set bar groups cannot be added to rebar assemblies, only entire rebar sets can be added.
4. Click the middle mouse button to finish selecting objects and to create the rebar assembly.

TIP Alternatively, you can first select the objects, then start the command, and finally click the middle mouse button. This may help in controlling that you have a correct set of objects selected for the rebar assembly.

Note that if you select components and want to include the component objects in the rebar assembly, the components need to be exploded first. Tekla Structures warns you about this and explodes the components when you click **Yes** in the warning dialog.

Note that if you select parts and assemblies, they are added as sub-assemblies, and the rebar assembly becomes a nested assembly (page 647).

To modify the rebar assembly structure, for example, by adding or removing objects and hierarchy levels, see Modify a rebar assembly (page 646) and Remove objects from a rebar assembly (page 649).

Rebar assembly properties

Use the **Rebar assembly** property pane to view and modify the properties of rebar assemblies. The file name extension of a rebar assembly property file is `.ras`.

Property	Description
Assembly numbering	Numbering prefix and start number (= numbering series) of the rebar assembly.

Property	Description
Name	User-definable name of the rebar assembly. By default, the name of the rebar assembly main object is shown in brackets [ ].
Rebar assembly type	Select the type for the rebar assembly. The options are (empty), **Cage**, **Mesh**, **Bent mesh**, **Braced girder**, **Embed**, and **Roll mat**.
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the rebar assembly. The available subtypes depend on the selected IFC entity. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
UDAs	User-defined attributes provide additional information about the rebar assembly, for example, workflow information. User-defined attributes can consist of numbers, text, lists, or dates. You can use the values of user-defined attributes in reports and drawings. You can also change the name of the fields, and add new ones, by editing the `objects.inp` file.

Modify a rebar assembly

You can modify the rebar assembly structure by adding more objects, or by changing the main object of a rebar assembly, for example.

NOTE The active selection switch controls whether you can select reinforcement objects or assemblies when you use the different rebar assembly commands.

Add objects to an existing rebar assembly

You can add more objects to a basic rebar assembly, or to any level of a nested rebar assembly.

Note that you can only add parts to rebar assemblies as sub-assemblies. See the 'Create a nested rebar assembly' instructions below.

1. Ensure that the  **Select objects in assemblies** or  **Select objects in components** selection switch is active.
2. Select the objects that you want to add.

3. Right-click and select **Assembly** --> **Add to assembly**.
4. Select the rebar assembly to which you want to add the objects.

Create a nested rebar assembly

You can add parts, assemblies, and rebar assemblies as sub-assemblies in a rebar assembly on a lower level in the assembly hierarchy.

1. Ensure that the  **Select assemblies** selection switch is active.
2. Select the parts and assemblies that you want to add to the rebar assembly.

   They will become sub-assemblies in the nested rebar assembly.
3. Right-click and select **Assembly** --> **Add as sub-assembly**.
4. Select the rebar assembly to which you want to add the parts and assemblies.

Alternatively, you can use the **Rebar** --> **Assembly** --> **Add as sub-assembly** command on the ribbon.

Check and highlight objects in a rebar assembly

Use the **Inquire** tool to check which objects belong to a particular rebar assembly.

1. On the ribbon, click the down arrow next to , and then select **Assembly objects** to inquire the rebar assembly objects.
2. Select a rebar assembly. Ensure that you are on the intended assembly hierarchy level.

   Tekla Structures highlights all the objects that belong to the same rebar assembly. The main reinforcement object is highlighted in orange and the other rebar assembly objects in yellow.

Change the main object of a rebar assembly

By default, the main object in a rebar assembly is the reinforcement object with the highest weight. You can change the main object in a rebar assembly.

1. If needed, check what is currently the main object of the rebar assembly.

   Use the **Inquire** tool to check which objects belong to a particular rebar assembly.
2. Ensure that the  **Select objects in assemblies** or  **Select objects in components** selection switch is active.

3. Select the new main object.
4. Right-click and select **Assembly** --> **Set as new main object of assembly**.

   Tekla Structures changes the main object.

Alternatively, you can use the **Rebar** --> **Assembly** --> **Set as main object** command on the ribbon.

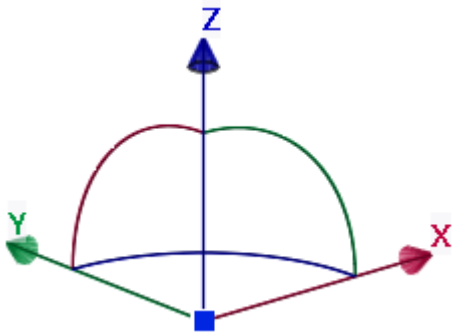
Note that if you remove the main object from a rebar assembly, the remaining reinforcement object with the highest weight is automatically set as the new main object.

Change the coordinate system of a rebar assembly

By default, the main reinforcement object of a rebar assembly sets the local coordinate system of the rebar assembly. The coordinate system then defines the orientation, length, width, and height of the rebar assembly for drawings and reports. If a rebar assembly is not shown the way you want in drawings and reports, even though you changed the main object, you can adjust the coordinate system of the rebar assembly.

1. Ensure that the  **Direct modification** switch is active.
2. Select a rebar assembly.
3. On the contextual toolbar, click  **Define or remove coordinate system**.

   Tekla Structures shows a blue direct modification handle at the origin, and the red, green, and blue axis handles and rotation handles that you can use to adjust the coordinate system of the rebar assembly.



4. To move the coordinate system to a new location, drag the  handle that is located at the origin.
5. To move the coordinate system along any of the coordinate axes, drag the relevant axis handle to a new location.

6. To rotate the coordinate system around any of the coordinate axes, drag the relevant rotation handle to a new location.

   Press **Tab** to rotate the coordinate system in 90-degree steps in the direction of the selected rotation handle.

7. To move or rotate the coordinate system by specifying a distance or angle:
   a. Select an axis handle or a rotation handle.
   b. Type the value by which you want the handle to move.

      When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog.

   c. Click **OK** or press **Enter** to confirm the value.

8. To revert to the original coordinate system, click  again on the contextual toolbar.

Remove objects from a rebar assembly

If you want to disassemble a rebar assembly, you can remove objects from it, or ungroup the objects belonging to the rebar assembly.

Remove an object from a rebar assembly

You can remove reinforcement objects or sub-assemblies from a rebar assembly one by one.

1. Select the object that you want to remove.
2. Right-click and select **Assembly** --> **Remove from assembly**.

Alternatively, you can use the **Rebar** --> **Assembly** --> **Remove from assembly** command on the ribbon.

To remove another object, restart the command.

Explode a rebar assembly or a sub-assembly

You can explode a rebar assembly back to single reinforcement objects, or just ungroup the objects belonging to a sub-assembly.

When you explode a nested rebar assembly, always start from the highest assembly hierarchy level. Tekla Structures breaks the assembly hierarchy level by level and ungroups the objects that belong to the highest level of the rebar assembly.

You can also explode sub-assemblies to single reinforcement objects without breaking the entire assembly hierarchy. The ungrouped sub-assembly objects remain within the higher-level, parent rebar assembly.

1. Ensure that the  **Select assemblies** selection switch is active.

2. Select the rebar assembly or sub-assembly that you want to explode.

   Ensure that you are on the intended assembly hierarchy level.

3. Do one of the following:

   - To explode a basic rebar assembly, or the highest level of a nested rebar assembly, right-click and select **Assembly** --> **Explode**.

     Use the **Explode** command several times to break a hierarchical nested rebar assembly back to single reinforcement objects.

   - To explode a sub-assembly, right-click and select **Assembly** --> **Explode sub-assembly**.

Alternatively, you use the **Explode assembly** and **Explode sub-assembly** commands on the ribbon, under **Rebar** --> **Assembly**.

2.10 Create construction objects and points

Points and construction objects help you to place other objects in the model.

When you want to place objects to a position where no lines or objects intersect in the model, you can create construction lines (page 651), planes (page 651), circles (page 652), arcs (page 653), and polycurves (page 655). For example, you can easily pick (page 88) the points at intersections of construction lines and circles. The snap priority of construction objects is the same as with the other lines.

Use the display settings (page 676) to control if points and construction objects are visible in model views. Points and construction objects remain in the model when you update or redraw views and windows. They do not appear in drawings.

Use the following selection switches to control if you can select points and construction objects in the model:



You can also create magnetic construction lines or planes to bind and move groups of objects (page 908). For example, rather than binding lots of handles and chamfers to part faces, simply create a construction plane that goes through all the handles and chamfers. Then make this plane magnetic and bind the plane to the appropriate face. When you move the plane, the attached handles and chamfers move with it.

See also

Create a construction line (page 651)

Create a construction plane (page 651)

Create a construction circle (page 652)

Create a construction arc (page 653)

Create a construction polycurve (page 655)

Copy a construction object with offset (page 656)

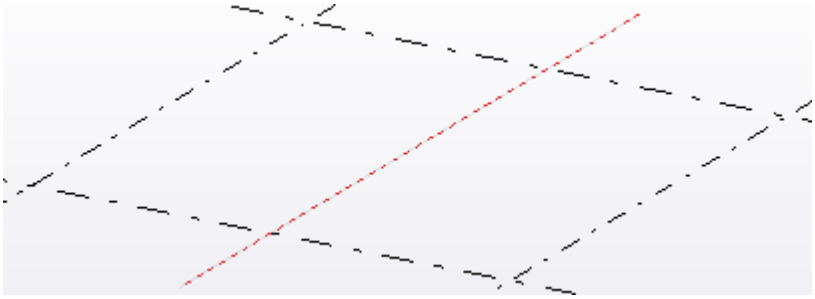
Modify a construction object (page 657)

Create points (page 660)

Create a construction line

1. On the **Edit** tab, click **Construction object** --> **Line** .
2. Pick the start point of the construction line.
3. Pick the end point of the construction line.

   Tekla Structures creates the line.



4. To end the command, press **Esc**.
5. To modify the construction line properties, double-click the line in the model.

   The line properties are shown in the property pane.

   a. If you want to make the line magnetic, select **Yes** in the **Magnetic** list.

   b. Select a color for the line.

   c. Define how far the line extends beyond the picked points.

   d. Select a line type for the line.

   e. Click **Modify** .

See also

Create construction objects and points (page 650)

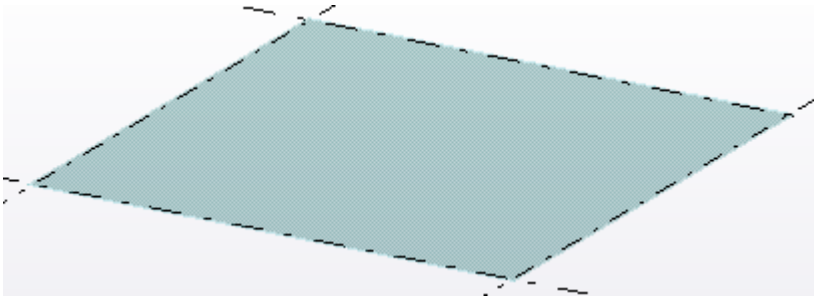
Copy a construction object with offset (page 656)

Modify a construction object (page 657)

Create a construction plane

1. On the **Edit** tab, click **Construction object** --> **Plane** .
2. Pick three points.
3. Click the middle mouse button.

   Tekla Structures draws the plane.

4. To end the command, press **Esc**.
5. To modify the construction plane properties, double-click the plane in the model.

   The plane properties are shown in the property pane.

   a. Enter a name for the plane.

   b. If you want to make the construction plane magnetic, select **Yes** in the **Magnetic** list.

   c. Click **Modify**.

See also

Create construction objects and points (page 650)

Modify a construction object (page 657)

Create a construction circle

You can create construction circles that are parallel to the view plane by picking two points, or you can create construction circles by picking three points in the 3D space in the model.

1. On the **Edit** tab, click **Construction object** --> **Circle** .
2. On the contextual toolbar that appears, click a button to specify which set of points you want to pick:

- Click , and then pick the center point and a point to define the radius of the circle.

  Tekla Structures creates the circle parallel to the view plane.

- Click , and then pick three points: the center point, a point to define the radius, and a point to define the plane of the circle.

- Click , and then pick three points along the arc of the circle.

  Tekla Structures creates the circle using the points you picked and using the current properties. Tekla Structures also indicates the center point of the circle with an X in the model.

3. To end the command, press **Esc**.
4. To modify the construction circle properties, double-click the circle in the model.

   The circle properties are shown in the property pane.

   a. Select a color for the circle.

   b. Select a line type for the circle.

   c. Click **Modify**.

See also

Create construction objects and points (page 650)

Copy a construction object with offset (page 656)

Modify a construction object (page 657)

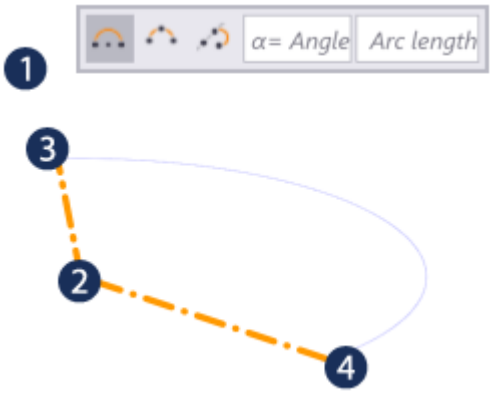
Create a construction arc

You can create construction arcs by picking three points in the 3D space in the model.

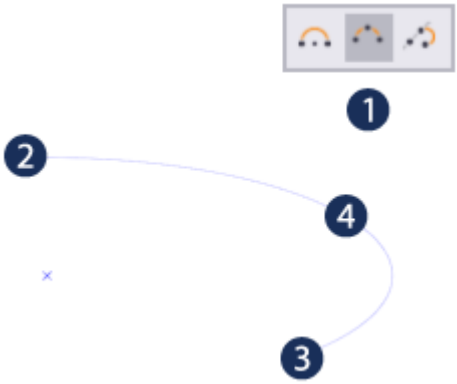
1. On the **Edit** tab, click **Construction object** --> **Arc** .
2. On the contextual toolbar that appears, click a button to specify which set of points you want to pick:

   - Click , and then pick three points: the center point, start point, and end point of the arc.

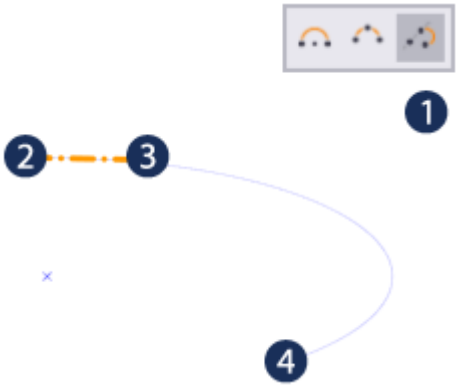
     You can also define the angle or the length of the arc.



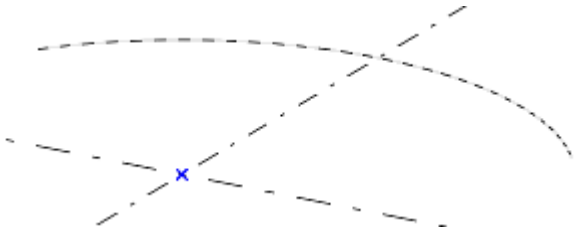
- Click , and then pick the start point, end point, and a point along the arc.



- Click , and then pick a point to define a tangent, and two points along the arc.



Tekla Structures creates the arc using the points you picked and using the current properties. Tekla Structures also indicates the center point of the arc with an X in the model.



3. To end the command, press **Esc**.
4. To modify the construction arc properties, double-click the arc in the model.

   The arc properties are shown in the property pane.

   a. Select a color for the arc.

   b. Select a line type for the arc.

   c. Click **Modify**.

See also

Create construction objects and points (page 650)

Copy a construction object with offset (page 656)

Modify a construction object (page 657)

Create a construction polycurve

You can create 3D construction polycurves that pass through the points you pick and that can have straight and curved segments.

1. On the **Edit** tab, click **Construction object** --> **Polycurve**.
2. On the contextual toolbar that appears, click a button to specify which set of points you want to pick to create a polycurve segment.

   You can switch between these picking modes each time you complete a segment.

   - For a straight segment, click  and then pick the start point and the end point of the segment.
   - For a curved segment, click  and then pick three points along the segment.

- For a curved tangential segment, click  and then pick a point on the tangent line, the start point, and the end point of the segment.
- For a straight segment that is tangential to the previous segment, click  and then pick a point on the tangent line.

Tekla Structures creates a polycurve segment.

3. Repeat step 2 for each polycurve segment that you want to create, but skip picking the first point of the segment as it is the same as the last point of the previous segment.
4. Click the middle mouse button to finish picking.

   Tekla Structures creates the polycurve through the points you picked, using the polycurve's current properties.
5. To end the command, press **Esc**.
6. To modify the construction polycurve properties, double-click the polycurve in the model.

   The polycurve properties are shown in the property pane.

   a. Select a color for the polycurve.

   b. Select a line type for the polycurve.

   c. Click **Modify**.

See also

Create construction objects and points (page 650)

Copy a construction object with offset (page 656)

Modify a construction object (page 657)

Copy a construction object with offset

You can copy construction lines, circles, arcs, and polycurves in the direction that you indicate, and using the offset values that you specify. For example, you can create new circles and arcs centered in the same location as the original circle or arc, and adjust the radiuses using the offset values.

1. On the **Edit** tab, click **Construction object** --> **Copy with offset** .
2. Select the construction object that you want to copy.

   You can copy lines (page 651), circles (page 652), arcs (page 653), and polycurves (page 655).
3. Enter the offset values in the box that appears, and then press **Enter**.

If you only enter one offset value, Tekla Structures creates one copy of the object.

To create several copies, enter multiple offset values. For example, `500 1000 1500`, or `4*800`.

4. Click in the direction where you want to copy the object.

Tekla Structures copies the selected object in the direction you indicated.

For example, if you selected a line, Tekla Structures makes a new copy of the line in the specified location. If you selected a circle or arc, Tekla Structures creates a new object that is centered in the same location as the original object, and adjusts the radius using the offset value that you specified.

See also

Create construction objects and points (page 650)

Modify a construction object (page 657)

Modify a construction object

You can modify construction points, lines, circles, arcs, polycurves, and planes using direct modification.

Before you start:

- To show (or hide) construction objects in model views, adjust their visibility (page 676) in the **Display** settings.
- Ensure that the  **Direct modification** switch is active.
- Select the construction object. Use the following selection switches:

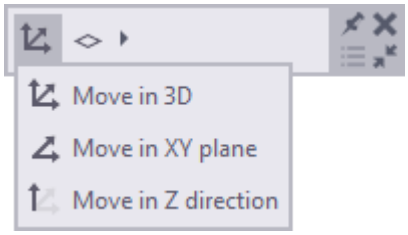
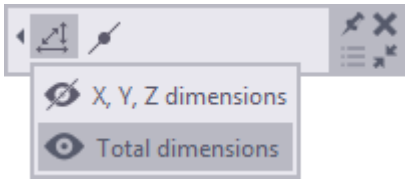
    

  Tekla Structures displays the handles and dimensions that you can use to modify the construction object.

  When you select a handle and move the mouse pointer over , Tekla Structures displays a toolbar with more modification options. The available options depend on the type of the construction object you are modifying.

To modify a construction object, do any of the following:

To	Do this	Available for
Set a reference point to move in one, two, or any direction	1. Select the handle in the reference point.	Construction points, lines, circle center points, planes

To	Do this	Available for
	2. To define in which directions the handle can move, select an option from the list on the toolbar:  You can also press **Tab** to cycle through the options. 3. To move the handle parallel to a certain plane only, click and select the plane.	
Move a point, a point on a line, circle, arc or polycurve, or a plane corner	Drag the handle in the reference point to a new location.	All construction objects
Move a circle or arc	Drag the handle in the center point to a new location.	Construction circles, arcs
Move a line or a plane edge	Drag the line handle to a new location.	Construction lines, planes
Move a plane	Drag the plane to a new location.	Construction planes
Show or hide diagonal dimensions	1. Select a handle. 2. On the toolbar, click . 3. Click the eye button to show or hide orthogonal and total dimensions:  	Construction lines, planes
Change a dimension	Drag a dimension arrowhead to a new location, or: 1. Select the dimension arrowhead which you want to move.	Construction lines, circles, planes Arcs (numeric input only)

To	Do this	Available for
	To change the dimension at both ends, select both arrowheads. To change the radius of a circle or arc, select the outer arrowhead. 2. Using the keyboard, enter the value with which you want the dimension to change. To start with the negative sign (-), use the numeric keypad. To enter an absolute value for the dimension, first enter $, then the value. 3. Press **Enter**, or click **OK** in the **Enter a Numeric Location** dialog.	
Chamfer a polycurve corner	1. Select a corner handle. 2. On the toolbar: • Click  to create a rounded chamfer, and then enter the chamfer radius. • Click  to create a straight chamfer, and then enter the chamfer dimensions X and Y. 3. Press **Enter** to confirm the chamfer dimensions.	Polycurves
Change an arc to a line Make a curved segment straight	Select the arc or segment midpoint handle (with an arc symbol)  and press **Delete**.	Arcs, curved polycurve segments
Change a line to an arc Make a straight segment curved	Drag the arc symbol  at the midpoint of the line or segment.	Lines, straight polycurve segments
Add a corner point and an intermediate segment to a polycurve	Drag a segment midpoint handle to a new location.	Polycurves
Remove a corner point and the	Select the corner point handle and press **Delete**.	Polycurves

To	Do this	Available for
two connected segments		
Remove the last polycurve segment	Select the end point handle and press **Delete**.	Polycurves
Change the radius of an arc and keep the end point locations	Click the radius dimension, enter a new value, and press **Enter**.	Arcs
Change the radius of a polycurve segment and keep the end point locations	Drag the arc symbol  at the segment midpoint handle.	Curved polycurve segments
Change the angle or length of an arc	Drag the start or end point to a new location.	Arcs
Copy a construction object using offset	See Copy a construction object with offset (page 656).	Lines, circles, arcs, polycurves
Change the modeling direction of the selected construction object	On the contextual toolbar, click  **Swap ends**. This may be needed when you create lofted plates (page 268) or lofted slabs (page 305) by using construction objects and if the geometry of the plate or slab would become self-intersecting.	Lines, arcs

See also

Create construction objects and points (page 650)

Create points (page 660)

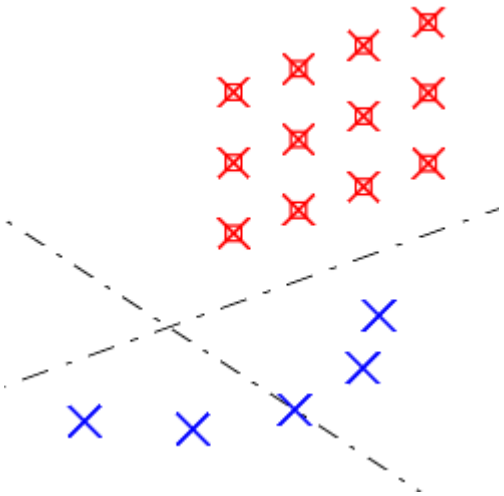
Resize and reshape model objects (page 119)

Create points

You can create points to make it easier to place model objects at positions where no lines or objects intersect.

There are many ways to create points in Tekla Structures. Which method is the most convenient at each time depends on what you have already created in the model and which locations you can easily pick.

When you create points, Tekla Structures always places them according to the work plane coordinate system. Points located on the view plane are by default blue and points outside the view plane are by default red. You can change the color of points in the point properties.



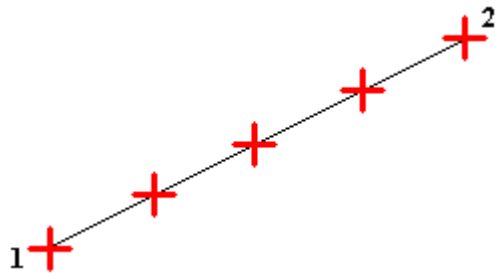
Use the display settings to control if points are visible in model views, and to adjust the point size if needed. For more information, see Adjust the display settings (page 676).

Create points on a line

You can create points at equal intervals along a line that is defined by two points.

1. On the **Edit** tab, click **Points** --> **On line**.

   The **Divided Line Points** dialog appears.
2. Define the number of points to be created.
3. Click **OK**.
4. Pick the start point of the line (1).
5. Pick the end point of the line (2).



Create points on a plane

You can create several equally spaced points in the desired area in the model. The points are created in relation to the picked origin position.

A point array consists of several points in a rectangular xy(z) pattern relative to the current work plane. The x, y, and z coordinates of the points define the array pattern. The x and y coordinates are relative distances between the points on the work plane. The z coordinates are absolute distances perpendicular to the work plane.

1. On the **Edit** tab, click **Points** --> **On plane**.

   The **Point Array** dialog appears.

2. Define the array point coordinates.

   Use positive or negative values to define the direction of the array.

   Use a zero at the beginning of the row to represent a point in the array origin. Separate multiple values with spaces. For example, `0 4000 4000`, or `0 5*1000`.

3. Pick the origin of the array in the view.

   Alternatively, you can define the origin in the **Point Array** dialog.

4. Click **Create**.

Create points parallel to two points

You can create offset points that are parallel to a line between two points you have picked.

1. On the **Edit** tab, click **Points** --> **Parallel to two points**.

   The **Point Input** dialog appears.

2. Define the distances at which the points are created.

   If you want to create multiple pairs of offset points, enter multiple values separated with spaces.

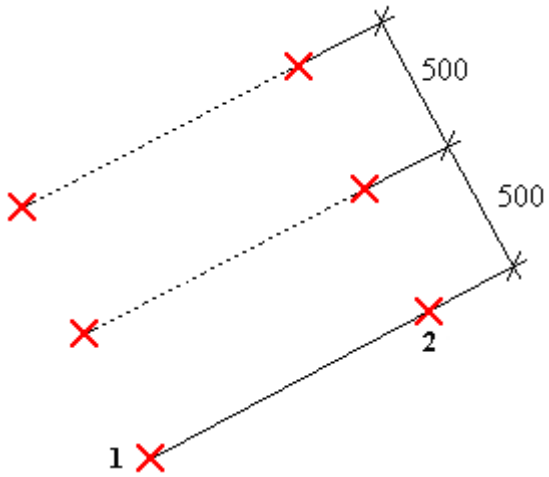
3. Click **OK**.
4. Pick the start point of the line (1).
5. Pick the end point of the line (2).

The picking order of the start point and the end point defines the offset direction of the new points.

When you look from the start point to the end point, Tekla Structures creates the new points to the left of the picked points. If you enter negative values to the **Point Input** dialog, Tekla Structures creates points to the right of the picked points.

When you pick points, Tekla Structures uses arrows to indicate the offset direction.

For example, if you enter `500 500` to the **Point Input** dialog, the first pair of new points is created at a 500 mm distance from the picked points, and the second pair of points is created at a 500 mm distance from the first pair of points.



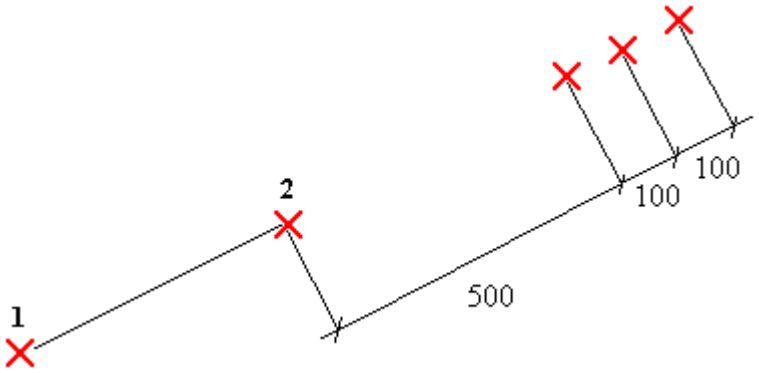
Create points along the extension line of two points

1. On the **Edit** tab, click **Points** --> **Along extension of two points**.

   The **Point Input** dialog opens.
2. Define the distances at which the points are created.

   Separate multiple values with spaces.
3. Click **OK**.
4. Pick the start point of the line (1).
5. Pick the end point of the line (2).

   For example, if you enter `500 100 100` to the **Point Input** dialog, the first point is created at 500 mm distance from the end point of the line, and the second and the third point are each created at 100 mm distance from the previous point.

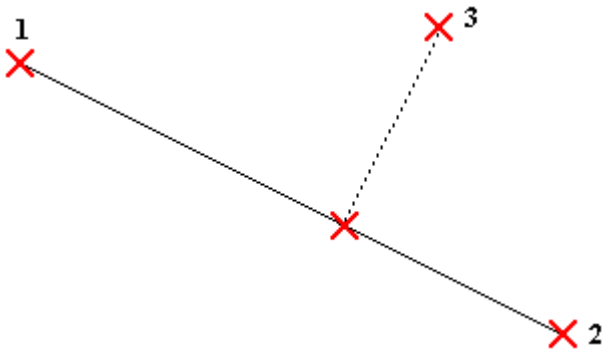


TIP Enter a negative value to the **Point Input** dialog to create a point between the start point and the end point.

Create projected points on a line

You can project a point onto a selected line or its extension.

1. On the **Edit** tab, click **Points** --> **Projected points on line**.
2. Pick the first point on the line (1).
3. Pick the second point on the line (2).
4. Pick the point to be projected (3).



Create points along an arc using center and arc points

You can create points along an arc.

1. On the **Edit** tab, click **Points** --> **Along arc using center and arc points**.

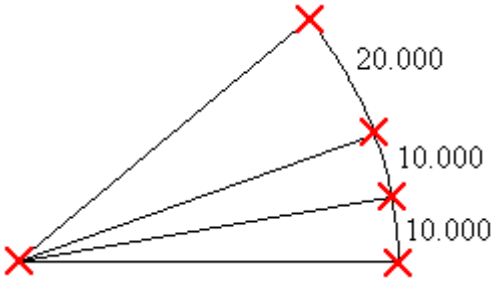
   The **Arc Points** dialog appears.
2. Select either **Angles** or **Distances** and enter the angles or distances between the points along the arc.

   Give the angle values in degrees.

   Separate multiple angle and distance values with spaces.

3. Click **OK**.
4. Pick the center point.
5. Pick the start point of the arc.

   Tekla Structures creates the arc points counterclockwise from the start point.



Create points along an arc using three arc points

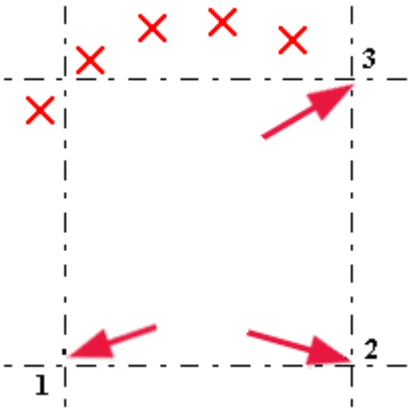
You can create points as an extension of an arc.

1. On the **Edit** tab, click **Points** --> **Along arc using three arc points**.

   The **Arc Points** dialog appears.
2. Select either **Angles** or **Distances** and enter the angles or distances between the points along the arc.

   Give the angle values in degrees.

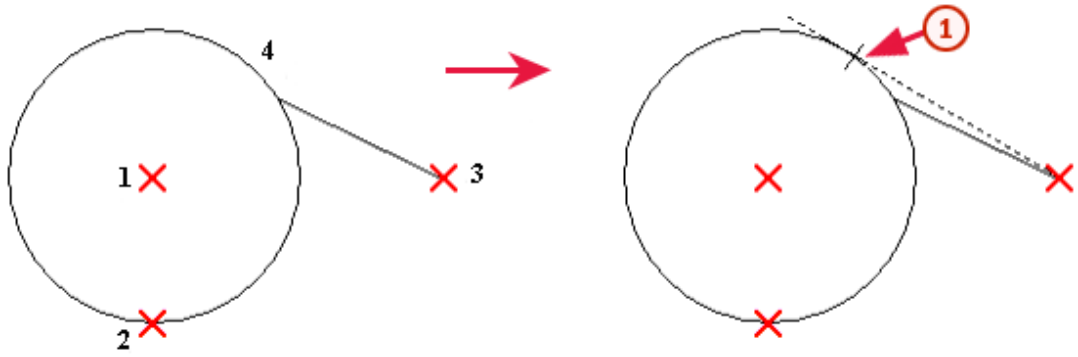
   Separate multiple angle and distance values with spaces.
3. Click **OK**.
4. Pick three points along the arc (1-3).



Create points tangent to a circle

You can create a point along the tangent of a circle.

1. On the **Edit** tab, click **Points** --> **Tangent to circle**.
2. Pick the center point of the circle (1).
3. Pick a point on the circle to define the radius (2).
4. Pick the end point of the tangent (3).
5. Pick a side to indicate the side on which Tekla Structures creates the tangent point (4).



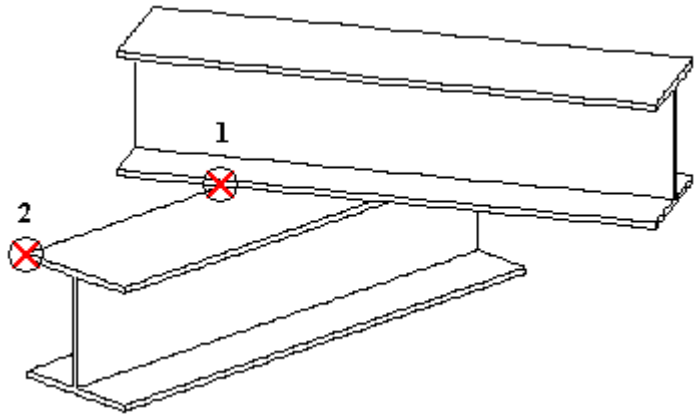
(1): Tangent point

Create points at any position

You can create a point at the position you pick. Snap switches determine the positions you can pick.

You can also use temporary reference points and numeric snapping to create a point, for example, to a certain distance from an existing corner or a point.

1. On the **Edit** tab, click **Points** --> **At any position**.
2. Pick the intersection of two part edges (1), or the corner of a part (2).



Create bolt points

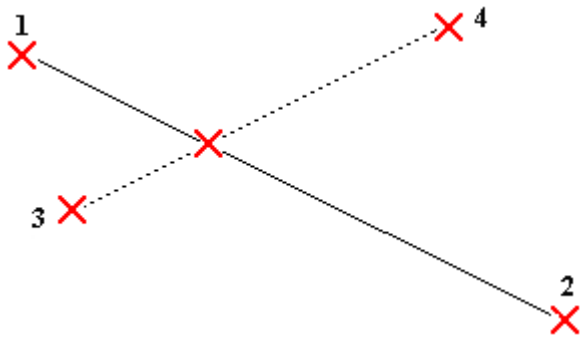
You can create points on the view plane at the center points of single bolts and bolt group bolts.

1. On the **Edit** tab, click **Points** --> **Bolt points**.
2. Select a bolt or a bolt group.

Create points at the intersection of two lines

You can create a point at the intersection of two lines in the view plane. The lines are treated as infinite length. The extensions of the lines must intersect at some point.

1. On the **Edit** tab, click **Points** --> **At intersection of two lines**.
2. Pick the start point of the first line (1).
3. Pick the end point of the first line (2).
4. Pick the start point of the second line (3).
5. Pick the end point of the second line (4).



Create points at the intersection of a plane and a line

You can create a point where a line intersects with a plane.

1. On the **Edit** tab, click **Points** --> **At intersection of plane and line**.
2. Pick three points to define the plane.
3. Pick the first point of the line.
4. Pick the second point of the line.

Create points at the intersection of a part and a line

You can create points where a line intersects with the surface of a part.

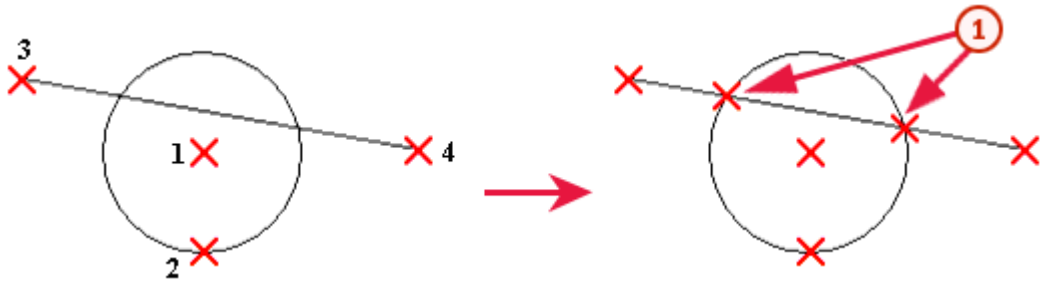
1. On the **Edit** tab, click **Points** --> **At intersection of part and line**.
2. Select the part.
3. Pick the first point on the line.

4. Pick the second point on the line.

Create points at the intersection of a circle and a line

You can create points where a circle and a line intersect.

1. On the **Edit** tab, click **Points** --> **At intersection of circle and line**.
2. Pick the center point of the circle (1).
3. Pick a point on the circle to define the radius (2).
4. Pick the first point on the line (3).
5. Pick the second point on the line (4).



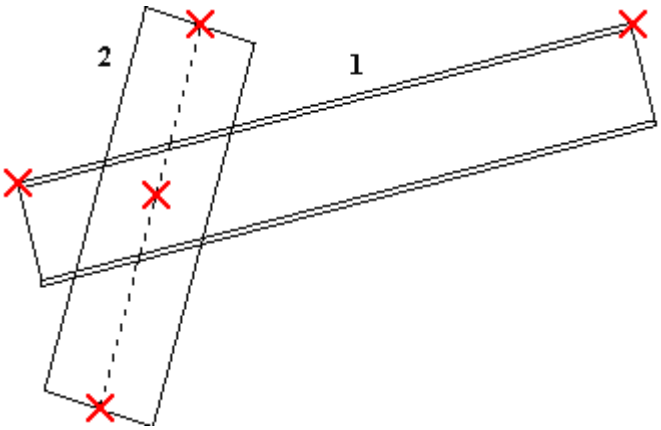
(1): New points

Create points at the intersection of two part axes

You can create points where the axes of two parts intersect, and project the points onto the axis of the part that you select first.

1. On the **Edit** tab, click **Points** --> **At intersection of two part axes**.
2. Select the first part (1).
3. Select the second part (2).

   Tekla Structures projects the point onto the axis of the first part.



Import points

You can import points to specific locations in an open Tekla Structures model using the **Point Creation Import (8)** component. You need to specify the point coordinates in a text file. In some cases this file is generated by another software package.

1. Create a point import file.
   a. Create a text file that consists of single lines for each point.

      Use commas or tabs as delimiters for the three point coordinates on a line. For example:

      ```
      100,500,1000
      300,700,1500
      ```

   b. Save the file.

   During the import process, Tekla Structures ignores all lines in the import file which do not consist of valid values delimited by tabs or commas.

2. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
3. Enter `point` in the **Search...** box.
4. Click **Point Creation Import (8)**.
5. Enter the ASCII file name.

   Include the full path and the file name extension (for example `.txt`). If you do not specify the path, Tekla Structures looks for the file in the current model folder.
6. Define the origin of the imported points by entering the coordinates.
7. Click **Create**.

Point properties

Use the **Point** properties to view and modify the properties of a point.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
General	
Color	Change the color of the points. Note that if you change the point color, the changed color is not applied the next time you create points. The points are created using the default point color.
Location	

Setting	Description
X **Y** **Z**	The local (work plane) and global x, y, and z coordinates of a point. Indicates the correct location of the point.

TIP You can change the point size in display settings (page 676).

3 Adjust how model objects are displayed

You can use different methods to adjust how model objects are displayed:

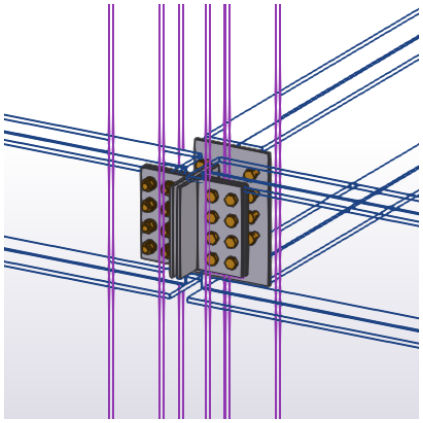
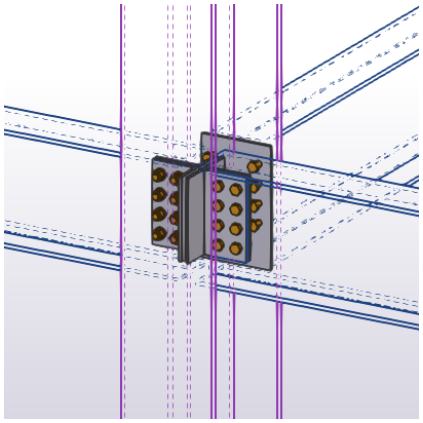
- To change the rendering of parts and components and select whether the object surfaces or the object outlines are displayed in the views, see Change the rendering of parts, components and reference models (page 671).
- To change the general display settings, see Adjust the display settings (page 676).
- You can temporarily change the part representation to show parts with exact lines or with high accuracy as explained in Change the part representation to show parts with exact lines or with high accuracy (page 681).
- You can temporarily hide selected objects or show only the selected objects as explained in Temporarily hide model objects or show only selected model objects (page 682).
- Detailed instructions for changing model object color and transparency can be found in Change the color and transparency of model objects by using object representation (page 686).
- You can group model objects using various criteria (for example, profile) to handle them as a single unit when you define display settings, see Use object groups in object representation and in filters (page 692).

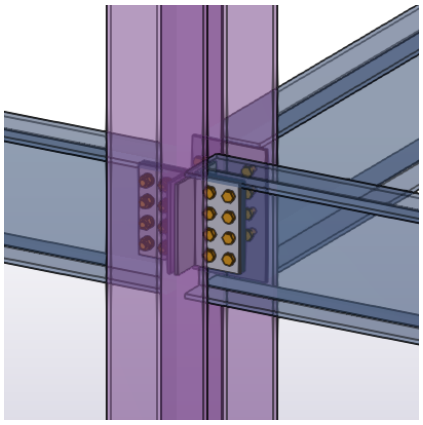
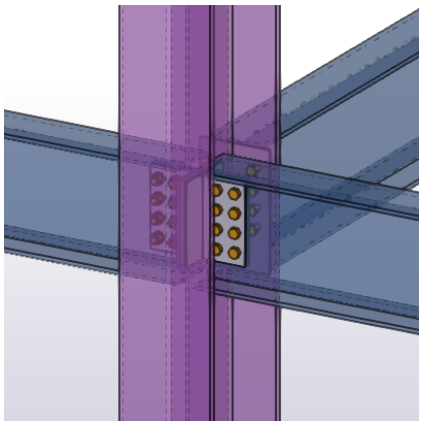
3.1 Change the rendering of parts, components, reference objects and building hierarchy

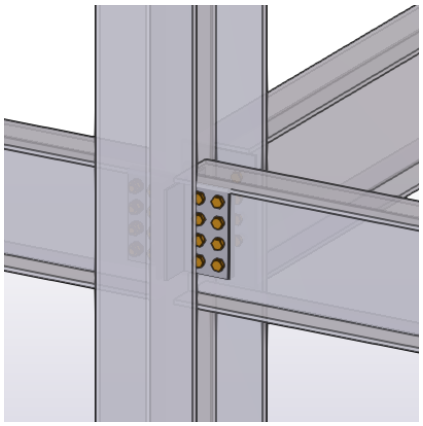
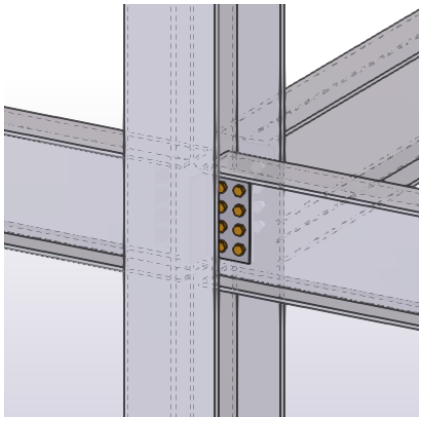
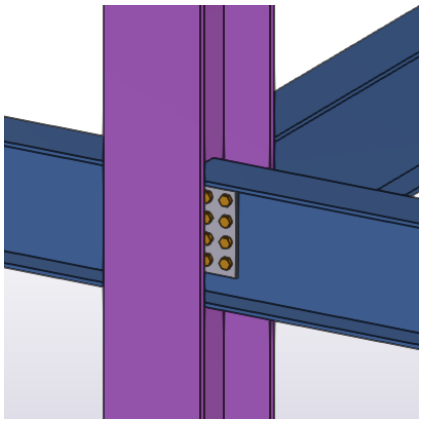
You can easily change the rendering of parts, components, reference objects, and building hierarchy in model views.

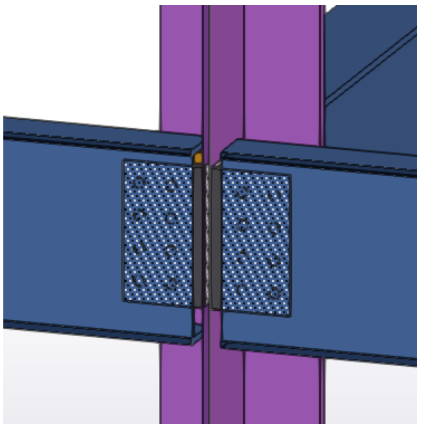
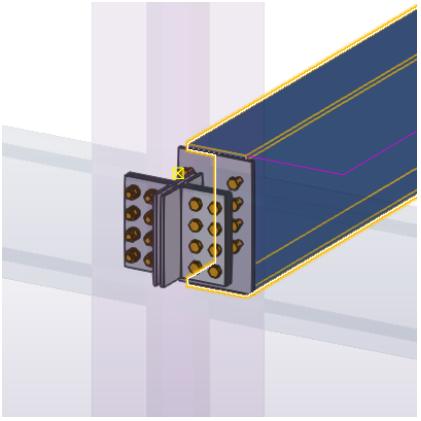
1. On the **View** tab, click **Rendering**.

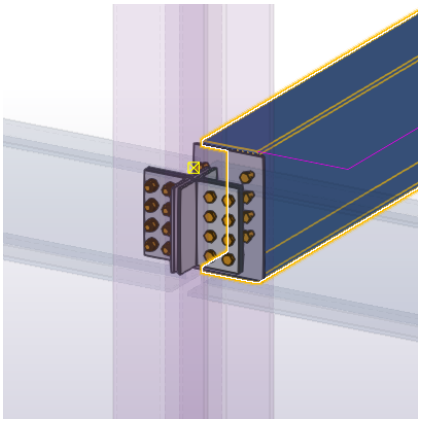
2. Select one of the rendering options for parts, components, reference objects, or building hierarchy:

Option	Description	Example
Wireframe: **Parts**, **Components**, **Reference objects**, **Building hierarchy**	Object outlines are displayed, surfaces are not. Objects are transparent.	In this example, component objects are displayed as rendered. 
	The part edge lines hidden behind another part are visualized with dashed lines if you have switched the **Dashed line for hidden line** option **on** in **File menu** --> **Settings** --> **Switches**. The dashed lines are visualized only in DirectX rendered (page 76) views.	In this example, component objects are displayed as rendered. 

Option	Description	Example
Shaded wireframe: **Parts**, **Components**, **Reference objects**, **Building hierarchy**	Object outlines are displayed. Objects are transparent, and their surfaces are rendered.	In this example, component objects are displayed as rendered. 
	The part edge lines hidden behind another part are visualized with dashed lines if you have switched the **Dashed line for hidden line** option **on** in **File menu** --> **Settings** --> **Switches**. The dashed lines are visualized only in DirectX rendered (page 76) views.	In this example, component objects are displayed as rendered. 

Option	Description	Example
Grayscale: **Parts**, **Components**, **Reference objects**	Objects are shown in grayscale.	In this example, component objects are displayed as rendered. 
	The part edge lines hidden behind another part are visualized with dashed lines if you have switched the **Dashed line for hidden line** option **on** in **File menu** --> **Settings** --> **Switches**. The dashed lines are visualized only in DirectX rendered (page 76) views.	In this example, component objects are displayed as rendered. 
Rendered: **Parts** ,**Components**,	Object surfaces are displayed. Objects are not transparent.	

Option	Description	Example
Reference objects	Any duplicate objects or overlapping parts on the same plane are visualized with a hatch if you have switched the **Hatching of overlapping surfaces** option **on** in **File menu** --> **Settings** --> **Switches**. The hatches are visualized only in DirectX rendered (page 76) views.	In this example, the overlapping surfaces are visualized with a hatch. 
Selected: **Show only selected part/Show only selected component/ Show only selected reference**	Selected objects are displayed. Other objects are almost completely transparent. This option is useful, for example, when viewing clash check results in a large model.	In this example, component objects are displayed as rendered. 

Option	Description	Example
	The part edge lines hidden behind another part are visualized with dashed lines if you have switched the **Dashed line for hidden line** options **on** in **File menu** --> **Settings** --> **Switches**. The dashed lines are visualized only in DirectX rendered (page 76) views.	In this example, component objects are displayed as rendered. 

TIP Alternatively, you can use the keyboard shortcuts **Ctrl+1...5** for parts and **Shift+1...5** for components to switch between the rendering options. **Ctrl+Shift+1...5** works for references, and **Ctrl+Alt+1...2** for building hierarchy.

TIP To display component content (page 803), click the **Content** command and select the component.

See also

Change the rendering mode for model views (page 76)

Adjust the display settings (page 676)

3.2 Adjust the display settings

Use the **Display** dialog to define how parts and other model objects appear in the model.

Set the visibility and representation of model objects

You can define the visibility and representation separately for different types of model objects in the **Display** dialog.

1. Double-click anywhere on the model background to open the **View properties** dialog.

   Alternatively, on the **View** tab, click **View properties**.

2. Click the **Display...** button to open the **Display** dialog.

3. To specify which objects are visible in the model, select or clear the checkboxes.

   You can specify separately the object visibility for the objects in model and for the objects in components.

4. To specify how parts, bolts, holes, welds, construction planes, and reinforcing bars are shown in the model, select a suitable representation option.

   You have the following options:

   - **Fast**
   - **Exact**
   - **Reference line** (only for parts)
   - **Exact slotted holes** (only for holes)
   - **Exact - no weld mark** (only for welds)

5. If you are working with cast-in-place (page 471) concrete structures, and the pour management functionality is enabled (page 469):

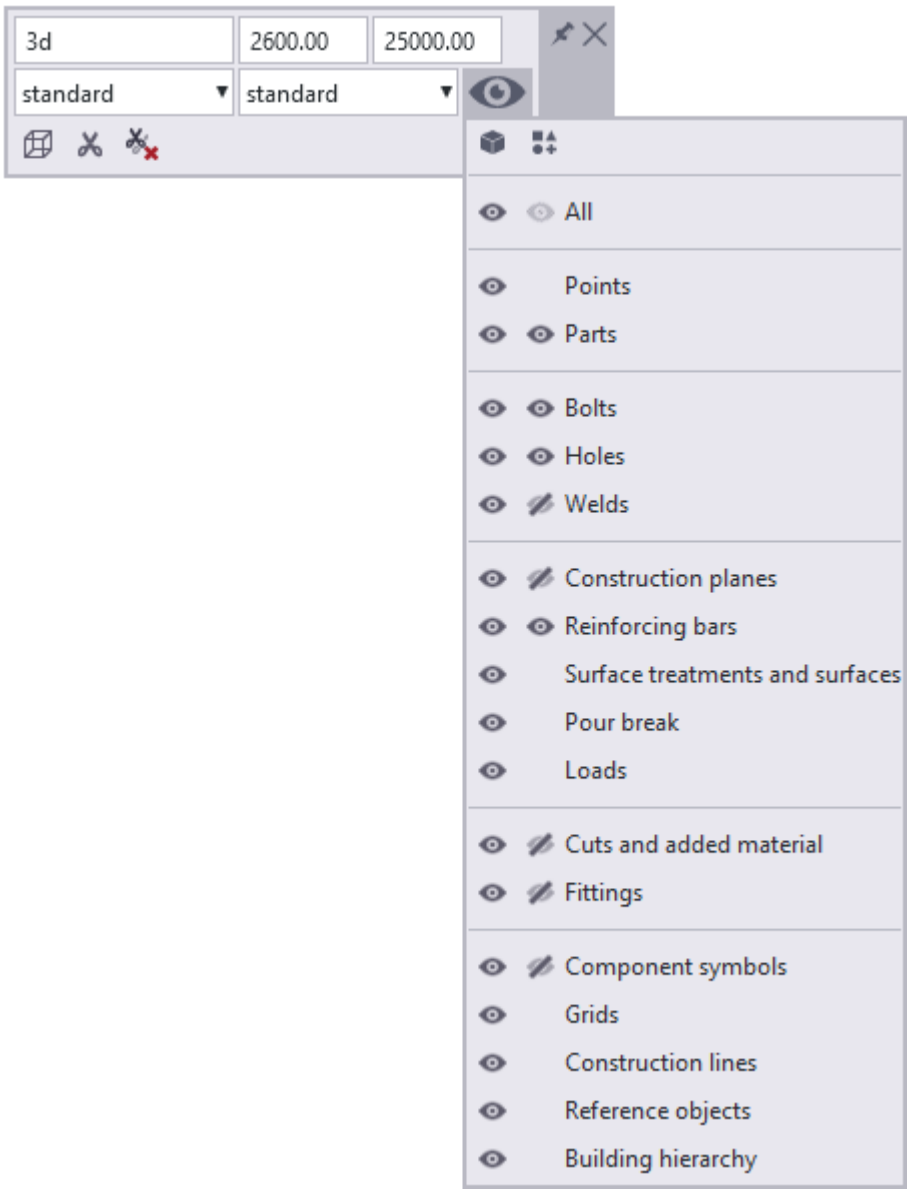
   a. In the **Cast in place** list, select whether you want to show the structures as **Parts** or as **Pours**.

   b. If you selected **Parts** for the cast-in-place concrete structures, select whether you want to show the parts as **Merged** or as **Separated**.

6. Ensure that the view is selected.

7. Click **Modify** to apply the changes.

To quickly modify the visibility of objects in model and in components, use the contextual toolbar. Click the view, and use the eye icon on the contextual toolbar to set the visibility.



Visibility and representation settings

Use the **Display** dialog to control the visibility and representation settings of model objects. Note that some of the settings may affect system performance.

Option	Description
Settings tab	

Option	Description
Parts	Select how parts are displayed. **Fast** uses a rapid drawing technique that displays internal hidden edges, but skips cuts. The setting does not automatically affect already modeled parts. When you switch this setting on, the fast representation mode will be applied only to newly created parts and to parts that are displayed with the **Show with exact lines** command. **Exact** displays the cuts, but hides the internal hidden lines of parts. **Reference line** shows parts using part reference lines (page 116). This option increases display speed significantly, when viewing the entire model, or large parts of it. Cast-in-place concrete structures can be displayed as **Pours**, or as **Parts** that can be **Merged** or **Separated**. For more information, see View cast-in-place concrete structures (page 471).
Bolts	Select how bolts are displayed. **Fast** displays the axis and a cross to represent the bolt head. This is the recommended representation mode for bolts, because it increases display speed significantly and consumes less system memory. **Exact** shows bolts, washers, and nuts as solid objects.
Holes	Select how holes are displayed. **Fast** only displays the circle in the first plane. When using this option, Tekla Structures always displays fast holes on the first part (counting from the head of the bolt). If there are slotted holes in any of the parts, a slotted hole is displayed on the first part, even if the hole in that part is not slotted. The new slotted hole has the same size and rotation as the first slotted hole (counting from the head of the bolt). Holes that are outside a part are always displayed as fast holes. **Exact** shows holes as solid objects. **Exact slotted holes** only displays slotted holes in exact mode and ordinary holes in fast mode.

Option	Description
Welds	Select how welds are displayed. **Fast** displays a symbol for welds. **Exact** shows welds as solid objects and displays the weld symbols. When you select welds, the weld marks are displayed. **Exact - no weld mark** shows welds as solid objects but does not display the weld symbols, nor the weld marks when you select welds. For more information, see Set the visibility and appearance of welds (page 393).
Construction planes	Select how construction planes are displayed.
Reinforcing bars	Select how reinforcement objects are displayed. **Fast** displays the shape of reinforcement meshes using an outline polygon and a diagonal line. Single reinforcing bars and bar groups are displayed as solid objects. **Exact** shows reinforcing bars, bar groups, and reinforcement meshes as solid objects.
Advanced tab	
Part label **Rebar label**	See Show part and reinforcement information by using part and rebar labels (page 744).
Point size	Select the appearance of points and part handles, and change the point size. The **In model** option increases the point size on the screen when you zoom in. The option shows points (on the view plane and outside the view plane) and part handles as 3D cubes.  The **In view** option does not increase the point size. The option shows points (on the view plane and outside the view plane) and part handles as flat 2D objects.  To change the size of the part handles, use the advanced option XS_HANDLE_SCALE. Other point properties can be modified in point properties (page 660).

3.3 Change the part representation to show parts with exact lines or with high accuracy

Even if you have set the part representation to **Fast** in the display settings, you can temporarily view the parts using different representation options.

Show parts with exact lines

Use the **Show part with exact lines** command to temporarily display a part with exact lines even if you are using the **Fast** representation option for parts.

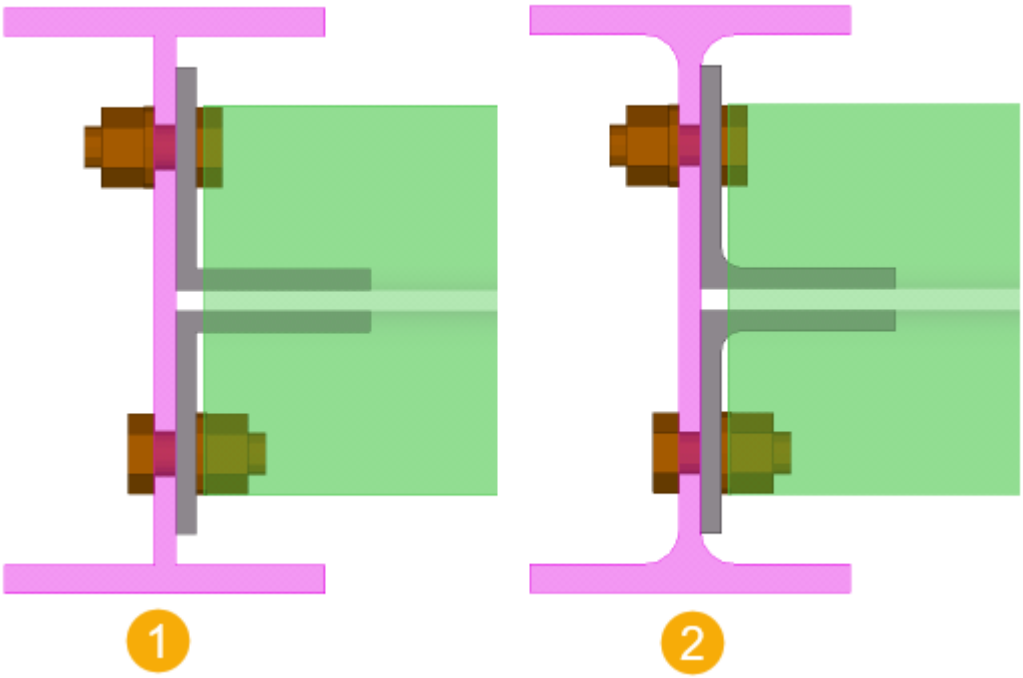
1. Select the part.
2. Go to **Quick Launch**, start typing `show part with exact lines`, and select the **Show part with exact lines** command from the list that appears.
3. Click the view in which you want to display exact lines.
4. To clear the exact lines effect, on the **View** tab, click .

Show parts with high accuracy

You can temporarily display parts with the highest possible level of accuracy. This can be useful, for example, when checking a large model, because the entire model can still be displayed in the **Fast** or **Exact** representation mode but the individual part can be shown in more detail.

1. Select the parts.
2. Right-click, and then hold down the **Shift** key while selecting **Show with exact lines**.

   Tekla Structures displays the selected parts with the highest possible level of accuracy.
3. To clear the high accuracy effect, right-click and select **Show with exact lines**.



(1) Normal display mode

(2) High accuracy mode

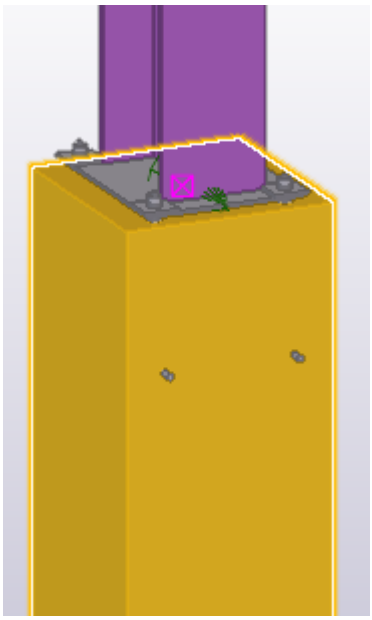
3.4 Temporarily hide model objects or show only selected model objects

You can temporarily show, hide, or show hidden model objects and assemblies in model views.

Hide parts or other objects in a model view

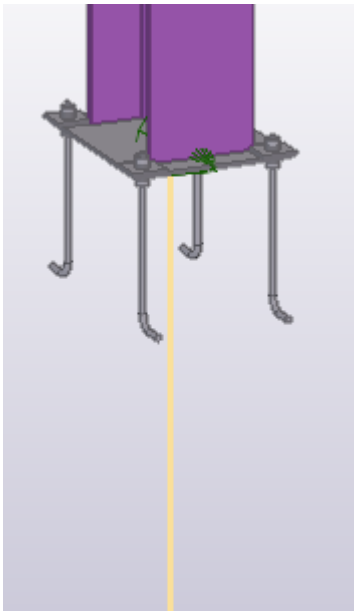
You can quickly hide selected parts or other objects in a model view. This can be useful, for example, when you want to temporarily hide some parts to see the parts behind them.

1. Select the parts or objects you want to hide.



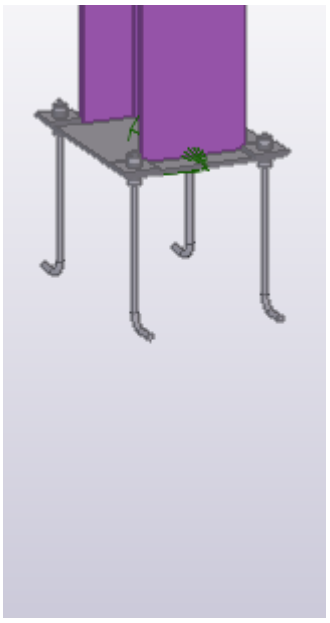
2. Right-click and select **Hide**.

   Tekla Structures shows the hidden parts as part reference lines.



   To completely hide the selected parts, hold down the **Shift** key when selecting the command.

   The selected parts become invisible.



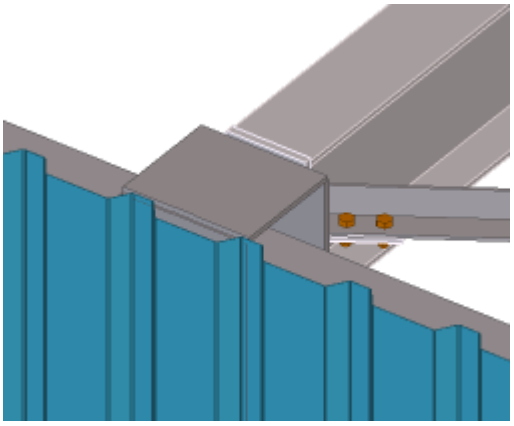
3. To make the hidden parts or objects visible again, click  on the **View** tab.

   Alternatively, right-click in the view and select **Redraw view**.

Show only selected parts or other objects in a model view

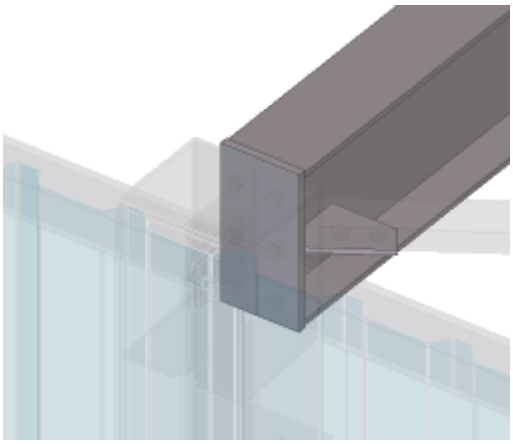
As an alternative to hiding single parts or other objects in a model view, you can define which parts you want to keep visible. All of the other, unselected parts will be hidden.

1. Select the parts or objects that you want to keep visible.



2. Right-click and select **Show only selected**.

Tekla Structures shows the unselected parts almost as transparent.



TIP To completely hide the unselected parts, hold down the **Shift** key when selecting the command.

To show the unselected parts as part reference lines, hold down the **Ctrl** key when selecting the command.

3. To make the unselected parts or objects visible again, click  on the **View** tab.

   Alternatively, right-click in the view and select **Redraw view**.

Temporarily show assembly and component objects in a model view

You can temporarily show the contents of an assembly or a component even if some of the assembly or component objects are not visible in a model view.

To	Do this
Show the contents of an assembly	1. Right-click an assembly, or a part in the assembly. 2. Select **Assembly** --> **Show assembly**. For a concrete part, select **Show assembly**. Tekla Structures shows an orange box around the assembly, and displays all parts, bolts, welds, and other details (not cuts or fittings) belonging to the assembly, even if you had defined them as hidden in the display settings (page 676). For concrete parts, Tekla Structures displays reinforcement and surface treatment (not

To	Do this
	surfaces), even if you had defined them as hidden in the display settings.
Show the contents of a component	1. On the **View** tab, click **Rendering** --> **Show component content**. 2. Select a component. Tekla Structures displays all bolts, welds, and other details belonging to the component, even if you had defined them as hidden in the display settings (page 676).
Reapply the display settings (page 676) and make assembly or component objects hidden again	On the **View** tab, click .

3.5 Change the color and transparency of model objects by using object representation

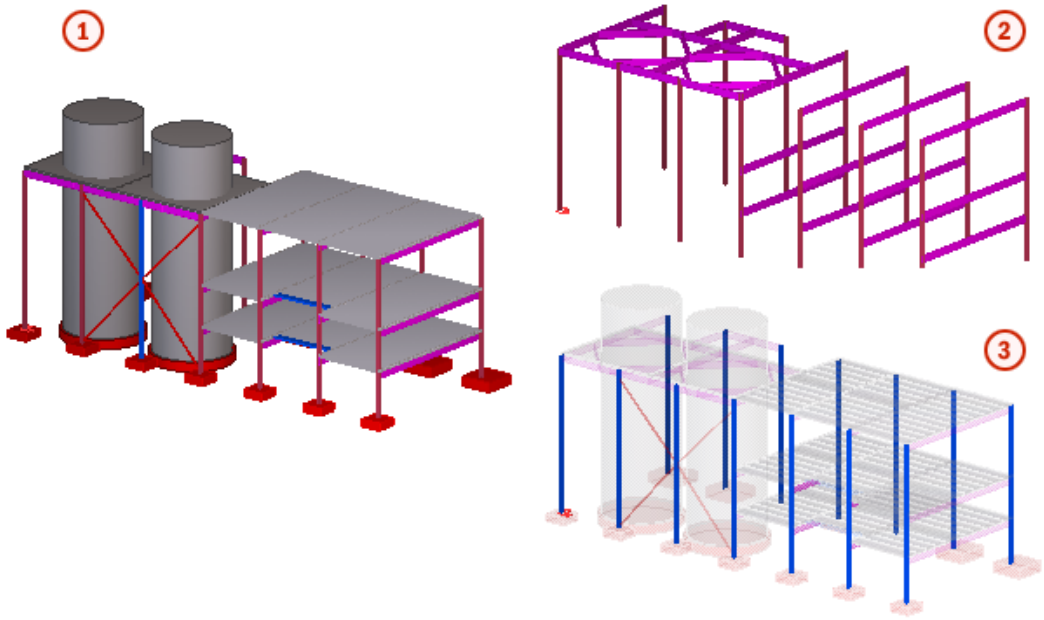
Use *object representation* to set the color and transparency of model objects in model views and to create customized presentations of the model by using defined *object groups*.

With object representation settings and object groups you can define that, for example, only some model objects are visible or that some model objects are of certain color.

To open the object representation settings, on the **View** tab, click **Representation**. The **Object Representation** dialog opens.

Note that changing the settings in the **Object Representation** box affects all model views.

The following images show the same model with different transparency settings:



1. Standard color and transparency settings
2. Only parts whose profile name starts with IPE* or HEA* are visible
3. Parts whose user-defined attribute **Planned erection date** is set to a certain date are displayed in blue, while all other parts are 90% transparent

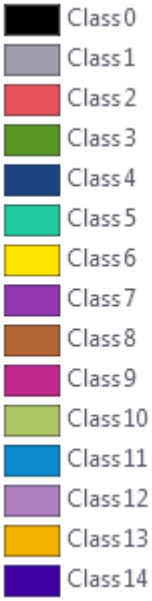
Use the Class property to change the color of parts and reinforcement

Use the object representation settings **Color by class** and **Visible** to enable the changing of the color of parts and reinforcement in the part and reinforcement properties.

1. To open the object representation settings, on the **View** tab, click **Representation**. The **Object Representation** dialog opens.
2. In the dialog, ensure that
   - **Object group** is set to **All**
   - **Color** is set to **Color by class**
   - **Transparency** is set to **Visible**

   With these settings, all parts and reinforcement are colored according to their **Class** property in the part and reinforcement properties.

   The possible class numbers range between 0 and 14, and result in different colors as follows:



Class numbers above 14 produce the same colors as 1...14. For example, class numbers 2, 16, 30, 44, and so on, all result in red.

3. To change the part or reinforcement color in the part or reinforcement properties:

   a. In the property pane, in the **Class** box, select a new class.

   b. Click **Modify**.

   Alternatively, you can change the **Class** property on the contextual toolbar.

NOTE You can also use class numbers to define the default color of pour objects and pour breaks.

Define color and transparency settings for object groups

1. On the **View** tab, click **Representation** to open the **Object Representation** dialog.

   The settings **Color by class** and **Visible** are on by default.

2. To define the color and visibility of a selected object group, click **Add row**.
3. Select an object group (page 692) from the **Object group** list.
4. Use the **Color** list to define the color for the objects in the object group.
5. Use the **Transparency** list to define the transparency for the objects in the object group.
6. Repeat steps 3–5 for each row you add.

7. Use the **Move up** and **Move down** buttons to change the order of the rows.

   The reading order of the rows is from bottom to upwards. If an object belongs to several groups, the upmost row defines the color and transparency setting defined for the objects.
8. Enter a unique name in the box next to the **Save as** button.
9. Click **Save as** to save the settings.

NOTE If your setting does not contain the group **All**, Tekla Structures adds that row to the bottom of the list when you click **Modify**, **Apply** or **OK**.

Color settings in object representation

Option	Description
As is	The current color is used. If the object belongs to one of the object groups defined in the following rows, its color is defined by the settings that the object group in question has on that row.
Colors	Select color from the list.
Color by class	All parts are colored according to their **Class** property.
Color by lot **Color by phase**	Parts belonging to different lots or phases get different colors according to the lot or phase number: 
Color by analysis type	Displays parts according to the member analysis class.

Option	Description
Color by analysis utility check	Displays parts according to the utilization ratio in analysis.
Color by attribute	Displays parts in different colors according to the values of a user-defined attribute.

Transparency settings in object representation

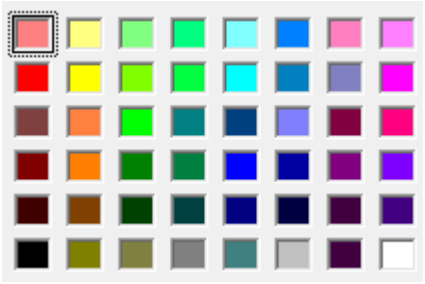
Option	Description
As is	The current visibility. If the object belongs to any object group whose visibility and color settings have been defined, the settings will be read from that object group.
Visible	Object is shown in the views.
50% transparent	Object is transparent in the views.
70% transparent	
90% transparent	
Hidden	Object is not shown in the views.

Define your own colors for object groups in object representation

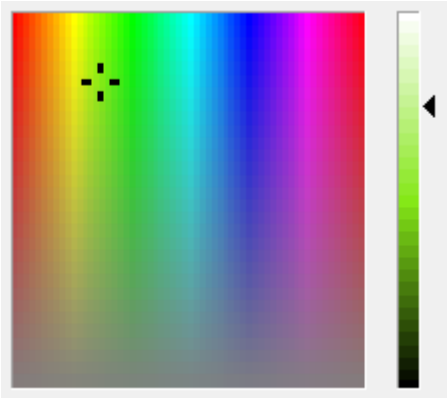
If the color properties are not sufficient, you can define your own colors.

1. On the **View** tab, click **Representation** to open the **Object Representation** dialog.
2. Select an object group from the **Object group** list.
3. In the **Color** list, select **Choose color...**.
4. Do one of the following:

- Click a color in the **Basic colors** palette.



- Click **Define Custom Colors** and create a color of your own:
  a. Click a color in the color window.

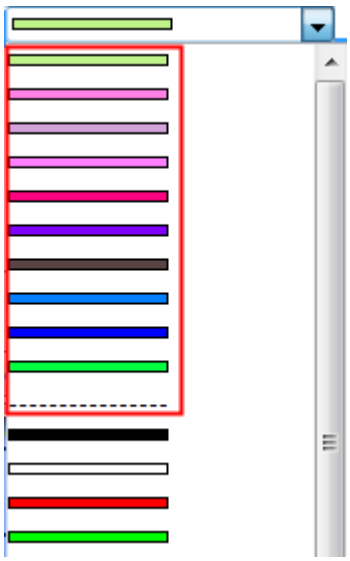


  b. Define the color depth by using the color bar on the right, or enter the exact RGB values.
  c. Click **Add to Custom Colors**.
  d. Click the color in the **Custom colors** palette to select it.



5. Click **OK**.
6. Click **Save** to save the changes.

When you open the **Object Representation** dialog the next time, the **Color** list shows a maximum of 10 colors that you last defined. Customized colors are shown above the dashed line:



Information about the colors that you define for the object groups is stored in the `used_custom_colors.clr` file, which is located in the current model's `\attributes` folder.

Information about the colors that you add in the **Custom colors** palette is stored in the `xs_user.xxx` file in the model folder (`xxx` is your user name).

Copy object representation settings to another model

1. Select the settings you want to copy.

   The settings you have created are located in the current model's `\attributes` folder, and they have the file name extension `.rep`.

2. Select where you want to copy the settings.

   - To make the settings available in another model, copy them to the `\attributes` folder of the destination model.
   - To make the settings available in all models, copy them to the project or firm folder, defined by the advanced option or .

3. Restart Tekla Structures.

If you want to delete the settings in the `\attributes` folder, select the `.rep` file and press **Delete**.

3.6 Use object groups in object representation and in filters

In *object groups* the objects are grouped by a set of rules and conditions. Use the object groups to control the transparency and coloring of model objects in the model views, and in filters to define which objects are displayed or can be selected.

Where object groups are used

Object groups are sets of rules with which you can group objects by selected properties and conditions.

Use object groups in

- object representations (page 686) to control the transparency and coloring of model objects in all views
- model view filters (page 172) to define which objects are displayed in the selected view
- selection filters (page 174) to define which objects can be selected
- Organizer to filter the content of categories
- IFC property set definitions to limit for which objects to export the property set in the IFC export.
- Project status visualization to review the status of model objects in a specific time frame

Object groups for different purposes use different object group files. Object group files are saved in the current model's `attributes` folder.

Object group	File name extension
Object group - Representation	`.PObjGrp`
Object group - View filter	`.VObjGrp`
Object group - Select filter	`.SObjGrp`
Object group - IFC property export	`.IFCObjGrp`
Object group - Organizer	`.OrgObjGrp`

Create an object group for object representation

1. On the **View** tab, click **Representation** to open the **Object Representation** dialog.
2. Click **Object group...** to open the **Object group - Representation** dialog.
3. Select an existing object group from the **Save/Load** list to create a modified version or click **New filter** to start without existing settings.

4. Click **Add row** or continue to modify the settings on an existing row.
5. Select options from the **Category**, **Property**, and **Condition** lists.

   You can use the same object properties (page 181) and techniques (page 198) as in filtering.
6. In the **Value** list, enter a value or select one from the model.

   The values can be complete strings, such as the profile name `UC310*97`. You can also use incomplete strings together with wildcards (page 202). For example, the value `UC*` will match with all parts whose profile name begins with the characters `UC*`. Empty values are matched to empty object properties.

   If you use multiple values, separate the strings with blank spaces (for example, `12 5`). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, `"custom panel"`), or use a question mark (for example, `custom?panel`) to replace the space.
7. Use the **And/Or** options and brackets (page 198) to define how multiple rows work together.
8. To temporarily disable rules without deleting them, you can deselect the check boxes in the first column of the row. Select the checkbox to re-enable the rule.
9. Enter a unique name in the box next to the **Save as** button.
10. Click **Save as** to save the object group.

Object group - Representation settings

When you create object groups in the **Object group - Representation** dialog, you can use the same object properties and techniques as in filtering. The following table introduces the options shortly, for more detailed information see Object properties in filtering (page 181) and Filtering techniques (page 198).

Column	Description
Checkboxes	Select the checkbox to include the row in the rule. By default each new row is disabled
Parentheses	Use parentheses to create nested rules.
Category	Use the **Object** category with user-defined attributes or, for example, if you need to create a selection filter based on GUIDs.
Property	Select an available property. All user-defined attributes can be selected as well.
Condition	Numeric, textual, and date properties all have different set of conditions.

Column	Description
Value	You can enter the value manually, or select the value from the model by clicking an object. For example, if you select **Part** as **Category**, and **Name** as **Property**, and click a part in the model, you get its **Name** property in the object group rule.
And/Or	Use the **And/Or** column when you create rules with several rows. An empty field is the same as if it had the **And** option selected.

Copy object groups to another model

1. Select the object group you want to copy.

   The object groups you have created are located in the model's `\attributes` folder, and they have the file name extension `.PObjGrp`.
2. Select where you want to copy the object group.
   - To make an object group available in another model, copy the file to the `\attributes` folder of the destination model.
   - To make an object group available in all models, copy the file to the project or firm folder, defined by the advanced option XS_PROJECT or XS_FIRM.
3. Restart Tekla Structures.

If you want to delete the object group file in the `\attributes` folder, select the `.PObjGrp` file and press **Delete**.

3.7 If you cannot see all objects

The visibility of objects in a view depends on a number of different settings. If you cannot see all the desired objects in a model view, check the following settings:

- work area
- view depth
- view filter
- view and representation settings
- color and transparency settings

Note that work area and view depth are like two virtual boxes. Objects that have their handles partially or totally inside both boxes are visible. Newly

created objects are also visible outside the view depth but never outside the work area. When you redraw a view, only the objects inside the view depth are displayed.

See also

Define the work area (page 52)

Model view properties (page 50)

Filter objects (page 168)

Change the color and transparency of model objects by using object representation (page 686)

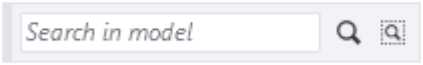
4 Check the model

You can use a variety of tools to check and view the model, and to ensure the model does not contain errors.

- To quickly search for objects in the entire model or within the selected model objects, see Search for model objects (page 697).
- To take snapshots and to create animations that demonstrate the design and build options of your model and use them in your presentations, see Visualize the model (page 700).
- To travel through the model by using the **Fly** command, see Fly through the model (page 720). If you need to focus in on required details in the model, see Create clip planes (page 713). To show parts in a selected view angle, see Show parts, components, or assemblies in a selected view angle (page 719).
- Use the different **Inquire** commands to get particular information about model objects, as described in Inquire object properties (page 721).
- Use the **Measure** commands to measure, for example, distances, as described in Measure objects (page 728).
- To find colliding objects in a model, see Detect clashes (page 731).
- To check the inconsistencies in a model, you can compare selected parts or assemblies, list and locate solid errors, or diagnose and repair the model to check and repair errors, or to find distant objects. For instructions, see Compare parts or assemblies (page 743), Check solid errors (page 746), Diagnose and repair the model (page 748), and Find distant objects (page 750).

4.1 Search for model objects

You can quickly search for objects in the entire model or within the selected model objects by using the **Model search** toolbar.



When you run a search in the model, Tekla Structures searches for objects whose property values contain the search term, and then highlights and selects the objects that have matching property values. Tekla Structures searches for the following objects and checks the listed object properties for matches:

- **Parts and items**: name, profile or shape, material, part position number, assembly position number or cast unit position number, GUID
- **Pour objects**: pour type (if pour management is enabled (page 469))
- **Reinforcement**: name, grade, position number, cast unit position number, GUID
- **Surfaces**: name, GUID
- **Components**: name, running number (shown in the **Inquire object** dialog box), GUID
- **Assemblies**: name, assembly position number, GUID
- **Cast units**: name, cast unit position number, GUID
- **Pour units**: name (if pour management is enabled)
- **All other objects**: GUID

You can use the following wildcards (page 202) in the search term: *, ?, or [ ], or " " for exact match.

The search term may consist of several words. When a property value of an object contains all of those words, Tekla Structures finds that object.

If there is only one word in the search term, Tekla Structures finds all the objects whose property values contain that word. For example, with the search term `Plate` you can find objects with the name `Plate` or `End plate`, but objects with the name `Plate1` are not found.

If you enclose the search term in " ", Tekla Structures only finds the exact matches. For example, `"end plate"` does not find objects with the name `Stiffened end plate` or `Two sided end plate`.

The search is not case-sensitive, which means that you can use both lower-case and upper-case letters. For example, the search terms `beam` and `BEAM` give the same search results.

If the **Select assemblies**  selection switch is active, Tekla Structures searches for assemblies, cast units, and pour units with matching names or position numbers. Otherwise Tekla Structures searches for other objects with matching property values. Other selection switches do not affect the search results.

Search in the entire model

When you run a search in the entire model, Tekla Structures also searches for the hidden objects whose property values match the search criteria.

1. If pour management is enabled (page 469) and you are searching for pour objects or pour units, ensure that you are using a pour view (page 471).
2. On the **Model search** toolbar, enter the search term in the box.

   You can use wildcards *, ?, or [ ], or " " for exact match.

3. Click , or press **Enter**.

Tekla Structures highlights and selects the model objects whose property values match the search criteria, and shows the object properties in the property pane.

Search within the selected model objects

1. Select the objects within which you want to search.

   You can use area selection or a filter (page 168) to focus on certain objects.
2. On the **Model search** toolbar, enter the search term in the box.

   You can use wildcards *, ?, or [ ], or " " for exact match.

3. Click .

Tekla Structures highlights and selects the model objects whose property values match the search criteria, and shows the object properties in the property pane.

Review the search results

You may find these tips useful when you examine the search results and the selected model objects.

- To hide the model objects whose property values do not match the search criteria, do any of the following:
  - Press **Ctrl+5** to show the matching objects only.
  - Press **Shift+5** to show the matching component objects only.
  - Right-click and select **Show only selected (page 684)**.
- To narrow down your search further, you can run nested searches. Keep the first search result objects selected and then run another search within them by clicking .

- To review the matching objects as a list, use **Organizer**. **Object Browser** lists the selected objects and shows their properties in columns.
- To review or modify the properties of the selected objects, use the property pane.

Show or hide the Model search toolbar

By default, the **Model search** toolbar is visible and located at the bottom of the Tekla Structures main window.

If the toolbar is not visible, do one of the following:

- Go to the **File** menu, click **Settings**, and in the **Toolbars** list, select the **Model search toolbar** checkbox.
- Go to **Quick Launch**, start typing `toolbar`, and then select the **Model search toolbar** checkbox in the list that appears.

If you want to hide the **Model search** toolbar, clear the **Model search toolbar** checkbox in **File** --> **Settings** --> **Toolbars** or by using **Quick Launch**.

4.2 Visualize the model with Trimble Connect Visualizer

Trimble Connect Visualizer creates a basic rendering of an open Tekla Structures 3D model. You can use Trimble Connect Visualizer to take snapshots and create animations that demonstrate the design and build options of your model. You can then use the snapshots and animations in your presentations.

You can access Trimble Connect Visualizer directly from Tekla Structures. The Trimble Connect Visualizer commands are located on the **View** tab.

Visualize all model objects

- On the **View** tab, click  **Visualize** and select **Visualize all**.

  A rendering of the entire model opens in Trimble Connect Visualizer.

  Note that parts that are not visible in the Tekla Structures model view are also rendered.

Visualize the selected model objects

1. In the model, select the objects that you want to visualize.
2. On the **View** tab, click  **Visualize** and select **Visualize selected**.

   A rendering of the selected model objects opens in Trimble Connect Visualizer.

Working in Trimble Connect Visualizer

Before you take snapshots or create animations in Trimble Connect Visualizer, you can zoom, drag, and pan the model to get the desired views. You can also adjust the scene settings to control, for example, the brightness of the scene and the position of the sun.

Zoom, rotate, or pan the rendered model

- Do any of the following:

To	Do this
Zoom in	Do either of the following: • Scroll forward with the mouse wheel. • Press the **W** key on the keyboard.
Zoom out	Do either of the following: • Scroll backward with the mouse wheel. • Press the **S** key on the keyboard.
Rotate the model	1. Place the mouse pointer over the point around which you want to rotate the model. 2. Hold down the left mouse button. 3. Drag the model with the left mouse button. You can also press the arrow keys on the keyboard to rotate the model.
Pan the model	1. Hold down the middle mouse button. 2. Drag the model with the middle mouse button. You can also press the **A** and **D** keys on the keyboard to pan the model.

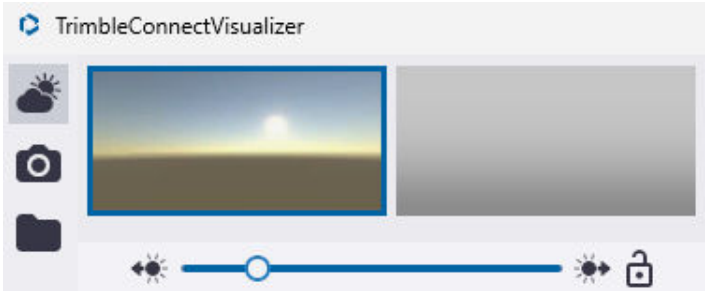
Adjust the scene

In Trimble Connect Visualizer, you can modify the following scene settings:

- The used skybox, or background, and its rotation
- The position of the sun
- The brightness of the scene and other camera settings
- The visibility and position of the ground plane

1. Click  in the Trimble Connect Visualizer side pane.

2. Do any of the following:

To	Do this
Change the skybox	The skybox that is currently in use is highlighted with blue. To change the skybox: • Click another skybox in the skybox selector.  You can also use your own images as skyboxes. Note that all skybox images need to be 360° images with a ratio of 2:1. Save the skyboxes in the `..\Users\<user>\Pictures\TrimbleConnectVisualizer\Skyboxes` folder. The skyboxes will be available in the skybox selector the next time you open Trimble Connect Visualizer.
Rotate the skybox	• Move the rotation slider, located at the top of the rendered model.  You can lock the position of the skybox and the direction of the sun together by clicking . After locking the direction of the sun and the position of the skybox, moving the rotation slider also moves the sun direction slider by the same amount.
Adjust the position of the sun	Do any of the following: • To change the direction of the sun, move the sun direction slider,located at the top of the rendered model.  You can lock the direction of the sun and the position of the skybox together by clicking . After locking the direction of the sun and the position of the skybox, moving the rotation slider also moves the sun direction slider by the same amount.

To	Do this
	• To change the height of the sun above the horizon, move the sun height slider, located on the left side of the rendered model. 
Show or hide the ground plane	• To switch between showing and hiding the ground plane, click , located on the left side of the rendered model. When the ground plane is switched on, a slider appears below the button. You can move the slider to adjust the height of the ground plane. 
Adjust the brightness of the scene and the camera settings	Click in the Trimble Connect Visualizer side pane to further modify different brightness and camera settings.

Take and view snapshots

1. Adjust the position of the model and the scene as needed.
2. Click in the Trimble Connect Visualizer side pane.

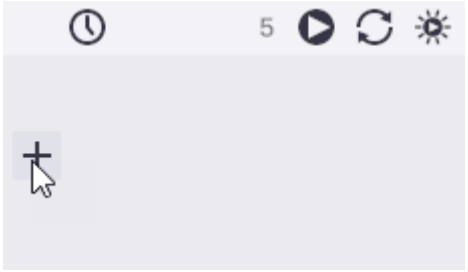
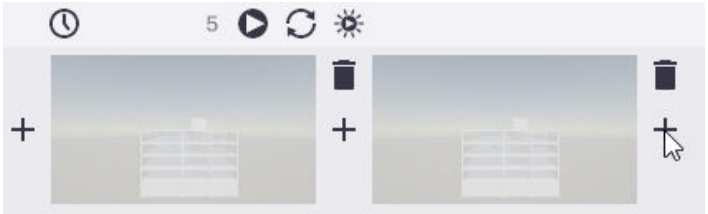
Trimble Connect Visualizer takes a snapshot of the current view of the rendered model. The snapshots are saved in the `C:\Users\<username>\Pictures\TrimbleConnectVisualizer` folder.

To open the `C:\Users\<username>\Pictures\TrimbleConnectVisualizer` folder and view the snapshots that you have taken, click in the Trimble Connect Visualizer side pane.

Create animations

You can combine different views of the rendered model to create animations that show the model from different angles. Do the following:

1. Click in the Trimble Connect Visualizer side pane.
2. Do any of the following:

To	Do this
Add a view	a. Move the model to get the view that you want to use. b. Click **+** on the toolbar below the view.  Repeat steps a to b to create all necessary views. Note that when you have added multiple views, you need to click **+** on the right side of the view after which you want to add the new view. Similarly, click **+** or on the left side of the view before which you want to add the new view. 
Remove a view from the animation	• Click the **Remove** button next to a view.

To	Do this
Set the time between views	The time between views determines the speed of the animation. • Type the desired time in seconds in the box below the rendered model ().
Play or stop the animation	• To play the animation, click  on the toolbar below the rendered model. • To stop the animation and return to the first view, click .
Play the animation continuously	You can choose to play the animation in a loop without stopping. a. Click  on the toolbar below the rendered model. b. To play the animation, click .
Freeze the sun position in the animation	In each view you create, the sun is in a different position. Animations can look strange if the sun moves, so it might be helpful to freeze the sun position. • Click  on the toolbar below the rendered model.

3. To save the animation, use a screen recorder.

Return to the initial view of the model

Click  in the Trimble Connect Visualizer side pane.

The model returns to the view that was opened when you started Trimble Connect Visualizer.

Enter or exit full screen mode

Click  in the Trimble Connect Visualizer side pane to enter the full screen mode.

Click  to exit the full screen mode.

Show or hide the Trimble Connect Visualizer side pane

Click  at the bottom of the Trimble Connect Visualizer side pane.

Use Trimble Connect Visualizer in VR mode

The VR mode in Trimble Connect Visualizer allows you to easily switch to a virtual reality mode. In the VR mode, you can move either with the keyboard or with an Xbox One controller that is connected to your computer. Note that moving in the VR mode imitates the speed of walking or running.

The VR mode uses the OpenVR API. Because of this, using the VR mode in Trimble Connect Visualizer has some prerequisites:

- A VR device that is compatible with OpenVR must be attached to your computer and configured according to the instructions of the hardware vendor. Among others, HTC Vive, Oculus Rift, Windows MR, and VarjoVR devices are compatible with OpenVR.

  The VR mode in Trimble Connect Visualizer is heavier to render than the regular 3D mode. Check the technical specifications of the device to see if the device has enough capacity for working in the VR mode in Trimble Connect Visualizer. For more information, contact the vendor of your device.
- Steam and SteamVR need to be downloaded and installed onto your computer:

  1. Download Steam and install it onto your computer.
  2. Create a Steam account and sign in to Steam.
  3. Download the SteamVR package onto your computer.
  4. Start SteamVR and configure it.

  You do not need to be signed in to Steam to work in the VR mode in Trimble Connect Visualizer. It is enough to set up Steam once, and then sign in for updates occasionally.

1. Click  in the Trimble Connect Visualizer side pane.

   If the VR mode icon () is not visible, your computer does not meet the requirements of the VR mode.

2. Move in the VR mode as you wish.

To move in the VR mode with	Do this
Your keyboard	The keyboard controls are the same as in the regular 3D mode: • To move forward, press the **W** key. • To move backward, press the **S** key. • To move left, press the **A** key. • To move right, press the **D** key. • To move up and down, press the **Q** and **E** keys. • To rotate around the global vertical axis, press the **X** and **Z** keys. • To move at running speed, hold down **Shift** .
An Xbox One controller	When an Xbox One controller is connected to your computer via Bluetooth, use the following controls: • To move the viewer relative to the view direction, use the left stick. • To rotate around the global vertical axis, use the right stick. • To move up and down, use the left and right triggers. • To move at running speed, hold down the left stick.

3. To switch back to the 3D mode, click  again.

Modify material mappings for Trimble Connect Visualizer

By default, Trimble Connect Visualizer attempts to map the materials used in Tekla Structures to the material types used in the Trimble Connect Visualizer material catalog. You can also set material type mappings manually to define how you want materials to look in Trimble Connect Visualizer. If necessary, you can override the materials of particular model objects, and use another material instead.

You can currently map the Tekla Structures materials to the following pre-defined material types in Trimble Connect Visualizer:

- Asphalt
- Concrete
- Dark gravel

- Default: renders objects with white matte material
- Glass
- Grass
- Ground
- Light Gravel
- Steel
- Timber
- Water

Pre-defined materials and their color and transparency information are read from the `materials_ifc.xml` file saved in the `..\TeklaStructures\<version>\bin\applications\Tekla\Tools\TrimbleConnectVisualizer\TrimbleConnectVisualizer_Data\StreamingAssets` folder.

You can also map to your own user-defined materials. For more information, see Create and modify user-defined materials in Trimble Connect Visualizer (page 709).

In Trimble Connect Visualizer, you can currently only render the material type of objects, not their material finish. This means that paints or other finishes are not visible in Trimble Connect Visualizer.

To set the material type mappings used in Trimble Connect Visualizer:

1. On the **View** tab, click  **Visualize** and select **Visualizer material type mapping**.
2. In the lists on the right side of the related Tekla Structures materials, select the Trimble Connect Visualizer material types that you want to map to.

   Selecting the blank option renders the color of the Tekla Structures class as a material that is similar to plastic. The blank option can be useful if you need to show the class colors to communicate an aspect of the structural design.
3. To override the Tekla Structures material of particular model objects in Trimble Connect Visualizer:
   a. In the model, select the objects.
   b. In the **Override Visualizer materials of selected objects** list, select the material that you want to use for visualizing the selected objects.
   c. Click **Set**.

   The selected override material is stored to the model objects' `VISUALIZER_MATERIAL` user-defined attribute, which you can see by inquiring the object properties (page 721).

4. Click **OK** to update the material type mapping.
5. To save the material type mapping as an XML file, do either of the following:

To	Do this
Save the material mapping to the model folder	• When Tekla Structures asks you if you want to save the material mapping to the model, click **Yes**. The material type mapping is saved in the model folder under the `\attributes` folder. The file name is `VisualizerMaterials.tcv11`. The material type mapping is saved for all users of the model, including the users that edit the model using Tekla Model Sharing or the multi-user mode. To avoid issues, do not modify the colors in the `VisualizerMaterials.tcv11` file manually.
Save the material mapping to local application data	• When Tekla Structures asks you if you want to save the material mapping to the model, click **No**. The material type mapping is saved in the `\Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\Trimble Connect Visualizer` folder. The file name is `VisualizerMaterials.tcv11`. The material type mapping now applies to all projects that you are working on. To avoid issues, do not modify the colors in the `VisualizerMaterials.tcv11` file manually.

Create and modify user-defined materials in Trimble Connect Visualizer

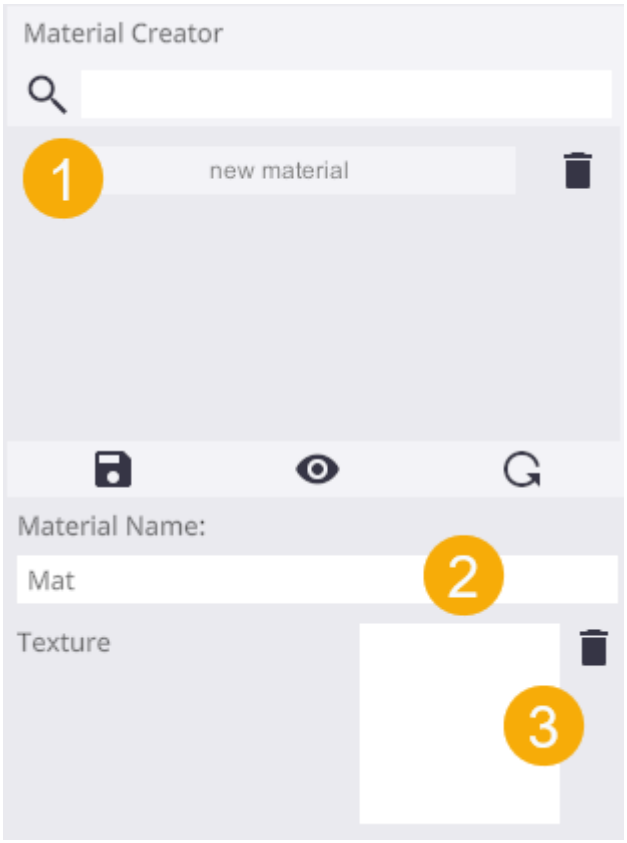
You can create your own customized materials in Trimble Connect Visualizer with the material editor. The customized materials can be used in Trimble Connect Visualizer material mapping in the same way as pre-defined materials.

NOTE User-defined materials are not shared to other users with Tekla Model Sharing. Any objects that use user-defined materials appear with their class colors to other users.

To create or modify user-defined materials:

1. Visualize the model. (page 700)

2. To open the material editor, click  in the Trimble Connect Visualizer side pane.

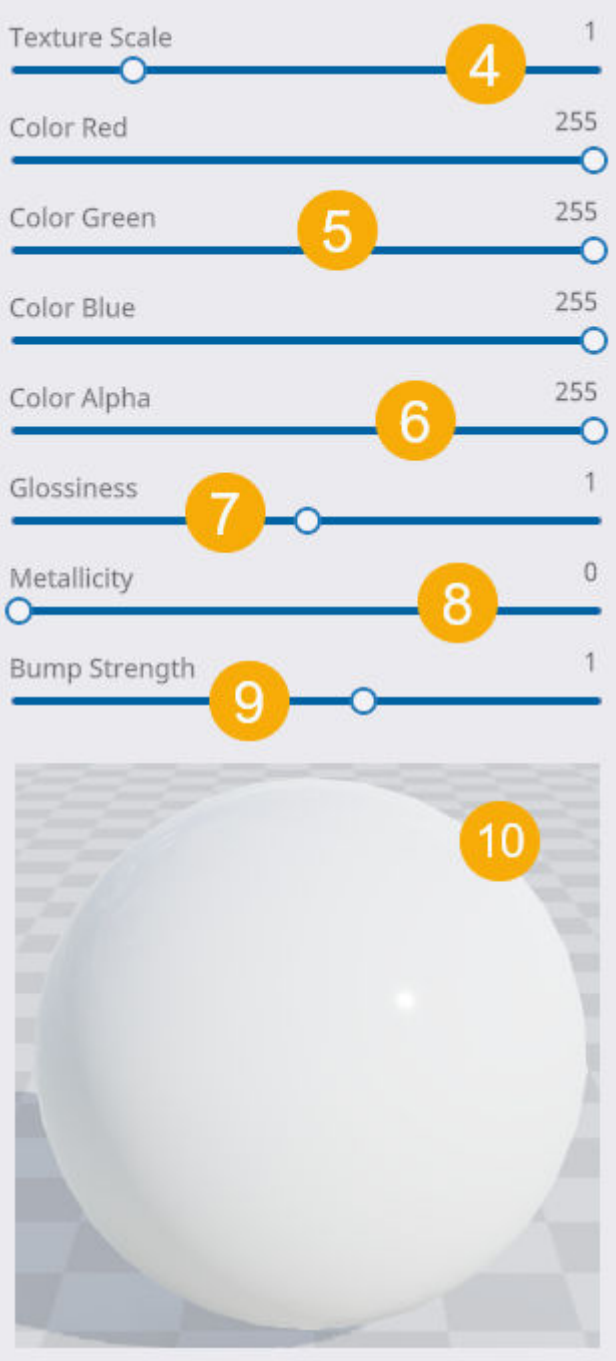


The material creator contains the following elements:

(1) List of user-defined materials

(2) Material name box

(3) Texture picker



(4) Scaling or tiling of the selected texture

(5) RGB color values

(6) Transparency

(7) Roughness

(8) Metallic look

(9) Bump strength

(10) Preview of the material

3. Do one of the following:
   - To create a new material, type a name in the material name box.
   - To modify an existing user-defined material, select the material in the list of user-defined materials.
4. In the material editor, do any of the following:

To	Do this
Add a texture	a. Click the texture picker. b. Click the texture that you want to use. Note that you can only use textures that are saved in the `..\Users\<user>\Pictures\TrimbleConnectVisualizer\Textures` folder as `.png` or `.jpg` images. To reset the material texture, the **Remove** button next to the texture picker.
Adjust scaling or tiling of texture, RGB colors, transparency, roughness, metallic look, or bump strength	Drag the related slider.
Preview all objects with the current user-defined material settings	Click . To reset the objects to use their original materials, click .

5. To save the material, click .

   User-defined materials are saved in the `..\Users\<user>\AppData\Local\TrimbleConnectVisualizer\CustomMaterials` folder.

   Each user-defined material is defined using the following files:

   - `<Material name>.xml`
   - `<Material name>_DiffuseMap.png`
   - `<Material name>_NormalMap.png`
   - `<Material name>_RoughnessMap.png`

Note that the name of the material file and the material name in the `.xml` file need to be the same.

To avoid issues, do not modify any of the material files manually.

See also

Modify material mappings for Trimble Connect Visualizer (page 707)

Working in Trimble Connect Visualizer (page 701)

4.3 Create clip planes and clip boxes

Clip planes and clip boxes enable you to focus on specific details in the model by isolating parts of the model for detailed examination.

With *clip planes* you can see inside the model by clipping out a part of the view. You can either create clip planes by selecting an object face, or create clip planes that are based on the view depth.

With *clip boxes* you can isolate a specific area within a model by hiding the rest of the model.

Note that you cannot create clip planes to a view which already has a clip box in it.

Create a clip plane

You can create up to six clip planes in each model view.

To create clip planes or clip boxes only to reference model objects and point clouds, select the **Clip only reference objects** checkbox in **View** --> **Clipping**. When selected, the native Tekla Structures objects are not clipped. When the checkbox is cleared, clip planes and clip boxes in all views clip all objects.

1. When you create clip planes, ensure that you are using a model view that shows object faces.

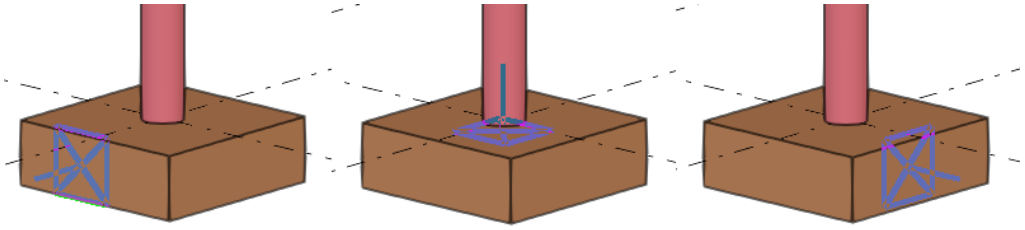
   On the **View** tab, click **Rendering**, and use any of the following options:

   - **Parts shaded wireframe** (Ctrl+2)
   - **Parts grayscale** (Ctrl+3)
   - **Parts rendered** (Ctrl+4)
   - **Components shaded wireframe** (Shift+2)
   - **Components grayscale** (Shift+3)
   - **Components rendered** (Shift+4)
   - **References shaded wireframe** (Ctrl+Shift+2)

- **References grayscale** (Ctrl+Shift+3)
- **References rendered** (Ctrl+Shift+4)

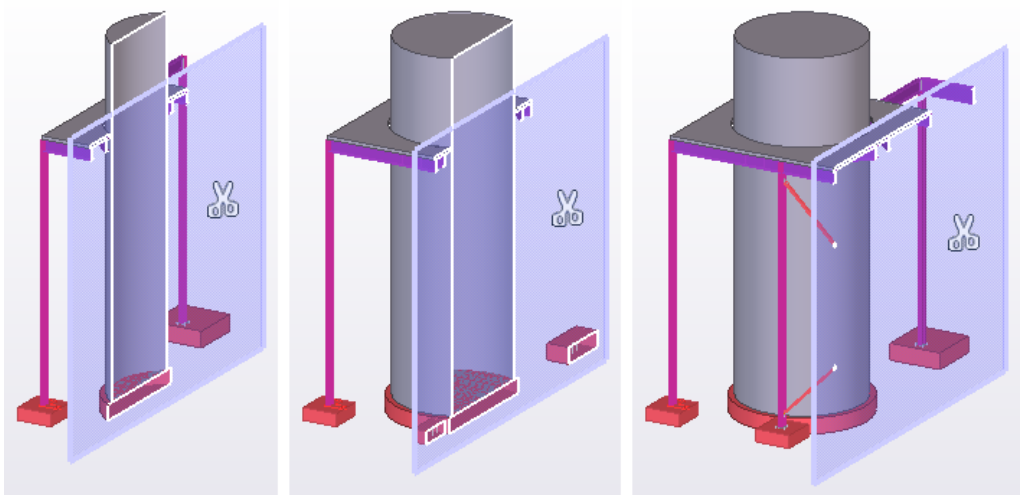
2. On the **View** tab, click **Clipping** --> **Clip plane**.
3. Move the mouse pointer over the model objects.

   A cross symbol indicates the object faces and planes that you can select and align the clip plane with. The line symbol indicates the side that will be clipped out. For example:



   Note that the snap depth setting on the **Snapping** toolbar affects the object faces you can select. Set snap depth to **3D** or **Auto** to select object faces in the entire 3D space.
4. Select an object face.

   The clip plane symbol appears in the model .
5. Repeat step 4 to create as many clip planes as needed.
6. To finish creating clip planes, press **Esc**.
7. To move a clip plane, select the clip plane scissor symbol and drag it to a new location.



8. If you want to move the clip plane scissor symbol to a new location on a clip plane, hold down **Shift** and drag the symbol.

   This does not move the clip plane, only the scissor symbol.

9. Delete the clip planes when you do not need them anymore.
   - To delete a single clip plane, select the clip plane symbol and press **Delete**.
   - To delete all the clip planes in all the open views, click **View** --> **Clipping** --> **Delete all clip planes and clip boxes**.

Create a clip box

Clip boxes enable you to focus on the required detail in the model, without the need to create multiple clip planes. You can isolate an area of the model for closer inspection with one click. You can create one clip box per one model view.

With clip boxes, you can either

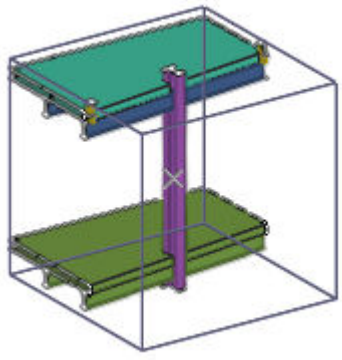
- create a clip box around a larger area of the model. The default size of a clip box is 5000 x 5000 x 5000 mm.
- create a clip box around a selected object in the model.

To create clip planes or clip boxes only to reference model objects and point clouds, select the **Clip only reference objects** checkbox in **View** --> **Clipping**. When selected, the native Tekla Structures objects are not clipped. When the checkbox is cleared, clip planes and clip boxes in all views clip all objects.

Create a clip box around a picked point

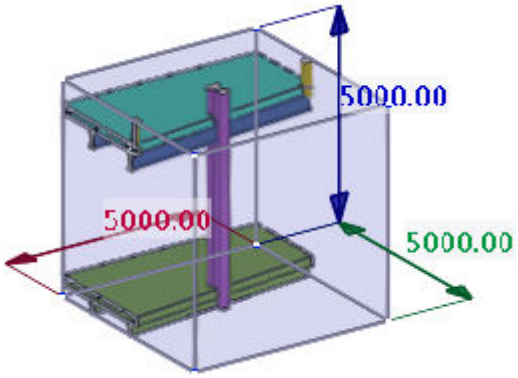
1. On the **View** tab, click **Clipping** --> **Clip box**.
2. In the model, pick the center point for the clip box.

   The clip box is created. The axis of the clip box follows the work plane.

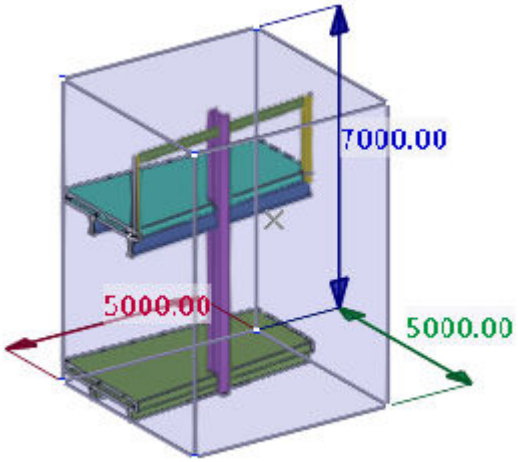


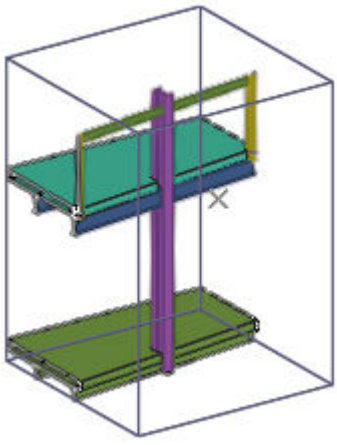
3. To change the position of the clip box, pick a new center point for the clip box.
4. To finish creating a clip box, press **Esc**.

5. To adjust the clip box size, select the clip box.
   The direct modification handles appear to the clip box.



6. Drag any of the handles, or modify the dimension values.





7. Delete the clip box when you do not need it anymore.
   - To delete a clip box, select the clip box and press **Delete**.

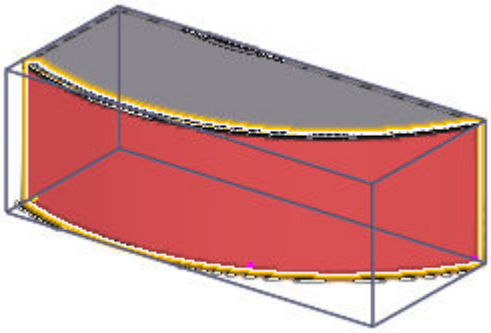
- To delete all clip boxes in all open views, click **View** --> **Clipping** --> **Delete all clip planes and clip boxes**.

Create a clip box around a selected object

In addition to the default sized clip box, you can create a clip box around a selected part, assembly, bolt, reinforcement, reference model, reference object, or buildings or spaces in building hierarchy. The clip box then contains only the selected object.

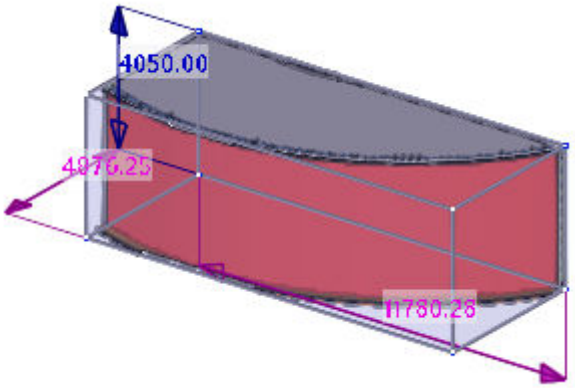
1. In the model, select the object you want to create the clip box around.
2. Right-click and select **Create clip box**.

   The clip box is created around the selected object, with its geometry defined by the object. The orientation of the clip box follows the coordinate system of the selected object.

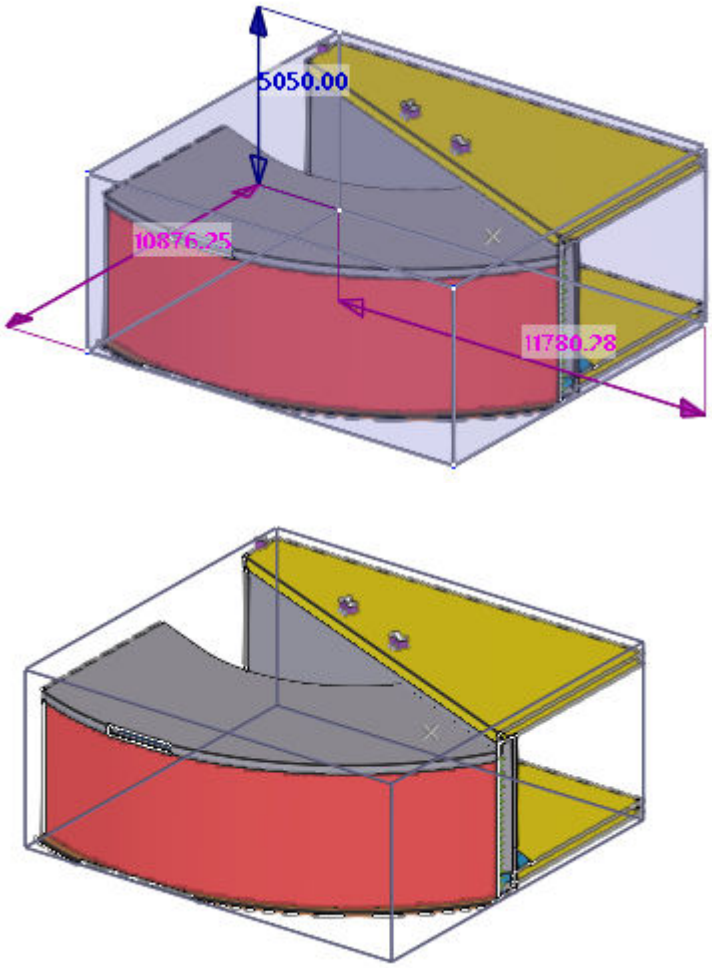


3. To adjust the clip box size, select the clip box.

   The direct modification handles appear to the clip box.



4. Drag any of the handles, or modify the dimension values.

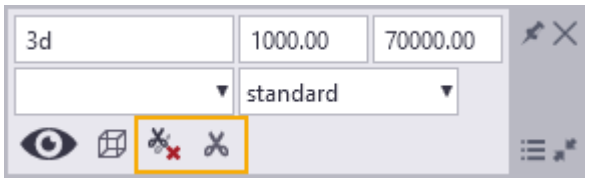


5. Delete the clip box when you do not need it anymore.
   - To delete a clip box, select the clip box and press **Delete**.
   - To delete all the clip boxes in all the open views, click **View** --> **Clipping** --> **Delete all clip planes and clip boxes**.

Create a view depth clip plane

If the model objects do not fit to the work area view depths, you can create view depth clip planes on the contextual toolbar.

Based on the view depth, you can, for example, easily isolate an entire floor from a building. View depth clip planes can be created both in native Tekla Structures models as well as in reference models and overlay models.



1. If needed, modify the view depth of the current view on the contextual toolbar.
2. On the contextual toolbar, click **Add or update view depth clip planes** .
3. If you want to remove the view depth clip planes, click **Remove view depth clip planes** on the contextual toolbar, or click **View** --> **Clipping** --> **Delete all clip planes and clip boxes**.

   Note that the work area box may have gaps to view depths, and when working with overlay models, you can clip models that are outside the work area.

Limitations:

- You cannot move the view depth clip planes by dragging the clip plane to a new location. This is because the view depth clip planes are strictly connected to the view depth.
- After you have adjusted the view depth, click the **Add or update view depth clip planes** button to manually update the view depth clip plane.

4.4 Show parts, components, or assemblies in a selected view angle

In some cases it is useful to examine parts, components, or assemblies in a selected view angle. For example, when you reinforce concrete elements, you can easily check the distances between the reinforcing bars.

Limitation:

The **View angle** option is not available for:

- detail system components
- detail custom components
- custom seams

1. Select an object.
2. Click **View angle** on the contextual toolbar.
3. Select a top, back, right, bottom, front, or left view.

Tekla Structures displays the object in the selected view angle. Note that the full object is not always displayed.

For parts, the view angle is based on the selected object's coordinate system, so that the top view is looking to the negative z-direction. For example, if you select the top view, Tekla Structures sets the current view angle from up to down in the coordinate system.

For assemblies and components, the view angle is based on the assembly main part's coordinate system. Because of that, the object may not always appear as expected in the selected view angle.

4. To return to the original 3D view, click the button in the middle of the view angle options.

NOTE Tekla Structures shows the object in the current view, which is the view where your mouse has last been. If the contextual toolbar is located, for example, on top of two views, the object is shown in the view where your mouse has last been and not in the view where the object was selected.

See also

Adjust how model objects are displayed (page 671)

4.5 Fly through the model

Using the **Fly** command, you can travel through a model, changing direction and varying the speed as you go. You can also adjust the field of view setting, which can be useful when flying in a tight space.

1. Set view projection to **Perspective**.
   a. Double-click the view to open the **View properties** dialog.
   b. In the **Projection** list, select **Perspective**.
   c. Click **Modify**.
2. If needed, adjust the field of view setting.

   The bigger the value, the more distance there is between the parts when you fly through the model.
   a. On the **File** menu, click **Settings** --> **Advanced options** , and go to the **Model views** category.
   b. Modify the advanced option XS_RENDERED_FIELD_OF_VIEW.
   c. Click **OK**.
3. On the **View** tab, click **Fly**.
4. Select a view.

The mouse pointer changes into an arrow and a cross. The arrow indicates the current flying direction.



5. Drag the mouse to move around in the model.
   - To fly forward, move the mouse forward.
   - To change the flying direction, drag the mouse in the desired direction.

     The flying speed grows exponentially when you are approaching the model from a distance.
   - To move up or down, hold down **Ctrl** and drag the mouse forward or backward.
   - To change the camera angle, scroll with the mouse wheel.
   - To fly in the in the direction of the camera angle, hold down **Shift** and scroll forward or backward.
6. To stop flying, press **Esc**.

4.6 Inquire object properties

Use the **Inquire** commands to get information about a particular object, or a group of objects, within the model.

Do any of the following:

To inquire	Do this
Object properties	1. On the ribbon, click **Object**. 2. Select an object. Tekla Structures displays the object properties in a separate window.
Point coordinates	1. On the ribbon, click the down arrow next to , and then select **Point coordinates**. The **Inquire point coordinates** dialog box is displayed.

To inquire	Do this
	2. Click **Pick**, and then pick a point in the model to see the coordinates of the point in: • Local coordinates • Model coordinates (global) • Project base point coordinates • Current base point coordinates
Center of gravity	1. On the ribbon, click the down arrow next to , and then select **Center of gravity**. 2. Select one or more parts or reinforcement objects. Tekla Structures creates a point at the center of gravity of the selected objects, and displays information about the center of gravity in a separate window.
Object properties using custom reports	See Custom inquiry (page 724).
Welded parts	1. On the ribbon, click the down arrow next to , and then select **Welded parts**. 2. Select a part. Tekla Structures highlights the selected part and all the parts that are welded to it.
Primary welded parts	1. On the ribbon, click the down arrow next to , and then select **Primary welded part**. 2. Select a part. Tekla Structures highlights the primary part when you select a secondary part.
Assembly, cast unit, or rebar assembly objects	See Check and highlight objects in an assembly (page 452), Check and highlight objects in a cast unit (page 454), or Check and highlight objects in a rebar assembly (page 647).
Component objects	1. On the ribbon, click the down arrow next to , and then select **Component objects**. 2. Select a component. Tekla Structures highlights all objects belonging to the selected component.

To inquire	Do this
Rebar set layers	1. On the ribbon, click the down arrow next to , and then select **Rebar set layers**. 2. Select a rebar set. Tekla Structures displays a separate window where you can check which rebar sets affect the selected rebar set, and which rebar sets are affected by the selected rebar set. For example, you can review the layer order number, concrete cover thickness, additional offset, and layer thickness at each leg face or leg surface of the selected rebar set. If you select a row with a GUID in the list, Tekla Structures highlights the corresponding rebar set in the model. Alternatively, you can first select a rebar set in the model, and then click  on the contextual toolbar.
Phases	On the ribbon, click the down arrow next to , and then select **Phases**. Tekla Structures displays information about objects in different phases in a separate window.
Model size	On the ribbon, click the down arrow next to , and then select **Model size**. Tekla Structures displays the quantity of all objects in the current model in a separate window.

See also

Object property report templates (page 723)

Object property report templates

When you view object properties using the **Inquire object** command, Tekla Structures uses the following report templates:

Object type	Template
Assemblies	TS_Report_Inquire_Assembly.rpt

Object type	Template
Bolts	TS_Report_Inquire_Bolt.rpt
Cast units	TS_Report_Inquire_Cast_Unit.rpt
Parts	TS_Report_Inquire_Part.rpt
Pour breaks	TS_Report_Inquire_Pour_Break.rpt
Pour objects	TS_Report_Inquire_Pour_Object.rpt
Pour units	TS_Report_Inquire_Pour_Unit.rpt
Rebar assemblies	TS_Report_Inquire_Rebar_Assembly.rpt
Reinforcement meshes	TS_Report_Inquire_Rebar_Mesh.rpt
Rebar sets	TS_Report_Inquire_Rebar_Set.rpt
Reinforcement strands	TS_Report_Inquire_Rebar_Strand.rpt
Reference models	TS_Report_Inquire_Reference.rpt
Reinforcement	TS_Report_Inquire_Reinforcement.rpt
Surfaces	TS_Report_Inquire_Surface.rpt
Welds	TS_Report_Inquire_Welding.rpt

When Tekla Structures is installed in the `..\Program Files` folder, these templates are by default available in the folder `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\system`.

You can copy these templates to your project or firm folder and then modify the copied templates according to your needs. For more information on how to use templates, see the Template Editor user documentation.

You can also create a custom template for connections and details by saving the template with the name `TS_Report_Inquire_Connection.rpt`.

See also

Inquire object properties (page 721)

Custom inquiry

You can use the **Custom inquiry** command to display information about the selected model object in the side pane. You can define what information you want to display.

Use the Custom Inquiry tool

1. Click the **Custom Inquiry** button  in the side pane.

Alternatively, you can click the down arrow next to  on the ribbon, and then select **Custom inquiry**.

The **Custom Inquiry** window opens in the side pane.

2. In the **Report type** list, select the report template you want to use for showing the object information.
3. Select a model object.

   Tekla Structures displays the object properties in the side pane.

   If you select several objects or object types, for example parts, bolts, and reinforcing bars, Tekla Structures displays the quantity of all the selected objects, regardless of the object types, or the report template used. For the object properties that differ, Tekla Structures shows **Varies**.

Define what information is shown by Custom Inquiry tool

You can define what information is displayed in the **Custom Inquiry** side pane window. You can add and modify report templates and the attributes in them.

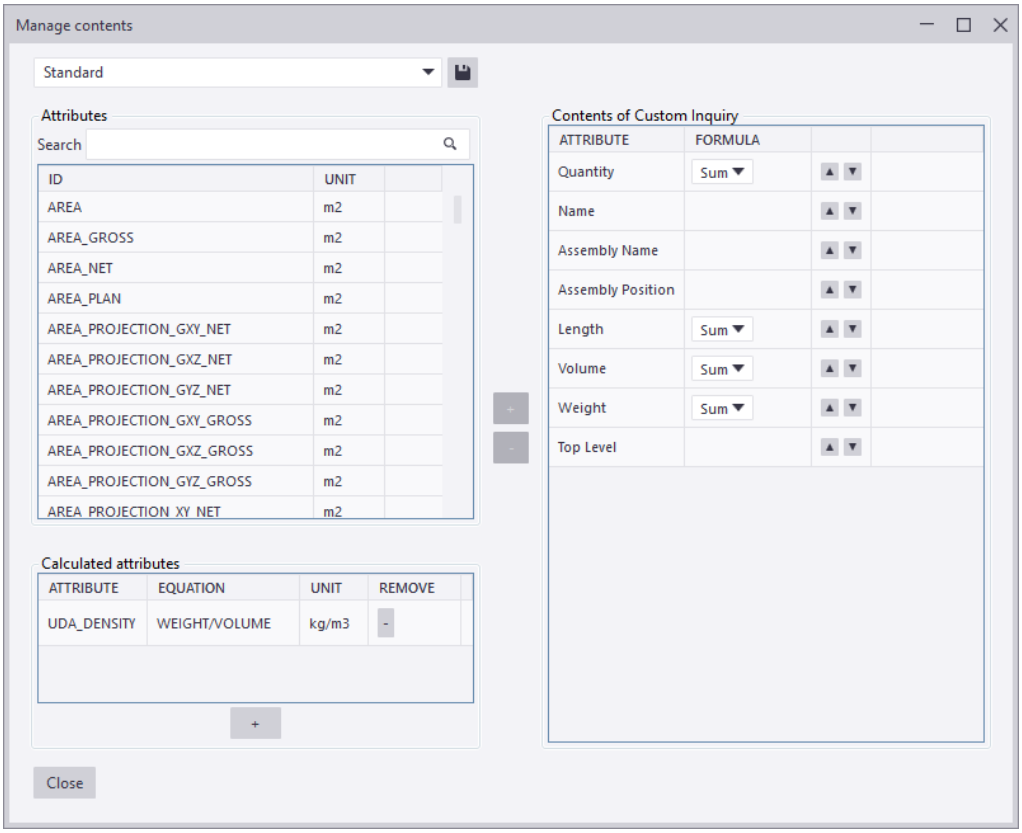
1. Click the **Custom Inquiry** button  in the side pane.

   Alternatively, you can click the down arrow next to  on the ribbon, and then select **Custom inquiry**.

   The **Custom Inquiry** window opens in the side pane.

2. Click the  button.

The **Manage contents** dialog appears.



The **Attributes** list contains the attributes that are available by default. In the **Calculated attributes** area you can create your own attribute formulas. The **Contents of Custom Inquiry** list contains the attributes whose values will be displayed in the side pane.

3. Define which report templates and attributes are available.

   - To modify an existing report template, select it from the top left-hand list in the **Manage contents** dialog.
   - To create a new report template, enter a name in the box next to the  button, and then click .
   - To modify the default attributes, edit the `InquiryTool.config` file.

     For example, you can change the units of the attributes.

   - To create a new calculated attribute, click the button below the **Calculated attributes** area. To modify a calculated attribute, double-click a cell in the **Calculated attributes** area. In the **Attribute** cell, type the name of the attribute. In the **Equation** cell, use attribute names and standard mathematical symbols (+, -, *, and /) to form equations.

4. Define which attributes are shown in the **Custom Inquiry** side pane window.
   - To add more attributes to the side pane, select an attribute in the **Attributes** or **Calculated attributes** list and then click the + button.
   - To remove attributes from the side pane, select an attribute in the **Contents of Custom Inquiry** list and then click the - button.
   - To change the order of the attributes, use the ▲ ▼ buttons.
   - To change the formula of an attribute, click the down arrow and select a different formula (**Sum**, **Average**, **Max**, or **Min**) from the list.
5. Click to save the changes.

Modify the default attributes in InquiryTool.config file

Use the `InquiryTool.config` file to control which attributes are shown as default attributes in the **Manage contents** dialog in the **Custom Inquiry** tool.

NOTE This section is for advanced users.

Tekla Structures searches for the `InquiryTool.config` file in the following folders in the following order:

1. `\attributes` folder in the model folder
2. `\CustomInquiry` subfolder in the folder defined by XS_PROJECT
3. `\CustomInquiry` subfolder in the folder defined by XS_FIRM
4. `\CustomInquiry` subfolder in the folder defined by XS_SYSTEM

If you specify several folders that have `\CustomInquiry` as a subfolder, Tekla Structures uses the first folder it finds.

To add new attributes to the `InquiryTool.config` file:

1. Open the `InquiryTool.config` file in any standard text editor.
2. Copy the entire contents of `[ATTR_CONTENT_??]` to the end of the file.
3. Change the position number of the new attribute.

   For example, change `[ATTR_CONTENT_??]` to `[ATTR_CONTENT_66]`.
4. Modify the `NAME`, `DISPLAY_NAME`, `DATATYPE`, `UNIT`, and `DECIMAL` values of the new attribute. Use the attribute names and definitions that are included in the `contentattributes_global.lst` or `contentattributes_userdefined.lst` file.

5. Change the `TOTAL_ATTR_CONTENT` value to reflect the total number of attributes in the file.

   For example, change `TOTAL_ATTR_CONTENT=65` to `TOTAL_ATTR_CONTENT=66`.

6. Save the file.

4.7 Measure objects

Use the **Measure** commands to measure angles, arcs, the distance between two points and between bolts in the model.

All measurements in the model views are temporary. The measurements appear in the model view window until you update or redraw the window. For instructions how to remove the measurements, see Update and refresh views (page 49).

The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.

Measure distances

You can measure horizontal, vertical and user-defined distances in the model. All measurements are temporary, and they appear in the model view window until you update or redraw the window.

1. Press **Ctrl+P** to switch to the plane view.
2. On the **Edit** tab, click **Measure** and select one of the following commands:
   - **Distance**

     This command measures the distance between any two points. Use this command to measure inclined or aligned distances. By default, the results contain the distance and the coordinates.

   - **Horizontal distance**

     This command measures the distance between two points in the direction of the view plane x axis.

   - **Vertical distance**

     This command measures the distance between two points in the direction of the view plane y axis.

3. Pick the start point.
4. Pick the end point.

5. To indicate on which side of the dimension line you want the measurement to appear, pick a point.

   The measurement appears in the model. It cannot be moved in the model.

6. To remove the measurements, refresh the view window by right-clicking the view and selecting **Update window**.

Measure angles

You can measure angles in the model. All measurements are temporary, and they appear in the model view window until you update or redraw the window.

1. On the **Edit** tab, click **Measure** --> **Angle**.
2. Pick the center point.
3. Pick the start point.
4. Pick the end point.

   The measurement appears in the model. It cannot be moved in the model.

5. To remove the measurements, refresh the view window by right-clicking the view and selecting **Update window**.

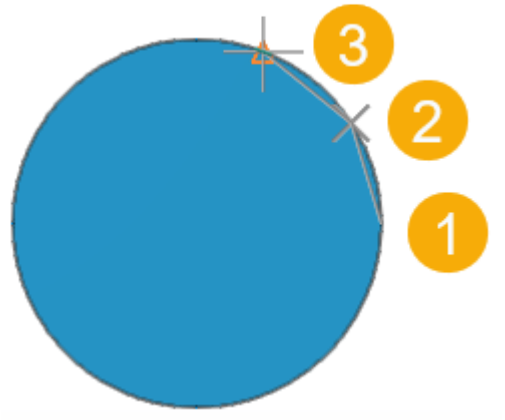
Measure arcs

You can measure the radius and length of an arc in the model. All measurements are temporary, and they appear in the model view window until you update or redraw the window.

1. On the **Edit** tab, click **Measure** --> **Arc**.
2. Pick the start point.
3. Pick the middle point.

   This can be any point along the arc between the start and end points.

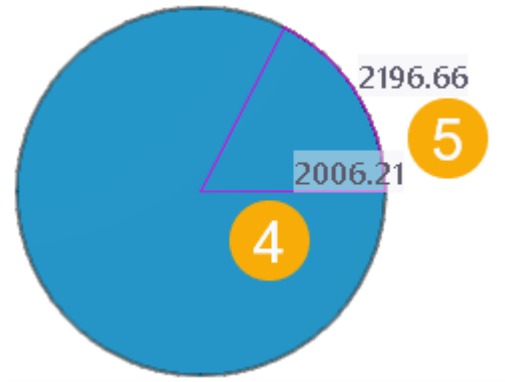
4. Pick the end point.



(1) Start point

(2) Middle point

(3) End point



(4) Arc radius

(5) Arc length

The measurements appear in the model. They cannot be moved in the model.

5. To remove the measurements, refresh the view window by right-clicking the view and selecting **Update window**.

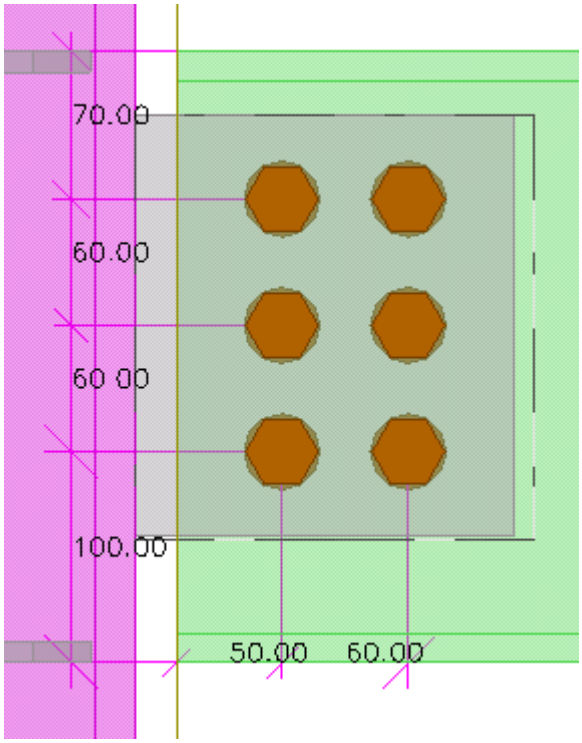
Measure bolt spacing

You can measure distances between bolts in a bolt group. Tekla Structures also gives you the edge distances between the bolts and a selected part. All measurements are temporary, and they appear in the model view window until you update or redraw the window.

1. On the **Edit** tab, click **Measure** --> **Bolt spacing**.
2. Select a bolt group.

3. Select a part.

   The measurements appear in the model. They cannot be moved in the model.



4. To remove the measurements, refresh the view window by right-clicking the view and selecting **Update window**.

4.8 Detect clashes

Use the **Clash Check Manager** tool to find objects that collide or that are too close to each other.

Clash Check Manager can check clashes within native Tekla Structures models, within reference models, and between the objects in these models.

Use the clash check settings to define clearances between different model objects.

For information on point cloud clash check, see the instructions about modifying point cloud properties and visualization settings.

You can also use the sections and floors created in **Organizer** to run a controlled clash check.

If you want to use another Tekla Structures model as a reference model, you must export it in the IFC format to be able to use it in clash check. The following reference model file types are supported in clash check:

- IFC
- IFC4
- DWG
- DGN
- SKP
- .tekla (overlay models excluded)

See also

Find clashes in a model (page 732)

Modify, view, and print clash check results (page 739)

Find clashes in a model

You can use clash check to find clashes between reference model objects and pour objects, and between these objects and native Tekla Structures parts, and inside reference models and Tekla Structures models.

Check the model for clashes

1. On the **Manage** tab, click **Clash check** .
2. In the **Clash Check Manager** dialog, select the appropriate clash check settings to determine what is included in the clash check.
   - **Between reference models**

     Clashes between reference models are included in the clash check.
   - **Objects in reference models**

     Clashes within reference models are included in the clash check (bolts and welds are not included).
   - **Between reference models and components**

     Clashes between reference models and components are included in the clash check.

     When you have the whole component and this setting selected, the component parts are included in the clash check.

     Note that if you already have the **Select objects in components** selection switch active to select the parts in the component, the **Between reference models and components** setting does not have any effect, even if you have the check box selected in **Clash Check Manager**.

- **Between parts**

  Clashes between native Tekla Structures parts are included in the clash check.

3. Set the **Minimum distance** between reference model objects and pour objects, and between these objects and native Tekla Structures parts.

   Objects that are closer to each other than the set minimum distance are reported in the clash check results. You can filter out minimum distance type clashes from the clash check results by setting the minimum distance value to `0.0`.

   You can define a minimum distance value within the range from `500 mm` to `-500.0 mm`. With a negative minimum distance, you can define how much overlap is allowed without reporting a clash.

   When checking the minimum distance, **Clash Check Manager** uses the Tekla Structures units set in **File** --> **Settings** --> **Options** --> **Units and decimals**.

   Note that Tekla Structures uses a hard-coded tolerance when checking clashes between native parts. If the native parts overlap less than 0.25 mm, clashes are not reported.

4. In the model, select the objects that you want to include in the clash check.

5. In the **Clash Check Manager** dialog, click  to check the objects.

   The status bar in **Clash Check Manager** indicates that clash check is in progress.

   Tekla Structures also displays a dialog with a clash check progress bar. The progress bar first shows the clash check progress for Tekla Structures objects, and then for reference model objects and pours. You can cancel the clash check in the dialog.

   When the clash check is finished, the status bar in the **Clash Check Manager** dialog shows a message about listing the clash check results and a progress bar that indicates how long it will take to list all the clashes. When the listing is complete, the status bar message changes to **Ready**.

6. If you want to include more objects in the clash check, select the desired model objects and re-run the clash check.

   New clashes are appended to the end of the list.

7. After removing or modifying objects, re-run the clash check to see if the clash still exists.

   a. Select the desired clashes in **Clash Check Manager**.

   b. Click  to re-run the clash check.

NOTE If you do not wish to check the entire model, you can use **Organizer** to select the sections and floors for which you want to run the clash check. Select a section or a floor in the category tree in **Organizer**, right-click and select **Select in the model**.

NOTE If you cannot find the clashes in the model, change the rendering of objects to **Show only selected part** (**Ctrl+5**) for better visibility.

Note that clashes are not detected if the profiles and positions are identical.

Symbols used in clash check

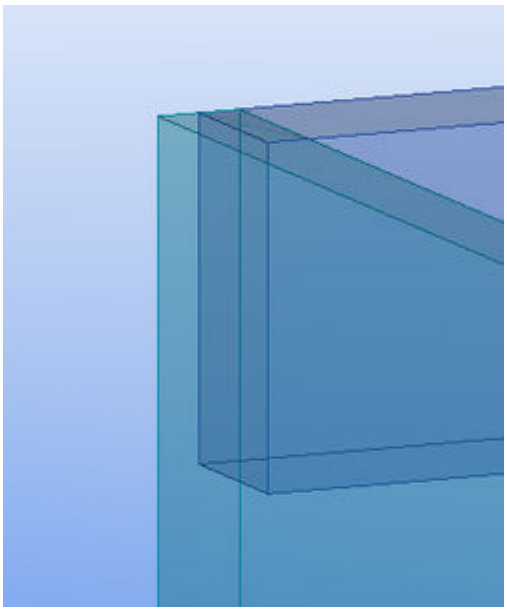
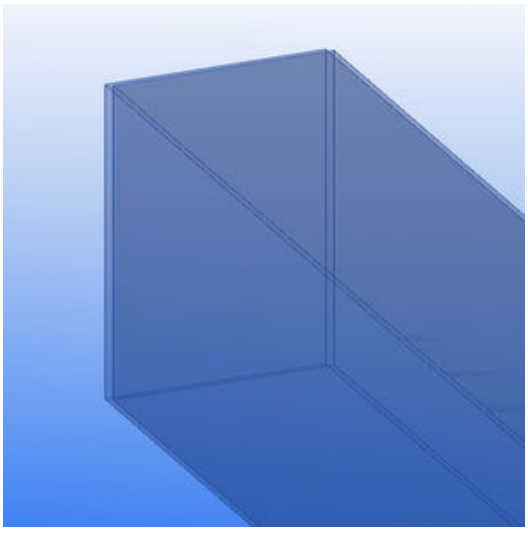
Clash Check Manager uses symbols to indicate the state of a clash.

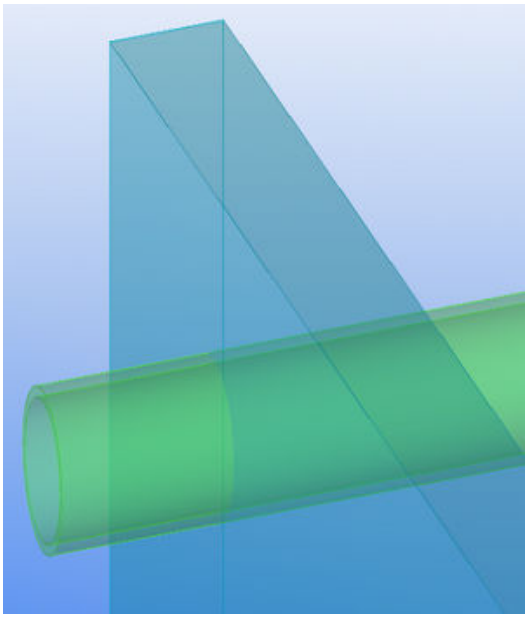
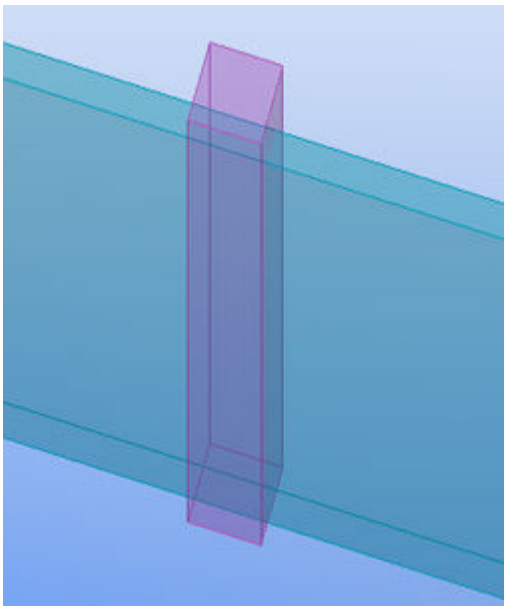
Flag symbol	State	Description
(none)	Active	The default state. The clash is not new, modified, resolved, or missing.
	New	All clashes are marked as new when they are found for the first time.
	Modified	If the object has been modified (for example, if the profile has been changed), the state changes to modified when you re-run the clash check. Only certain object properties affect this flag. To see which properties have an effect, right-click one of the column headings. Both visible and hidden properties affect the flag.
	Resolved	If the objects no longer clash, the state changes to resolved when you re-run the clash check.
	Missing	If one or both of the clashing objects have been removed from the model, the state changes to missing when you re-run the clash check.

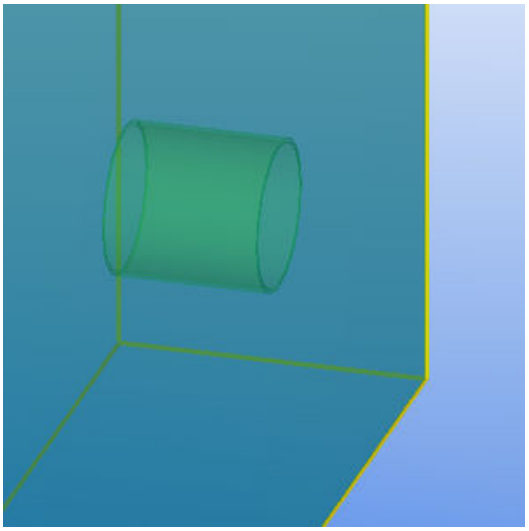
About clash types

Tekla Structures shows the type of each clash in the **Type** column in the **Clash Check Manager** dialog.

The following clash types can occur:

Type	Description	Example
Clash	The object partly overlaps with another object.	
Clash	Two identical objects overlap completely.	

Type	Description	Example
Clash	Objects intersect with each other in several locations.	
Clash	The object cuts through another object.	

Type	Description	Example
Is inside	The object is inside another object.	
Min distance	The objects are closer to each other than the defined minimum distance. The minimum distance is used between reference model objects and pour objects, and between these objects and native Tekla Structures parts. Minimum distance is not used between native Tekla Structures parts only.	

Open and save clash check sessions

Clash check sessions are saved as `.xml` files in the `..\TeklaStructuresModels\<model>\Clashes` folder. Tekla Structures creates the folder automatically when you open **Clash Check Manager** for the first time.

To open or save sessions in **Clash Check Manager**, do any of the following:

To	Do this
Save the current session	Click in the **Clash Check Manager** dialog.
Open a session	1. Click in the **Clash Check Manager** dialog. 2. In the **Open** dialog, select a session. 3. Click **Open**.
Start a new session	Click in the **Clash Check Manager** dialog. **Clash Check Manager** clears the list of clashes without running a clash check. If you already have clashes listed in the **Clash Check Manager** dialog, selecting or clearing the checkbox of the **Between reference models**, **Between reference models and components**, **Objects in reference models**, **Between parts**, or **Minimum distance** setting will start a new clash check session. Tekla Structures displays a dialog asking whether you want to save your current clash check session.
Save the current session with another name or in another location	1. Click the arrow next to the button in the **Clash Check Manager** dialog. 2. Click **Save as** . 3. In the **Save as** dialog, browse to the folder where you want to save the session. 4. In the **File name** box, enter a new name. 5. Click **Save**.
Save only selected clashes	1. In the **Clash Check Manager** dialog, select the clashes that you want to save.

To	Do this
	2. Click the arrow next to the button. 3. Click **Save selected** .

Modify, view, and print clash check results

You can modify, view, and print clash check results in the **Clash Check Manager** dialog.

To open the **Clash Check Manager** dialog, go to the **Manage** tab on the ribbon and click **Clash check** .

Modify clash check results

You can change the status and priority of clashes, group and ungroup clashes, and add comments for clashes in the **Clash Check Manager** dialog.

To	Do this
Change the status of a clash	1. Select the clash whose status you want to change. 2. Right-click on the **Status** column. 3. Select a suitable status.
Change the priority of a clash	1. Select the clash whose priority you want to change. 2. Right-click on the **Priority** column. 3. Select a suitable priority.
Group and ungroup clashes	You can combine several clashes into a group so that the clashes are treated as a single unit. 1. Select the clashes that you want to group. 2. Right-click and select **Group** --> **Group** . 3. If you want to add clashes to a group that already exists, select

To	Do this
	the clashes and the group, and then repeat step 2. Note that you cannot create nested groups of clashes. 4. If you want to ungroup clashes: a. Select the clash group to ungroup. b. Right-click and select **Group** --> **Ungroup** .
Add, modify, and remove comments for a clash	You can add comments for clashes and clash groups. For example, you might want to use comments as reminders to yourself and other users. You can modify the comments and remove them. 1. Select the clash or clash group for which you want to add a comment. 2. Right-click and select **Clash Information**. 3. Go to the **Comments** tab. 4. Click  to open the **Add comment** dialog. 5. Enter your comment in the **Comment** box. 6. Modify the author name and the date, if needed. 7. Click **OK**. To modify a comment, select the comment in the **Clash Information** dialog and click . To delete a comment, select the comment in the **Clash Information** dialog and click .

View clash check results

You can search for clashes, and view the history and other details of a clash in the **Clash Check Manager** dialog.

To	Do this
Search for clashes	Use the **Search** box to find clashes. The more search terms you enter, the more refined your search will be. For example, if you enter `column 8112`, only the clashes that match both of these terms are displayed. 1. Open the clash check session from which you want to find clashes. 2. In the **Search** box, enter the words to search for. The search results are displayed as you type. 3. To narrow down your search, enter more characters. 4. To display all clashes again, click ✕ next to the **Search** box.
To zoom in the active view	To zoom in the active view so that the selected objects are shown in the center of the view, double-click a clash in **Clash Check Manager**.
To zoom in on and highlight a clashing object	To zoom in on and highlight a clashing object when there is only one clash, right-click the clash in **Clash Check Manager** and select **Clash Information**.
View the history of a clash	You can view the history of a particular clash. For example, you can view when and by whom the clash was detected. 1. Select a clash or a clash group. 2. Right-click and select **Clash Information**. 3. Go to the **History** tab to view the history information.
View the details of a clash	You can view the details of a clash, such as the profile, material and class of the objects that collide. This can be

To	Do this
	useful especially when you view clash groups that contain more than two objects. 1. Select the clash or clash group whose details you want to view. 2. Right-click and select **Clash Information**. You can view the details of one clash or clash group at a time. If you select more than one clash or clash group, the **Clash Information** option is disabled.
Show or hide a column in **Clash Check Manager**	1. Right-click one of the column headings to open a menu. 2. Click any of the list items to show or hide them. A check mark in front of the item indicates that it is visible. Click the heading of the desired column to alternate between ascending and descending sort order.

Print clash check results

You can print the clash check results from the **Clash Check Manager** dialog. You can control the print settings in the same manner as in any standard Windows application.

To	Do this
Print the clash check results	1. Click to open the clash check session that you want to print. 2. Click **Print...**. 3. If needed, modify the print settings. 4. Click **Print...**.
Preview before printing	1. Click the down arrow next to and select **Print preview...**. 2. Click one of the page layout buttons to

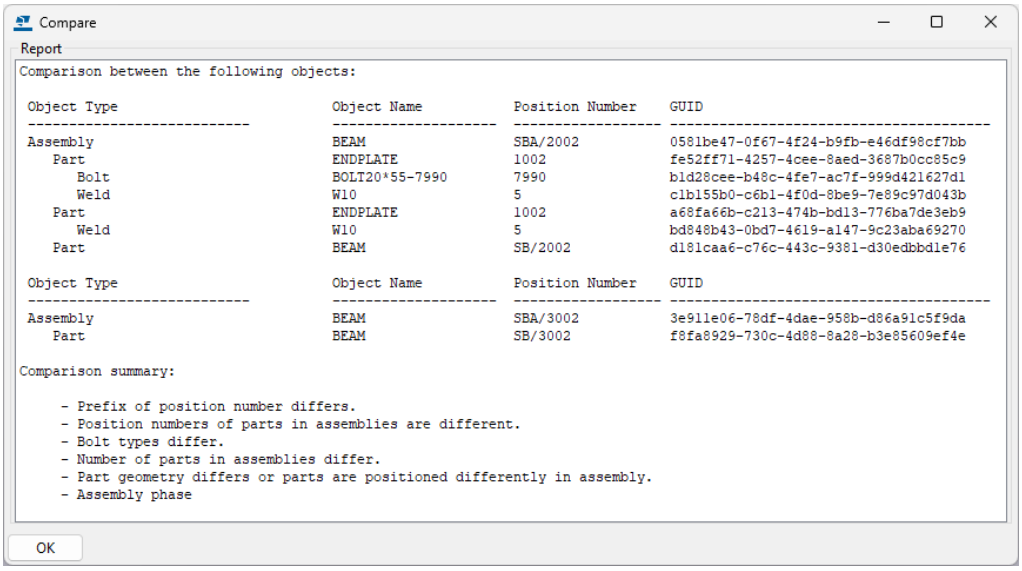
To	Do this
	select how many pages to view at a time. 3. Click  to print the current page.
Define the page setup	Click the arrow next to the  button and select **Page setup**.

4.9 Compare parts or assemblies

You can compare two selected parts or assemblies to check, for example, if they get the same position number.

1. Select the objects you want to compare.
   - To compare parts, select two parts in the model.
   - To compare assemblies, select a part in each assembly.
2. On the **Edit** tab, click **Compare**, and then select either **Compare parts** or **Compare assemblies**.

   Tekla Structures displays the results on the status bar and opens a report with more details. For example, you may get a message that the part geometry differs or parts are positioned differently in the assembly.



   If you select rows in the report, Tekla Structures highlights the corresponding objects in the model, and you can investigate the objects further. For example, you can select sub-assemblies in the previously

compared assemblies and use the **Compare assemblies** command again to compare the sub-assemblies.

If necessary, you can customize the comparison report templates to suit your needs. When Tekla Structures is installed in the `..\Program Files` folder, the templates are by default available in the folder `..\ProgramData\Trimble\Tekla Structures\<version>\Environments\common\system\`. Copy the `TS_Report_Assembly_Comparison.rpt` and `TS_Report_Part_Comparison.rpt` templates to your project or firm folder and then modify the copied templates as needed.

See also

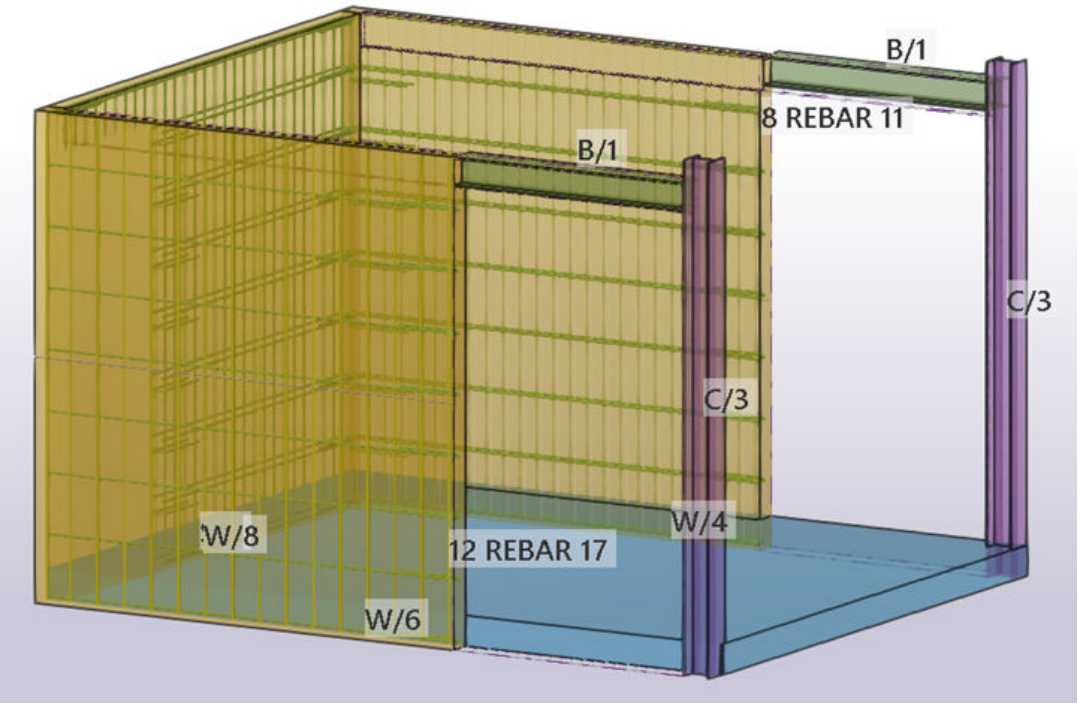
Check the model (page 697)

4.10 Show part and reinforcement information by using part and rebar labels

You can display selected part and reinforcement properties, user-defined attributes and template attributes in a model view by using *part labels* and *rebar labels*.

Part labels and rebar labels are textual descriptions that are displayed next to the object they represent. You can define what information to display in

the labels, such as the name, profile, and position number of the part, or the grade, diameter, and sequence number of the reinforcement object.



1. Double-click the view to open the **View properties** dialog.
2. Click **Display...** to open the **Display** dialog.
3. Go to the **Advanced** tab.
4. Next to **Part label** or **Rebar label**, in the **Properties** list, select a property.
5. Click **Add >** to add the property to the **Part label** or **Rebar label** list.

   The **Part label** or **Rebar label** checkbox is automatically selected when you add properties.
6. To remove a property from the **Part label** or **Rebar label** list, select a property and click **Remove**.

   The **Part label** or **Rebar label** checkbox is automatically cleared if you remove all the properties.
7. If needed, define which user-defined attribute or template attribute you want to display in part labels or rebar labels.
   a. Select **User-defined attribute** in the **Properties** list.
   b. Click **Add >**. The **Part label** or **Rebar label** dialog appears.

c. Enter the user-defined attribute name exactly as it appears in the `objects.inp` file, or the template attribute name. For example, `PRELIM_MARK`.

   For more information on the `objects.inp` file, see Define and update user-defined attributes (UDAs).

d. Click **OK**.

8. In the **Show for** lists, select for which objects you want to show the labels.
   - **All**: Labels are shown for all parts and/or reinforcement objects in the view.
   - **Selected**: Labels are only shown for the parts and/or reinforcement objects you select.
   - **Main part for selected**: Part labels are only shown for the main parts of the assemblies you select.
   - **Main part for all**: Part labels are shown for all main parts of all assemblies.

   Note that if you select the **Selected** or the **Main part for selected** option, you need to first apply the changes to the view when the view is selected. Then continue to select the objects for which you want to show the labels.
9. Click **Modify**.
10. If you want to change the color of the labels, use the advanced option `XS_VIEW_PART_LABEL_COLOR`.

4.11 Check solid errors

When working with Tekla Structures, you may encounter errors in solid objects in the model. You can use the **Solid checker** tool to check the model for solid errors. You can also view solid errors in a log file.

For example, when working with cast-in-place concrete parts, you should inspect the resulting pour objects regularly before starting the detailing process and before creating drawings and reports. Errors in solid pour objects can result in incorrect volumes, for example.

Check the model for solid errors using Solid checker

1. Go to the **File** menu, and click **Diagnose & repair** --> **Check solids**.

2. In the **Solid checker** dialog, click **Check solids**.

   Tekla Structures lists the errors, along with the objects causing them, in a table.

3. Select an error row in the table to highlight the error in the model.

   You can also highlight individual objects by clicking their identifiers in the table.

4. To zoom in on a problematic object in the active model view, select the **Zoom to selected** checkbox and click the object's identifier.

5. To fix an error in the model, you can try the following methods:

   - Modify touching part faces to either overlap fully, or not intersect at all. Moving the faces along their normal direction by a few millimeters (0.3 mm or more) is enough.
   - Move problematic parts slightly.
   - Model parts in a different order.

6. To see the updated situation, click **Check solids** again to refresh the list of errors.

   When an error has been resolved, its status changes to **Fixed** on the list.

7. If you do not want to fix an error, you can click **Ignore error** on the error row to hide it from the list.

   To see the ignored errors on the list again, select the **Show ignored errors** checkbox.

For more information about finding and fixing solid errors in pours, see Troubleshoot pours (page 488).

View solid errors in a log file

You can view solid errors in a log file. This may be needed, for example, if overlapping volumes and faces occur for parts and pour objects, and you need to analyze the error.

This is an older, alternative method to using the **Solid checker** tool.

1. On the **File** menu, click **Logs** --> **Session history log** .
2. Look for rows that start with the phrase `Solid error`.
3. Click the corresponding `Solid failure position` row to show the solid object error.

A diamond-shaped position locator is displayed in the model to point you to the error.



TIP When you click a `Solid error` row in the log file, hold down the **z** key to center the view to the error location.

4. Redraw the view to hide the position locator.

4.12 Diagnose and repair the model

Use the **Diagnose & repair** commands to check and to repair errors and inconsistencies in the structure of model objects and the library database (`xslib`). By diagnosing and repairing the model you can ensure, for example, that empty assemblies are removed and that unused points and attributes are deleted. Repairing the model also corrects invalid object relations and hierarchies. We recommend you to diagnose and to repair your model regularly to maintain the consistency and integrity of your model databases.

1. On the **File** menu, click **Diagnose & repair**.
2. Select the appropriate diagnose command.

   The errors and inconsistencies found in the model are listed in a report. Some of them are automatically corrected, some of them are warnings that you need to correct manually.

   If a profile, material grade, bolt element or assembly, or reinforcement seems to be missing from the corresponding catalog, your Tekla Structures environment or a catalog file may not be the same as the model's original one.

   If there are solid errors (page 746) in the model objects, these errors are listed in the report. The report is updated every time you run the **Diagnose model** command. (Solid errors are also listed in the session history log file `TeklaStructures_<user>.log`.)

   The following table lists the most common errors and inconsistencies found when diagnosing the model.

Diagnosis result	Description	Action needed
Empty assembly	The assembly does not contain any objects.	a. On the **File** menu, click **Diagnose & repair**.

Diagnosis result	Description	Action needed
		b. Under **Model**, click **Repair model** to delete the assembly.
Missing assembly	A part is not included in any assembly.	a. On the **File** menu, click **Diagnose & repair**. b. Under **Model**, click **Repair model** to create an assembly and to move the part to it.
Illegal profile	An unknown profile has been found.	a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original `profdb.bin` and `profitab.inp` files and save them in the model folder. c. Reopen the model.
Illegal material	An unknown material grade has been found.	a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original `matdb.bin` file and save it in the model folder. c. Reopen the model.
Illegal bolt	An unknown bolt element or bolt assembly has been found.	a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original `screwdb.db` and `assdb.db` files and save them in the model folder. c. Reopen the model.
Illegal reinforcing bar size or grade Illegal reinforcement mesh	Reinforcement with invalid properties has been found.	a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original `rebar_database.inp` and `mesh_database.inp` files and save them in the model folder. c. Reopen the model.
Invalid rebar geometry	Reinforcement with undefined	See Check the validity of reinforcement geometry (page 594).

Diagnosis result	Description	Action needed
	geometry has been found.	
Solid creation failed	A model object with invalid geometry has been found.	a. Ensure that you are using the correct Tekla Structures environment and profile catalog files. b. Try the following: • List and locate the solid errors. (page 746) • If there are cuts and/or fittings in the object, slightly adjust one of them or the object. • Check the pour geometry. (page 488) • Remodel the object.

If the model does not contain any errors or inconsistencies, a message is displayed on the status bar.

See also

Check the model (page 697)

4.13 Find distant objects

When the work area is huge, the model may contain some distant objects that are not easy to find. Use the **Find distant objects** command to find these objects.

1. On the **File** menu, click **Diagnose & repair**.
2. Under **Utilities**, click **Find distant objects**.

   Tekla Structures displays a list of object GUIDs. At the end of the list, Tekla Structures displays additional six objects that have the biggest and smallest x, y, or z coordinates.

   ```
   Min x: Guid: e32a7a28-40db-4597-b160-031d15c1944a
   Max x: Guid: 985a39e2-8097-4a9a-8706-9651d08f61c6
   Min y: Guid: 8ccb2748-cfe8-4a97-be80-abf453008567
   Max y: Guid: 08c8e02d-6a79-4b7e-be70-5370359a1ff5
   Min z: Guid: 95eec6e2-d22b-4ae8-8c31-ee8009c028a6
   Max z: Guid: f791c3d0-de62-4ced-8d79-03668296f862
   ```

3. Select an object in the list.

4. Right-click and select a command.
   You can, for example, inquire or delete the object.

See also

Check the model (page 697)

5 Number the model

This section explains how to change numbering settings and apply numbering in Tekla Structures.

Click the links below to find out more:

What is numbering and how to plan it (page 752)

Adjust the numbering settings (page 760)

Number parts and other model objects (page 767)

Preview the numbering results (page 771)

Check the numbering (page 777)

View the numbering history (page 780)

Repair numbering errors (page 781)

Change existing numbers (page 781)

Clear existing numbers (page 782)

Renumber the model (page 783)

Control numbers (page 783)

Number parts by design group (page 790)

Numbering examples (page 793)

Tips for numbering (page 798)

5.1 What is numbering and how to plan it

Before you can create fabrication drawings or accurate reports of parts, assemblies, or cast units, you need to number these objects first. You do not need to number the model before you create general arrangement drawings.

Numbering is the key to the production output, for example, drawings, reports, and NC files. Numbers are also needed when you export models.

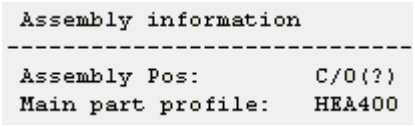
Part numbers are vital in the fabrication, shipping, and erection stages of construction. Tekla Structures assigns a mark to each part and assembly/cast unit in a model. The mark includes part or assembly prefix and position number, and other elements, such as profile or material grade. It is useful to identify the parts with numbers to see which parts are similar and which different. Identical parts within a numbering series have the same number, which makes the planning of the production easier.

We recommend that you plan the numbering in an early phase of the project. If other users are using the same model, it is even more important to make a numbering plan that everybody in the project follows. You should have the numbering ready before you create the first drawings and reports.

When planning the numbering, it can be useful to number the model in phases, for example first floor of the building first, then the second, and so on.

Give start numbers in wide ranges so that you do not run out of numbers within a numbering series, and that any numbering series does not overlap with another. For example, start the first floor with the start number 1000 and the second with the start number 2000.

If the numbering of a part or assembly is not up to date, a question mark (?) is displayed in the part label and in the **Inquire object** dialog, for example:



See also

Numbering series (page 753)

Identical parts (page 757)

Identical reinforcement (page 757)

Define what affects numbering (page 758)

User-defined attributes in numbering (page 759)

Inquire object properties (page 721)

Numbering series

Use numbering series to divide steel parts, reinforcement, assemblies, and cast units into groups. For example, you can allocate a separate numbering series to each phase or part type. Using separate numbering series for different objects and part types speeds up the numbering operation.

The name of a numbering series consists of a *prefix* and a *start number*. You do not always have to define a prefix (for example, you may want to omit the part prefix for minor parts).

When you run numbering, Tekla Structures compares model objects that belong to the same series with each other. All identical objects within tolerance and in the same numbering series are given the same position number. The oldest object in the series is considered as the origin object for comparison.

NOTE When the advanced option `XS_INHERIT_CONCRETE_PART_NUMBERING_SETTINGS_FROM_CAST_UNIT` is set to `TRUE`, which is the default value, concrete parts are numbered according to the cast unit numbering settings. For example, if the cast unit prefix is **C** and the start number is **1**, concrete parts will get the part prefix **Concrete_C-1**.

This applies also to concrete components whose part position prefix is **Concrete** and start number is **1**.

Example

For example, if you define a numbering series with the prefix P and start number 1001, Tekla Structures numbers that series P1001, P1002, P1003, ...

See also

Plan your numbering series (page 754)

Assign a numbering series to a part (page 755)

Assign a numbering series to an assembly (page 755)

Overlapping numbering series (page 756)

Plan your numbering series

Before you start modeling, it is a good idea to plan the numbering prefixes and start numbers you will use for the entire project. Careful planning prevents numbering conflicts.

To save time, include the numbering series in the default part properties for each type of part before you start modeling.

You may want to omit the part prefix for minor parts, such as plates. If you do this, ensure that you assign a **Start number** for that numbering series so that it will not overlap other parts.

Example

One way to plan the numbering series is to create a table:

Part type	Part Prefix	Part Start number	Assembly Prefix	Assembly Start number
Beam	PB	1	AB	1
Vertical brace	PVB	1	AVB	1

Part type	Part Prefix	Part Start number	Assembly Prefix	Assembly Start number
Horizontal brace	PHB	1	AHB	1
Rafter	PR	1	AR	1
Purlin	PP	1	AP	1
Column	PC	1	AC	1
Plate		1001	A	1

See also

Numbering series (page 753)

Overlapping numbering series (page 756)

Assign a numbering series to a part

1. Double-click a part to open the part properties in the property pane.
2. Go to the **Numbering series** section.

   If you are assigning a cast unit numbering series to a concrete part, go to the **Cast unit** section.
3. Define a part prefix and a start number.
4. Click **Modify**.

See also

Assign a numbering series to an assembly (page 755)

Numbering series (page 753)

Assign a numbering series to an assembly

To assign a numbering series to an assembly:

To	Do this
To assign a numbering series to an assembly according to its main part	1. Check what is the main part of an assembly. a. On the ribbon, click the down arrow next to , and then select **Assembly objects**.

To	Do this
	b. Select the assembly. Tekla Structures highlights the main part in orange. c. Press **Esc**. 2. Ensure that the **Select objects in assemblies** selection switch is active. 3. Double-click the assembly main part to open the part properties in the property pane. 4. Under **Numbering series**, define an assembly prefix and a start number. 5. Click **Modify**.
To assign a numbering series to an assembly by using the assembly properties	1. Ensure that the **Select assemblies** selection switch is active. 2. Double-click an assembly to open the assembly properties in the property pane. 3. In the **Assembly numbering** boxes, define an assembly prefix and a start number. 4. Click **Modify**.

See also

Assign a numbering series to a part (page 755)

Numbering series (page 753)

Overlapping numbering series

When you plan numbering, ensure that you reserve enough numbers for each series. If a series overlaps another, Tekla Structures numbers only one of the objects that would have overlapping numbers and leaves the other object unnumbered.

Tekla Structures warns you about series overlaps. Use the numbering preview or view the numbering history log to check which numbers overlap, and then adjust the numbering prefixes and start numbers so that the series will not overlap anymore.

See also

Numbering series (page 753)

Preview the numbering results (page 771)

View the numbering history (page 780)

Identical parts

Tekla Structures gives parts the same number if the parts are identical **in the fabrication or casting**. If a part is deformed after fabrication or casting (for example if the part is cambered, shortened, or warped), the final geometry on site and in the model may be different.

Tekla Structures treats parts as identical and gives them the same number if the following basic part properties are the same:

- Part geometry
- Casting direction
- Numbering series
- Profile
- Material
- Finish
- Shortening

You can set the degree of tolerance for part geometry in the **Numbering Setup** dialog. If the geometry of parts differs within this degree of tolerance, Tekla Structures treats the parts as identical for numbering purposes.

Class does not affect numbering. Tekla Structures gives the same number to identical parts that belong to different classes.

Pop marks and contour marking in parts affect numbering if you have enabled them in the NC file export settings.

See also

Casting direction (page 458)

Define what affects numbering (page 758)

User-defined attributes in numbering (page 759)

Identical reinforcement

Tekla Structures treats reinforcing bars and other reinforcement objects as identical, and gives them the same number, if the following properties are the same:

- Bar geometry
- Numbering series
- Size
- Grade
- Bending radius

To specify how the geometry of reinforcing bars is taken into account in numbering and which type of bar geometry (modeled, rationalized, or fabrication) is used when bars are compared with each other, use the advanced option `XS_REBAR_GEOMETRY_TYPE_IN_NUMBERING`.

You can set the degree of tolerance for bar geometry in the **Numbering Setup** dialog. If the bar geometry differs within this degree of tolerance, Tekla Structures treats the bars as identical for numbering purposes.

Class does not affect numbering. Tekla Structures gives the same number to identical reinforcing bars that belong to different classes.

The modeling direction of tapered reinforcing bar groups affects numbering. This means that identical bar groups with different modeling directions receive different numbers.

The bar length rounding, step tapering, and some other rebar set settings affect bar geometry and thus the numbering of rebar set bars. The rounding settings that are defined in the `rebar_config.inp` file for drawings and reports do not affect numbering.

The advanced option `XS_CONSIDER_REBAR_NAME_IN_NUMBERING` controls whether the names of reinforcement objects are taken into account in numbering. By default `XS_CONSIDER_REBAR_NAME_IN_NUMBERING` is set to `FALSE`, and reinforcement object names do not affect numbering.

See also

Number reinforcement (page 768)

Define what affects numbering

To define which properties affect numbering in your model, modify the settings in the **Numbering Setup** dialog.

You can have Tekla Structures compare the following properties:

- Holes (if created with the **Bolt** command)

- Part name
- Beam orientation
- Column orientation
- Assembly name
- Assembly phase (XS_ENABLE_PHASE_OPTION_IN_NUMBERING set to `TRUE`)
- Reinforcement
- Embedded objects (affect only cast units)
- Surface treatment (affects only assemblies)
- Welds (affect only assemblies)

If these properties differ, Tekla Structures treats objects as different, and therefore numbers them differently.

For example, if two otherwise identical parts have different names and you select the **Part name** checkbox, Tekla Structures gives the parts different numbers.

By default, a part retains its number, as long as only one part has that particular number, regardless of the settings in the **Numbering Setup** dialog.

See also

Adjust the numbering settings (page 760)

What is numbering and how to plan it (page 752)

Identical parts (page 757)

Identical reinforcement (page 757)

User-defined attributes in numbering (page 759)

General numbering settings (page 1057)

User-defined attributes in numbering

You can set in the `objects.inp` file whether a user-defined attribute affects numbering or not. Tekla Structures treats parts and reinforcing bars as different, and therefore numbers them differently, if the values of the user-defined attribute differ.

NOTE Only user-defined attributes of parts and reinforcement can affect numbering. User-defined attributes of other objects, such as phases, projects and drawings do not affect numbering.

If you want Tekla Structures to consider a user-defined attribute when numbering, set the `special_flag` option of the attribute to `yes` in the `Part attributes` section of `objects.inp`. For reinforcement, you need to set `special_flag` to `yes` **also** in the `Reinforcing bar attributes` section.

Tekla Structures assigns different numbers to parts or reinforcement that are otherwise identical but have different values for this user-defined attribute.

If you want Tekla Structures to ignore a user-defined attribute when numbering, set the `special_flag` option to `no` in `objects.inp`.

See also

Examples of user-defined attributes (UDAs) for parts (page 356)

Select and change the profile or material of a part (page 332)

Define what affects numbering (page 758)

5.2 Adjust the numbering settings

You can adjust the numbering settings to better suit your needs. This should be done early in the project, before creating any drawings or reports. Do not change your numbering conventions in the middle of a project.

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog.
2. Modify the settings (page 1057) if necessary.

   For example, you can define which part properties affect numbering in your model. Using the default settings is effective in most cases.
3. Click **Apply** or **OK**.

NOTE Always check and repair numbering after you have changed the numbering settings.

See also

Define what affects numbering (page 758)

Numbering settings during a project (page 798)

Repair numbering errors (page 781)

Family numbering

With family numbering you can group objects within the same numbering series into different “families”. This can be used, for example, to find similar precast cast units that can be cast in the same bed.

Family numbering is not available for cast-in-place cast units.

When you use family numbering, the cast unit position numbers consist of a *family number* and a *qualifier*. For example:



1. Family number
2. Qualifier

Assemblies and precast cast units that match the comparison criteria you define in the **Numbering Setup** dialog get the same family number. However, if they have the same family number but different part geometry or materials, they get unique qualifier numbers.

See also

Numbering series (page 753)

Assign family numbers (page 761)

Example: Use family numbers (page 794)

Assign family numbers

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog.
2. Go to the **Family numbering** tab.
3. Define which numbering series to assign family numbers to.
   a. Click **Add series** to open the **Add series** dialog.

      Tekla Structures displays all the assembly and cast unit numbering series in the model.

   b. Select a numbering series from the list, and then click **Add**.

      The numbering series appears in the family numbering list.
4. Under **Compare**, select the properties that need to be identical for the members of the same family.

   Define comparison criteria for each numbering series separately.

   Select at least one checkbox, but not all of them. If you select all the checkboxes, the family number will be the same as the normal assembly position, and the qualifier number will be 1 for all. If you do not select any checkboxes, only one family number per series is assigned.
5. Click **Apply**.

   Tekla Structures stores the settings in the numbering database file (`<model_name>.db2`) in the current model folder the next time you save the model.

6. If you are assigning family numbers to parts that have already been numbered, clear the existing numbers.
7. Update numbering in the model.

   Tekla Structures assigns a family number to all objects in the numbering series.

See also

Family numbering (page 760)

Clear existing numbers (page 782)

Reinforcement sequence numbering

With reinforcement sequence numbering you can assign running numbers to reinforcement within each cast unit or pour unit. The sequence numbers usually start from 1. With sequence numbers you can indicate the installation order of the bars, for example.

Reinforcement sequence number is an optional identifier that can be used in addition to the regular reinforcement position number (REBAR_POS), which remains as the basis for identifying identical reinforcement objects and cast units in the model. Reinforcement sequence numbering relies on position numbers to assign the same sequence number to identical bars within a cast unit or pour.

Either or both of these two numbers, model-specific reinforcement position numbers and cast unit or pour-specific reinforcement sequence numbers, can be shown in drawings and reports, and used in exports.

Reinforcement sequence numbers can be assigned to all reinforcement object types: single reinforcing bars, reinforcing bar groups, rebar sets, reinforcement meshes, and reinforcement strands. Reinforcement objects added to a rebar assembly are numbered within that assembly. However, reinforcement objects that are added to cast units as steel sub-assemblies, such as embeds, are excluded from sequence numbering. Also loose reinforcement objects that are detached from parts are excluded.

If you have defined a pour by setting the same pour number for several pour objects, they are treated as one entity in reinforcement sequence numbering. Pour objects without a pour number are treated individually, and they are not combined with other pour objects that do not have a pour number.

NOTE Sequence numbers are always assigned to all reinforcement objects within a cast unit or a pour. When using the **Number series of selected objects** command, not all reinforcement objects may be included in the selection. In such cases, the numbering process automatically expands to include all reinforcement objects within the cast unit or pour.

To use reinforcement sequence numbers in existing drawing and report templates created with Tekla Structures versions prior to 2025, you should modify the templates. This can be laborious if there are many such templates. Alternatively, you can set the following advanced options to show sequence numbers instead of position numbers for reinforcement in outputs:

- XS_REBAR_POSITION_NUMBER_FORMAT_STRING

  For example, set `XS_REBAR_POSITION_NUMBER_FORMAT_STRING` to %REBAR_SEQ_POS%.
- XS_REBARSET_TAPERED_GROUP_POSITION_NUMBER_FORMAT_STRING
- XS_REBARSET_TAPERED_REBAR_POSITION_NUMBER_FORMAT_STRING

NOTE
- If you set the above advanced options to show reinforcement sequence numbers in outputs, the original reinforcement position numbers (`REBAR_POS`) cannot be shown.
- These advanced options only control how the reinforcement position numbers are shown in outputs. In the model database, reinforcement numbers are still stored as position numbers.

Assign reinforcement sequence numbers

1. To define the default separator for the reinforcement sequence numbers, do the following:
   a. Go to **File menu** --> **Settings** --> **Options** --> **Numbering**.
   b. In the **Rebar sequence number separator** list, select dot (.), comma (,), slash (/), or hyphen (-) to be the default separator.

      You can also select the empty option, and then later include the separator in the numbering prefix and, for example, use a different separator for each object group.
   c. Click **OK** to save the setting and to close the **Options** dialog.
2. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog.
3. Go to the **Reinforcement sequence numbering** tab.
4. In the **Create reinforcement sequence numbers** list, select **Yes**.
5. Under **Use reinforcement sequence numbering for object groups**, define which object groups to assign reinforcement sequence numbers to.

   You can start either by modifying the default object group with the **All rebars** selection filter and the **standard** settings, or you can create new object groups.
   a. To create a new object group, click **Add row**.

b. Select an object group row.

c. On the selected row, in the **Filter** column, select a selection filter.

   To use the same settings for all reinforcement objects, you can use the predefined **All rebars** option.

   To create a new filter (page 172) or to modify an existing filter (`.SObjGrp`), click  on the **Selecting** toolbar and use the **Object group - Selection filter** dialog.

   The following content options can be used in the filter:

   - Cast units, rebar assemblies, or pours (**Object type** = **Assembly** or **Pour unit**)

     All reinforcement objects within a cast unit, rebar assembly, or a pour will get a sequence number.

   - Parts (**Object type** = **Part**)

     All reinforcement objects attached to a part will get a sequence number.

   - Reinforcement objects (**Object type** = **Reinforcing bar**)

     The filtered reinforcement objects will get a sequence number.

   - Combinations of the above three.

d. On the selected row, in the **Settings** column, select the settings that you want to use for the object group.

   To define or modify the numbering settings for the selected row, click **Edit settings** and follow the steps below in the "Modify reinforcement sequence numbering settings" section.

6. To add more rows for object group filters, repeat step 5.

   The filters work in the order they are in the list, starting from top. For each reinforcement object, the first matching filter will be used.

7. To arrange the rows in a different order, select a row and use the **Move up** and **Move down** buttons.

8. To define whether the reinforcement sequence numbers are applied within cast units or pour units, select an option in the **Number rebars in cast in place using** list.

   The list is only available when pour management (page 468) is enabled for the model (the advanced option `XS_ENABLE_POUR_MANAGEMENT` is set to `TRUE`).

   If you select the **Pour units** option, Tekla Structures automatically calculates the pour units (page 477) before numbering.

9. Modify the other numbering settings (page 1057) as required.

10. Click **OK** to save the numbering settings and to close the **Numbering Setup** dialog.

    Tekla Structures stores the settings in the numbering database file (`<model_name>.db2`) in the current model folder the next time you save the model.

11. If you want to assign sequence numbers to reinforcement objects that have already been numbered, or that have not been modified since the last numbering, clear the existing numbers (page 782).

    Use the **Change number** --> **Clear reinforcing bar numbers** command on the **Drawings & reports** ribbon tab.

12. Update numbering in the model.

    Tekla Structures assigns a reinforcement sequence number to new and modified reinforcement objects the next time you use a numbering command, such as **Number modified objects** or **Number series of selected objects**.

Modify reinforcement sequence numbering settings

In the **Reinforcement sequence numbering settings** dialog, you can define and modify the reinforcement sequence numbering settings of object groups.

1. In the **Numbering Setup** dialog, go to the **Reinforcement sequence numbering** tab.
2. Under **Use reinforcement sequence numbering for object groups**, select an object group row.
3. Click **Edit settings**.

   The **Reinforcement sequence numbering settings** dialog appears.
4. Select how you want to number new or modified reinforcement objects.

   The options are **Compare to old** and **Take new number**. You can also select to re-use old numbers.
5. Click the **+** button to add the needed elements to the prefix.

   In addition to adding elements, you can type text in the **Prefix** box.

   To remove an added element from the prefix, select the element in the **Prefix** box, right-click, and select **Delete**.
6. In the **Start with** list, select or enter the start character. The options are **Number** and **Letter**.

   If specific sequence numbering start characters are needed for different reinforcement objects in individual cast units or pour units, you can define the start characters on the **Reinforcement** tab in the cast unit or pour unit user-defined attributes. There are separate attributes for reinforcing bars, reinforcement meshes, and strands:

   - **Rebar start number or letter** (`RBR_SEQ_START_NO_R`)

- **Mesh start number or letter** (RBR_SEQ_START_NO_M)
- **Strand start number or letter** (RBR_SEQ_START_NO_S)

7. In the **Sort by** list, select the parameters that you want to use in sorting.

   For example, for reinforcement in slabs, you might want to sort by the center of gravity and indicate the installation order of bars with the sequence numbers: bottom bars with the lowest center of gravity first, then the edge bars, and finally the top bars.

   Note that you can search for parameters by starting typing a parameter name.

   The following table lists the available parameters and describes how they are used in sorting:

Parameter	Description
Bending shape	Bars are arranged by the number of bends, with the smallest first. The fabrication geometry of bars is used. When the advanced option XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION is set to FALSE, hooks are excluded when calculating the number of bends. When there are several shapes with an equal number of bends, they are arranged by the SHAPE attribute in alphabetical order.
Diameter	The nominal diameter of the bar is compared in sorting.
Length	The length of the bar is compared in sorting.
Center of gravity	Numbering is done first using the global z coordinate of the bar's center of gravity, then x and y, starting from the lowest value.
Layer	Numbering is done in three groups, in the following order: bottom bars, side bars, and top bars. Within each group, numbering is done in the order of layer numbers; bottom and side bars starting from the smallest layer number (outmost bars), top bars starting from the largest layer number (innermost bars). If the sorting order is changed from ascending to descending, the change is done for the whole outcome, not individually within bottom, side, and top groups. Reinforcement objects that do not have a layer prefix will be numbered last.

Parameter	Description
UDAs	User-defined attributes whose type is 'integer' can be used as sorting parameters. In the list of UDAs, only the ones with the type 'integer' are displayed.

8. To define in which order the selected parameters are used in sorting, drag the parameter rows up or down, so that they are in the correct order.
9. To define the sorting order for the values of a parameter, click the sorting arrow button next to the parameter name.

   A down arrow  indicates that the parameter values are sorted in descending order, and an up arrow  denotes ascending order.
10. To save the settings, enter a file name in the box at the top of the **Reinforcement sequence numbering settings** dialog, and then click **Save**.

    The settings are saved as an `.snj` file in the `\attributes` folder in the current model folder.
11. Click **X** in the upper-right corner to close the **Reinforcement sequence numbering settings** dialog.

5.3 Number parts and other model objects

Use the **Number modified objects** command to number all parts, assemblies, cast units, and reinforcement that have been created or modified since the last numbering. If this is the first time you run numbering for this model, all objects in the model are new and will therefore be numbered.

To number new and modified objects:

1. On the **Drawings & reports** tab, click **Perform numbering** --> **Number modified objects** .

   Tekla Structures opens the **Numbering preview** dialog where you can check the numbering results (page 771) before assigning position numbers to the objects.
2. If you are **not** happy with the proposed position numbers, click **Reject numbers**. Then adjust the model objects and the numbering settings as needed, and start the command again.
3. When you are happy with the proposed position numbers, click **Assign numbers**.

Tekla Structures numbers the objects according to the numbering settings (page 1057).

See also

Number a series of parts or other model objects (page 768)

Number reinforcement (page 768)

Number assemblies, cast units, and rebar assemblies (page 769)

Number welds (page 770)

Save preliminary numbers (page 771)

Number parts by design group (page 790)

Number a series of parts or other model objects

Use the **Number series of selected objects** command to only number objects that have a certain prefix and start number. This enables you to limit the numbering to certain series of objects only, which can be useful in large models.

Before you start, we recommend that you plan the numbering series carefully, and split the model into smaller numbering series, for example by area or phase, and object type.

1. Select the objects that have the desired prefix and start number.

   Only objects that have the same prefix and start number as the selected object will be numbered.

2. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.

   Tekla Structures opens the **Numbering preview** dialog where you can check the numbering results (page 771) before assigning position numbers to the objects.

3. If you are **not** happy with the proposed position numbers, click **Reject numbers**. Then adjust the model objects and the numbering settings as needed, select the required objects, and start the command again.

4. When you are happy with the proposed position numbers, click **Assign numbers**.

Tekla Structures numbers all new and modified objects in the specified numbering series.

See also

Example: Number selected part types (page 795)

Example: Number parts in selected phases (page 796)

Number reinforcement

To number reinforcement, use the same numbering commands as for numbering parts.

To specify how the geometry of reinforcing bars is taken into account in numbering and which type of bar geometry is used when bars are compared with each other, use the advanced option XS_REBAR_GEOMETRY_TYPE_IN_NUMBERING.

To use cast unit or pour unit specific reinforcement sequence numbers (page 762) for reinforcement objects, adjust the settings on the **Reinforcement sequence numbering** tab in the **Numbering Setup** dialog.

Before numbering rebar assemblies, you may want to modify the sort order settings similarly as when numbering assemblies or cast units (page 769).

Note that reinforcement may affect the numbering of cast units. To force Tekla Structures to give otherwise identical cast units different numbers if they have different reinforcement, select the **Reinforcing bars** checkbox in the **Numbering Setup** dialog.

Part numbering and cast unit numbering do not affect the numbering of reinforcement.

See also

Number parts and other model objects (page 767)

Identical reinforcement (page 757)

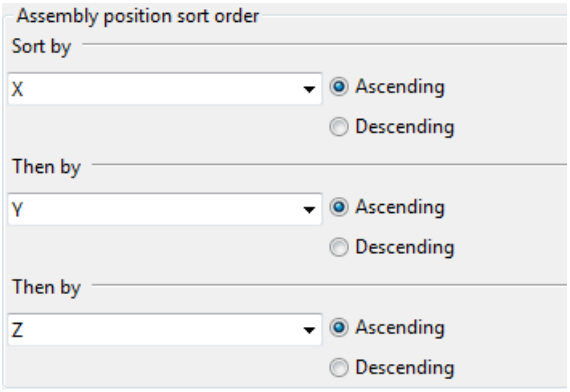
Define what affects numbering (page 758)

User-defined attributes in numbering (page 759)

Number assemblies, cast units, and rebar assemblies

To number assemblies, cast units, and rebar assemblies, use the same numbering commands as for numbering parts. Before numbering, you can modify the sort order, which defines how assemblies, cast units, and rebar assemblies are given their position numbers. Sorting does not affect part position.

1. If needed, modify the sort order of assemblies and cast units.
   a. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog.
   b. Modify the sort order by selecting options from the **Sort by** and **Then by** lists.



The default sort order is XYZ. You have the following options:

- The x, y or z coordinates of the main part of the assembly or cast unit, or the main reinforcement object of the rebar assembly.

  The sorting is based on the center of gravity (COG) location of the assembly or cast unit. Tekla Structures finds the center of gravity for each assembly and cast unit and compares them in the order you defined.

- The user-defined attribute of an assembly or the main part

  If your sorting is based on user-defined attributes, Tekla Structures displays a list that includes all the available user-defined attributes.

c. Click **Apply** or **OK** to save the changes.

2. If needed, modify the other numbering settings (page 1057).
3. On the **Drawings & reports** tab, click **Perform numbering** --> **Number modified objects** to number the model.

NOTE If you add new parts in the model, parts that have already been numbered are **not** renumbered to suit the sorting order. In this case you should check and repair the numbering of those parts.

See also

Repair numbering errors (page 781)

Number welds

Use the **Number welds** command to assign numbers to welds. Weld numbers are displayed in drawings and reports.

1. On the **Drawings & reports** tab, click **Perform numbering** --> **Number welds** to open the **Weld Numbering** dialog.

2. If needed, modify the weld numbering settings (page 1059).

   For example, you can define whether to assign numbers for **All welds** or **Selected welds**.
3. If you selected to assign numbers to certain welds only, select the welds.
4. Click **Assign numbers** to start numbering welds.

See also

Number parts and other model objects (page 767)

Save preliminary numbers

Preliminary mark is a user-defined attribute that defines the part position number. You can save the current part position numbers as preliminary numbers for selected parts. The previous preliminary numbers are overridden.

1. Select the parts.
2. On the **Drawings & reports** tab, click **Numbering settings** --> **Save preliminary numbers** .

The preliminary numbers appear on the **Parameters** tab in the parts' user-defined attributes.

See also

Number parts and other model objects (page 767)

5.4 Preview the numbering results

After you have started a numbering command, but before you accept the numbering results, you can preview the position numbers that Tekla Structures would assign to objects. You can go through the proposed numbers and the related objects in the model, and you can work on the numbering overlaps, for example. You also have the option to cancel the numbering and reject the proposed numbers, if needed, and go back to adjust the numbering settings and the model.

Previewing the numbering results is useful when you are creating an intermediate submittal, for example.

The **Numbering preview** dialog opens when you start a numbering command, **Number modified objects** or **Number series of selected objects** from the ribbon, or either of the **Diagnose & repair** --> **Numbering** commands from the **File** menu.



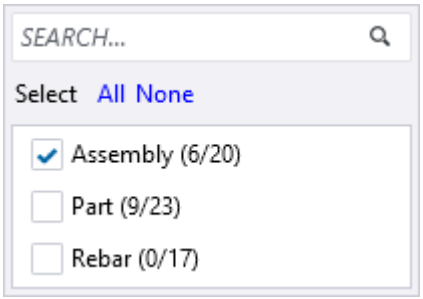
In the **Numbering preview** dialog, you can use the upper, main table to display the object properties that affect numbering. You can select rows in the main table, and then, in the **Individual objects** table, review the related objects individually. This is useful when the objects have different values for properties that do not affect numbering, such as the **Comment** user-defined attribute. The **Varies** value is shown for these object properties in the main table, but in the **Individual objects** table you can see the values for each individual object.

Note that the **Numbering preview** dialog is not shown if you use a standard-part model (page 799), or if **Synchronize with master model (save-numbering-save)** is selected in the **Numbering Setup** dialog.

The objects can have the following statuses in numbering, indicated with different colors in the **Numbering preview** dialog:

Status	Color	Description
Overlapping	Red	Objects from different numbering series would get the same position number. This is not allowed, and objects with overlapping numbers will be excluded when assigning the numbers.
Changes number	Yellow	Object has been modified and will get a new number.
New	Green	Object has not been numbered yet, or the prefix or start number has been changed.
Keeps number	Gray	Object has been modified but it can keep its previous number.

This is how you work with the numbering preview:

To	Do this
Change the sorting order of rows by property values in one or more columns	To change the sorting order of rows using one column, click the column header. The arrow symbol in the column header indicates if the sorting order is ascending or descending . Columns without an arrow symbol (with when you move the mouse pointer over a column header) are not sorted. The "not sorted" state is useful when you have several sort columns and you want to disable only one of them, without starting all over again. To sort rows using several columns: 1. Sort by one column. 2. Hold down **Shift** and then sort by the other columns.
Filter (show or hide) rows by property values in a column	1. Click in the column header. 2. In the list that appears, select the values you want to filter by. For example:  You can search for values. You can also click **All** or **None** to first select all or none of the available values, and then remove or add some values. The numbers next to each value indicate how many times the value is found with the current filter and in total, for example, (6/20). The symbol in a column header indicates that a filter is active in the column.

To	Do this
	To clear all filters in all columns, click the  **Clear all filters** button next to the table where you want to clear the filters.
Select related rows and objects	• To automatically select those objects in the model that are related to the selected rows in the **Numbering preview** dialog, click the  button. When the button is blue  and you select rows, Tekla Structures automatically selects the corresponding objects in the model. • To select rows that are related to certain objects in the model, first select the objects in the model, and then click the  button.
Select and review identical objects individually	1. In the main table of the **Numbering preview** dialog, select a row where **Quantity** is more than 1. 2. If the **Individual objects** table is not open under the main table, open it by clicking the **Individual objects** text or the right arrow in front of the text. 3. In the **Individual objects** table, select one or more rows. When this button  is blue and you select rows, Tekla Structures automatically selects the corresponding objects in the model. You can now, for example, compare or modify the objects as needed.
Add a new column	1. Click the  button next to the table where you want to add the column. 2. Depending on the object type whose values you want to show in the new column, select **Add column for assembly**, **Add column for part**, or **Add column for rebar**. 3. Search for and select an attribute in the list. The list shows the template attributes and user-defined attributes that are available for the selected object type.

To	Do this
	4. If needed, enter a customized name for the column. By default, the name of the selected attribute is used automatically. 5. Click **OK** to add the new column. Note that you cannot add columns if you have modified the model while the **Numbering preview** dialog has been open.
Remove an added column	Right-click the column header, and then select **Delete**.
Change the order of the columns	Drag a column header to a new location.
Show or hide columns	• To hide a column, right-click the column header, and then select **Hide**. • To show a hidden column, right-click a column header, select **Show column**, and then select the name of the column that you want to show. • To show all hidden columns, right-click a column header, and then select **Show all columns**.
Save the column setup	1. Enter a file name in the box at the top of the dialog. 2. Click the  button. The `Settings.PreviewTable.xml` file is saved in the `\attributes` folder in the current model folder. The file can contain several setups. You can place the file, and Tekla Structures searches for it, in all usual locations: the model folder, project folder (XS_PROJECT), firm folder (XS_FIRM), and system folder (XS_SYSTEM). If several files are found, their contents are not merged. Note that the applied filters and sorting order are not saved.
Check, select, and fix objects with overlapping numbers	1. Select a red, **Overlapping** row.

To	Do this
	2. Right-click, and then use any of the following options: • Click **Select rebar sets**. The rebar sets where the selected bars belong to are selected in the model and the properties are shown in the property pane. You can then adjust the prefix and/or the start number of the rebar sets. • Click **Select this series in model**. To make more numbers available for the selected series, adjust the start number of the series. • Click **Select overlapping object in model**. Then adjust the prefix and/or the start number of the overlapping object. • Click **Select overlapping series in model**. Then adjust the prefix and/or the start number of the entire series. Note that you can use **Select this series in model** also on rows other than **Overlapping**.
Reject the proposed numbers	If you are not happy with the proposed position numbers, do the following: 1. If you do not want to close the **Numbering preview** dialog, select the **Keep preview open after rejecting** checkbox. This is useful if you want to open drawings or compare objects while fixing the numbering issues. 2. Click **Reject numbers**. 3. Adjust the model objects and the numbering settings as needed. 4. Close the **Numbering preview** dialog. 5. Start a numbering command and preview the numbering results again.
Accept the numbering results and assign the proposed numbers to objects	When you are happy with the proposed position numbers, click **Assign numbers**. Tekla Structures numbers the objects according to the numbering settings (page 1057).

To	Do this
	Note that the **Assign numbers** button is not available if you have modified the model.
Give feedback on the **Numbering preview** feature	Click the **Submit feedback** button at the upper-right corner of the dialog.

5.5 Check the numbering

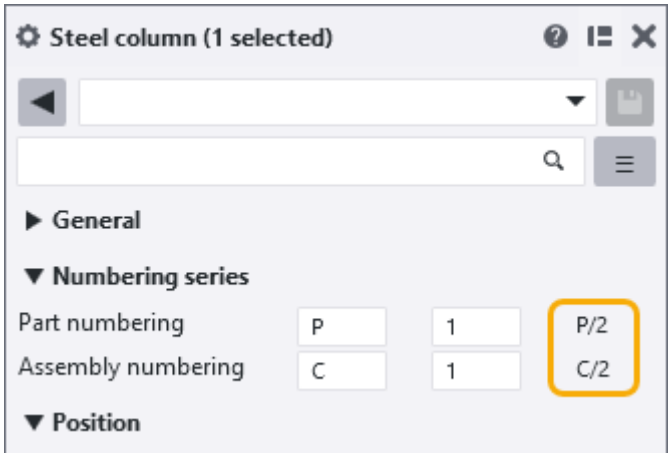
After you have started a numbering command, use the **Numbering preview** dialog to check the proposed position numbers (page 771).

After accepting the numbering results and assigning position numbers to objects, you can also check the actual position numbers in many places:

- You can see the position number of a steel part, assembly, cast unit, or rebar assembly in the property pane.

  1. Double-click the object in the model to open its properties in the property pane.
  2. Go to the **Numbering series** section for steel parts, the **Cast unit** section for concrete parts, or the **General** section for assemblies, cast units, or rebar assemblies.

  The position number appears next to the **Prefix** and **Start number** boxes. For example:



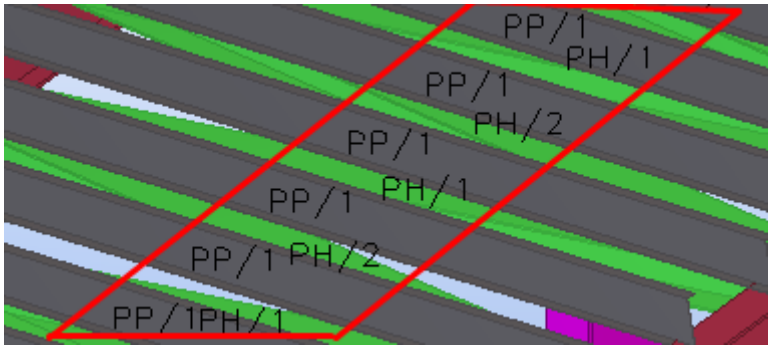
  If there is a question mark (?) after the position number, the numbering is not up to date.

  If you have several objects with different position numbers selected in the model, the text **Varies** is shown. Also, a list then shows the different position numbers within the selection, and in parentheses how many objects have those position numbers.



- You can add the position numbers to the part labels.
  1. In the model, make sure the **Select views** selection switch is active.
  2. Double-click the background to open the **View properties** dialog.
  3. Click **Display...** to open the **Display** dialog.
  4. Go to the **Advanced** tab and add **Part position** to the **Part label**.
  5. Click **Modify**.

  The part labels now contain the position numbers.



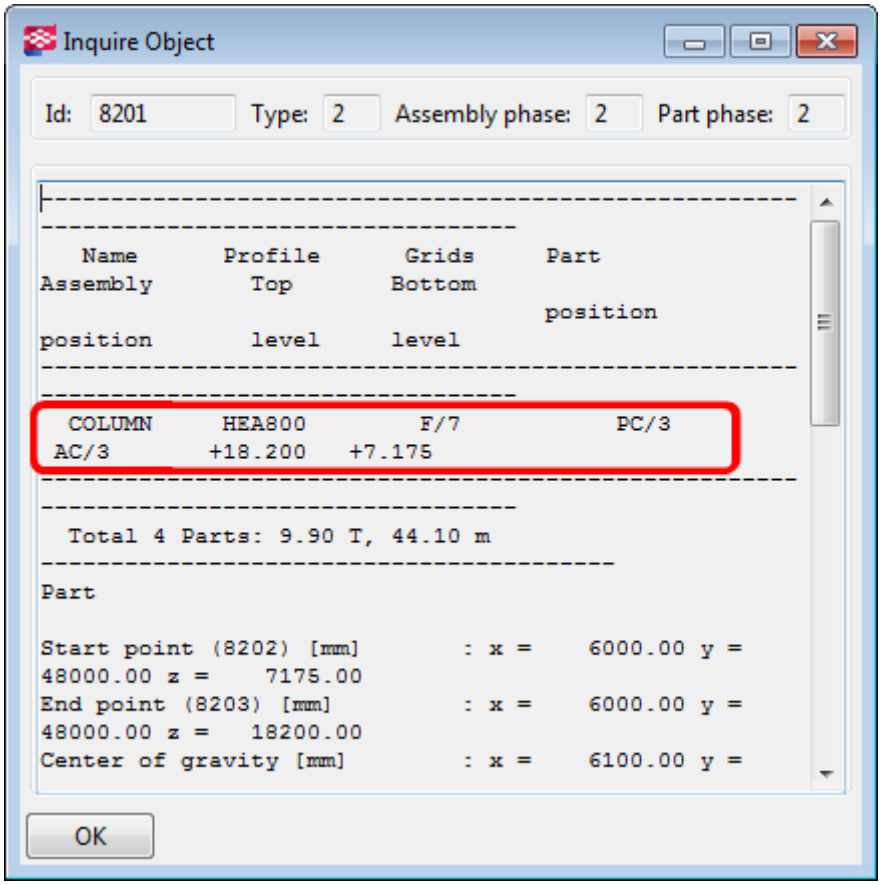
- You can check the part number in Document manager.

[AP.1] STANDARD
[AR.1] STANDARD
[AV.1] STANDARD

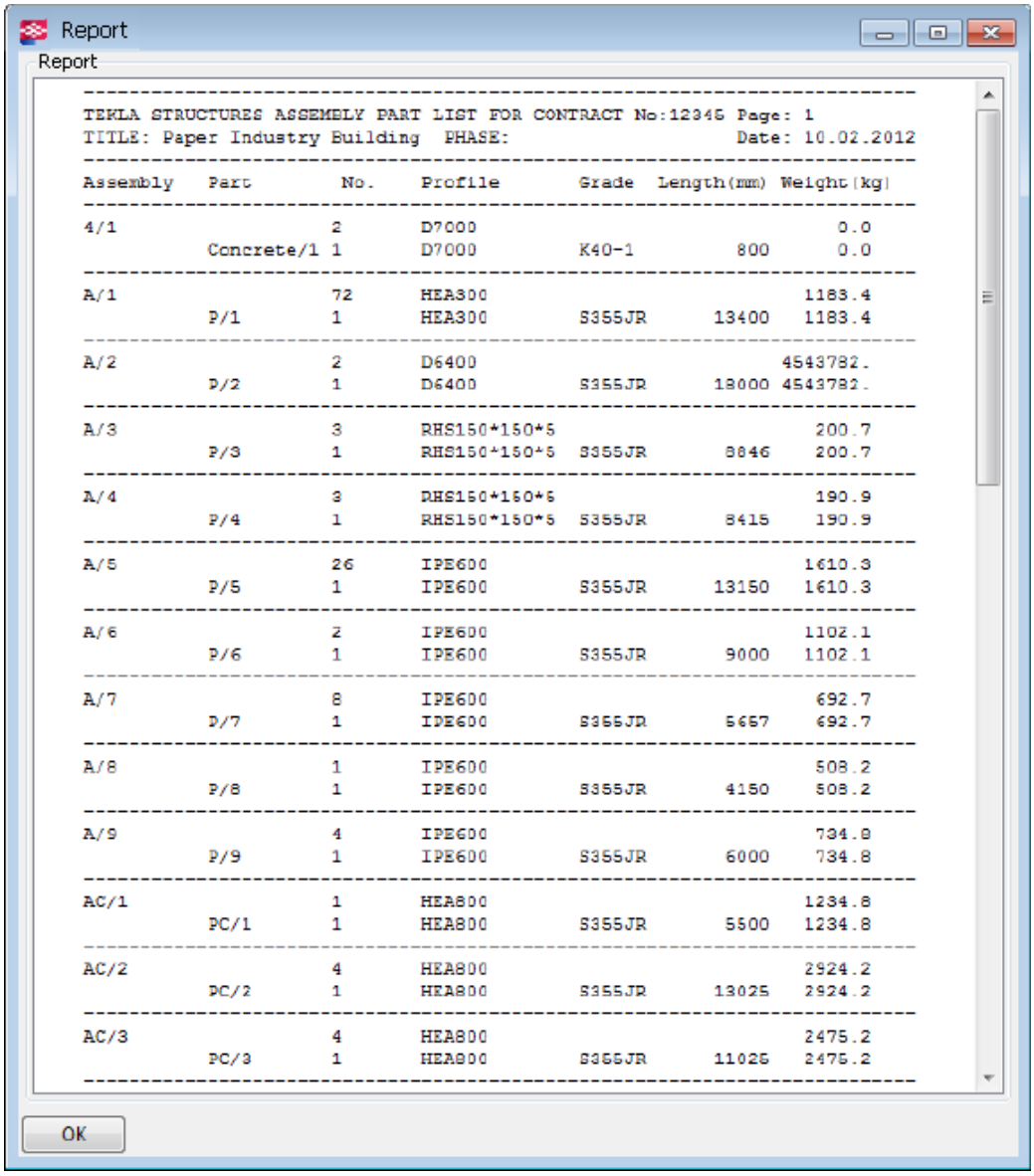
- The drawing label can show the position number and how many identical parts have the same number.

GENERAL NOTES:	ALL HOLES ARE 0.0 mm UNLESS NOTED ALL WELDS ARE 0.0 mm F.W UNLESS NOTED					
MATERIAL LIST FOR ASSEMBLY MK'D			AC/5		3 No. Required	
Mark	Profile	Material	No.	Length	Area	Weight
PC/5	HEA800	S355JR	1	18200	49.1	4086.1
				Total	49.1	4086.1

- You can use the **Inquire** commands.



- You can create reports that list your assembly and part positions.



See also

Repair numbering errors (page 781)

5.6 View the numbering history

To view the numbering history:

- On the **File** menu, click **Logs** --> **Numbering history log** .

  Tekla Structures displays the numbering log file.

5.7 Repair numbering errors

We recommend that you check and repair numbering in the model every now and then, especially before producing drawings or reports.

NOTE If you work with Tekla Model Sharing or in the multi-user mode, it is very important that you repair numbering regularly.

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog.
2. Ensure that the option **Compare to old** is selected for **New** parts.
3. Ensure that one of the following options is selected for **Modified** parts:
   - **Compare to old**
   - **Keep number if possible**
4. Click **OK** to save the changes.
5. Unless you want to repair the entire model, select the objects whose numbering you want to repair.
6. On the **File** menu, click **Diagnose & repair** and select one of the following commands under **Numbering**:
   - **Diagnose and repair numbering: all**

     This command numbers all parts and assemblies, even the unmodified ones.
   - **Diagnose and repair numbering: series of selected objects**

     This command numbers all parts and assemblies that have the same prefix and start number as the selected part.

     Note that Tekla Structures assigns the position number of the oldest part or assembly to all identical parts, even if a newer part or assembly has a smaller position number.

     TIP To manually assign a certain position number on a part or assembly, use the **Change number** command after repairing numbering in the model.

See also

Change existing numbers (page 781)

5.8 Change existing numbers

Use the **Change number** commands to change the existing part, assembly, multi-position, or family numbers into something you have defined yourself. These commands do not change the numbering series of parts. To prevent drawing, modeling, and fabrication errors, Tekla Structures does not allow you to use identical numbers for two different assemblies or parts.

1. On the **Drawings & reports** tab, click **Change number** and select one of the following commands:
   - **Change part number**
   - **Change assembly number**
   - **Change part multinumber**
   - **Change assembly multinumber**
   - **Change family number**

   The corresponding dialog appears. For example, if you select the **Change assembly number** command, the **Assign Assembly Number** dialog appears.
2. Select a part in the model.
3. In the dialog that appeared (for example **Assign Assembly Number**), click **Get** to view the current numbering properties of the part.
4. Enter the numbering properties you want to use for this part.

   Note that the position numbers you enter here are not absolute numbers. For example, if the start number of the series is `100`, the position numbers refer to numbers in that series. Therefore, position number `1` is actually `100`, position number `2` is `101`, position number `3` is `102`, and so on.
5. If you are changing the assembly number of selected parts, ensure that the **Assign to** option is set to **Selected objects only**.

   Otherwise, all parts with the same original number will be renumbered.
6. Click **Assign** to change the number.

   If the number you specified is already in use, Tekla Structures displays a warning and keeps the original number.

   Tekla Structures also displays a warning if the position number is higher than the highest current number. This is for information only and the number is still changed.

See also

Number parts and other model objects (page 767)

5.9 Clear existing numbers

Use the **Clear** commands to permanently remove the current position numbers of parts. Next time you run numbering, Tekla Structures assigns new numbers to these parts, irrespective of what their previous numbers were.

1. Select the parts whose numbers you want to clear.
2. On the **Drawings & reports** tab, click **Change number** and select one of the following commands:
   - **Clear part and assembly numbers**
   - **Clear part numbers**
   - **Clear assembly numbers**
   - **Clear reinforcing bar numbers**

   Tekla Structures removes the position numbers of the selected parts.

See also

Number parts and other model objects (page 767)

5.10 Renumber the model

Use the **Renumber all** option when the numbering needs to be started over. This option permanently removes the existing position numbers and resets them with new ones. Any existing drawings will also be removed.

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog.
2. Select the **Renumber all** checkbox.
3. Click **Apply** or **OK**.
4. On the **Drawings & reports** tab, click **Perform numbering** --> **Number modified objects** .
5. When you are asked to confirm the renumbering of the model, click **Yes**.

   Tekla Structures renumbers the entire model.

See also

Change existing numbers (page 781)

Clear existing numbers (page 782)

5.11 Control numbers

Control numbers are additional numbers that can be used to identify parts in a model. Use control numbers if you need to give additional, unique numbers to assemblies or cast units, regardless of their position numbers.

Control numbers can be useful, for example, when delivering a large number of similar wall elements to the site. To successfully pack and unpack the load, you need to plan the order of the wall elements already when the order is shipped. Although all of the wall elements may have the same cast unit position number, you can assign a unique control number to each individual wall element.

See also

Assign control numbers to parts (page 784)

Control number order (page 785)

Display control numbers in the model (page 786)

Remove control numbers (page 787)

Lock or unlock control numbers (page 788)

Example: Use control numbers to indicate the erection order (page 788)

Assign control numbers to parts

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Assign control numbers** to open the **Create control numbers** dialog.
2. Indicate which parts to assign control numbers to.
   - To number the entire model, do not select any parts.
   - To only number specific parts, select the parts you want to number.
3. If you want to assign control numbers only to parts in a specific numbering series:
   a. In the **Numbering** list, select **By numbering serie**.
   b. Enter the **Prefix** and **Start number** in the corresponding boxes.
4. Define the control numbers to be used.
   a. In the **Start number of control numbers** box, enter the first control number to be used.
   b. In the **Step value** box, define the control number interval.

      For example, to assign the control numbers 2, 5, 8, 11, and so on, enter 2 in the **Start number of control numbers** box and 3 in the **Step value** box.

5. Use the **Renumber** list to specify how to treat parts that already have control numbers.
   - Select **No** to keep the existing control numbers.
   - Select **Yes** to replace the existing control numbers with new ones.
6. Use the **First direction**, **Second direction**, and **Third direction** lists to define the order of control numbers.
7. In the **Write UDA to** list, select where to save the control numbers. The control number will appear on the **Parameters** tab in the user-defined attributes dialog either for:
   - **Assembly**
   - **Main part**
8. Click **Apply** to save the changes.
9. Click **Create** to number the parts.

See also

Control number order (page 785)

Control number settings (page 1060)

Control number order

When you assign control numbers, you must define in what order to assign them. The order is based on the location of each part on the global coordinate system.

The options are:

- **None**
- **X**
- **Y**
- **Z**
- **-X**
- **-Y**
- **-Z**

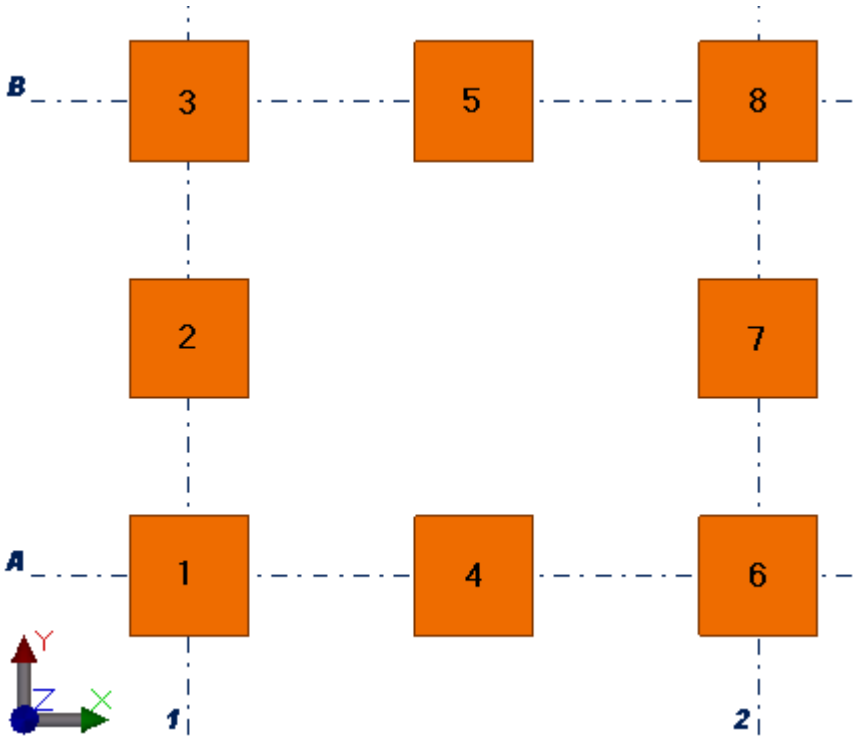
With positive directions (X, Y, and Z), the parts with the lowest coordinate value are numbered first. With negative directions (-X, -Y, and -Z), the parts with the highest coordinate value are numbered first.

For example, if the first direction is X, the second direction is Y, and the third direction is Z, numbering starts from the parts that have the lowest x coordinate value. If multiple parts have the same x coordinate, also

their y coordinates are compared. If multiple parts have the same x and y coordinates, also their z coordinates are compared.

Example

In the following example, the first direction is X and the second direction is Y. The numbers 1–8 indicate the control numbers.



See also

Assign control numbers to parts (page 784)

Display control numbers in the model

If the control numbers are not displayed in the model, you can set them visible using the display settings.

1. Double-click the view to open the **View properties** dialog.
2. Click **Display...** and go to the **Advanced** tab.
3. Select the **Part label** checkbox.
4. In the **Properties** list, select **User-defined attributes**, and then click **Add >**.

   The **Part label** dialog is displayed.

5. Enter `ACN` and click **OK**.

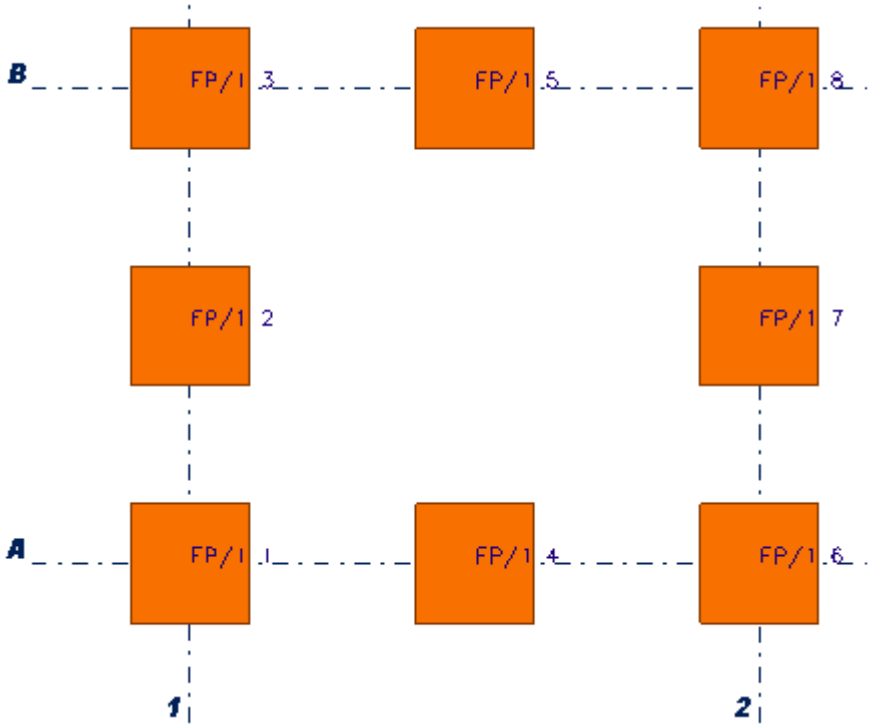
   The property is moved to the **Part label** list.

6. Click **Modify**.

   The control numbers are displayed in the model, right after the part position numbers.

Example

In the following example, the numbers 1–8 indicate the control numbers.



See also

Control numbers (page 783)

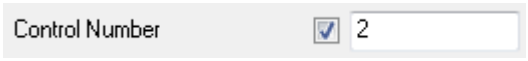
Remove control numbers

If needed, you can remove existing control numbers from all or some of the parts. Do not remove control numbers unless you are absolutely sure that they are no longer needed.

NOTE **Removing** of control numbers is different than **reassigning** of control numbers. If you only want to reassign new control numbers to parts that already have control numbers, use the **Renumber** option in the dialog.

1. Double-click a part to open the part properties in the property pane.
2. Click the **User-defined attributes** button.

   The current control number of the part is displayed on the **Parameters** tab, in the **Control Number** box. For example:

3. Remove the existing control number from the box.
4. Click **Modify** to apply the change.

See also

Control numbers (page 783)

Lock or unlock control numbers

To prevent other users from changing the control numbers of some or all of the parts in the model, use the **Lock/unlock control numbers** command. Later on, if the control numbers need to be changed, you can use the same command for unlocking the numbers.

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Lock/unlock control numbers** to open the **Lock/Unlock control numbers** dialog.
2. Define which parts' control numbers to lock or unlock.
   - To lock or unlock the control numbers of all parts, do not select any parts in the model.
   - To lock or unlock the control numbers of specific parts only, select the parts in the model.
3. In the **Status** list, select **Lock** or **Unlock**.
4. Click **Apply** to save the changes.
5. Click **Create** to lock or unlock the numbers.

See also

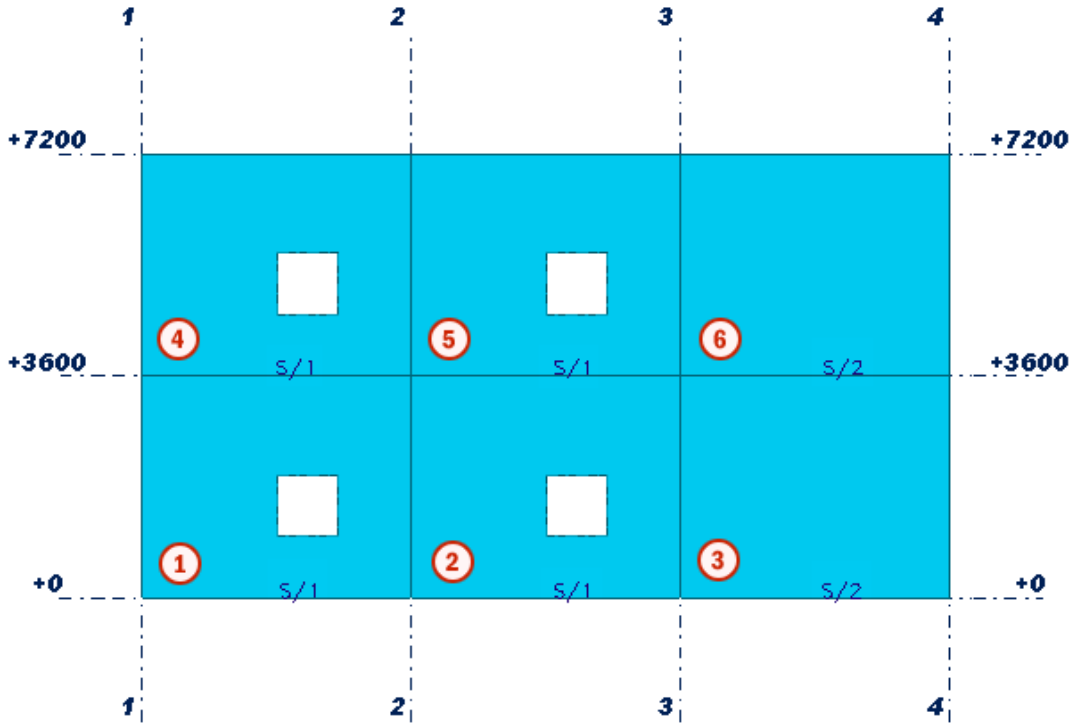
Control numbers (page 783)

Example: Use control numbers to indicate the erection order

This example shows how to assign control numbers to six concrete walls. As four of these walls have the same cast unit position, you cannot make a clear distinction between the cast units based on their position number. That is why each wall will get a unique identifier that indicates their erection order at

the site. The erection order also affects the order of shipping. For example, the wall number 1 needs to be shipped on top of the delivery, because it will be erected first on the site; the wall number 2 should be the second on the bundle, because it will be erected next; and so on.

The following image represents the desired end result.



(1) Erected as 1st

(2) Erected as 2nd

(3) Erected as 3rd

(4) Erected as 4th

(5) Erected as 5th

(6) Erected as 6th

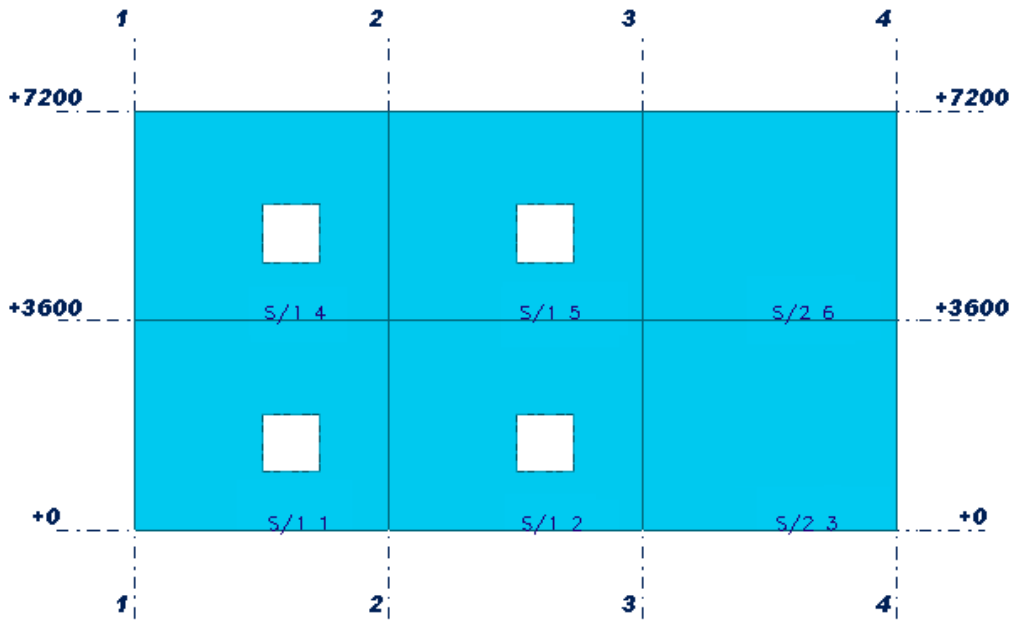
To assign control numbers to the concrete walls:

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Assign control numbers** to open the **Create control numbers** dialog.
2. Select the six concrete walls.
3. Define that you want to assign control numbers only to parts in the numbering series S with the start number 1.
   a. In the **Numbering** list, select **By numbering serie**.
   b. In the **Prefix** box, enter `S`.
   c. In the **Start number** box, enter `1`.

4. Define that you want to use the numbers 1–6 as the control numbers for these concrete walls.
   a. In the **Start number of control numbers** box, enter `1`.
   b. In the **Step value** box, enter `1`.
5. Define that you want to first number the walls with identical z coordinates in the order they appear in the positive x axis.
   a. In the **First direction** list, select **Z**.
   b. In the **Second direction** list, select **X**.
6. Click **Apply** to save the changes.
7. Click **Create** to number the concrete walls.

   Each concrete wall gets a unique control number, as shown in the following image.

TIP If you cannot see the control numbers in the model, modify the display settings. For more information on the required settings, see Display control numbers in the model (page 786).



5.12 Number parts by design group

You can number parts by design groups so that you can differentiate the parts from each other in drawings and reports. The design group numbers can be used in engineering documents or as preliminary numbers.

Design group numbers consist of a prefix, a separator, and a number or a letter.

Use the **Design group numbering** application to assign prefixes and numbers or letters to parts on the basis of design groups. The **Design group numbering** application groups parts that match a selection filter into a design group, numbers them, and optionally compares the part lengths. The application also compares the user-defined attributes of parts that are set to affect numbering.

NOTE The **Design group numbering** application only numbers parts that have a profile extruded to create the length of the part, such as beams, columns, panels, and footings. Contour plates, slabs, or items are not numbered.

Before you start:

- Create the needed selection filters that define the design groups.
- If you want to use certain letters in design group numbering, list the allowed letters using the advanced option XS_VALID_CHARS_FOR_ASSEMBLY_POSITION_NUMBERS. By default, letters A–Z are allowed.
- In multi-user or Tekla Model Sharing models, make sure that only one of the users runs the **Design group numbering** application.

To number parts by their design group:

1. In the model, Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **Design group numbering** to start the application.
4. In the **Design group numbering** dialog:
   a. Click **Add group** to create design group numbering settings for parts matching a selection filter.
      - Select the filter in the **Group filter** column.

        The selection filters are read from specific folders in the standard folder search order.
      - Enter the design group prefix and start number or start letter you want to use for the parts in this group.
      - In the **Compare length** column, define whether the part lengths are compared or not, or if gross length is used in comparison.

   b. Repeat step 4a for all part groups that you want to number by design group.

c. If needed, change the order of the groups by using the **Move up** and **Move down** buttons.

   If a part belongs to several groups, the last group filter in the list overrides the previous ones.

d. If you want to compare part lengths, define the length tolerance.

   For example, if you enter 0, parts must be exactly the same length to receive the same design group number (or letter). If you enter 2, part lengths can differ 2 mm from each other.

   The default tolerance is 0.05 mm.

e. Enter a number separator that is used to separate the design group prefix and number or letter in drawing marks and in reports. For example, enter - .

   We recommend that you do not change the separator during the project.

f. To reuse old, unneeded numbers or letters, select the **Reuse old numbers** checkbox.

g. In **Number using letters**, select whether to use letters or not.

h. In **Renumber all**, select whether to renumber all parts or not.

i. In **Compare UDA**, select whether to compare the user-defined attributes of parts.

j. In **Compare finish**, select whether to compare the **Finish** property of parts.

k. To number the parts by design group, click **Perform numbering**.

   A design group number is saved as the user-defined attribute `DESIGN_GROUP_MARK` of each part.

   By default, the user-defined attribute `DESIGN_GROUP_MARK` is available in the `objects.inp` file in the Engineering configuration in the default and US environments.

l. To create a report to show the numbering results, select whether to create a report of all or the selected parts, and then click **Create report**.

   Tekla Structures shows the report in the **List** dialog and also saves the report. The report is saved as `dgnReport.txt` in the folder defined by the advanced option XS_REPORT_OUTPUT_DIRECTORY. In the Default environment, the report is saved in the `\Reports` folder under the current model folder.

   When you select a row in the **List** dialog, Tekla Structures highlights and selects the corresponding part in the model.

If the numbering of a part is not up to date, meaning that the part has been modified after numbering, a question mark (?) is added after the design group number.

TIP Alternatively, you can click **Open Multi report generator** and create reports using Multi report generator.

5. To show the design group numbers in drawing marks or in reports, use the user-defined attribute `DESIGN_GROUP_MARK`.

See also

Create new filters (page 172)

5.13 Numbering examples

This section gives some examples of numbering the model.

Click the links below to find out more:

Example: Number identical beams (page 793)

Example: Use family numbers (page 794)

Example: Number selected part types (page 795)

Example: Number parts in selected phases (page 796)

Example: Number identical beams

This example explains how different numbering settings result in different part numbers when you modify a part.

To number identical beams:

1. Create three identical beams with the numbering series prefix P and start number 1.
2. Number the model. All the beams have the part position number P1.
3. Modify one of the beams.
4. Number the model. You should now have two beams P1 and one P2.
5. Change beam P2 to be identical to the others.
6. Number the model.

Depending on the numbering settings in the **Numbering Setup** dialog, Tekla Structures assigns one of the following part position numbers to the modified part:

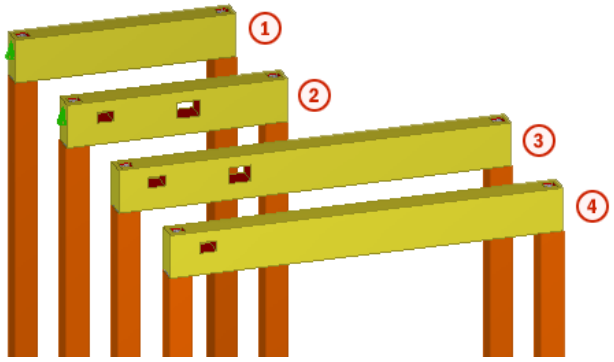
- **Compare to old**: P1
- **Keep number if possible**: P2
- **Take new number**: P3

See also

Number parts and other model objects (page 767)

Example: Use family numbers

In this example, the following four beams have the numbering series prefix B and the start number 1. The parts have the same main profile, and each pair has the same length, but the holes are different.



(1) Assembly position: B/1
(2) Assembly position: B/2
(3) Assembly position: B/3
(4) Assembly position: B/4

We use the following family numbering settings:

- **Use family numbering for series**: add series **B/1**
- **Compare**: select options **Main part profile** and **Overall length**

With the given family numbering criteria, Tekla Structures divides the beams into two families. All beams have the same profile, but each pair has a different length. Within both families the beams get different qualifiers because they have different holes.

- The first beam gets the assembly position number B/1-1
- The second beam gets the assembly position number B/1-2
- The third beam gets the assembly position number B/2-1
- The fourth beam gets the assembly position number B/2-2

See also

Family numbering (page 760)

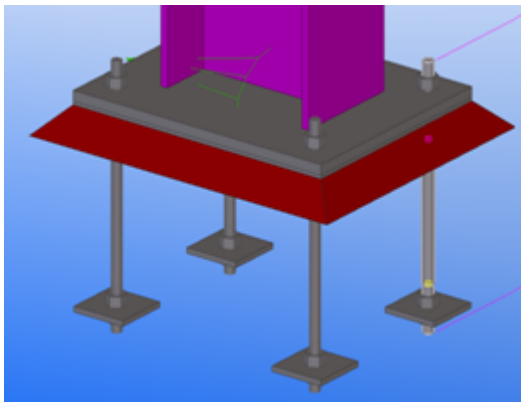
Example: Number selected part types

This example shows how different numbering settings can be used for different part types. We will use one set of numbering settings for steel anchor rods, and another one for steel columns. Note that the **Number series of selected objects** command numbers all parts that have the same assembly prefix.

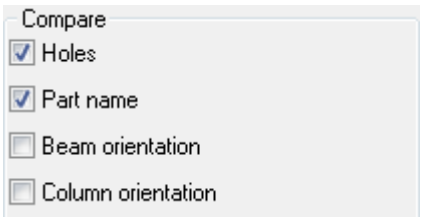
To number anchor rods and columns:

1. Create steel columns.
2. Create anchor rods with the numbering series prefix AR and start number 1.

   Ensure that this numbering series is different to any other parts or assemblies in the model.



3. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog.
4. Ensure that the **Column orientation** checkbox is not selected, and then click **Apply**.



5. Select one of the anchor rods in the model.
6. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.

   All parts with the AR prefix and start number at 1 are numbered.

7. Wait until the anchor rod numbering is complete.
8. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog.
9. Select the **Column orientation** checkbox, and then click **Apply**.
10. Select one of the steel columns in the model.
11. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.

    All columns belonging to the same numbering series as the selected column are numbered.

See also

Number a series of parts or other model objects (page 768)

Example: Number parts in selected phases

This example shows how to number a model that consists of multiple phases, each phase having a different detailing and submittal schedule. This enables you to release drawings for a particular phase at any time.

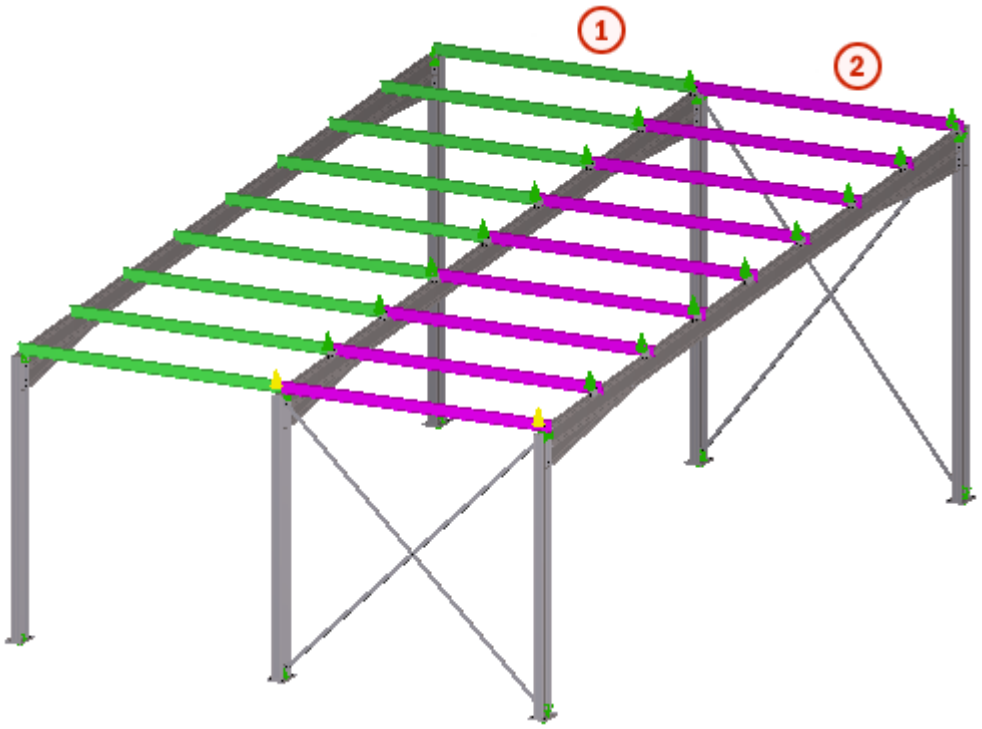
Before you start, divide the model into phases.

To number parts in selected phases:

1. Apply a specific numbering series prefix and start number for parts in each phase.

   For example:

   - The beams in phase 1 get the numbering series prefix B and start number 1000.
   - The beams in phase 2 get the numbering series prefix B and start number 2000.



(1) Phase 1: green

(2) Phase 2: magenta

2. Ensure that the numbering series are not overlapping.

   For example, to avoid numbering overlaps with the beams in phase 2, phase 1 should not contain more than 1000 position numbers.

3. Select the parts you want to number.

   TIP Use selection filters to easily select parts belonging to a certain phase or parts with a specific start number series. You can also use selection filters to ignore specific phases that are already completed or phases that are not ready for numbering.

4. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog.
5. Modify the numbering settings, and then click **Apply**.
6. Select one of the parts you want to number.
7. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.

   All parts belonging to the same numbering series as the selected part are numbered.

See also

Number a series of parts or other model objects (page 768)

Numbering settings during a project (page 798)

5.14 Tips for numbering

- It is a good idea to have some kind of a routine in numbering. For example, number the model when you start your working day or when you finish for the day.
- To save time, include the numbering series in the default part properties for each type of part before you start modeling.
- Numbering is not just another way of classifying parts. To classify, use **Organizer**, user-defined attributes, or colors.
- If you have overlapping position numbers, Tekla Structures will warn you about it.

  Use the **Numbering preview** dialog (page 771) to find and fix objects with overlapping numbers.

  It is important to solve the numbering overlaps by changing the objects' numbering series, because the objects with overlapping numbers will not be numbered.

See also

Numbering settings during a project (page 798)

Numbering examples (page 793)

Create a standard-part model (page 799)

Numbering settings during a project

You can use different numbering settings at different times in a project.

For example:

- Before releasing a phase of the project for fabrication, you might use the **Re-use old numbers** option for numbering the entire model.
- If a phase has already been released for fabrication in a project, you might use the **Take new number** option for new and modified parts.
- If you are numbering other phases of the project at earlier stages of detailing, you might use the **Compare to old** option and try to combine as many position numbers as possible.

See also

Example: Number parts in selected phases (page 796)

General numbering settings (page 1057)

Create a standard-part model

A standard-part model contains only standard parts with specific part prefixes. You can use these prefixes when numbering parts in another model. The prefixes that you define will be used as actual part position numbers in the other model.

The standard-part model is only used for comparing parts when you number parts in a project model. It cannot be used for creating parts in the project model.

NOTE This functionality only applies to steel parts. Assemblies are not affected.

---

1. Create a new model and give it a descriptive name.

   For example, `StandardParts`.

2. Create the objects you want to use as standard parts.
3. Explode all components.

   You can explode the components if you plan to delete the unnecessary parts such as duplicate angles and main parts.

4. Delete all unnecessary items.
5. Give the objects part prefixes that are not used elsewhere (for example, STD1, STD2, and so on).

   Ensure that the standard-part model does not contain duplicate part prefixes. You do not need to define the assembly prefix or the part or assembly start numbers.

6. Save the standard-part model.

   To use the standard-part model with Tekla Model Sharing, save the standard-part model in a separate folder under the current model folder.

   To use the standard-part model with a multi-user model, save the standard-part model so that all users have access to it.

7. Open a project model that you wish to number.
8. On the **File** menu, click **Settings** --> **Advanced options** --> **Numbering** .
9. Check that the advanced option XS_STD_PART_MODEL points to the correct standard-part model.

For example:

```
XS_STD_PART_MODEL=C:\TeklaStructuresModels\StandardParts\
```

10. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to display the **Numbering Setup** dialog.
11. If you have selected the **Part name** checkbox, ensure that the project model has the same part names as the standard-part model.
12. Select the **Check for standard parts** checkbox.
13. Click **Apply** to save the changes.
14. On the **Drawings & reports** tab, click **Perform numbering** --> **Number modified objects** to number the project model.

    As the parts are being numbered, Tekla Structures compares all parts in the project model to the standard-part model. Any part prefixes found in the standard-part model are applied to all identical parts found in the project model. The numbering series of the oldest numbered standard part in the project model is applied to all identical parts found in the project model.

See also

Number parts and other model objects (page 767)

6 System components and applications

All available system components, applications, macros, and drawing plugins are located in the **Applications & components** catalog.

Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.

System components

Tekla Structures contains a wide range of predefined system components by default. System components are tools that you can use to connect parts in the model. System components automate tasks and group objects so that Tekla Structures treats them as a single unit. You can save the properties of a component and use the properties in other projects.

There are three types of system components:

- **Connection** components connect two or more parts, and create all the required objects, such as cuts, fittings, parts, bolts, and welds. For example, end plates, clip angles, and bolted gussets are connections.

  In the **Applications & components** catalog, the connection symbol is .
- **Detail** components add a detail or a reinforcement to the main part. A detail is only connected to one part. For example, base plates and lifting hooks are steel details, and beam reinforcement and pad footing reinforcement are concrete details.

  In the **Applications & components** catalog, the detail symbol is .

- **Detailing** components automatically create and assemble the parts to build a structure, but do not connect the structure to existing parts. For example, stairs, frames, and towers are detailing components.

  In the **Applications & components** catalog, the detailing symbol is .

You can also create your own components, custom components (page 863), if you do not find a system component that suits your needs. You can use custom components in the same way as system components.

Macros

Macros (page 811) are available as `.cs` files in the `\drawings` or `\modeling` folder under the folders that have been defined with the advanced option XS_MACRO_DIRECTORY. By default, this advanced option is set to `..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\macros`.

We recommend that you do not change the default folder in the advanced option. You can add one or more additional folder locations after the default folder, for example, when company or project-specific folders are needed to store macros.

You can also record macros of your own. When you create your own macros, the macros will be saved under the model folder, in the `\macros\drawings` or `\macros\modeling` folder depending on the mode (drawing or modeling) you are using while recording the macro. Tekla Structures creates the `\macros\drawings` and `\macros\modeling` folders automatically.

Macros are first read from the model folder, then from the folders defined in `XS_MACRO_DIRECTORY`. We recommend that you keep the default folder `\common\macros` as the first folder in the advanced option.

Example of the definition for `XS_MACRO_DIRECTORY`:

```
%XSDATADIR%environments\common\macros;%XSDATADIR%environments\uk\General\user-macros
```

Extensions (.tsep)

You can download Tekla Structures extensions that have the `.tsep` file extension from Tekla Warehouse and import these extensions to the **Applications & components** catalog. When you restart Tekla Structures, the imported extensions are installed and added to the **Ungrouped items** group in the catalog. You can move them to a suitable group. Find out more about extensions in .

Publish groups in the Applications & components catalog

You can collect content to a group that you create in the **Applications & components** catalog. You can then publish the group as a catalog definition file to make it available for other Tekla Structures users.

6.1 Work with system components

Use system components to connect parts in the model and to automate tasks.

System components adapt to changes in the model, which means that Tekla Structures automatically modifies the component if you modify the parts it connects. When you copy or move objects, Tekla Structures automatically copies or moves all the associated components with the objects.

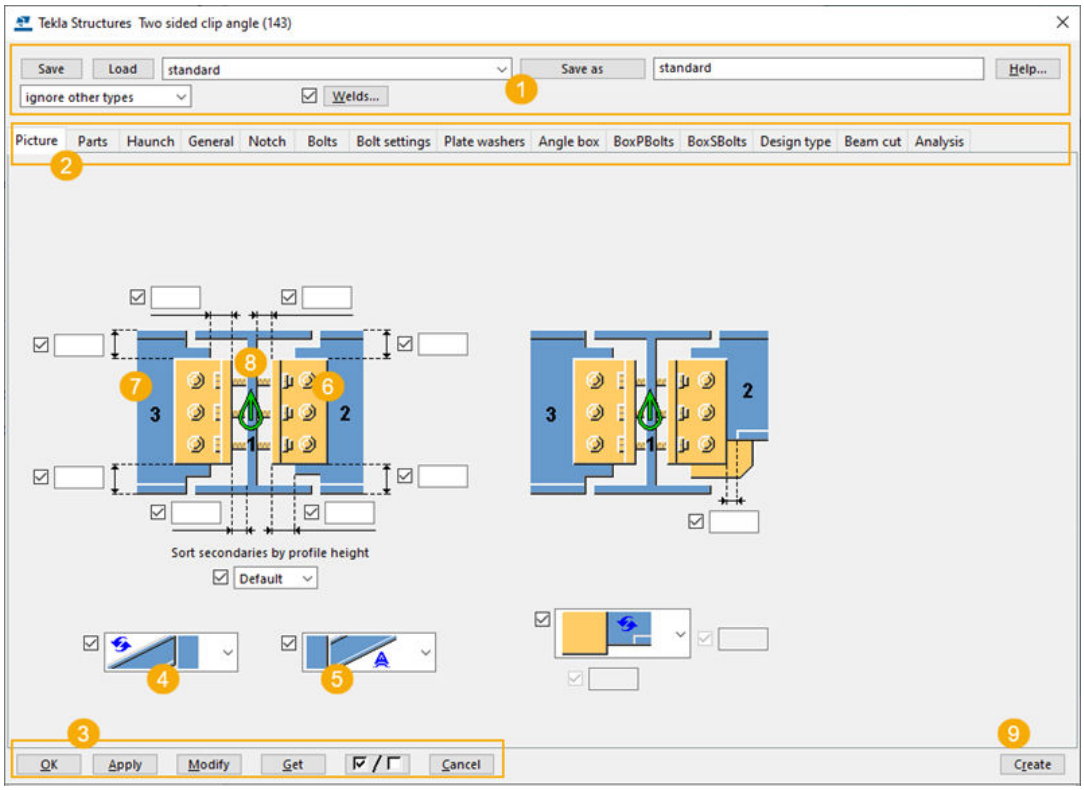
All system components are stored in the Applications & components catalog.

Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.

Define component properties

Each component has a dialog where you can define the properties of the component. You can open the dialog by double-clicking the component in the **Applications & components** catalog.

The image below shows a typical example of a steel connection, the **Two sided clip angle (143)** connection. Concrete and reinforcement component dialogs may have different options.



	Description
1	In the upper part of the dialog, you can save and load predefined settings. Some components have buttons for accessing bolts, welds, and DSTV properties. When modifying connections and details, you can select whether Tekla Structures ignores other types of connections and details, or modifies all selected connections and details irrespective of their type. With the modify option, the type of the selected components is changed to match the type of the component you are currently modifying. For more information, see and Model folder files and file name extensions.
2	On the tabs, you can define the properties of the parts and bolts that the component creates. You can enter values manually, use system default values, AutoDefaults values, automatic values, or for some steel connections, the values in the `joints.def` file. Manually entered values, AutoDefaults, automatic values and the properties defined in the `joints.def` file all override the system defaults. System default values are used if you do not manually enter a value or select any other type of property value. You cannot change the system default values. For more information on `joints.def`, see Define connection properties in the joints.def file (page 839).
3	For information on the dialog buttons, see .
4	If you select an AutoDefaults option, Tekla Structures uses the property defined in the AutoDefaults rules. The image in the AutoDefaults option is an example and does not necessarily match the outcome in the model. For more information on AutoDefaults, see AutoDefaults (page 826).
5	If you select an automatic option, Tekla Structures automatically determines which option to use for a property. For example, when you use the automatic option for the stiffener in **End plate (144)**, the connection automatically adds the stiffener to a beam-to-column connection, but does not add it to a beam-to-beam connection. For more information on AutoConnection, see AutoConnection (page 820).
6	The parts that are yellow in the component dialog are created by the component.
7	The parts that are blue in the component dialog should already exist in the model before you create the component.

	Description
8	Up direction indicates how the connection is rotated around the secondary part, relative to the current work plane. The  symbol on the **Picture** tab of the component dialog indicates the correct up direction. If there are no secondary parts, Tekla Structures rotates the connection around the main part. The options are: `+x, -x, +y, -y, +z, -z`. You can change the default up direction on the **General** tab in the component dialog. Try changing the positive directions first.
9	Click the **Create** button to to start creating a new component in the model using the settings applied in the component dialog. Then follow the instructions on the status bar.

Add a component to a model

When you add a component to a model, you either attach the component to existing parts in the model, or pick positions to indicate the location or length of the component.

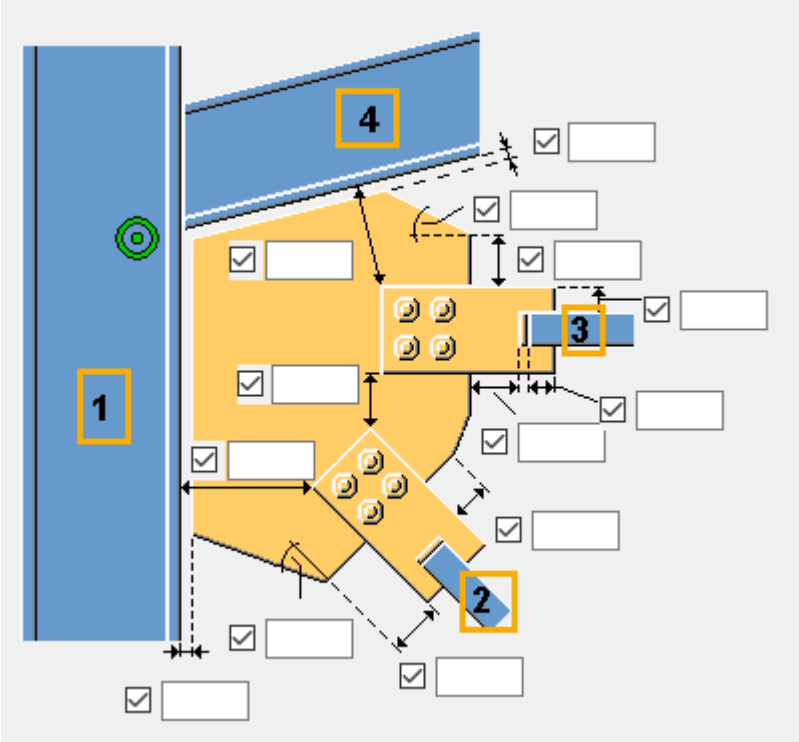
Connections and details have a main part that you select first. Connections also have one or more secondary parts that you select after you have selected the main part. Detailing components do not always have a main part and secondary parts. Instead, they automatically create and assemble the parts to build a structure when you pick a position in the model.

If you use a component you are unfamiliar with, use the default properties of the component. Then check what needs to be modified, and modify only few properties at a time to see how the modifications affect the component. This is quicker than trying to set all the component properties before seeing what the component actually creates.

You can get information about adding and modifying the component in the model in a command prompt window. This information can be useful in problem situations. Set the `XS_DISPLAY_RPC_COMPONENT_CONSOLE_WINDOW` advanced option to `TRUE` to display the window. By default, the value is `FALSE` and the window is not displayed.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.

   You can also press **Ctrl + F**.
2. Select a component and do one of the following:

To	Do this
Add a connection	1. Select the main part. 2. Select the secondary part or parts. • If there is one secondary part, the connection is automatically created when you select the secondary part. • If there are several secondary parts, click the middle mouse button to finish selecting the parts and to create the connection. In the example image below, the numbers from 1 to 4 show the selection order of parts. The blue parts should already exist in the model before you create the component. 
Add a detail	1. Select the main part. 2. Pick a position in the main part to determine the location of the detail.
Add a detailing component	Pick one to three positions to determine the location of the objects that the detailing component creates.

To	Do this
Add a new a component using the **Create new** command	Select a component in the model, right-click and select **Create new**. Then follow the instructions on the status bar. If you have a component dialog open, you can also use the **Create** button in the dialog to create a new component.

3. Check the status of the component.

   When you have added the component, Tekla Structures shows the component status using the symbols shown in the table below.

Color	Status
	The green symbol shows that the component was created successfully. The detailing component symbol in the model is  .
	The yellow symbol shows that the component was created, but it has problems. This often occurs when bolts or holes have an edge distance that is less than the default value.
	The red symbol shows that the component was not created. Common reasons are that the properties are incorrect or that the up direction is not appropriate.

4. If the component is not created correctly or not added at all, you can check the following, for example:
   - If the component is not added to the model, check the status bar. For example, you may need to click the middle mouse button to stop selecting parts before Tekla Structures creates the component.
   - Some components work with certain profiles only. If a component is not created successfully, try entering a valid profile.

- If a component does not by default create the parts that you need, look for options to create them. If there are no options, try entering a thickness value for the parts.

  If a component creates parts that you do not need, look for options to remove them. If there are no options, enter zero (0) as the thickness of the parts.
- If you are using a connection that only allows one secondary part, you may see the message `Many parts found` on the status bar. This means that Tekla Structures cannot determine which parts to connect. You may have several parts in the same location, or the view may be set too deep.

Modify a component in a model

You can modify the properties of a component after you have added the component in the model, for example, if you need to change the number of bolts or plate dimensions.

1. Double-click the component symbol in the model to open the component dialog.
2. Modify the properties.
3. If needed, define which connections and details are modified:
   - Ignore other types: Tekla Structures only modifies the connections and details that are of the same type as the connection or detail you are modifying.
   - Modify connection type: If you have selected several connections and details, click **Modify** to modify all the selected connections and details irrespective of their type. If the connection type of a selected connection is not the same as in the connection dialog, Tekla Structures changes the connection type.
4. Click **Modify**.

When you have added components to the model, you can use the property pane to list the components:

- If you select one component in the model, the property pane shows the name and number of that component. You can open the component properties dialog by clicking the **Component properties** button in the property pane.
- If you select several different components in the model, the property pane shows lists that have the text **Varies**. Open the lists to show the names and numbers of the selected components.

- If you select components and other model objects, click the **Object type list** button  in the property pane to open a list of the selected object types, and select **Component** to list the components.

View a component in a model

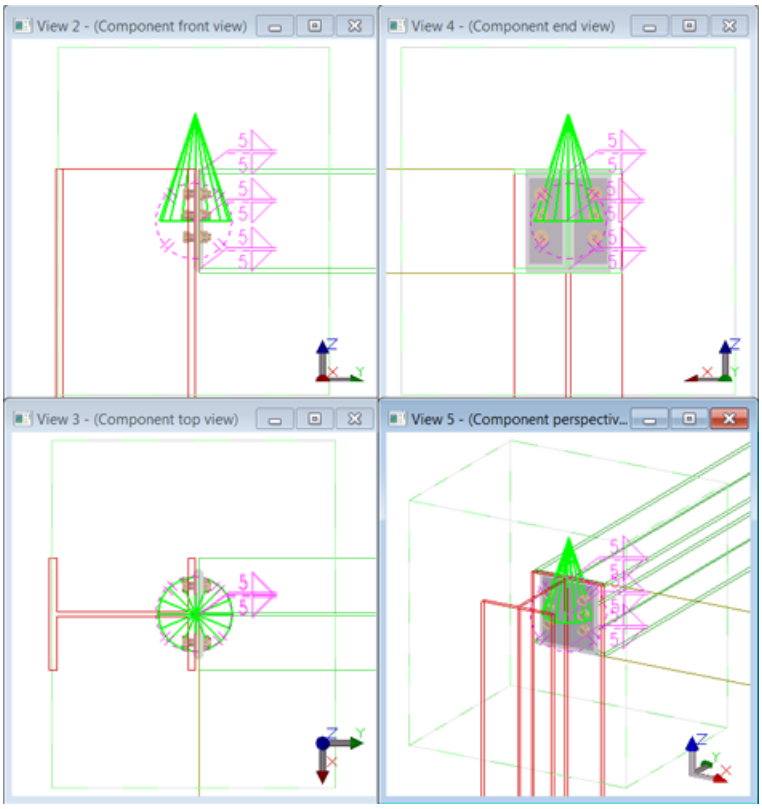
You can create several views of a component to view it from different viewpoints.

1. Click the component symbol in the model to select the component.
2. Right-click and select **Create view** --> **Default views of component** .

   Tekla Structures creates four views: front, end, top, and perspective.

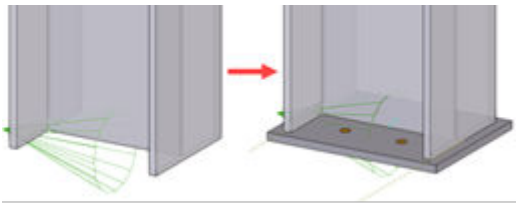
   You can check the dimensions, such as bolt locations and edge distances, using the **Measure** tool in the **Component front view**.

   The example image shows the default views of the **End plate (144)** connection.



TIP You can display the contents of selected components, such as parts, bolts, welds and other details, even if **Visibility in components** is not selected in the **View properties** --> **Display** dialog. To display the contents, go to the **View** tab on the ribbon, then click **Rendering** --> **Content** and select the components. You

can also type `content` in the **Quick Launch** box to start the **Show component content** command.



When you click **View** --> **Redraw** --> **Redraw all views**, the contents are no longer visible.

Detailing and conceptual components

Depending on the Tekla Structures configuration you are using, you can create either detailing or conceptual components.

- Detailing components include all the information needed for fabrication, such as assemblies, cast units, and reinforcing bars.

  You can create detailing components in the **Tekla Structures Diamond** and **Tekla Structures Graphite** configurations, and in the **Full**, **Primary**, **Steel Detailing**, and **Precast Concrete Detailing** legacy configurations.

  Detailing components have a round symbol in the model:  or .

- Conceptual components look similar to detailing components but do not include the option to change part numbering or assembly numbering settings. Conceptual components are meant to be used as reference information for further fabrication detailing.

  You can create conceptual components in the **Engineering**, **Rebar Detailing**, and **Construction Modeling** legacy configurations. In the **Tekla Structures Graphite** configuration, if needed, administrators can use the advanced option to define that only conceptual components can be created.

  Conceptual components have a rectangular symbol in the model:  or .

Convert components

When you make changes in the model, for example, modify the size of the main part for the component, this does not automatically convert a conceptual component to a detailing component, or vice versa.

You can convert components in the **Applications & components** catalog. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.

You can edit conceptual components and convert them to detailing components in the **Tekla Structures Diamond** and **Tekla Structures Graphite** configurations, and in the **Full**, **Primary**, **Steel Detailing**, and **Precast Concrete Detailing** legacy configurations.

In the **Engineering**, **Rebar Detailing**, and **Construction Modeling** legacy configurations, detailing components will adapt to the changes you make in the model, but you cannot modify any properties using the component dialog. Detailing components will also remain as detailing components unless you convert them one by one.

Do one of the following:

To	Do this	Configuration
Convert a conceptual component to a detailing component	1. Click  > **Convert to detailing component**. 2. Select the component symbol.	**Tekla Structures Diamond**, **Tekla Structures Graphite** **Full**, **Primary**, **Steel Detailing**, **Precast Concrete Detailing**
Convert a detailing component to a conceptual component	1. Click  >**Convert to conceptual component**. 2. Select the component symbol.	**Engineering**, **Construction Modeling**, **Rebar Detailing**

6.2 Work with applications

You can run, add, edit, rename, save as and delete applications, macros and plugins in the **Applications** section of **Applications & components** catalog. You can also record and edit macros.

Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog. Then do one of the following:

To	Do this
Record a macro	1. Click the **Access advanced features** button  and then click **Record macro**. 2. Enter a name for the macro in the **Macro name** box.

To	Do this
	3. Click **OK** and perform the actions you want to record. 4. Click the **Stop recording** button in the **Applications & components** catalog to stop recording. The recorded macro is saved under the model folder, in the `\macros\drawings` or `\macros\modeling` folder depending on the mode (drawing or modeling) you are using while recording the macro. Tekla Structures creates the `\macros\drawings` and `\macros\modeling` folders automatically.
Create a macro file and add contents later	1. Click the **Access advanced features** button  and then click **New macro**. 2. Enter a name for the macro in the **Macro name** box. 3. Click **OK**. This creates an empty macro file that is displayed in the **Applications** list. The macro file is saved under the model folder, in the `\macros\drawings` or `\macros\modeling` folder depending on the mode (drawing or modeling) you are using while creating the macro file. Tekla Structures creates the `\macros\drawings` and `\macros\modeling` folders automatically. 4. Right-click the empty macro file and select **Edit**. 5. Add the macro content, for example, by copying commands

To	Do this
	from other macro files and save the file.
View or edit a macro	1. Click the arrow next to **Applications** to open the applications list. 2. Right-click the macro you want to edit and click **Edit**. The macro can be opened in any text editor. 3. If needed, edit the macro and save the macro file.
Run an application	1. Click the arrow next to **Applications** to open the applications list. 2. Double-click the application you want to run.
Save an application with another name	1. Click the arrow next to **Applications** to open the applications list. 2. Right-click the application that you want to save with another name and click **Save as**. 3. Enter a new name for the application and click **OK**. The application will be added in the list.
Rename an application	1. Click the arrow next to **Applications** to open the applications list. 2. Right-click the application you want to rename and click **Rename**. 3. Enter a new name for the application and click **OK**. The name of the application changes.
Delete an application	1. Click the arrow next to **Applications** to open the applications list.

To	Do this
	2. Right-click the application you want to delete and click **Delete**. The application is removed from the list.

See also

System components and applications (page 801)

6.3 How to use the Applications & components catalog

Components are stored in the **Applications & components** catalog and organized into two different types of groups: default groups are automatically available and predefined groups depend on your environment.

Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog. You can also press **Ctrl + F**.

To use a component (page 803), select it in the catalog and follow the instructions on the status bar to add the component to the model. Double-click the component in the catalog to open the component properties dialog.

Groups in the catalog

Default groups and predefined groups are shown against different background color in the catalog.

Default groups are automatically available:

- **Recent** contains the 12 components and applications that you have most recently used in the model.
- **Ungrouped items** contains the components and applications that are not in any predefined group.

  Ungrouped items can be, for example, imported components that have not been moved to any other group yet.
- **Applications** contains applications (page 801), macros and drawing plugins.

  If you create your own macros, you can add them to this group.
- **Connections** contains connections and seams.
- **Detailing** contains detailing components.
- **Details** contains details.

- **Parts** contains custom parts.
- **Legacy catalog** shows the folder structure of the **Component Catalog** used in previous Tekla Structures versions if the catalog definition files are found in the standard folder search paths.

Depending on your environment, the catalog may also contain **predefined groups** for specific usage, such as **Steel** --> **Beam to beam connections**. You can create your own groups according to your needs, for example, for your own favorite connections. This way you can find these connections quickly and easily. You can also hide the groups that you are not using so that only the groups you use are visible in the catalog.

Model-specific components are shown in the modeling mode and drawing-specific components are shown in the drawing mode.

Search for a component in the catalog

To search for a component in the catalog, enter the search term in the search box. The search is case-insensitive.

Note that the search does not find catalog content that has been hidden. Select the **Show hidden items** checkbox to show the hidden content.

The search uses the following rules:

- Non-numeric search terms find partial matches, for example, `bolt` shows both `bolt` and `bolted` in the search result.

  If you have more than one word in the search term, for example `bolt plate`, the words are automatically combined so that the search result shows the components that contain both bolt and plate in their name, description, or tags.
- Numeric (integer) search terms find the exact match, for example, `121` shows component number **121** in the search result.

  You can use the `*`, `?` and `[ ]` wildcards to search for partial numeric matches. For example, `10*` finds components number **10**, **110**, **104**, **1040**, and so on.
- You can restrict the search to specific tags, groups and types of components by using the keywords `tag`, `group`, or `type`. For example, `10 tag:advanced` finds the number **10** components that have `advanced` in the tag, and `type:custom` finds all custom components.

Change the view in the catalog

- Click  to show the thumbnail view.

- Click  to show the list view.
- Click  to show the compact view.

  The compact view shows the thumbnail images of the group that you select from the list above the search box. You can use the compact view to have more space on the screen.
- Click  to show the normal view.

Show selected components in the catalog

Click **Show selected** to show a **Selected components** group that contains the components that are selected in the model or in the drawing.

Click **Show selected** again to hide the **Selected components** group.

The **Show selected** button is not available when you are using the search in the catalog.

TIP You can use the property pane to list the components that you have selected in the model. If you select one component, the property pane shows the name and number of the component. If you select several different components, the property pane shows lists that have the text **Varies**. Open the lists to show the names and numbers of the selected components.

If you select components and other model objects, click the **Object type list** button  in the property pane to open a list of the selected object types, and select **Component** to list the components.

View and modify component information in the catalog

Each component has an information box that shows the type of the component and the groups the component belongs to. You can add a description for the component and tags that can be used in the search.

1. Select a component in the catalog and click the small arrow on the right to open the component information box.
2. Type a description in the **Description** box.
3. Click  to add a tag and enter a tag in the box.
4. If needed, click  again to add more tags. You can also remove tags.

5. Click outside the information box to close it.

The descriptions and tags that you add are by default saved in the `ComponentCatalog.xml` file in the model folder.

Add a thumbnail image for a component in the catalog

Components have a default thumbnail image that shows a typical situation where the component can be used. You can add several thumbnails for a component and select which thumbnail is shown in the thumbnail view in the **Applications & components** catalog.

1. Select a component in the catalog.
2. Right-click and select**Thumbnails**.
3. Click **Add thumbnail**.
4. Select an image and click **Open**. You can use any standard image format, for example, `.png`, `.jpeg`, `.gif`, `.tiff`, and `.bmp`.
5. Select the checkboxes of the thumbnails that you want to show in the component information box. You can also remove thumbnails, except for the default thumbnail.
6. Click **Close**.

The thumbnail information that you add is by default saved in the `ComponentCatalog.xml` file in the model folder.

Publish a component in the catalog

You may need to use the same component with different settings in different situations. To easily use the component, you can define the settings for each situation and publish the component in the catalog.

For example, you may need **End plate (144)** in three different situations. Add **End plate (144)** once to each situation in the model. Define the settings needed and then publish each **End plate (144)** in the catalog. You will then have **End plate (144)** saved in the catalog as three separate components, each with different settings. You can use these components from the catalog in the same way as other components.

1. Add a component once to all needed situations in the model.
2. Define the desired settings for each situation.
3. Select one of the components you added in the model, right-click and select **Publish in Catalog...**.
4. Enter a descriptive name for the component and click **OK**.
5. Repeat steps 3-4 for each component you added.

The components are first placed in the **Ungrouped items** group in the catalog. They have the name you entered and the thumbnail image of the original component.

You can move the components to a more appropriate group in the catalog and change the thumbnail image. For example, you can create a basic component view in each situation, and use an image of the view as the thumbnail.

Create and modify groups in the catalog

You can create groups and subgroups, and move groups to different locations in the predefined groups section in the catalog. You can add and remove components from the groups, rename the groups, and add descriptions for the groups.

To	Do this
Create a group	Right-click in the catalog and select **New group...**. Drag the group to the desired location.
Create a subgroup	Right-click a group in the catalog and select **New group...**.
Name a group	Right-click a group, select **Rename...** and type the name.
Add components to the group	• Select components in the catalog and drag them to another group. • Select components in the catalog, right-click and select **Add to group**. Then select the group to which the components are added. • Right-click a group, select **Add all to group** and select the group to which all the components in the group are added. Note that the components are copied, not moved, to the other groups.
Remove a group Remove a component from a group	Right-click a group or a component in a group, and select **Remove from group**.

The groups that you create are by default saved in the `ComponentCatalog.xml` file in the model folder.

NOTE You cannot add or remove groups in the default groups, and you cannot modify the content of the default groups. However, you can hide the default groups and the individual items in the groups.

Change the order of groups in the catalog

You can change the order of the predefined groups in the **Applications & components** catalog. Predefined groups depend on your environment, for example, **Steel** and **Concrete** can be such groups. Note that you cannot change the order of the default groups, for example, **Applications**, **Connections**, and **Detailing**.

You can control the order with a sort index. The **Sort index** option is available in the group information of each predefined group in the **Applications & components** catalog. Sort indexes are saved in the catalog definition files.

You can change the sort index by entering either a negative or a positive integer number, or `0`, in the **Sort index** option box. A negative sort index moves a group towards the top and a positive sort index moves a group towards the bottom in the predefined groups section. Enter `0` or clear the value to revert to the default order. By default, the groups are in alphabetical order.

The sort index changes you make are model specific and they are saved in the `ComponentCatalog.xml` file in the `\model` folder. Administrators can define the order of groups for an environment or a project using the catalog definition files in the environment, firm and project folders. Do not edit these files if you are not an administrator.

Note that even if administrators have defined the order, you can still make model-specific changes to the order of the groups by entering a different sort index value for a group. If you need to revert to the default order, enter `0` as the sort index.

To change the order:

1. Select a predefined group.
2. Click the small arrow on the right to open the group information box.
3. Enter a number in the **Sort index** box.

   The group is immediately moved.
4. Save the model to keep the order.

Hide groups and components in the catalog

1. Select a group or a component in the catalog.
2. Right-click and select **Hide / Unhide** to hide the group or the component.
3. To view the hidden group or component again, select the **Show hidden items** checkbox at the bottom of the catalog. The hidden group or component is shown as dimmed.
4. To show the hidden group or component normally, right-click it and select **Hide / Unhide**.

Show the catalog message log

If there are errors or warnings, for example, in the catalog definition files, the **Message Log** button is shown in the lower right corner in the catalog. The button is not shown if there are no errors or warnings.

To view the error log, click the **Message Log** button.

Errors and warnings are also written to the `ComponentCatalog_<user>.log` file in the `\logs` folder under the model folder.

Catalog definitions

The commands in **Access advanced features**  > **Catalog management** are used for modifying catalog definitions. Generally, there is no need to modify catalog definitions. Do not modify the definition files if you are not an administrator. For more information on administrator tasks, see Customize the Applications & components catalog.

6.4 Automate connection creation

This section describes the tools that you can use to automate connection creation in the model.

Click the links below to find out more:

AutoConnection (page 820)

AutoDefaults (page 826)

AutoConnection and AutoDefaults rules (page 831)

AutoConnection

Use the AutoConnection tool to automatically select and apply connections with predefined properties to selected parts in a model. With AutoConnection, Tekla Structures automatically creates similar connections for similar framing conditions.

You can use AutoConnection to quickly add connections individually, in phases, or project-wide. This is useful when you are working on a large project using many connections, modifying a model, or importing modified profiles.

NOTE Before using AutoConnection in a working model, we recommend that you create a test model, and create all the connection conditions in it that you need for a particular project. You can then use the test model to check the

rules and properties of various connection types. The model also acts as a quick reference for connection information.

See also

Define AutoConnection settings and rules (page 821)

Create a connection using AutoConnection (page 824)

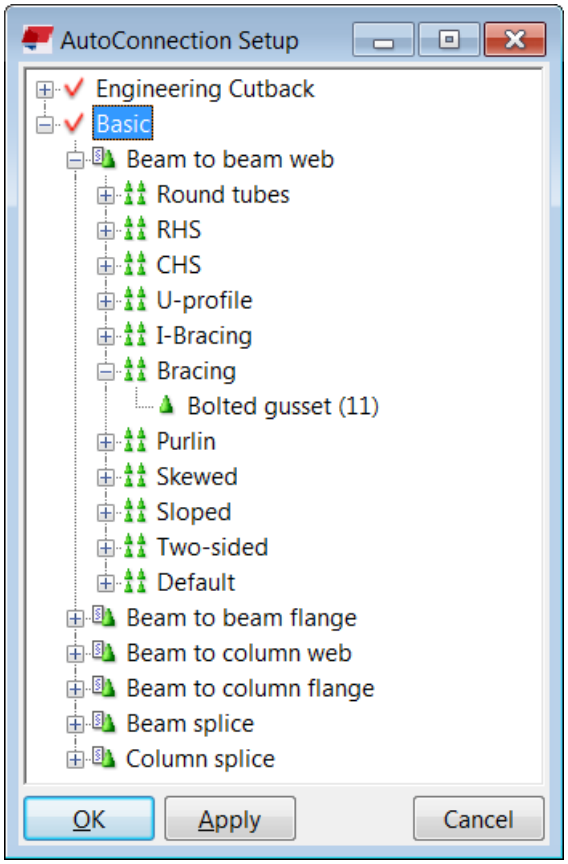
AutoConnection and AutoDefaults rules (page 831)

Define AutoConnection settings and rules

With AutoConnection you can define groups of rules which Tekla Structures automatically applies when creating connections in a model. By using a rule group to select connections and connection properties you do not have to select each connection and define its properties separately. For example, you can create separate rules for different standards, projects, manufacturers, and even individual models.

AutoConnection settings

To open the **AutoConnection Setup** dialog, on the **File** menu, click **Catalogs** --> **AutoConnection settings**.



Icon	Setup level	Description
	Rule group	You can use rule groups to organize connections and connection properties according to different standards, projects, manufacturers, and models. You can create, modify and delete rule groups.
	Framing condition	Framing conditions are predefined connection types that you cannot change. Tekla Structures creates the framing conditions automatically: • Beam to beam web • Beam to beam flange • Beam to column web • Beam to column flange • Beam splice • Column splice
	Rule set	You can use rule sets to define which connection to use in a certain situation. You can create additional rule sets.
	Connection	The connection to apply if the rule set criteria are met. To apply a particular connection, the conditions in the model have to match all the rules in the branch that contains the connection.

Rules.zxt file

When you use AutoConnection, Tekla Structures saves the AutoConnection information in a zipped `rules.zxt` file in the `\attributes` folder under the current model folder.

You can copy the `rules.zxt` file to the project or firm folder to make it available in other models. Each time you modify the AutoConnection setup you need to recopy this file to the firm and project folders. To use the modified setup in other models, restart Tekla Structures.

Limitation

You can have a maximum of two secondary parts in the connection (for example, you cannot use complex gussets with several secondary parts). AutoConnection uses the profile height and ID number as the criteria for determining the first secondary and second secondary parts.

Create a rule group for AutoConnection

You can define rule groups for AutoConnection to organize connections and connection properties according to different standards, projects, manufacturers, and models.

1. On the **File** menu, click **Catalogs** --> **AutoConnection settings**.
2. Right-click an existing rule group and select **New Rule Group**.
3. Click the **New** group and enter a name.

   Give the rule group a name that reflects the group of connections that you want to create (page 824). For example, use the fabricator's name, the project name, or any name that clearly identifies the connection rules that you want to use for a specific model.

When you create a new rule group, Tekla Structures automatically adds the existing framing conditions in the group.

Create a rule set for AutoConnection

You can create AutoConnection rule sets under framing conditions to specify which connection properties to use when specific conditions in the model are met.

You only need to create AutoConnection rule sets if you plan to use different connections (page 824) to connect similar framing conditions. For example, in the model, some beam-to-beam connections require clip angles, others need shear tabs. You need to define rule sets to determine where each connection type should be used.

1. On the **File** menu, click **Catalogs** --> **AutoConnection settings**.
2. Click the plus icon in front of the rule group to open the tree structure.
3. Right-click the relevant framing condition and select **Create Additional Rule Sets**.
4. Right-click the new rule set and select **Edit Rule Set...**.
5. Enter a name for the rule set.
6. Select a rule from the **Available rules** list.
7. Click the right arrow button to move the selected rule into the list of **Rules in rule set**.
8. Enter the values used in the rule: either an exact value, or minimum and maximum values.
9. Click **OK**.

NOTE The order of the rules in the tree structure is important. Tekla Structures uses the first rule that matches the conditions in the model so you should place the most limiting rule highest in the tree, and the most generic rule lowest.

You can change the priority of a rule set by right-clicking the rule set and selecting **Move up** or **Move down**.

Change a connection in an AutoConnection rule set

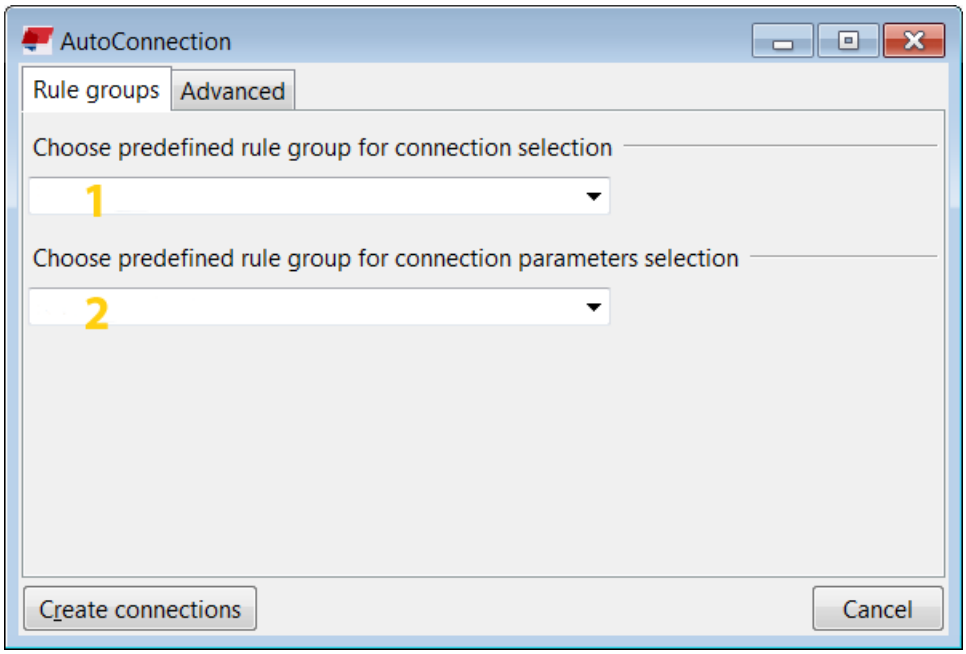
You can change the connection in a rule set by selecting a connection in the **Applications & components** catalog.

1. On the **File** menu, click **Catalogs** --> **AutoConnection settings**.
2. Click the plus icon in front of the relevant framing condition  and rule set  to find the connection that you want to change.
3. Right-click the connection and select **Select Connection Type...**.
4. Double-click a connection in the **Select component** dialog.
5. Click **OK** in the **AutoConnection Setup** dialog.

Create a connection using AutoConnection

Use AutoConnection to have Tekla Structures automatically create connections using the properties of predefined rules. When you use AutoConnection, Tekla Structures ignores the properties in the connection dialogs. Tekla Structures does not modify the existing connections.

1. In the model, select the parts to connect.
2. On the **Edit** tab, click **Components** --> **Create AutoConnections**.
3. Select the rule groups from the lists on the **Rule groups** tab.



1	Rule group for AutoConnection
2	Rule group for AutoDefaults

4. If needed, go to the **Advanced** tab to change the rules used in the framing conditions:
   a. Select the connection in the **Connection selection** option:
      - **AutoConnection** applies the connection defined in the rule group that you have selected in the first list on the **Rule groups** tab.
      - **None** does not create a connection.
      - Click **Select...** to select a connection from the **Applications & components** catalog. Tekla Structures creates the connection using the default properties.
   b. Select the connection properties in the **Parameters selection** option:
      - **Autodefaults** applies the properties of the rule group that you have selected in the first list on the **Rule groups** tab.
      - **No autodefaults** applies the default connection properties.
5. Click **Create connections**.

TIP You can also use the **Auto connect selected parts** macro to automatically create connections using the current properties without opening the **AutoConnection** dialog.

Macros are located in the **Applications** group in the **Applications & components** catalog.

See also

Define AutoConnection settings and rules (page 821)

AutoDefaults

Use AutoDefaults to set up properties for existing connections. AutoDefaults allows you to modify the default connection properties and save them for use in specific circumstances. When you use AutoDefaults, Tekla Structures automatically creates the connections with the predefined AutoDefaults properties. You can also use AutoDefaults for a single connection.

For example, you can use AutoDefaults to automatically adjust the thickness of each base plate you create, according to the main part profile. If the main part profile changes, Tekla Structures automatically adjusts the thickness of the base plate.

NOTE Before using AutoDefaults in a working model, we recommend that you create a test model, and create all the connection conditions in it that you need for a particular project. You can then use this test model to check the rules and properties of various connection types. It also acts as a quick reference for connection information.

See also

Define AutoDefaults settings and rules (page 826)

Modify a connection using AutoDefaults (page 831)

AutoConnection and AutoDefaults rules (page 831)

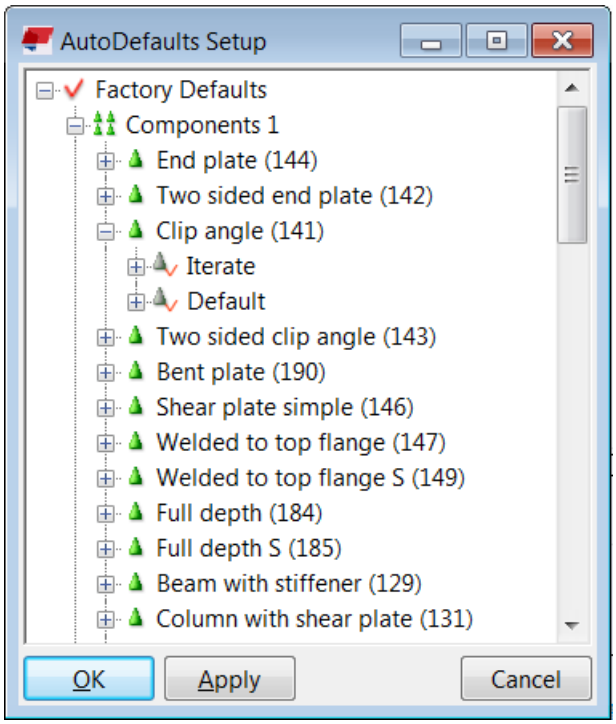
Define AutoDefaults settings and rules

Use AutoDefaults to set up properties for existing connections. AutoDefaults chooses connection properties based on the framing condition. With AutoDefaults you can create rules that define the situations where the predefined properties are used.

To apply the configured AutoDefaults settings and rules to connections, see Modify a connection using AutoDefaults (page 831).

AutoDefaults settings

To open the **AutoDefaults setup** dialog, click **File** --> **Catalogs** --> **AutoDefaults settings**.



Icon	Setup level	Description
	Rule group	You can use rule groups to organize settings according to different standards, projects, manufacturers, and models. You can create, modify and delete rule groups.
	Components	The component tree structure shows the connections that are available on component toolbars in Tekla Structures.
	Rule set	Rule sets control which properties to use in certain situations. You can create additional rule sets. Tekla Structures processes AutoDefaults rule sets in the order in which they are in the tree, so you can control the selection of properties.
	Properties file	The properties files are under the rule sets. By default, each connection has a standard properties file that defines the standard

Icon	Setup level	Description
		properties, for example, `standard.j144` or `standard.j1042`. You can create additional properties files for the properties that you want to use again and give the files distinctive names.

Defaults.zxt file

When you use AutoDefaults, Tekla Structures saves the AutoDefaults rules in a zipped `defaults.zxt` text file in the `\attributes` folder under the current model folder.

You can copy the `defaults.zxt` file to the project or firm folder to make it available in other models. Each time you modify the AutoDefaults setup, you need to recopy this file to the firm or project folder. To use the modified setup in other models, restart Tekla Structures.

NOTE We do not recommend that you edit the `defaults.zxt` file using a text editor, but if you do, ensure that you are using the right syntax. The easiest way to unzip the `.zxt` file is to change the file extension `.zxt` to `txt.gz` and unzip the file. Change the extension back to `.zxt` when you have finished. You do not need to zip the file after editing it, Tekla Structures can also read the unzipped file.

Limitation

AutoDefaults only affects connection parts (clip angles, shear tabs, end plates, ...), bolts, and welds. AutoDefaults cannot change beam profiles or the connection number.

Create a rule group for AutoDefaults

You can define rule groups for AutoDefaults to group the rules according to different standards, projects, or manufacturers, for example.

1. On the **File** menu, click **Catalogs** --> **AutoDefaults settings**.
2. Right-click an existing rule group and select **New Rule Group**.
3. Click the **New** group to rename it.

   Give the rule group a name that reflects the contents of the group. For example, use the fabricator's name, the project name, or any name that clearly identifies the rules that you want to use for a specific model.

When you create a new rule group, Tekla Structures automatically adds the existing components to the group.

Create a rule set for AutoDefaults

You can create rule sets to define which connection properties are used when specific conditions in the model are met.

1. On the **File** menu, click **Catalogs** --> **AutoDefaults settings**.
2. Click the plus icon in front of the rule group to open the tree structure.
3. Click the plus icon in front of the relevant group of components and connection .
4. Right-click an existing rule set and select **New Rule Set**.
5. Right-click the new rule set and select **Edit Rule Set...**.
6. Enter a name for the rule set.
7. Select a rule from the **Available rules** list.
8. Click the right arrow button to move the selected rule into the list of **Rules in rule set**.
9. Enter the values used in the rule: either an exact value, or minimum and maximum values.
10. Select from the **Parameter files selection in rule set** list how the properties are selected in the rule set.

Option	Description
Use combination of first parameters	Tekla Structures uses the properties files it finds in the first matching sub-rule set and does not check other rule sets.
Iterate until connection symbol is green	Tekla Structures checks sub-rule sets until it finds matching properties.
Iterate until connection symbol is yellow	Tekla Structures checks sub-rule sets until it finds matching properties.
Use combination of all parameters	Tekla Structures checks all rule sets and uses the properties files in all matching rule sets. The order of the properties files is important. When Tekla Structures combines the properties files, the most recent files (the lowest in the tree) override previous ones. If you do not enter any values for the properties, Tekla Structures does not override the previous property values.

11. Click **OK**.

NOTE The order of the rules (page 834) in the tree structure is important. Tekla Structures uses the first rule that matches the conditions within the model so you should place the most limiting rule highest in the tree, and the most generic rule lowest.

You can change the priority of a rule set by right-clicking the rule set and selecting **Move up** or **Move down**.

Modify connection properties for AutoDefaults

Each connection has a default standard property file that defines the properties for the connection. You can modify the properties that the standard file uses. Save the connection properties that you want to use and set the standard file to use these properties (page 831) in the AutoDefaults settings.

1. On the **File** menu, click **Catalogs** --> **AutoDefaults settings**.
2. Click the plus icon in front of the rule group to open the tree structure.
3. Click the plus icon in front of the relevant group of components and connection .
4. Right-click the `standard.j` connection file that you want to modify, for example, `standard.j144` and select **Edit Connection Parameters...**.
5. In the connection dialog, set the properties that you want to save.

   Such properties could be, for example, bolt properties, profiles, and materials.
6. Enter a descriptive name for the properties in the box next to the **Save as** button.
7. Copy this name in the **Connection code** option on the **General** tab.

   Using the same name allows you to check which properties Tekla Structures used in specific situations. Tekla Structures does not automatically show the AutoDefaults values in the connection dialog.
8. Click **Save as**.

   Tekla Structures saves the properties file in the `\attributes` folder under the current model folder. The filename consists of the name you entered in **Save as** and the file extension `.jXXX`, where `XXX` is the connection number, for example, `sec_0-190.j144`.
9. Click **Cancel** to close the connection dialog and return to the **AutoDefaults setup** dialog.

   If you click **OK** to close the connection dialog, you need to load the default properties the next time you use the connection. Using the default properties ensures that AutoDefaults can modify the properties.

10. Right-click the `standard.j` file again and select **Select Connection Parameters...**.

    The **Attribute file list** dialog that opens contains the properties that have been set and saved in the connection dialog.

11. Select a file in the **Attribute file list** dialog.
12. Click **OK**.

Modify a connection using AutoDefaults

When you use a connection that you are unfamiliar with, first use the default properties. Then use AutoDefaults to modify the properties.

1. Double-click a connection symbol in the model to open the connection dialog.
2. On the **General** tab, select a rule group from the **AutoDefaults rule group** list.
3. On all tabs, select the AutoDefaults options marked with the arrow symbol  for the properties in which you want to use AutoDefaults.
4. Click **Apply**.

If you manually modify the properties after using AutoDefaults, Tekla Structures uses the manually modified properties.

For example, you have manually set the base plate thickness of a connection to 20 mm. AutoDefaults is active and sets the plate thickness according to the main part profile. If you modify the main part profile, Tekla Structures does not update the base plate thickness. It remains at 20 mm.

NOTE You can view which AutoDefault rules and properties are used:

- To view AutoDefaults rules, select the connection symbol in the model, right-click and select **Inquire** .

  Tekla Structures shows the rule group, rule sets and properties files used.

- To view the AutoDefaults properties, double-click the connection symbol in the model, select `<AutoDefaults>` in the list box next to the **Load** button and click **Load**.

See also

Define AutoDefaults settings and rules (page 826)

AutoConnection and AutoDefaults rules

You can create your own AutoConnection and AutoDefaults rules for project and company defaults. By defining rules you can accurately select connections and connection properties when using AutoConnection and AutoDefaults.

General rules

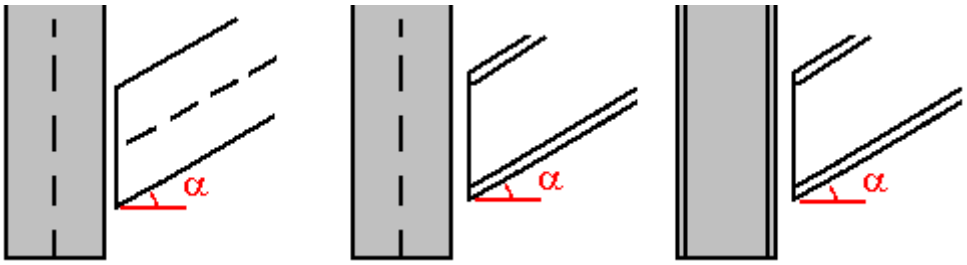
- **Profile name** is the name in the profile catalog.
- **Profile type**
  - **I** = 1
  - **L** = 2
  - **Z** = 3
  - **U** = 4
  - Plate = 5
  - Round bar = 6
  - Pipe = 7
  - Square pipe = 8
  - **C** = 9
  - **T** = 10
  - **ZZ** = 15
  - **CC** = 16
  - **CW** = 17
  - Polygon plate = 51
- Number of secondary parts
- Number of main parts
- Material name

Orientation rules

Depending on the relative angle of a beam, the connections can be classified as sloped, skewed, or cant. The angle value can be between - 90 and 90 degrees.

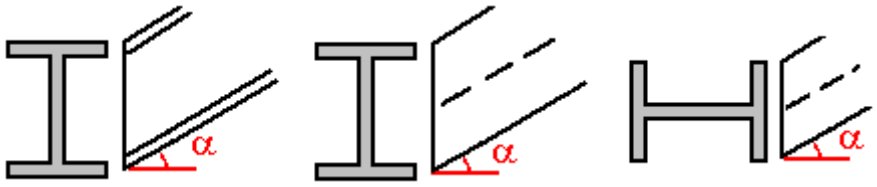
- **Sloped** angle (relative to main part cross section)

  The longitudinal axis of the secondary part follows the slope of the longitudinal axis of the main part.



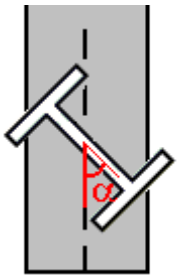
- **Skewed** angle (relative to main part longitudinal axis)

The longitudinal axis of the secondary part is skewed according to the main part cross section. The angle is the smaller of the angles between the longitudinal axis of the secondary part and the main part Z or Y axis.



- **Cant** angle

  For rotated secondary parts



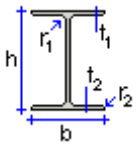
Dimension rules

- **Profile depth**
- **Web depth**

  For profiles with an upper and lower flange, the web depth is: `h-t1-t2-2*r1`

  Or, if t2 is zero: `h-2*t-2*r1`

  For profiles with one flange, the web depth is `h-t-r1-r2`.



- **Web thickness**
- **Flange thickness**

Forces and strengths

- Shear force
- Axial force
- Bending moment

See also

Combining and iterating properties for AutoDefaults (page 834)

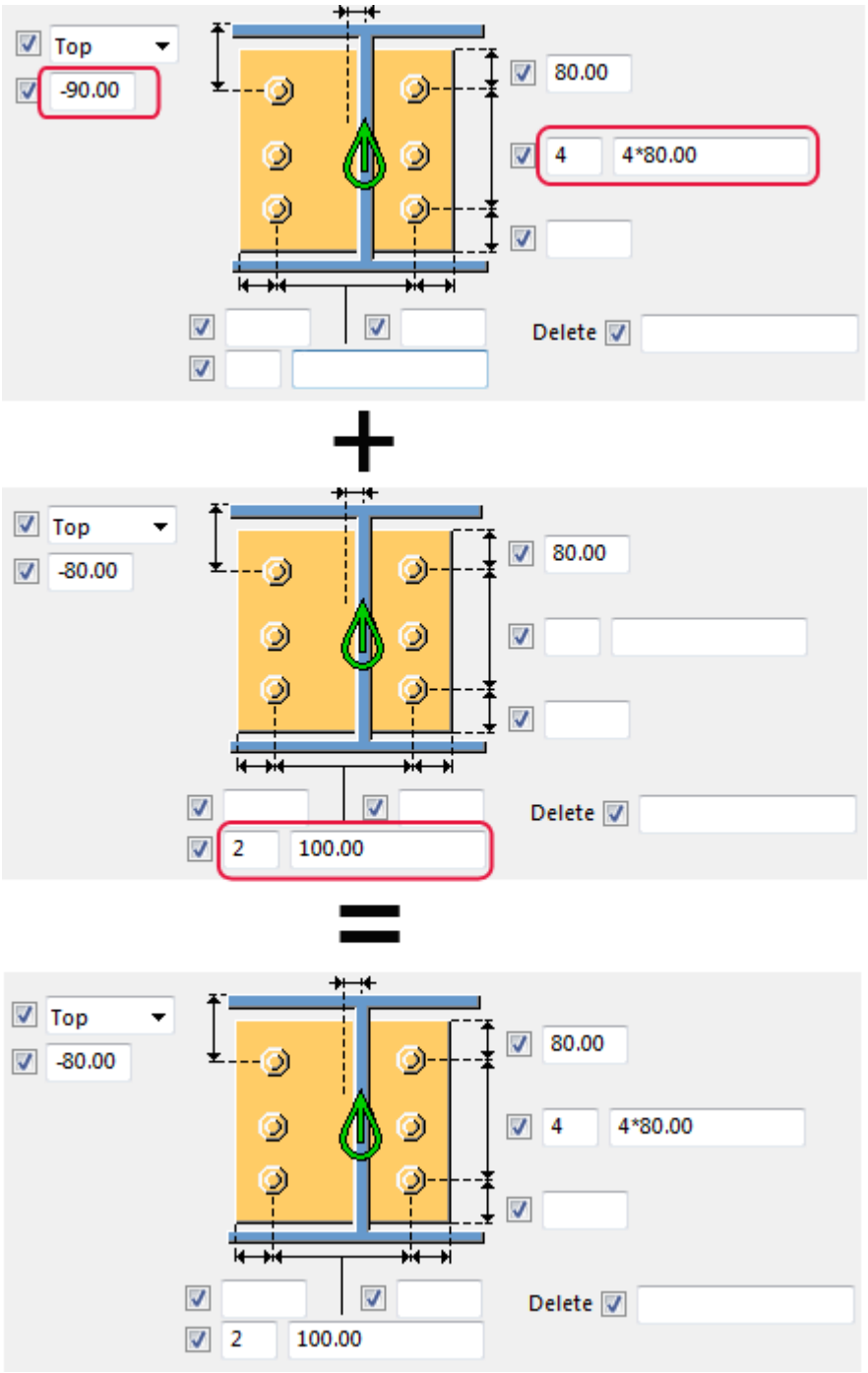
AutoDefaults example: Using iteration with connection check (page 836)

Using reaction forces and UDLs in AutoDefaults and AutoConnection (page 838)

Combining and iterating properties for AutoDefaults

Combining properties

You can save properties files that cover different groups of properties and use these files to define many rules. For example, you can have one file for bolt properties and another for profile properties. AutoDefaults combines the separate files into one file. This means that you can define fewer files because you can use one file for several rules. If the files contain different values for the same property, Tekla Structures uses the last property it finds, see the example image below.



Iterating properties

Tekla Structures tests the properties until the connection symbol is yellow or green. Iteration changes the properties automatically if the connection is not created successfully, even if the rules match. If connection check is active, the iteration results in properties that have passed the check.

Limitations

- Tekla Structures cannot iterate property files directly. Use a single iteration rule set with sub-rule sets.
- You cannot have many parallel iteration rule sets. Use a single iteration rule set and place it just before the default rule set.
- Place the combination rule sets above the iteration rule set in the AutoDefaults tree structure.
- Combination rule sets can only be one level deep.
- Tekla Structures disregards empty rule sets, so include at least one rule in each rule set.

See also

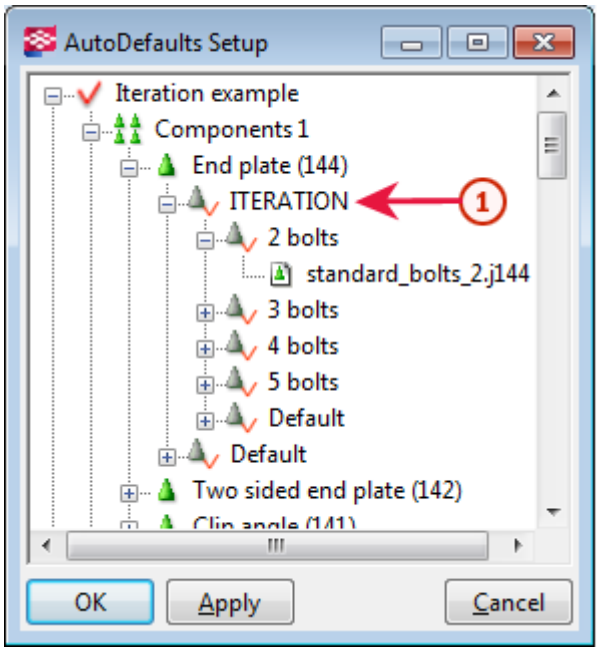
Define AutoDefaults settings and rules (page 826)

AutoDefaults example: Using iteration with connection check

You can use the connection check result when using AutoDefaults with iteration. If an iteration rule matches, but the connection does not pass the connection check and the connection symbol remains red, AutoDefaults continues testing other rules and properties until the connection symbol is green.

In this example, you will create iteration rules to set the number of bolts according to the result of the connection check. After this, you will use the

rules group and connection check together for a connection. The example image below shows the rules in the **AutoDefaults setup** dialog.



To create iteration rules for use with connection check:

1. On the **File** menu, click **Catalogs** --> **AutoDefaults settings**.
2. Right-click the tree and select **New Rule Group**.
3. Click the new rule group and rename it to `Iteration example`.
4. Browse the `Iteration example` tree to find **End plate (144)**, right-click it, and select **Create Additional Rule Sets**.
5. Right-click the **New** rule set and select **Edit Rule Set**.
6. Change the rule set name to `ITERATION`.
7. Set the **Parameter file(s) selection in rule set** option to **Iterate until connection symbol is green**.
8. Click **OK**.
9. Right-click the `ITERATION` rule set and select **Create Additional Rule Sets**.
10. Right-click the **New** rule set and select **Edit Rule Set**.
11. Change the rule set name to `2 bolts`.
12. Select the rule **Secondary 1 depth** and set the minimum and maximum depth values for two bolts.
13. Set the **Parameter file(s) selection in rule set** option to **Use combination of first parameters**.
14. Click **OK**.

15. Right-click the connection properties file `standard.j144` under `2 bolts` and select **Select Connection Parameters**.
16. Select a properties file for two bolts in the **Attribute File List** and click **OK**.

    TIP If there is no suitable properties file, you can create a new file. Right-click the `standard.j144` file and select **Edit Connection Parameters**. Save the needed properties and click **Cancel** to close the dialog. The saved properties are now available in the **Attribute File List**.

17. Click **Apply** to have the changes available in the connection dialog.
18. Repeat steps 9 to 16 for other rule sets.
19. Open the **End plate (144)** dialog.
20. Select `<Defaults>` from the list next to the **Load** button and click **Load**.
21. On the **General** tab, set the **AutoDefaults rule group** option to the `Iteration example` you created.
22. On the **Design type** tab, set the **Check connection** option to **Yes**.
23. Enter the load from secondary members in the **Shear**, **Tension**, and **Moment** options.
24. Click **OK**.

See also

Define AutoDefaults settings and rules (page 826)

Combining and iterating properties for AutoDefaults (page 834)

Using reaction forces and UDLs in AutoDefaults and AutoConnection

You can set reaction forces for AutoConnection and AutoDefaults in the user-defined attributes of a part, and for AutoDefaults also on the **Design** tab in the connection dialog.

Reaction forces

When you use reaction forces in a rule and AutoDefaults is activated, Tekla Structures first searches for reaction forces in the corresponding connection's properties. If the properties do not contain reaction forces, Tekla Structures searches the user-defined attributes of the secondary part of the connection. If Tekla Structures does not find forces there, you cannot use reaction force rules.

Shear force calculation

If you have not given any reaction force values, shear force is calculated using the UDL (uniformly distributed load) shear force routine. The UDL calculation

is mainly intended for use with imperial units. It uses the yield stress value, profile dimensions, and UDL percentage to calculate the maximum shear force allowed.

- Yield stress is defined in the material catalog.
- Profile dimensions come from the profile catalog.
- UDL percentage is taken either from the connection dialog or from an advanced option.

Tekla Structures compares the result with the shear force rule in AutoDefaults.

To use UDLs for AutoConnection and AutoDefaults:

To	Do this
Use UDL for AutoConnection	1. On the **Design** tab in the connection dialog, set the UDL option to **Yes**. 2. Enter the UDL percentage in the **UDL%** box. If you do not enter any value, Tekla Structures uses a default percentage set with the `XS_AUTODEFAULT_UDL_PERCENT` advanced option.
Use UDL for AutoDefaults	1. On the **Design** tab in the connection dialog, set the **Use UDL** option to **Yes**. 2. Enter the UDL percentage in the **UDL %** box. If you do not enter any value, Tekla Structures uses a default percentage set using the `XS_AUTODEFAULT_UDL_PERCENT` advanced option.

6.5 Advanced component settings

This section describes how to set default properties for different connection types, how to use Excel spreadsheets in connection design, and advanced component properties, such as analysis and design properties.

Click the links below to find out more:

Define connection properties in the joints.def file (page 839)

Excel spreadsheets in connection design (page 852)

Define connection properties in the joints.def file

The `joints.def` file contains general connection settings and connection-specific settings for different connection types. You can use the `joints.def`

file to set the default properties for different connection types. `Joints.def` is a text file that you can open and edit in any standard text editor.

Tekla Structures uses the values defined in the `joints.def` file for the properties that do not have values in the connection dialogs. If you manually enter values in the connection dialogs, the manually entered values are used instead of the values in the `joints.def` file. AutoDefaults also override the values defined in the `joints.def` file.

Tekla Structures stores the `joints.def` file in the system folder. Tekla Structures searches for the `joints.def` file in the standard search order: model, project, firm, and system folder.

How to use the joints.def file

The `joints.def` file contains general connection settings and connection-specific settings for different connection types in separate sections. You can modify the `joints.def` file using any standard text editor.

When you modify the file:

- Enter absolute values or names.
- Do not use feet and inch symbols.
- Ensure that the profiles exist in the profile catalog.
- Ensure that the bolts exist in the bolt catalog.
- You can set the measurement units at the beginning of the file.
- You can define in the `JOINTDEFAULT` line whether Tekla Structures uses the default values in the `joints.def` file or the system default values, for example, as follows:

```
// is default file available (1) or not (0)
JOINTDEFAULT  1
```

  - Value `1` means that the default values defined in the `joints.def` file are used.
  - Value `0` means that the system default values are used.
  - The `//` characters at the beginning of a line mean that the line is a comment line. Tekla Structures does not use the information on these lines.
- You can force Tekla Structures to use the system default for a particular property by entering the value `-2147483648` for the property.

Connection-specific properties

The properties for clip angles, shear tabs, end plates, gusset connections and diagonal connections are in separate sections. Each section begins with a header row that contains the column labels, for example as follows:

joints.def

```
// name            part     lproflength    diameter  number_of_bolts
BOLTHEIGHT         GUSSET   100            20.0      2
```

Do not add columns to the file. If Tekla Structures cannot find a property in the connection-specific section, it searches for the default property in the general defaults section.

Connections that use the joints.def file

The following connections use the `joints.def` file:

- **Welded gusset (10)**
- **Bolted gusset (11)**
- **Bracing cross (19)**
- **Tube gusset (20)**
- **Tube crossing (22)**
- **Two sided angle cleat (25)**
- **Corner tube gusset (56)**
- **Corner bolted gusset (57)**
- **Wraparound gusset (58)**
- **Hollow brace wraparound gusset (59)**
- **Wraparound gusset cross (60)**
- **Wrapped cross (61)**
- **Gusseted cross (62)**
- **Corner wrapped gusset (63)**
- **Beam with stiffener (129)**
- **Column with shear plate (131)**
- **Bolted moment connection (134)**
- **Clip angle (141)**
- **Two sided end plate (142)**
- **Two sided clip angle (143)**
- **End plate (144)**
- **Shear plate simple (146)**
- **Welded to top flange (147)**
- **Welded to top flange S (149)**
- **Moment connection (181)**
- **Column with stiffeners W (182)**
- **Full depth (184)**

- **Full depth S (185)**
- **Column with stiffeners (186)**
- **Column with stiffeners S (187)**
- **Column with stiffeners (188)**
- **Shear plate tube column (189)**
- **Bent plate (190)**

Example: How Tekla Structures uses the joints.def file

This example explains how Tekla Structures calculates the bolt diameter and other properties of the **Bolted gusset (11)** connection using the `joints.def` file.

The height of the diagonal profile is 10". Tekla Structures calculates the bolt size and the number of bolts according to the profile height. It searches the `BOLTHEIGHT` rows for a profile height of 10".

As the profile height is greater than `8.0` but under `12.0`, Tekla Structures uses the row with profile height `8.0`. This sets the bolt diameter to `0.75`.

```
// DIAGONAL JOINTS
// diagonal default boltdiameters depending on prof height, higher prior than
//
// name              part          profileheight     diameter  number_of_bolts
BOLTHEIGHT           DIAGONAL      3.0               0.75      1
BOLTHEIGHT           DIAGONAL      8.0               0.75      2
BOLTHEIGHT           DIAGONAL      12.0              0.75      3
BOLTHEIGHT           DIAGONAL      16.0              0.75      4
BOLTHEIGHT           DIAGONAL      18.0              0.75      5
```

Tekla Structures uses the bolt diameter to assign the bolt and part properties. It searches the `DIAGBOLTPART` rows for bolt diameter `0.75`.

```
//                bolt        angle         conn.plate  |      horizontal bolts         |  vertical bolts
// name           diameter    profile       thickness   number    pitch     edge_dist  number          pitch           edge
DIAGBOLTPART      0.5         L4X3X1/2      0.375       2         1.5       1.0        -2147483648     -2147483648     1.0
DIAGBOLTPART      0.75        L4X4X1/2      0.375       2         2.5       1.5        -2147483648     -2147483648     1.5
DIAGBOLTPART      1.0         L5X5X1/2      0.375       2         3.0       2.0        -2147483648     -2147483648     2.0
```

The following property values are used:

Bolt diameter	0.75
Number of bolts horizontally	2
Edge distance horizontally	1.5
Edge distance vertically	1.5
Distance between bolts horizontally	2.5

Distance between bolts vertically	System default is used.

Tekla Structures does not use the connection plate thickness or angle profile properties in this connection.

General defaults in the joints.def file

Tekla Structures uses the general defaults in the `joints.def` file if it cannot find a connection property in the connection-specific section.

For example, for clip angles, Tekla Structures determines the bolt diameter and the number of bolts according to the secondary beam height. If the secondary beam is higher than the highest value in the clip angle section in the `joints.def` file, Tekla Structures uses the default bolt diameter in the general defaults.

The properties in the general defaults section in the `joints.def` file are:

Property	Description
`boltdia`	Bolt diameter
`pitch`	Distance from the center of one bolt to the center of the next bolt
`clipweld`	Weld size
`angle-cc-inc`	Tekla Structures adds bolt to bolt distance and web thickness, then rounds up the result using this value. Complies with the US AISC standard.
`lprofgapinc`	Tekla Structures rounds up the angle profile gap using this value. Complies with the US AISC standard.
`lsize`	Size of the angle profile
`copedepth`	Notch size
`copelength`	Notch size
`boltedge`	Edge distance
`webplatelen`	Haunch plate height (h)
`webplatewid`	Haunch plate width (b)
`beamedge`	Setback distance between the end of the beam and the main part
`knifeclr`	No longer used
`clipedge`	Edge distance for bolts (clip angles only)
`gap`	No longer used
`shearplatethk`	Shear tab thickness
`endplatethk`	End plate thickness

Property	Description
shearweld	Size of weld
cliplsize	Size of angle profile (clip angles only)
flangecutclear	Flange cut clearance
slotsize	Size of slotted hole
clipslots	Part with slotted holes: • 1 = beam • 2 = angle profiles • 3 = both This property is the **Slots in** option on the **Bolts** tab.
clip_attac	Clip angle attached to the main part and secondary parts: • 1 = both parts bolted • 2 = main part bolted/secondary part welded • 3 = main part not welded • 4 = main part welded/secondary part bolted • 5 = both parts welded • 6 = main part not bolted • 7 = secondary part not welded • 8 = secondary part not bolted • 9 = both parts bolted/welded This property is the bolt attachment option on the **Bolts** tab where the location of bolts is defined.
copedepth_inc	Tekla Structures rounds up notch depth using this value.
copelength_inc	Tekla Structures rounds up notch length using this value.

Bolt diameter and number of bolts in the joints.def file

In the joints.def file, the BOLTHEIGHT rows in each connection-specific section show the default bolt diameter and default number of bolt rows for the connection type.

Tekla Structures determines the bolt diameter and the number of bolts according to the connection type based on the following properties:

For	According to
Clip angles	Secondary beam height
Shear tabs	Secondary beam height

For	According to
End plates	Secondary beam height
Gusset connections	Angle profile length
Diagonal connections	Profile height

Clip angle, shear tab, and end plate connections

Tekla Structures calculates the default bolt diameter and the number of bolt rows vertically according to the height of the secondary beam. You can enter the following properties:

Property	Description
`name`	`BOLTHEIGHT`
`part`	`ANGLECLIP`
`sec.beam.height`	Minimum height of the secondary beam for a certain number of bolts
`diameter`	Bolt diameter. The diameter must exist in the bolt catalog.
`number_of_bolts`	Number of bolts vertically

Gusset connections

Tekla Structures calculates the default bolt diameter and the number of bolt rows horizontally according to the length of the angle profile. You can enter the following properties:

Property	Description
`name`	`BOLTHEIGHT`
`part`	`GUSSET`
`lproflength` or `angleproflength`	Length of the angle profile
`diameter`	Bolt diameter. The diameter must exist in the bolt catalog.
`number_of_bolts`	Number of bolts horizontally

Diagonal connections

Tekla Structures calculates the default bolt diameter and number of bolt rows horizontally according to the profile height. You can enter the following properties:

Property	Description
`name`	`BOLTHEIGHT`
`part`	`DIAGONAL`

Property	Description
conn.pl.height or profileheight	Profile height
diameter	Bolt diameter. The diameter must exist in the bolt catalog.
number_of_bolts	Number of bolts horizontally

Bolt and part properties in the joints.def file

Once Tekla Structures has used the joints.def file to calculate the bolt diameter, it uses the result to assign other properties to bolts and parts, according to the connection type.

In clip angle connections, for example, the default properties for bolts and parts are in the rows that begin with ANGLECLBOLTPART in the CLIP ANGLE section of the joints.def file.

The table below lists the properties that you can assign for bolts and parts in each connection type.

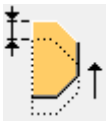
Property	Description	Clip angle	Shear tab	End plate	Gusset	Diagonal
name	Identifies the connection type. For example, GUSSETBOLTPART for gusset connections.	*	*	*	*	*
bolt diameter	The bolt diameter must exist in the bolt catalog.	*	*	*	*	*
shear plate thickness	Thickness of the shear tab		*			
end plate thickness	Thickness of the end plate			*		
gusset thickness	Thickness of the gusset plate				*	
conn. plate thickness	Thickness of the connection plate					*

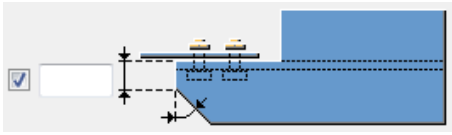
Property	Description	Clip angle	Shear tab	End plate	Gusset	Diagonal
angle profile or L profile	The name of the used angle profile must exist in the profile catalog. Enter the exact profile, for example: L100*100*10.	*			*	*
number	Number of bolts in each row vertically and horizontally.	*	*	*	*	*
pitch	Distance between the bolts from the center of each bolt for vertical and horizontal bolts	*	*	*	*	*
edge distance	Distance from the center of a bolt to the edge of the part for vertical and horizontal bolts	*	*		*	*
vert. bolt firshole	Position of the first vertical row of bolts	*	*		*	

Gusset connection properties in the joints.def file

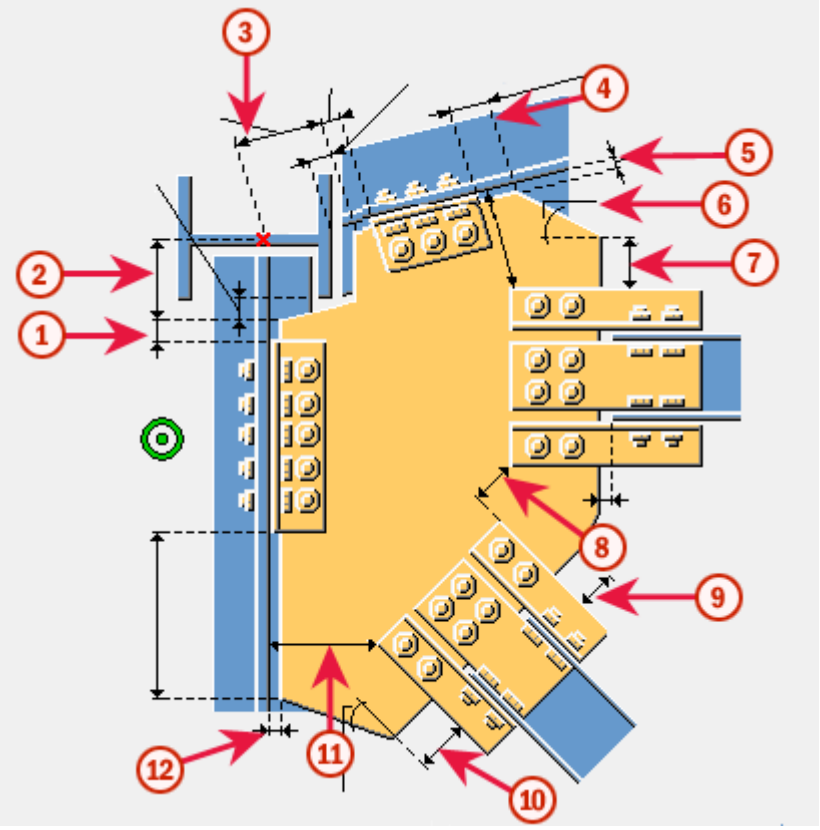
Enter the additional default properties for gusset connections in the GUSSETDEFDIM row . All gusset connections do not use all the properties.

Property	Description	Affects plate shape
name	GUSSETDEFDIM	
boltdia_def	Bolt diameter for all bolt groups Tekla Structures uses this value if the **Bolt size** box is empty in the connection dialog.	
tol_prim	Tolerance between the gusset and main part web	
tol_sec	Tolerance between the gusset and secondary part web	
dist_diag_prim	Clearance between the first secondary part selected and the main part	
dist_diag_sec	Perpendicular distance from the last secondary part selected to the nearest secondary part	

Property	Description	Affects plate shape
angle_first_corner	Corner angle dimension	Yes
angle_sec_corner		
dist_between_diag	Clearance between braces	
first_bolt_from_line	Bolt edge distance for the bolt groups on the **Gusset** tab	
corner_dx	Corner dimension	
corner_dy	Corner dimension	
movey	 option on the **Gusset** tab	
movez	 option on the **Gusset** tab	
dist1	Edge length of the gusset plate perpendicular to the lowest brace	Yes
dist2	Edge length of the gusset plate perpendicular to the braces	Yes
dist3	Edge length of the gusset plate perpendicular to the uppermost brace	Yes
tol_lprof	Edge tolerance from gusset plate to connection plate	
tol_stiffener	Stiffener tolerance	
chamfer_dx	Stiffener chamfer dimension on the **Gusset** tab	
chamfer_dy	Stiffener chamfer dimension on the **Gusset** tab	
chamfer_corner_dx		
chamfer_corner_dy		
side_length	Side length	
diafit_length	Fit length in the **Bracing cross (19)** connection.	

Property	Description	Affects plate shape
	Tekla Structures uses this value if the option on the **Parameters** tab is empty. 	

The example image below shows the properties of the **Wraparound gusset (58)** connection on the **Picture** tab.

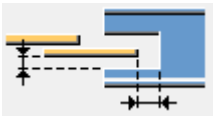
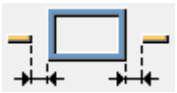
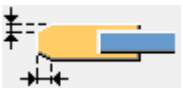


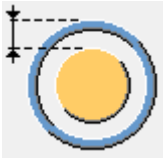
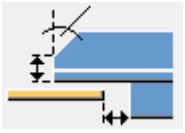
1. `tol_lprof`
2. `corner_dy`
3. `corner_dx`
4. `dist_diag_sec`
5. `tol_sec`
6. `angle_sec_corner`
7. `dist3`
8. `dist_between_diag`

9. `dist2`
10. `dist1`
11. `dist_diag_prim`
12. `tol_prim`

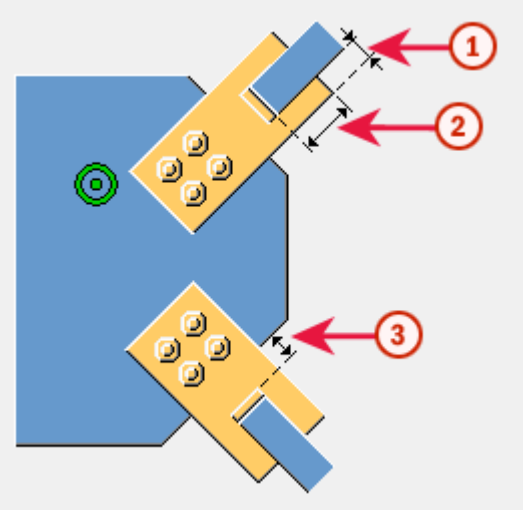
Diagonal connection properties in the joints.def file

Enter the additional default properties for bolts and parts in the `DIAGDEFDIM` row. All diagonal connections do not use all the properties.

Property	Description
`name`	`DIAGDEFDIM`
`boltdia_def`	Bolt diameter for all bolt groups Tekla Structures uses this value if the **Bolt size** box is empty in the connection dialog.
`dist_gus_diag`	Gap between the gusset plate and the brace If the tube profiles are closed with end plates, `dist_gus_diag` is the gap between the gusset plate and the end plate. See the **Tube crossing (22)** image below.
`dist_in`	Cut depth in the brace. Enter a negative value to prevent the connection plate from being inside the tube brace. See the **Tube crossing (22)** image below.
`dist_dv`	Brace edge distance to the edge of the connection plate. This dimension changes the width of the connection plate. See the **Tube crossing (22)** image below.
`sec_cut_tol` `slot_length_tol`	On the **Brace conn** tab: 
`tube_cut_tol`	On the **Brace conn** tab: 
`conn_cut_dx` `conn_cut_dy`	On the **Brace conn** tab: 
`round_plate_tol`	On the **Brace conn** tab:

Property	Description
	
flanges_cut_angle	On the **Brace conn** tab: 
dist_flanges_cut	
dist_skew_cut	
end_plate_thk	End plate thickness

The example image below shows the properties of the **Tube crossing (22)** connection on the **Picture** tab:

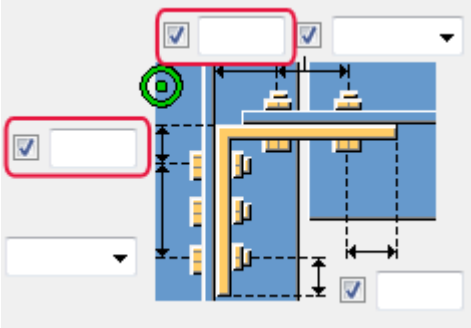


1. dist_dv
2. dist_in
3. dist_gus_diag

Profile dependent bolt dimensions in the joints.def file

For some connections, such as **Clip angle (141)** and **Two sided clip angle (143)**, Tekla Structures calculates the bolt size according to the profile size.

For these connections, Tekla Structures takes the bolt size from the PROFILEBOLTDIM rows of the PROFILE TYPE-DEPENDENT BOLT DIMENSIONS section in the joints.def file if you leave the corresponding options empty on the **Bolts** tab.



Property	Description
`width`	Profile width
`one bolt firsthole`	For single bolts, distance from the edge of the profile angle to the first hole
`two bolts firsthole`	For two bolts, distance from the edge of the profile angle to the first hole
`pitch`	Distance between bolts from the center of each bolt, for vertical and horizontal bolts

For example, to find the bolt dimensions to be used with an `L6X6X1/2` profile in a clip angle connection:

1. Tekla Structures first searches the `PROFILEBOLTDIM` rows for `L6X6X1/2` in the `PROFILE TYPE-DEPENDENT BOLT DIMENSIONS` section.
2. If there is no match, Tekla Structures then searches the `ANGLECLBOLTPART` rows in the `CLIP ANGLE` section.

Excel spreadsheets in connection design

You can use Excel spreadsheets in connection design for all steel connections that have the **Design** or **Design type** tab in the connection dialog.

You can link connections to Excel spreadsheets by selecting **Excel** in the **External design** option on the **Design** or **Design type** tab. The connection information is transferred to the connection type-specific spreadsheet where the needed calculations are made. The calculated properties are saved to an output file and the modified component property values are transferred back to the connection. The connection is modified according to the changes.

You can create an Excel spreadsheet for a connection type using the `component_template.xls` file available in the `..\Tekla Structures\<version>\Environments\common\exceldesign` folder, or use a predefined file.

Files used in Excel spreadsheet connection design

The following files are used in the connection design with Excel spreadsheets:

File	Description
Visual Basic script file	The `Excel.vb` file links Tekla Structures with the external software and defines the Excel spreadsheet file names and the locations. The file is located in the `..\Tekla Structures\<version>\Environments\common\exceldesign` folder.
Component type-specific Excel spreadsheet	The component type-specific spreadsheet contains predefined calculations. When you run the connection design, the connection properties and information of the main and secondary parts are transferred to the **Input** and **Component** sheets of the Excel spreadsheet. Excel searches for the relevant spreadsheet file in the following order: 1. From the `\exceldesign` folder in the current model folder: file named as `component_` + `number or name` + `.xls`, for example, `..\test_model\exceldesign\component_144.xls`. 2. From the location defined with the `XS_EXTERNAL_EXCEL_DESIGN_PATH` advanced option as follows: `XS_EXTERNAL_EXCEL_DESIGN_PATH (=%XSDATADIR% \environments\common\exceldesign\) + "component_" + number + ".xls"`
Connection specific result file	The result file contains the modified connection properties. • The result file is created automatically from the **Calculation** sheet. • The file is stored in the `\exceldesign` folder in the model folder and named with the GUID, Globally Unique Identifier. • The file is updated each time you modify the connection. • The calculation results can be stored as an Excel spreadsheet, or in HTML or PDF format, depending on how the calculation spreadsheet is configured.

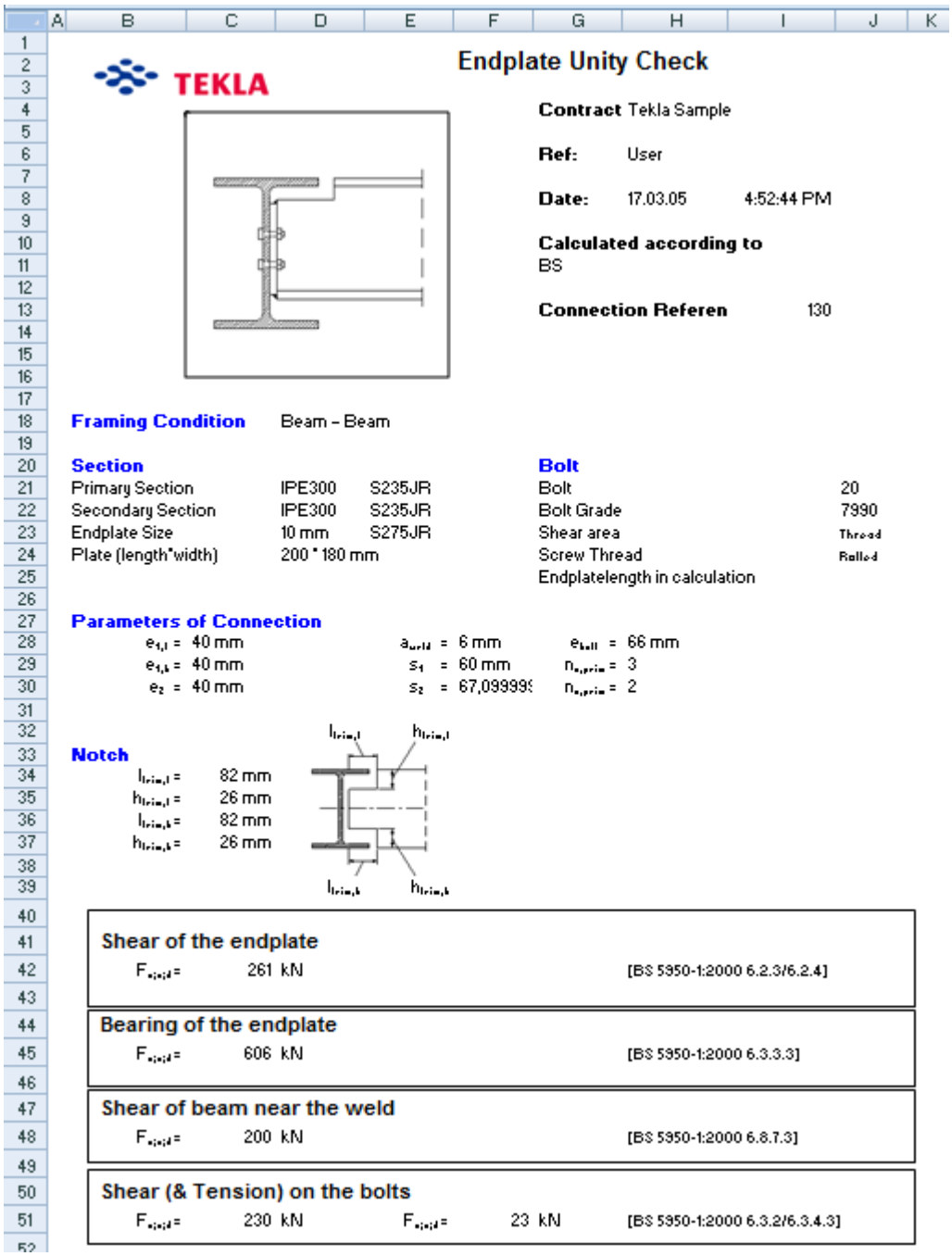
File	Description
Template spreadsheet	The `..\Tekla Structures\<version>\Environments\common\exceldesign` folder contains a `component_template.xls` spreadsheet you should use to create your own spreadsheet applications to use with Tekla Structures components.

Example of an Excel spreadsheet in connection design

The images in this example show the Excel spreadsheet that is used for the **End plate (144)** connection.

The sample spreadsheet has the following sheets:

The **Calculation** sheet contains a report of the calculations.



TEKLA

Endplate Unity Check

Contract Tekla Sample

Ref: User

Date: 17.03.05 4:52:44 PM

Calculated according to
BS

Connection Referen 130

Framing Condition Beam – Beam

Section

Primary Section	IPE300	S235JR
Secondary Section	IPE300	S235JR
Endplate Size	10 mm	S275JR
Plate (length*width)	200 * 180 mm	

Bolt

Bolt	20
Bolt Grade	7990
Shear area	Thread
Screw Thread	Rolled
Endplatelength in calculation	

Parameters of Connection

$e_{1,t}$ =	40 mm	a_{weld} =	6 mm	e_{bolt} =	66 mm
$e_{1,b}$ =	40 mm	s_1 =	60 mm	$n_{x,prim}$ =	3
e_2 =	40 mm	s_2 =	67,09999	$n_{z,prim}$ =	2

Notch

$l_{trim,t}$ =	82 mm
$h_{trim,t}$ =	26 mm
$l_{trim,b}$ =	82 mm
$h_{trim,b}$ =	26 mm

Shear of the endplate

$F_{x,t,d}$ = 261 kN [BS 5950-1:2000 6.2.3/6.2.4]

Bearing of the endplate

$F_{x,t,d}$ = 606 kN [BS 5950-1:2000 6.3.3.3]

Shear of beam near the weld

$F_{x,t,d}$ = 200 kN [BS 5950-1:2000 6.8.7.3]

Shear (& Tension) on the bolts

$F_{x,t,d}$ = 230 kN $F_{x,t,d}$ = 23 kN [BS 5950-1:2000 6.3.2/6.3.4.3]

The **Inputs** sheet contains the properties of the connection from the connection dialog.

	A	B	C	D	E	F
1				**Attribute**	**Value**	**Type**
2		**Plate**				
3			Material	mat		string
4			Thickness	tpl1	10	double
5			Depth	hpl1	-2147483648	double
6			Width	bpl1	180	double
7						
8		**Bolt**				
9			Diameter	diameter		double
10			Grade	screwdin		string
11				lbd	-2147483648	string
12				lwd	-2147483648	string
13				lba	-2147483648	double
14				nb	-2147483648	int
15				nw	-2147483648	int
16				rb1	-2147483648	double
17				rb2	-2147483648	double
18				rw1	-2147483648	double
19				rw2	-2147483648	double
20						
21		**Weld**				
22				w3_size	-2147483648	double
23						
24		**Notch**				
25				t_cut_length	-2147483648	double
26				t_cope_length	-2147483648	double
27				b_cut_length	-2147483648	double
28				b_cope_depth	-2147483648	double
29						
30		**Loading**				
31				designcode	0	int
32				**END**		

The **Outputs** sheet contains the design results. These values are transferred to the connection and the connection in the model is modified accordingly.

The **Component** sheet contains calculations, information on the connection geometry, and on the main part and the secondary parts. The component attributes in the spreadsheet are the same as in the corresponding `.inp` file. See more about `.inp` files in .

	A	B	C	D
1	**Connection**	**Attribute**	**Value**	
2	Connection id in model	id	130	
3	Connection class	group	99	
4		flags	50	
5	Number of the connection	jointnumber	144	
6	Local x-coordinate of Connection up directio	up.x	0	
7	Local y-coordinate of Connection up directio	up.y	0	
8	Local z-coordinate of Connection up directio	up.z	1000	
9	Model Directory	ModelDirectory	C:\TeklaStructuresModels\	
10		END		
11			**Primary**	**Secondaries**
12		attribute	value	value 1
13	Primary and secondary ids	id	108	70
14	**PartCoordinateSystem**	x.x	-9,11626E-13	6000
15	y-coordinate of part origin (first end) point	x.y	8000	-9,13758E-13
16	z-coordinate of part origin (first end)point	x.z	-150	-150
17	x-coordinate of second end point of part	y.x	12000	6000
18	y-coordinate of second end point of part	y.y	8000	8000
19	z-coordinate of second end point of part	y.z	-150	-150
20	x-coordinate of parts up direction point	z.x	-9,11626E-13	6000
21	y-coordinate of parts up direction point	z.y	8000	-9,13758E-13
22	z-coordinate of parts up direction point	z.z	850	850
23	**PartExtrema**			
24	Minimum x value of primary or secondary pa	min.x	-9,11626E-13	5925
25	Minimum y value of primary or secondary pa	min.y	7925	-9,13758E-13
26	Minimum z value of primary or secondary pa	min.z	-300	-300
27	Maximum x value of extrema	max.x	12000	6075
28	Maximum y value of extrema	max.y	8075	8000
29	Maximum z value of extrema	max.z	0	0
30	**FramingCondition**			
31	Member type (Column, Beam)	Type	1	1
32	Profile name	Name	COLUMN	BEAM
33	Profile type	ProfileType	1	1
34	Skew angle between primary/sec	SkewAngle		0
35	Slope angle between primary/sec	SlopeAngle		0
36	Cantilever angle between primary/sec	AngleCant		90
37		Offset		0
38	Shear force at connection end of the beam	ShearForce		-2147483648
39	Axial force at connection end of the beam	AxialForce		-2147483648
40	Moment at connection end of the beam	BendingMoment		-2147483648
41	Use uniformly distributed load	UseUDL		0
42	How many percents from maximum uniformly	UDLPercent		0

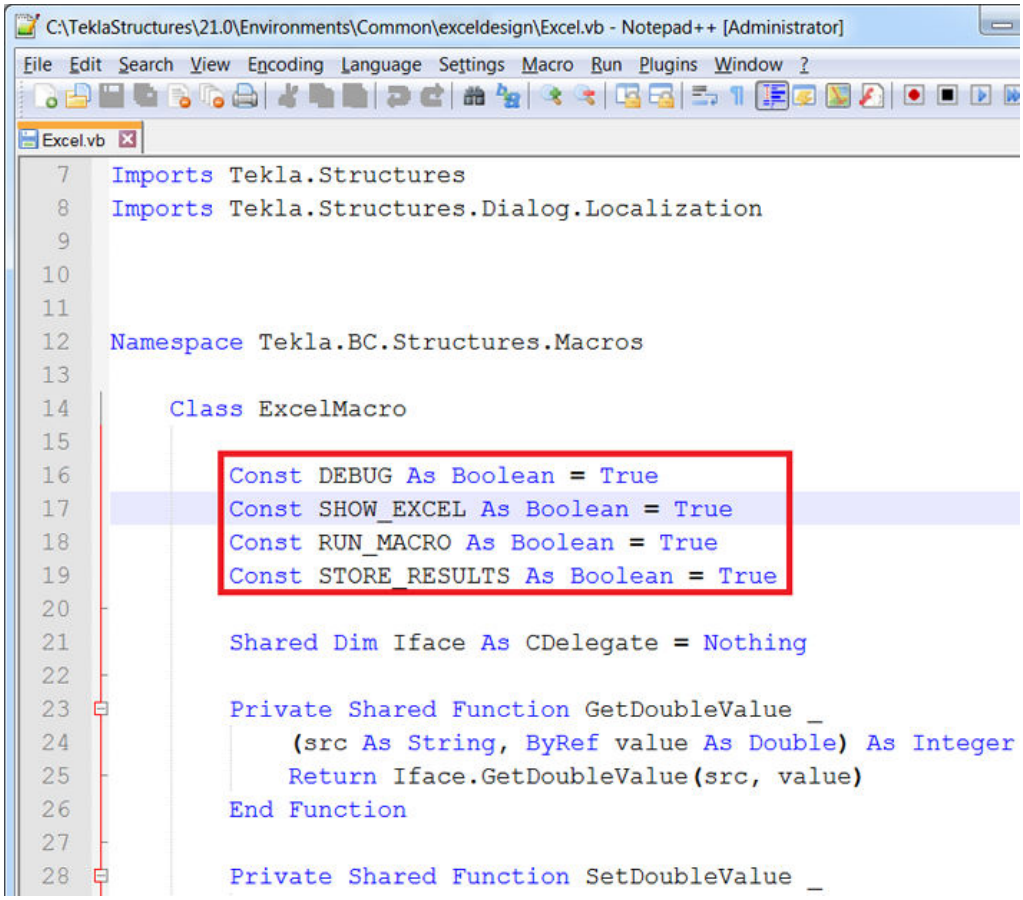
The following sheets are also included:

- **Data** shows catalog information.
- **Norm** shows the code selection and references to codes.
- **Language** shows translations.

Example of visualizing the Excel connection design process

You can define in the `Excel.vb` file how the Excel connection design process is visualized. The `Excel.vb` file links Tekla Structures with the external software and defines the Excel spreadsheet file names and the locations.

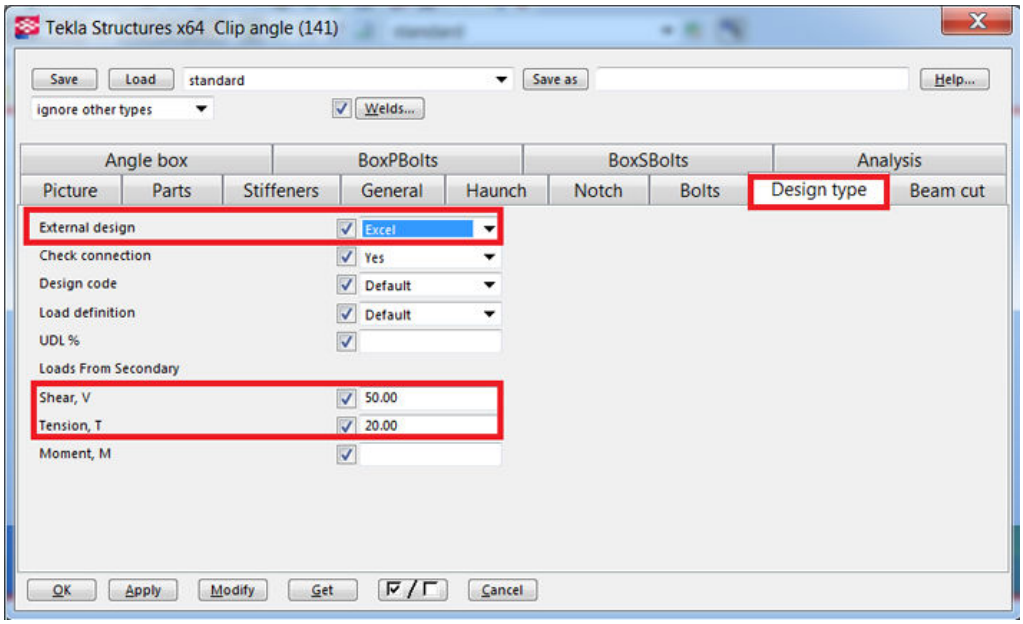
1. Open the `Excel.vb` file located in `..\Tekla Structures\<version>\Environments\common\exceldesign`.
2. Configure the `Excel.vb` file as follows:



   - Visualizing control - `Const DEBUG As Boolean= True`
   - Visualizing using Excel - `Const SHOW_EXCEL As Boolean = True`
   - Storing the output - `Const STORE_RESULTS As Boolean = True`
3. Save the file.
4. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
5. Search for **Clip angle (141)** and double-click it to open the properties dialog.
6. On the **Design type** tab:

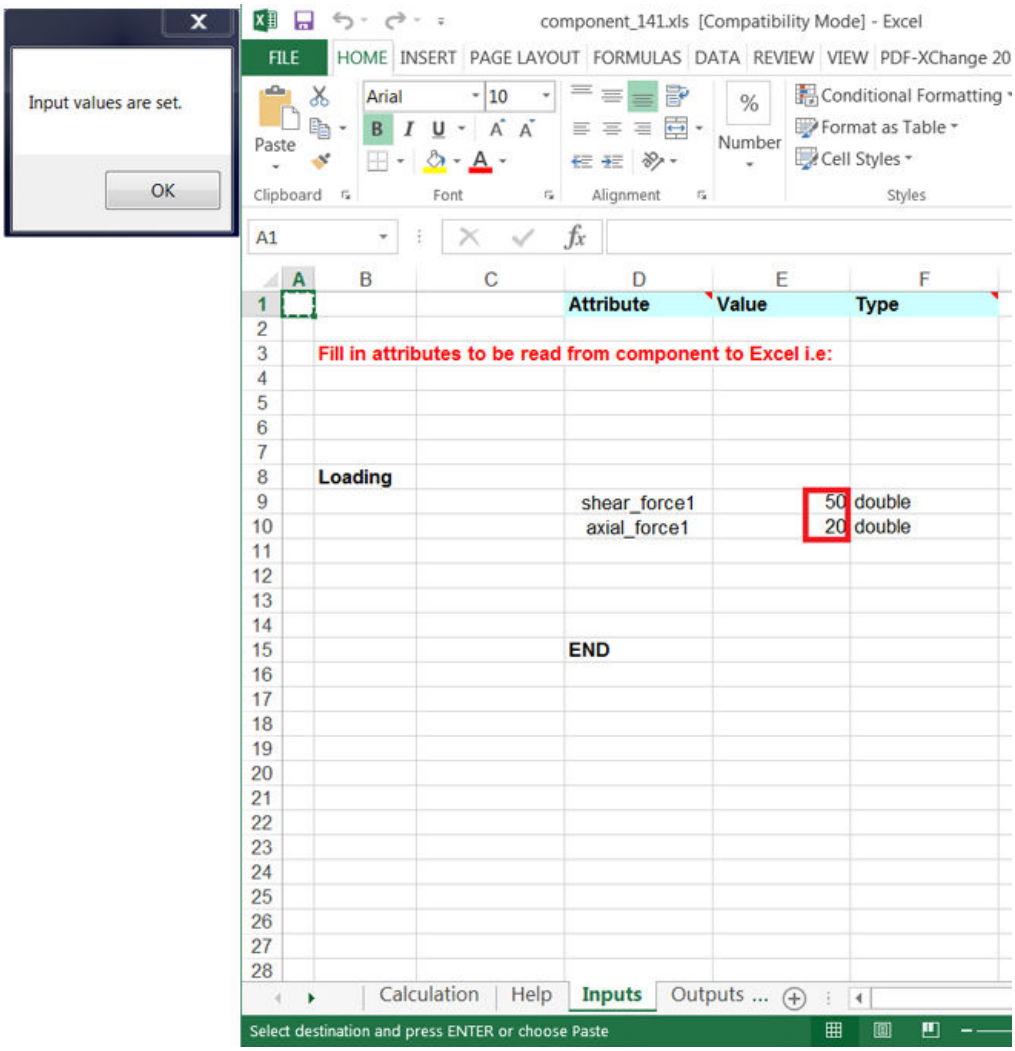
a. Select **Excel** in the **External design** option.

b. Enter the load values.



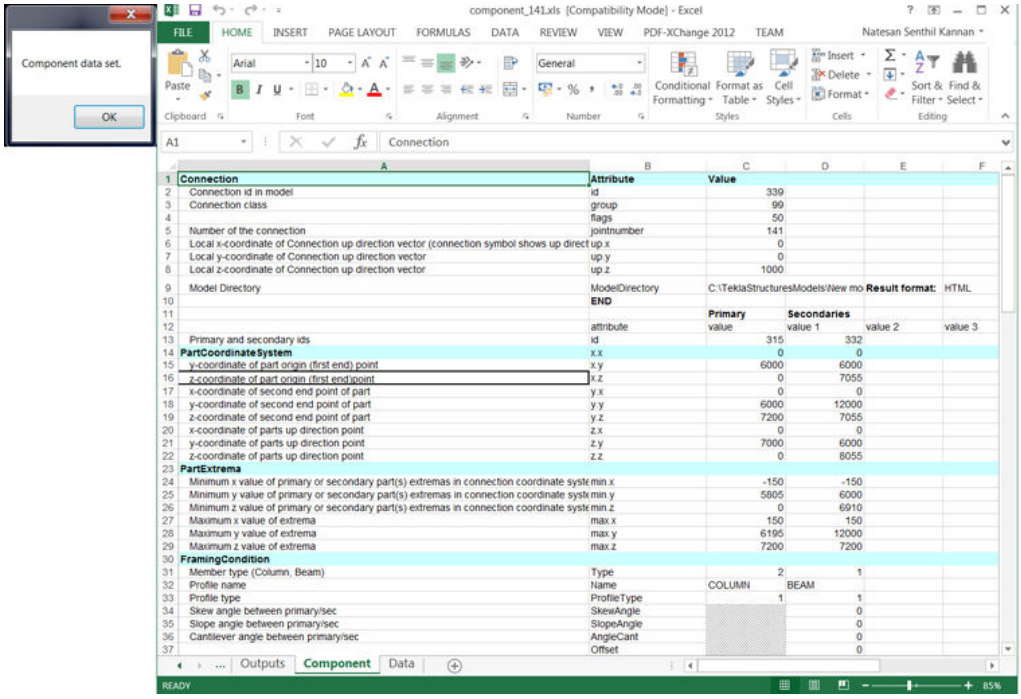
7. Click **Modify**.

The Excel design file opens and shows the **Inputs** sheet.



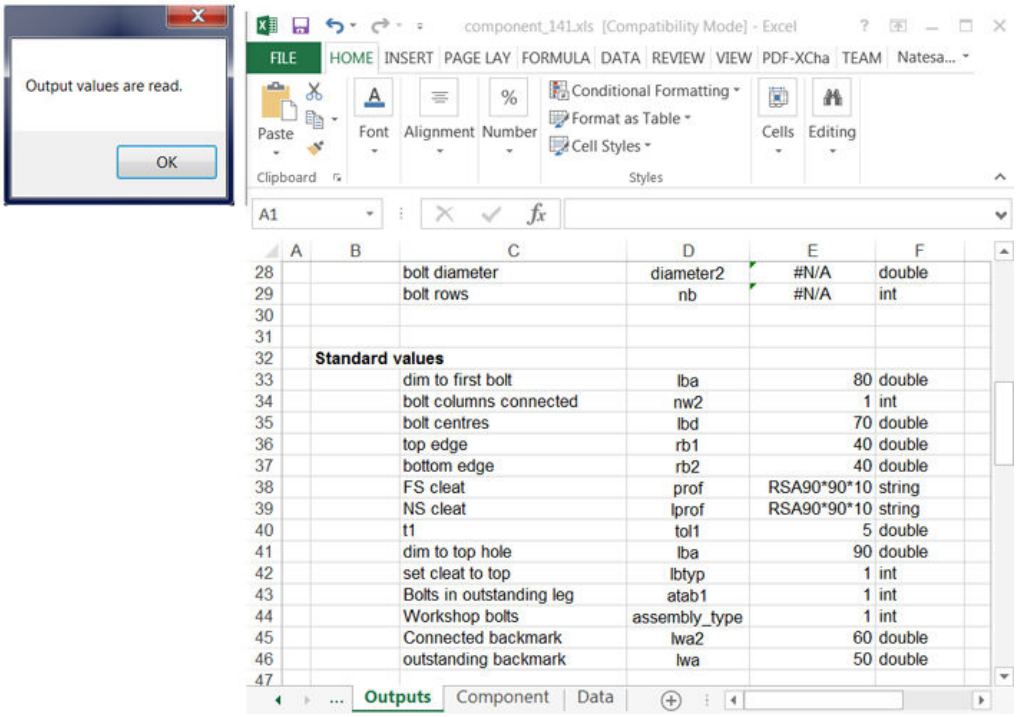
8. Click **OK** to continue.

The Excel design process now calculates the data that is shown on the **Component** sheet, and then opens the **Component** sheet.



9. Click **OK** to continue.

   The Excel design process now calculates the result output values and then opens the **Outputs** sheet. The calculated results values are transferred to the connection.



	A	B	C	D	E	F
28			bolt diameter	diameter2	#N/A	double
29			bolt rows	nb	#N/A	int
30						
31						
32		**Standard values**				
33			dim to first bolt	lba	80	double
34			bolt columns connected	nw2	1	int
35			bolt centres	lbd	70	double
36			top edge	rb1	40	double
37			bottom edge	rb2	40	double
38			FS cleat	prof	RSA90*90*10	string
39			NS cleat	lprof	RSA90*90*10	string
40			t1	tol1	5	double
41			dim to top hole	lba	90	double
42			set cleat to top	lbtyp	1	int
43			Bolts in outstanding leg	atab1	1	int
44			Workshop bolts	assembly_type	1	int
45			Connected backmark	lwa2	60	double
46			outstanding backmark	lwa	50	double
47						

10. Save the file in the model folder.
11. When you click **OK**, the design process is completed and the Excel design file is closed.

Showing connection status in Excel connection design

When you use Excel spreadsheets in connection design, you can have Tekla Structures use different colors in component symbols to show the status of a component in the model.

You can do this by including an error attribute on the **Outputs** sheet of the component's Excel spreadsheet. The type of the attribute is `int`.

The possible values are:

Value	Color	Status
1	Green	Bolt edge distances are sufficient. The connection passes the connection design check using the UK and US design codes embedded in the system.
2	Yellow	Bolt edge distances are insufficient according to the value defined in the **Components** settings in **File** --> **Settings** --> **Options**.
3	Red	Tekla Structures cannot calculate the component properties. The possible reasons are: • The connection direction is not correct. • The work plane is not correct. • The selected connection is not appropriate for the situation. • The connection design check was carried out using the embedded UK and US design codes and the connection cannot support the loading you have defined.

NOTE The component symbol color can only be controlled for system components, not for custom components.

7 Custom components

You can define customized connections, seams, details, and parts for your project. These are called *custom components*. You can use custom components in the same way as any Tekla Structures system component. By editing the custom components you can create parametric custom components that automatically adjust to changes in the model.

When to use

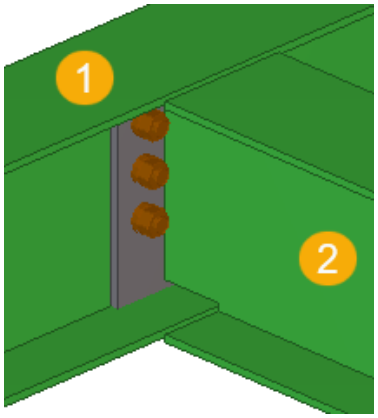
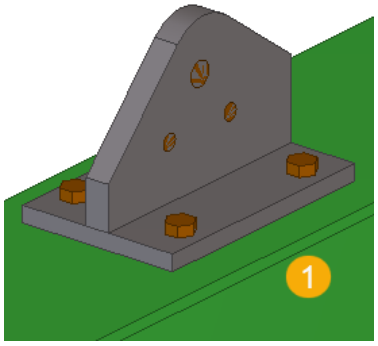
Define a custom component if you cannot find a predefined system component (page 803) that meets all your needs. Custom components are especially useful if you need to create a large number of complex model objects and copy them across several projects.

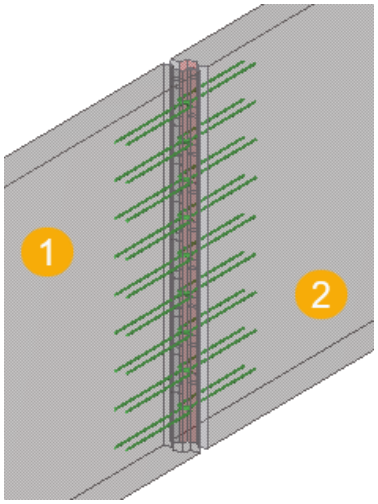
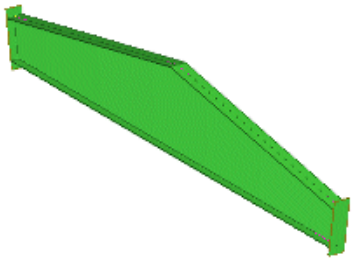
Benefits

Once you define and store a custom component in the **Applications & components** catalog, you can easily access it from the catalog and use it in another location in the same model. If you need to modify the custom component, you only need to make the changes once. When you save the changes, they will be automatically applied to all copies of that custom component in the model. You can also import and export custom components as `.uel` files between models and share the custom components with your colleagues, or store the custom components in a model template (page 211) so that they are available with each new model that is based on the used template.

Custom component types

You can create four types of custom components:

Type	Description	Example
Custom connection	Custom connections create connection objects that connect the secondary parts to the main part. The main part may be continuous at the connection point. Custom connections can be used to connect up to 30 secondary parts to the main part. The connection is made between the main part and the ends of the secondary parts. The component symbol is green.  **(1)**: Main part **(2)**: Secondary part	
Custom detail	Custom details create objects and connect them to the main part at the location you have picked. Custom details can be used to add more information to a single part, such as extra plates or cut-outs. The component symbol is green.  **(1)**: Main part	

Type	Description	Example
Custom seam	Custom seams create seam objects and connect the parts along a line defined by two picked points. The parts are typically parallel, and the seam is created along the length of the part. Custom seams can be used to connect up to 30 secondary parts to the main part, or they can be used on one main part only. The component symbol is green.  **(1)**: Main part **(2)**: Secondary part	
Custom part	Custom parts create a group of objects that may contain connections and details. Custom parts may consists of a single part or a group of parts, and they often have a complex composition. **Note:** Unlike other custom components, custom parts are **not** marked with a component symbol  in the model. Custom parts have the same position properties as beams.	

See also

Define custom components (page 866)

Edit custom components (page 887)

Add variables to a custom component (page 893)

Examples of parametric variables and variable formulas in custom components (page 934)

7.1 Define custom components

You can define customized components that have all the details you need.

Start by defining a simple custom component which you can modify later. Defining a simple custom component typically takes only few minutes. You can invest more time in defining your custom components if you are planning to use them in future projects.

By further editing custom components, you can define self-adjusting, parametric custom components that automatically adjust to changes in the model. This is more time-consuming but can be beneficial later when you have a group of parametric custom components which you can use across multiple models or projects.

To create your own components:

1. Start by applying a similar system component in the model and then explode it.
2. Once the objects are detached, use the objects to define the desired custom component in the **Custom component wizard**.
3. Use the newly created custom component in a suitable situation in the model.

See below for detailed instructions.

Explode an existing component

When you start defining a custom component, we recommend that you first apply a similar system component in the model and then explode it. Exploding means that you ungroup the objects of an existing component. Once the objects are detached, you can modify, remove or add objects to suit your needs, and then create new custom components using these objects. Exploding a component and using the detached objects as a basis for a new custom component can be useful when you want to create custom components more quickly.

Alternatively, you can individually create the component objects that are needed in a custom component, such as parts, cuts, fittings, and bolts.

1. In a Tekla Structures model, select the component you want to explode.
2. Right-click and select **Explode component**.

   Tekla Structures separates the component objects. You can modify the objects and use them when you define new custom components.

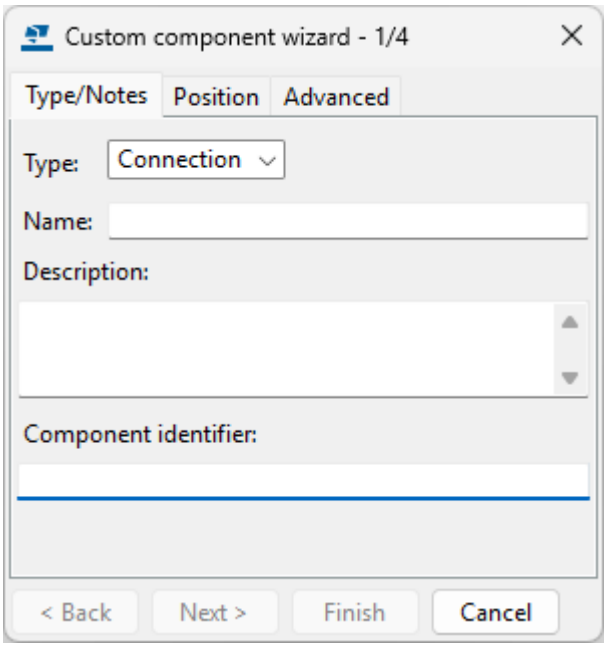
Define a custom component in the Custom component wizard

The following example shows how to define a simple custom connection.

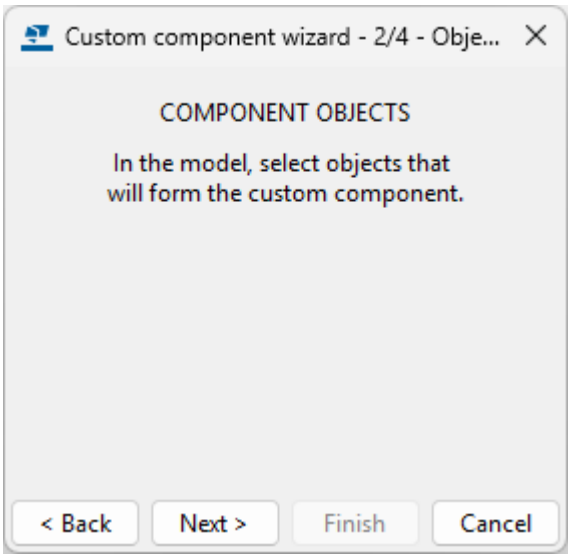
1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

2. Click the **Access advanced features** button and select **Define custom component...**.

   The **Custom component wizard** dialog opens.



3. In the **Type** list, select the component type: connection, detail, seam, or part.
4. In the **Name** box, enter a unique name for the component.
5. Modify the other properties on the **Type/Notes** tab, **Position** tab, and **Advanced** tab, and then click **Next >**.
6. In the model, select the objects you want to include in the custom component.



You can use area selection to select multiple objects simultaneously. The main parts, secondary parts and grids are ignored when you select objects for the custom component.

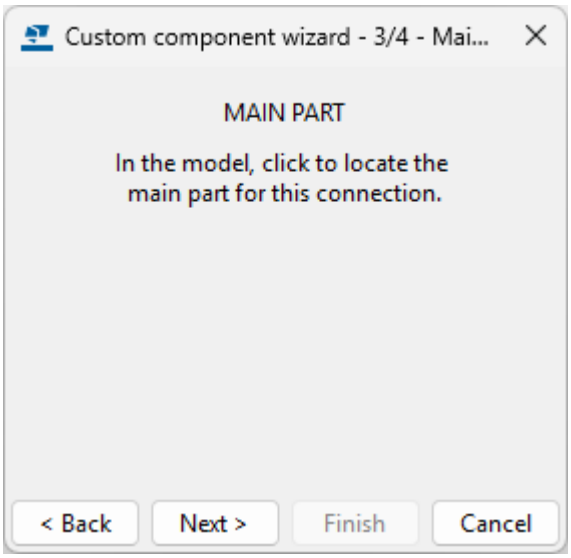
NOTE If you cannot select desired objects in the model, check the selection switches and the selection filter settings (page 171).

If you want to include rebar set modifiers (page 557) in the custom component:

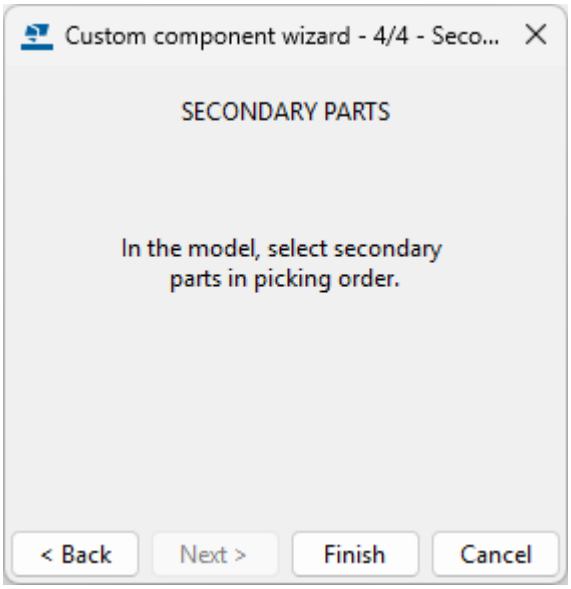
- Ensure that **Direct modification** is switched off when you select the modifiers.
- Hold down **Shift** and select the modifiers one by one. Area selection does not select modifiers.

---

7. Click **Next >**.
8. Select the main part for the component.



9. Click **Next >**.
10. Select the secondary parts for the component.



To select multiple secondary parts, hold down **Shift** when you select them. The maximum number of secondary parts in a custom component is 30.

NOTE Pay attention to the order in which you select secondary parts. Tekla Structures will use the same picking order when you use the custom component in a model.

---

11. Define any other properties required for this custom component, such as detail or seam position.

    The properties depend on the component type that you selected in step 3.

12. If you want to adjust any of the settings at this stage, click **< Back** to return to the previous page of the **Custom component wizard**.
13. When you are happy with the settings, click **Finish** to create the custom component.

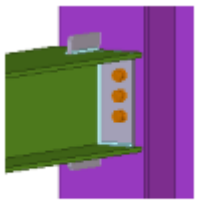
    The custom component is added in the model and in the **Applications & components** catalog.
14. If needed, add a thumbnail image for the custom component.

    The thumbnail image is displayed in the **Applications & components** catalog. In the thumbnail image you can show a typical situation where the component can be used.

    a. Take a screenshot of the custom component.

    b. Add a thumbnail image (page 817) of the custom component in the **Applications & components** catalog.

    Tekla Structures shows the thumbnail image in the **Applications & components** catalog:



15. If you want to modify custom component settings afterwards:

    a. On the **Custom component editor** toolbar, click the **Custom component settings** button.

    b. Change the settings.

    c. Click **OK**.

The defined custom component does not automatically adjust dimensions to reflect the changes in the model. To have the component adapt to changes in the model, edit the component in the custom component editor, where you can build dependencies between component objects and model objects.

Add a custom component to a model

1. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
2. To search for a component, browse the catalog or enter a search term in the search box.

Custom components have the following symbols in the catalog:

Type	Symbol
Custom part	
Custom connection or seam	
Custom detail	

3. Select the custom component you want to add.
4. Follow the instructions on the status bar to add the custom component in the model.
5. To modify the properties, double-click the custom component in the model.

Properties in the Custom component wizard

Use these properties when you define new custom components in the **Custom component wizard**. You can change some of these properties when you modify an existing custom component.

Type/Notes tab

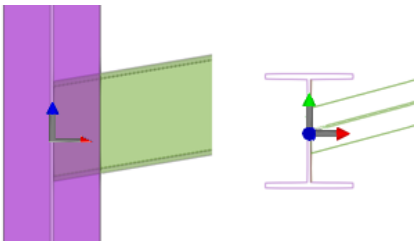
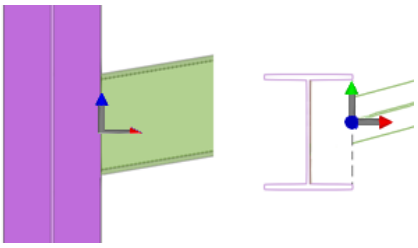
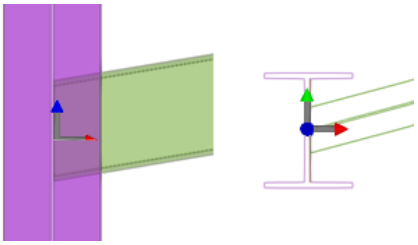
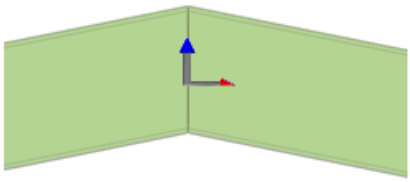
Option	Description
Type	Select the type of the custom component. Type affects how you insert the custom component in the model. Type also defines if the custom component connects to existing parts.
Name	Enter a unique name for the custom component.
Description	Enter a short description for the custom component. Tekla Structures shows the description in the **Applications & components** catalog.
Component identifier	Enter an additional name or reference for the component, for example a design code reference. This can be shown in general arrangement and assembly drawings, and in lists. To show this in drawings, include **Code** in the connection mark.

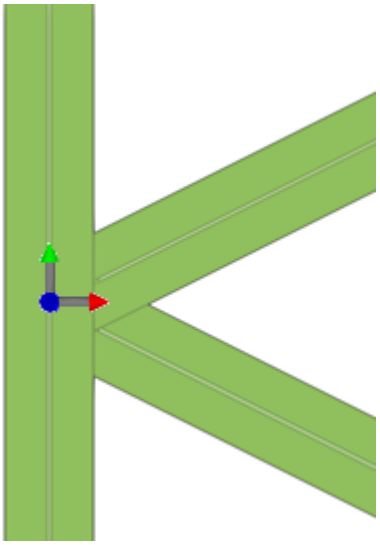
Position tab

Option	Description	Note
Up direction	Sets the default up direction.	Not available for parts.

Option	Description	Note
Position type	The position (or origin) of the component, relative to the main part.	Not available for details and parts.

You can define the position type for custom connections and seams. Position type determines the location of the objects that the custom component creates, relative to the main part. Position type affects which type of main parts you can use the custom component with, and defines the main part component plane used in bindings.

Position type	Description	Example
Middle	Where the center lines of the main and secondary parts intersect.	
Box plane	Where the main part bounding box and the center line of the secondary part intersect.	
Collision plane	Where the main part and the center line of the secondary part intersect.	
Endend plane	Where the center line of the secondary part hits the end of the main part.	

Position type	Description	Example
Gusset plane	Where the center lines of the main part and the first secondary part intersect. The x direction is perpendicular to the center line of the main part.	

Advanced tab

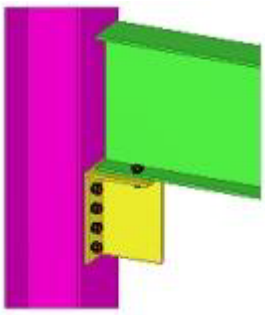
Option	Description	Note
Detail type	Determines on which side of the main part the component is located. The options are: • **Intermediate detail** Tekla Structures creates all components on the same side of the main part • **End detail** Tekla Structures creates all components on the side of the main part closest to the details Only affects asymmetric components.	Only available for details and seams
Definition point position in relation to primary part	Determines the position you pick to create the detail, relative to the main part.	Only available for details

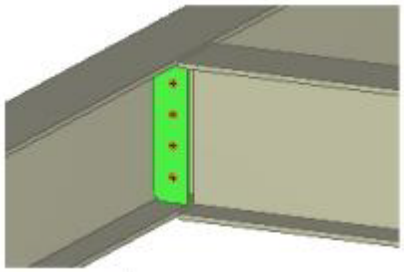
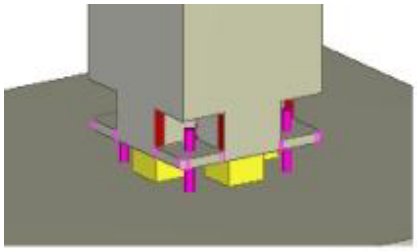
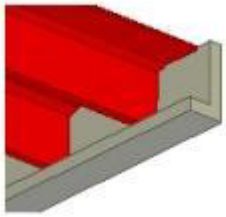
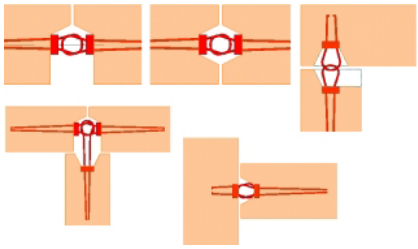
Option	Description	Note
Definition point position in relation to secondary part	Determines where the component is created, relative to the secondary part.	Only available for seams
Allow multiple instances of connection between same parts	Select this option to create many components to the same main part, in different locations.	Only available for connections and seams
Exact positions	Select this option to position the seam based on the positions that you pick in the model. Clear the checkbox to let Tekla Structures use automatic seam recognition to position the seam. This is useful especially with warped seams.	Only available for seams
Use the center of the bounding box in positioning	Select this option to position the custom part based on the center of its bounding box (the box that surrounds the actual part profile).	Only available for parts

Typical examples of custom components

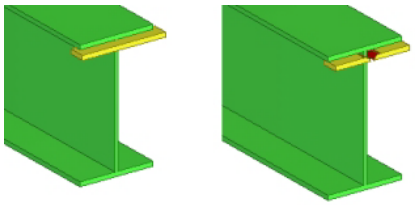
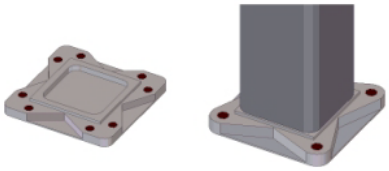
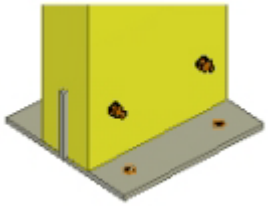
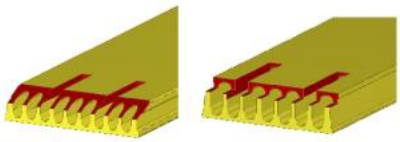
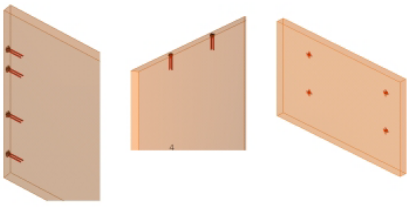
Here are some typical examples of the different types of custom components that you can create.

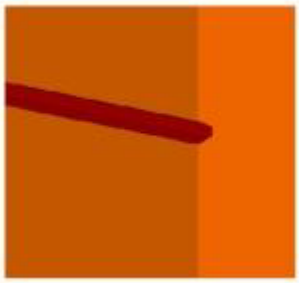
Examples of custom connections

Steel	Built-up plate seat	

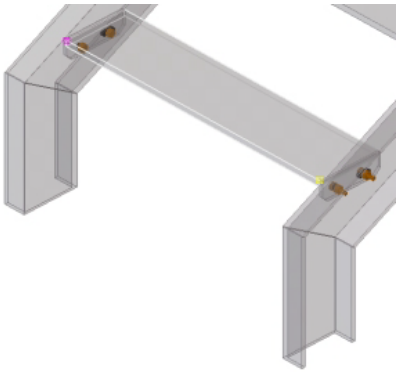
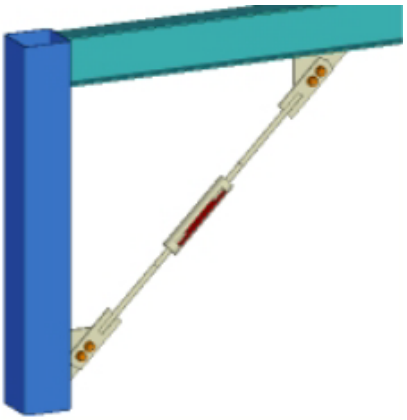
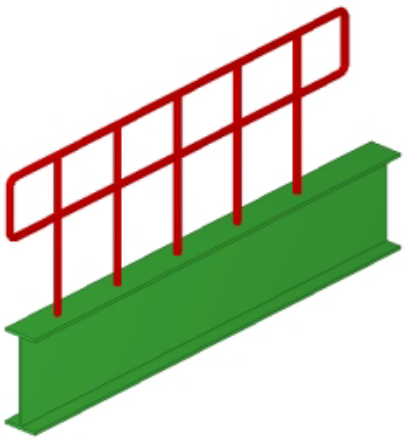
	Shear plate	
Precast concrete	Base detail	
	Double tee to L profile	
	Column cut-out	
	Wall panel connections	

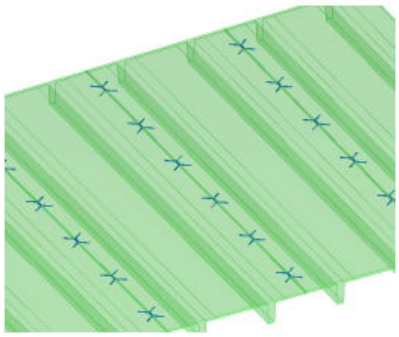
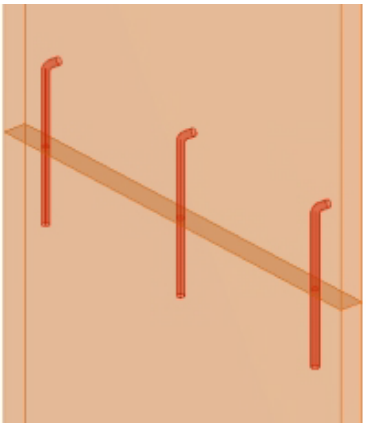
Examples of custom details

Steel	Backing plates	
	Cast base	
	Timber base	
Precast concrete	Door and window	
	Hollow core end details	
	Lifting details	

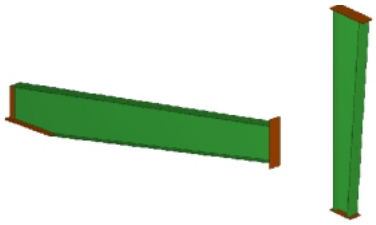
	Mock joint/reveal	

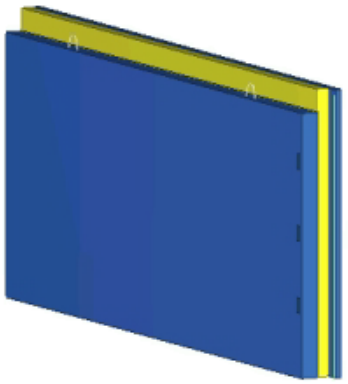
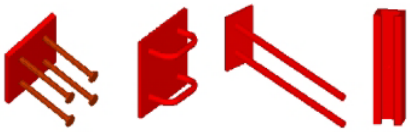
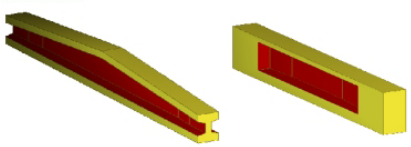
Examples of custom seams

Steel	Steel stair step	
	Turnbuckles	
	Handrail	

Precast concrete	Double tee connection	
	Panel to panel grout tube connection	

Examples of custom parts

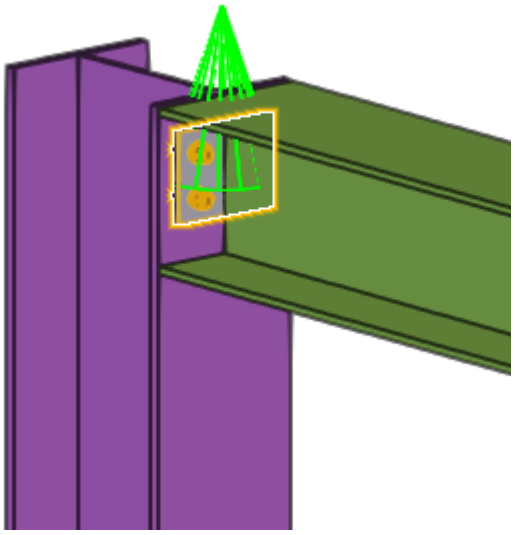
Steel	Company standard bracing plates	
	Castellated beam and cell beam	
	Built-up beams/columns	
	Standard glazing fixings	

Precast concrete	Sandwich panel	
	Lifters	
	Standard embeds/inserts	
	Standard beams	

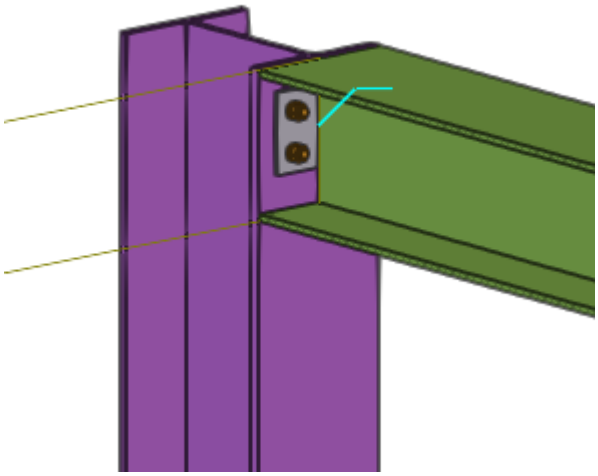
Custom component example: define an end plate custom component

This example shows how to define a simple custom component based on an existing end plate component.

1. Start by exploding an existing end plate component:
   a. Insert an end plate system component in the Tekla Structures model.
   b. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
   c. Click the **Access advanced features** button and select **Explode component**.
   d. Select the end plate component in the model.



Tekla Structures separates the objects in the component.



Alternatively, in the model, select the component you want to explode. Then right-click and select **Explode component**.

2. To define a new custom component based on the exploded system component:

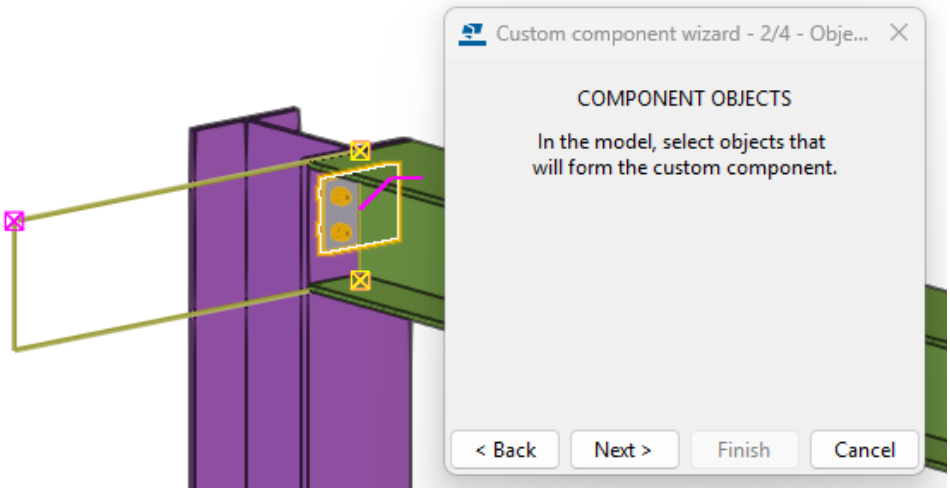
   a. Click the **Access advanced features** button and select **Define custom component...**.

   b. In the **Type** list, select **Connection**.

   c. In the **Name** box, enter a unique name for the custom component.

   d. Click **Next >**.

e. Select the objects you want to use in the custom component, and then click **Next >**.



You can use area selection (left to right) to select the objects. Tekla Structures ignores the main part and secondary parts and the grids when you are selecting objects to include in the custom component.

f. Select the column as the main part, and then click **Next >**.

The main part supports the secondary part.

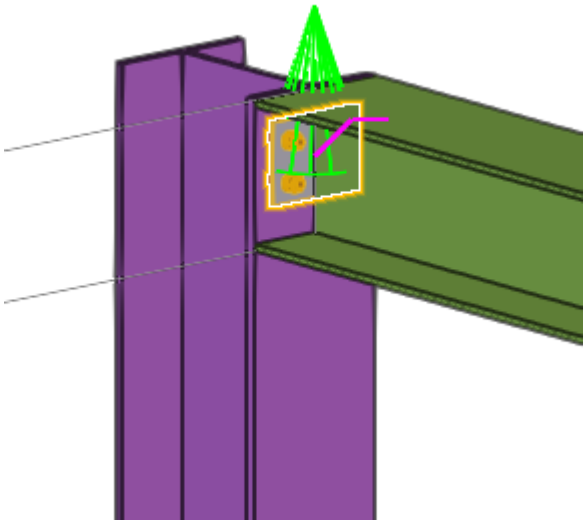
g. Select the beam as the secondary part.

The secondary part is supported by the main part.

When you select multiple secondary parts, pay attention to the order of selection. The custom component will use the same selection order when you add the component in a model. The maximum number of secondary parts in a custom component is 30.

h. Click **Finish**.

Tekla Structures displays a component symbol for the new component.



You have now defined a simple custom component, which you can use in locations similar to where it was originally created. The defined custom component does not automatically adjust dimensions to reflect the changes in the model. To have the component adapt to changes in the model, edit the component in the custom component editor, where you can build dependencies between component objects and model objects.

3. To insert the newly created custom end plate in the model:

   a. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

   b. Search for `end plate`.

   c. Select the new end plate custom connection.

      Custom connections have the following symbol in the **Applications & components** catalog: .

      Because you have not modified the end plate connection to adapt to changes in the model, you need to add the custom connection to a similar location where the connection was created. Otherwise the end plate connection may not work as required.

   d. Select the column as the main part.

   e. Select the beam as the secondary part.

      Tekla Structures adds the end plate connection to the model.

Custom component example: define a nested custom component

You can define more complex custom components by joining two or more components together as a nested component. This allows you to create smaller, simpler components and wrap them up into a single component. The original components become sub-components in the nested component. Nested components are typically used with precast and cast-in-place components, such as with embeds.

Do not nest components more than necessary. If you nest more than two levels, you might run across some limitations. To view the different component levels in a nested custom component (page 133), hold down the **Shift** key and scroll with the mouse wheel.

1. In the model, create the system components that you want to include in the nested component.
2. In the model, create the other component objects that you want to include in the nested component.

   You can create the component objects either by creating a component and then exploding it, or by creating the component objects manually.

3. Click the **Applications & components**  button in the side pane to open the **Applications & components** catalog.
4. Click the **Access advanced features**  button and select **Define custom component...**.

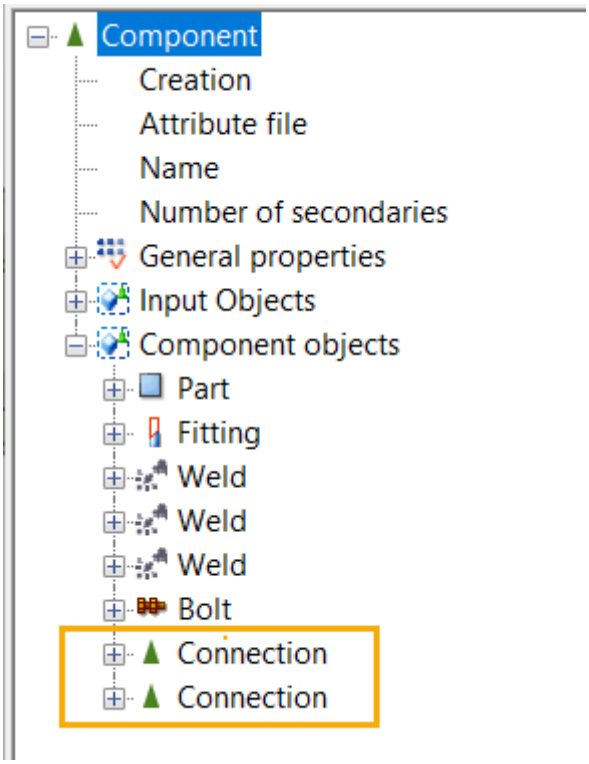
   The **Custom component wizard** dialog opens.
5. In the **Type** list, select the type for the nested custom component.
6. In the **Name** box, enter a unique name for the nested component.
7. If needed, modify the other properties on the **Type/Notes** tab, **Position** tab, and **Advanced** tab, and then click **Next >**.
8. In the model, select the components and any other objects you want to include in the nested component, and then click **Next >**.
9. Follow the instructions in the **Custom component wizard** to continue.

   You will be asked to select the main and secondary parts for the nested component. Depending on the component type that you selected in a step 5, you may also have to define other properties, such as detail or seam position.
10. When you are happy with the settings, click **Finish** to create the nested component.

    The component is added in the model and in the **Applications & components** catalog.

11. In the model, select the nested component you just created, right-click and select **Edit custom component**.

    The custom component editor opens. The sub-components are shown in the **Custom component browser**, together with the other component objects:



12. If you want to adapt the custom component to changes in the model, you can start building dependencies between component objects and model objects.

WARNING If you use a component of the type plug-in as a sub-component of a nested component, and change the sub-component's properties in the custom component editor, note that those changes may be lost when you save the nested component and use it in a model.

To prevent losing any properties, link a variable to each plug-in property that you want to keep. You can also use component attribute files to do this. For more information, see Examples of variable formulas (page 934).

See also

Edit custom components (page 887)

Add variables to a custom component (page 893)

Example of a variable formula: Create a nested custom connection with stiffeners (page 945)

Custom component example: add or move a custom part in the model

Use the direct modification handles and dimensions when you add or move custom parts. If you are unable to select custom parts in the model, ensure that the **Select components** selection switch is active.

NOTE This method cannot be used when adding custom parts to faces that have cuts or edge chamfers. You need to hide the cutting parts and edge chamfer objects from the view before you add custom parts on cut or chamfered faces using direct modification.

We do not recommend using this method with custom parts that are parametric, and in which the input points define the dimensions of the custom part. The preview is simplified, based on the default custom part dimensions, and snapping has a different focus than usually.

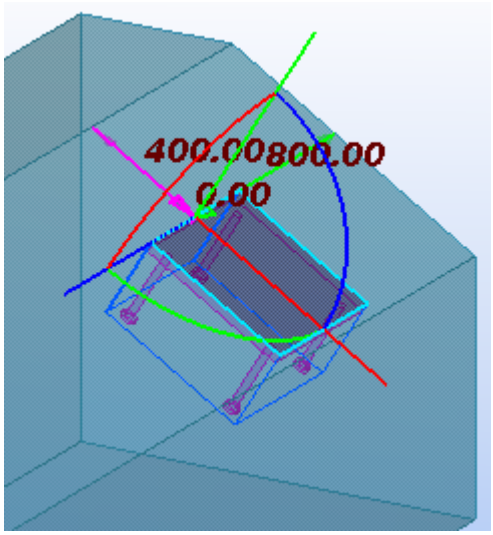
1. Ensure that **Direct modification** is switched on.
2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.
3. In the catalog, select the custom part you want to add.
4. Move the mouse pointer over part faces and edges in the model, and see how the custom part turns over and adjusts to the part faces.

   If you are adding the custom part to another object, Tekla Structures shows location dimensions to the nearest object edges.

   If you are adding a custom part that has only one input point, press **Tab** to rotate it in 90-degree steps around the work plane Y axis.
5. Depending on the number of custom part input points, pick one or two points to place the custom part in the model.

   Tekla Structures shows the coordinate axes, rotation handles, and location dimensions that you can use to fine-tune the location and rotation of the

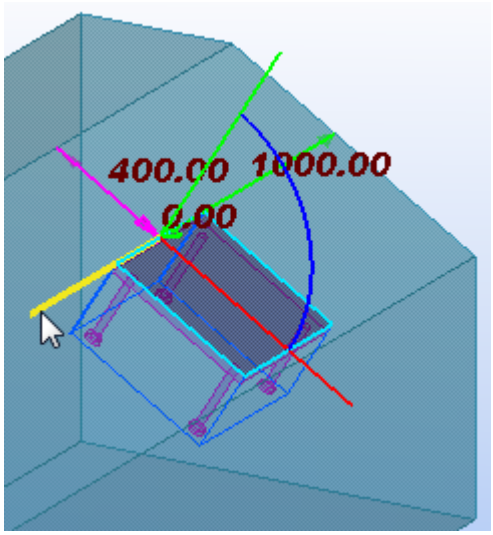
custom part. The handles are red, green, and blue, according to the local coordinate system of the custom part.



6. Click the middle mouse button to confirm the location and rotation.

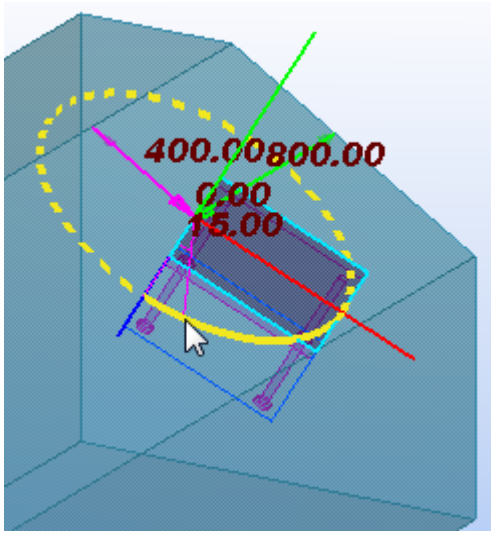
   Tekla Structures adds the custom part to the model.

7. To move the custom part along any of its coordinate axes, drag the relevant axis handle to a new location.



8. To rotate the custom part around any of its coordinate axes, drag the relevant rotation handle to a new location.

Press **Tab** to rotate the custom part in 90-degree steps in the direction of the selected rotation handle.



9. To move or rotate the custom part by specifying a distance or angle:
   a. Select an axis handle, a rotation handle, or a dimension arrowhead.
   b. Type the value by which you want the dimension to change.

      When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog.
   c. Click **OK** to confirm the new dimension.
10. To stop modifying, press **Esc**.

See also

Resize and reshape model objects (page 119)

Define custom components (page 866)

7.2 Edit custom components

Use the custom component editor to fine-tune existing custom components, and to modify a simple custom component so that it adapts to changes in the model.

In the custom component editor you can build dependencies between component objects and model objects. For example, you can specify that the size of a stiffener depends on the size of the beam. If you change the size of the beam, the size of the stiffener also changes.

You can also add distance variables, for example, to specify the gap between a plate and a beam. You can modify only the component objects, not the main or secondary parts, in the custom component editor. When you have edited a

custom component, Tekla Structures updates all instances of that component throughout the model with the changes you have made.

1. In the model, select the custom component by clicking the green component symbol .

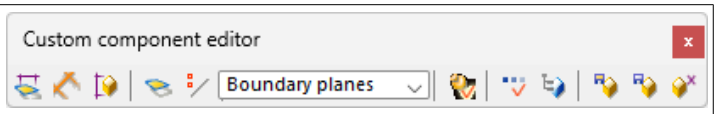
   NOTE Custom parts do not have a component symbol in the model.

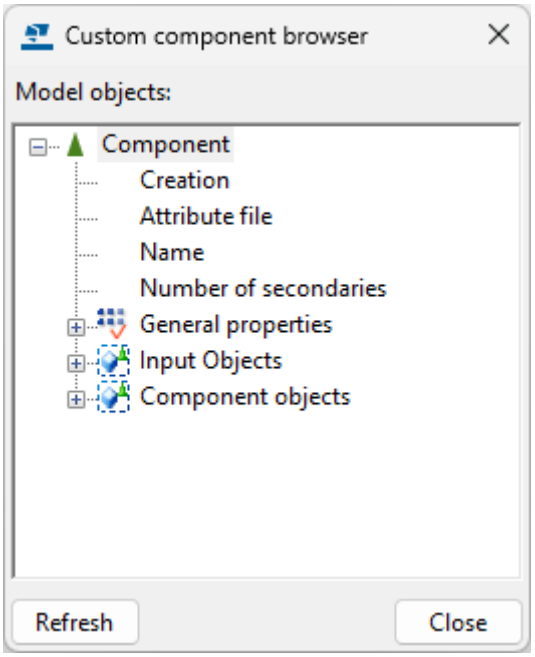
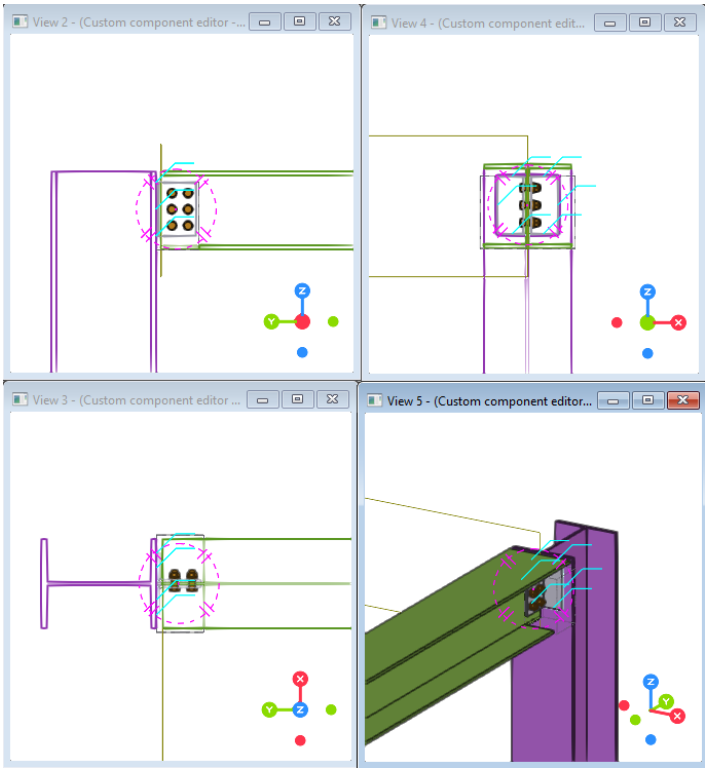
   To select custom parts, ensure that the **Select components** selection switch is active.

2. Right-click and select **Edit custom component**.

   Alternatively, click the **Applications & components** button in the side pane to open the **Applications & components** catalog, and click the **Access advanced features** button and select **Edit custom component**.

   The custom component editor opens. It shows the **Custom component editor** toolbar, the **Custom component browser** and four views of the custom component.

Custom component editor toolbar	

Custom component browser	
Four different **views** of the custom component	

3. Modify the custom component in one of the four custom component views. Only component objects, not the main or secondary parts, can be modified in the custom component editor.

   You can, for example:

- Add or remove component objects.

  For example, add extra bolts or stiffeners to the component.
- Bind component objects to model objects so that component objects adjust for new beam size or profile.
- Create fields for material grade, profile, bolt size, and other properties that will be visible in the custom component dialog box.
- Create formulas to calculate values, for example, for calculating bolt edge distance using bolt size.

4. Save the custom component.

To	Do this
Save changes to all copies of the custom component	a. On the **Custom component editor** toolbar, click the **Copy component to library** button. b. In the **Save confirmation** dialog, click **Yes**. Tekla Structures saves the changes and applies them to all copies of the custom component in the model.
Save the component with a new name	a. On the **Custom component editor** toolbar, click the **Copy component to library with new name** button. b. Enter a new name for the component.
Save and close the component	a. On the **Custom component editor** toolbar, click the **Close** button. b. In the **Close custom component editor** message, click **Yes**. If you click **No**, the custom component editor closes without saving the changes.

Custom component editor toolbar

Use the tools on the **Custom component editor** toolbar, for example, to create distances, select planes, and to save the component.

Icon	Description
Create distance	Create a distance. Select the plane first and then the handle or chamfer where to bind.

Icon	Description
Create reference distance 	Create a reference distance.
Automatically create distances for picked components' handle points 	Create distances automatically. Tekla Structures binds the selected part to their handles' contact planes. Tekla Structures selects planes in the following order: 1. construction planes 2. planes of the main and the secondary parts
Create construction plane 	Create a construction plane.
Add construction line 	Add a construction line.
Plane types	Show the plane types you can use when defining distance variables.
Custom component settings 	Edit the type or the position of a custom component, or add notes after you have created the component.
Display variables 	Show all created variables (page 893).
Display custom component browser 	Open the **Custom component browser**.
Copy component to library with new name 	Save the custom component with another name. Enter a new name for the component.
Copy component to library 	Save changes to all copies of the custom component. In the **Save confirmation** dialog, click **Yes**. Tekla Structures saves the changes and applies them to all copies of the custom component in the model.

Icon	Description
Close editor 	Save and close the editor. In the **Close custom component editor** message, click **Yes**. If you click **No**, the custom component editor closes without saving the changes.

Tekla Structures saves the custom components in the current model folder, in the `xslib.db1` file, which contains custom components and sketched profiles.

The custom component dialogs are saved separately in the `CustomComponentDialogFiles` folder in the current model folder.

Custom component browser

Custom component browser shows the contents of a custom component in a hierarchical, tree-like structure.

The **Custom component browser** works with the custom component editor views. When you select an object in the **Custom component browser**, Tekla Structures highlights the object in the views, and vice versa.

The **Custom component browser**

- lists the model objects, to which the custom component is attached
- lists the objects the custom component creates
- enables creating links between custom component variables and component object properties.

By selecting a property in the **Custom component browser** and right-clicking the property you can copy names, values, and references (page 916) from the main and secondary parts in the component, and then use them in the **Variables** dialog to define the properties of custom components.

Protect a custom component with a password

You can protect a custom component with a password to prevent unauthorized modifications. Password-protected custom components can still be added to models as usual.

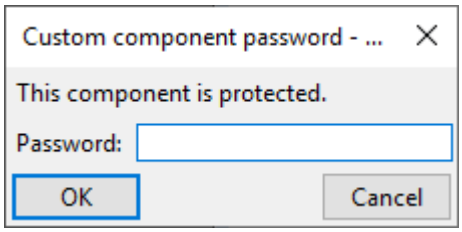
1. Select the custom component in the model, right-click and select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog opens.
3. To create a new parametric variable, click **Add**.
4. Modify the parametric variable as follows:
   a. In the **Name** box, enter `Password`.

b. In the **Formula** box, enter the desired password.

5. Save the custom component.

   The next time someone attempts to edit this custom component, they will be prompted for the password.



NOTE If you have a password-protected custom component and you explode it in the **Custom component editor**, you have two options:

- Use the **Explode component** command to explode the custom component without creating component parameters and mappings.
- Use the **Explode component with parameters** command, which prompts you for the password. The component will only be exploded, and its parameters and mappings created, after you enter the correct password.

See also

Define custom components (page 866)

Edit custom components (page 887)

Add variables to a custom component (page 893)

Customize the custom component dialogs (page 995)

7.3 Add variables to a custom component

Variables are the properties of a custom component. You can create variables in the custom component editor and use them to adapt custom components to changes in the model. Some of the variables appear in the custom component's dialog, while others are hidden and are only used in calculations.

Variable types

There are two types of variables:

- **Distance variable**: The distance between two planes, or between a point and a plane. A distance variable binds parts together, or works as a reference distance.

- **Parametric variable**: Controls all other properties in a custom component, such as name, material grade, and bolt size. Parametric variables are also used in calculations.

Distance variables

Use distance variables to bind custom component objects to a plane (page 898), so that the component objects stay at a fixed distance even if the surrounding objects change. You can create distance variables manually or automatically.

Use the handles of the following custom component objects to bind them to a plane:

- parts (only custom component objects)
- bolt groups
- reinforcing bars
- reinforcement meshes and strands
- chamfers
- part cuts, polygon cuts, line cuts
- fittings
- construction planes

You can decide which distance variables are shown in the custom component's dialog. Show the variables if you want to edit their values in the dialog. Hide the variables if you only use them to bind objects to a plane.

Parametric variables

Use parametric variables to set properties for any object the custom component creates (page 913). After creating a variable, you can change its value directly in the custom component's dialog.

You can also create formulas (page 934) to calculate values. For example, you can calculate the position of a stiffener relative to the beam length.

You can decide which parametric variables are shown in the custom component's dialog. Show the variables if you want to edit their values in the dialog. Hide the variables if you only use them in calculations.

NOTE There are some limitations concerning the variable names.

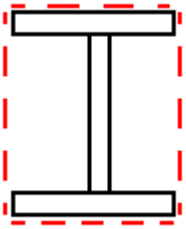
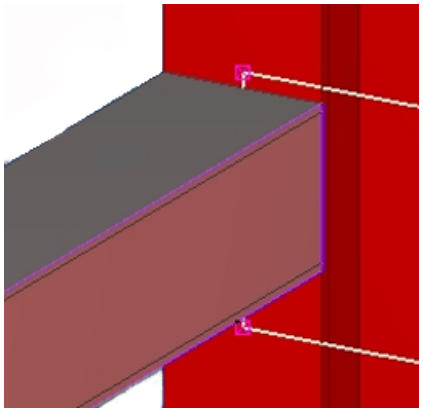
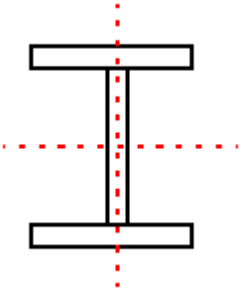
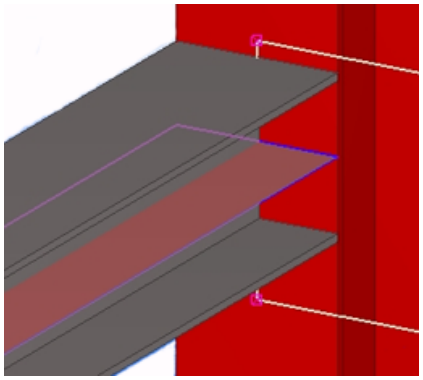
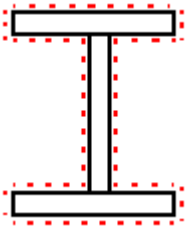
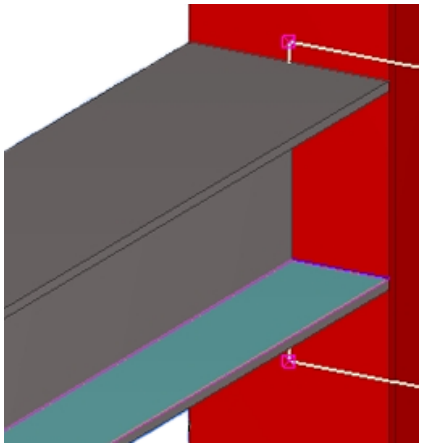
- To be able to correctly reference a variable in your formula, the variable name must be 19 characters or shorter. Variables with longer names will not work correctly when referenced.
- Variable names cannot contain mathematical operators (`+`, `-`, `*`, `/`).
- You cannot use a mathematical constant, such as `PI` or `e`, as a variable name.

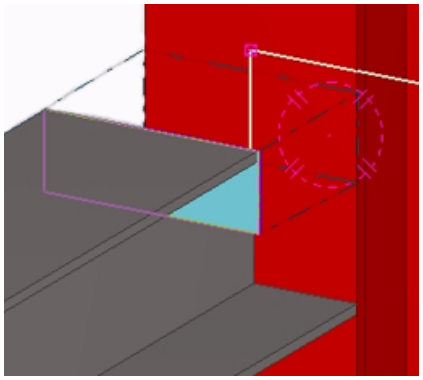
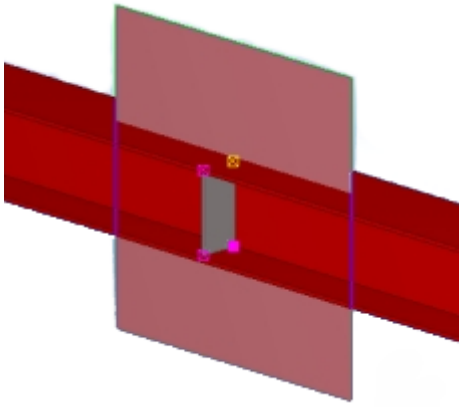
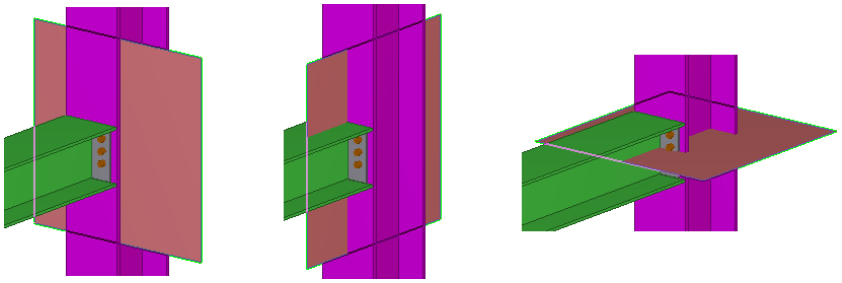
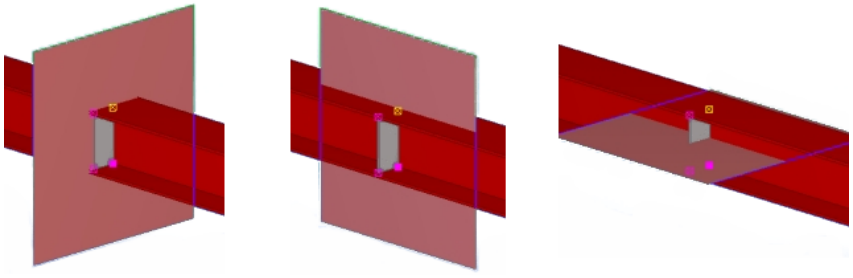
Plane types in distance variables

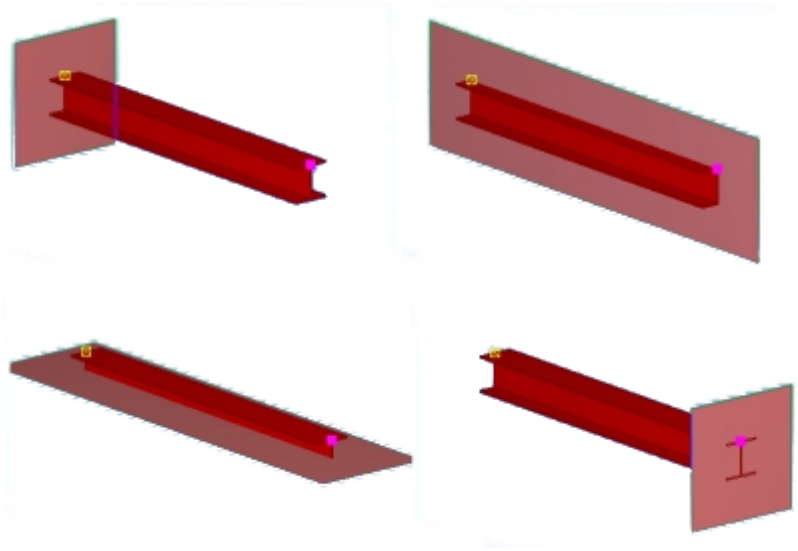
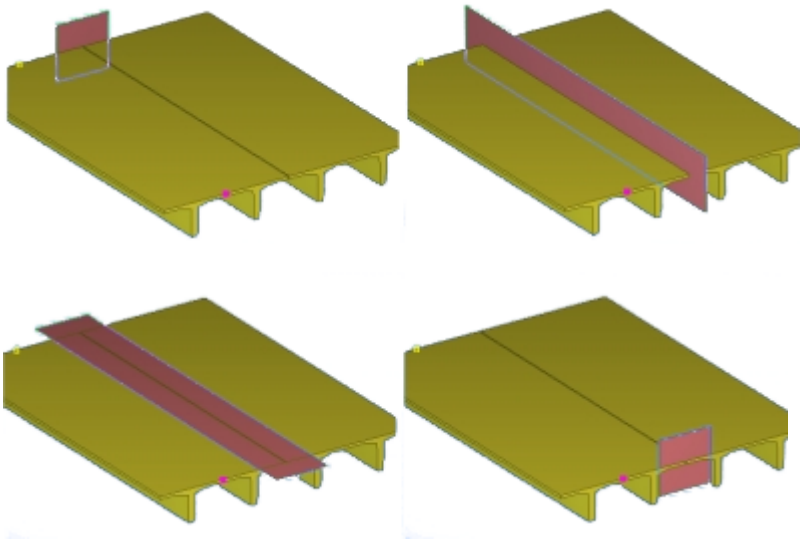
When you add distance variables to a custom component, or to a model, you must select a plane type. The plane type defines which planes you can select.

You can change the plane type when the **Create distance** command is active.

Boundary planes and component planes work for most profile types, so try to use them rather than outline planes whenever you can.

Plane type	Description	Example
Boundary planes	You can select the edges of a bounding box that surrounds the profile. 	
Center planes	You can select the center planes of a profile. 	
Outline planes	You can select the outer and inner surfaces of a profile. 	

Plane type	Description	Example
Cut planes	If the part contains line, part, or polygon cuts, this option enables you to select cut surfaces. Fittings cannot be selected.	
Component planes	What you can select depends on the component type and the **Position type** of the custom component.  **Connection component planes**  **Detail component planes** 	

Plane type	Description	Example
	Part component planes  **Seam component planes** 	

Note that when you create distance variables in a model and bind model object handles to different planes, select the plane type from the third list on the **Snapping** toolbar. Most of the plane type options there are the same as above, but **Grid planes** is available instead of **Component planes**.

See also

Bind component objects to a plane (create distance variables) (page 898)

Bind component objects using magnetic construction planes or lines (page 908)

Bind component objects to a plane (create distance variables)

Use *distance variables* to bind component objects to a plane. Binding keeps the custom component at a fixed distance from the plane even if the surrounding objects change. Distance variables automatically get the prefix **D** (distance), which is shown in the **Variables** dialog.

You can create distance variables manually or automatically.

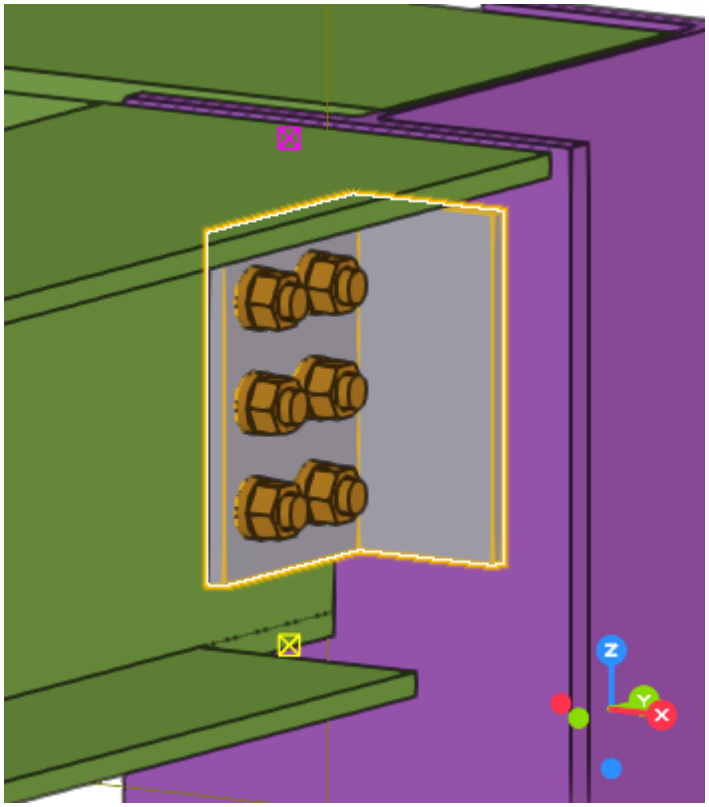
- Manual bindings require you to select the component object handles that you want to bind and select the plane you want to bind the handles to.
- With automatic bindings Tekla Structures automatically creates dependencies between the selected component objects and existing planes, if the objects or their handles are located exactly on the plane.

Bind objects manually (create distances)

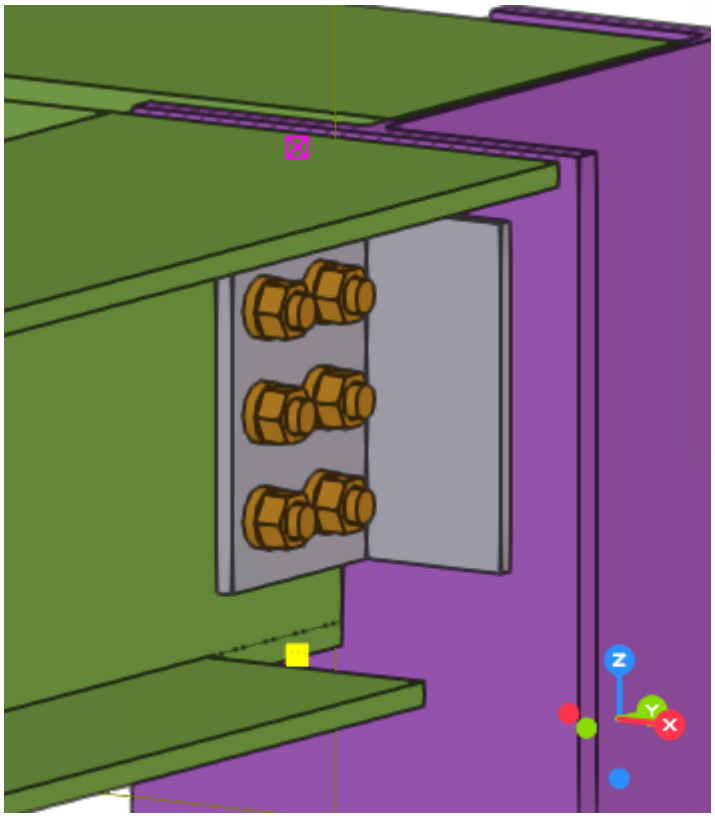
When you create distances by binding objects manually, you can bind a custom component object from specific handles only. You can bind an object to a maximum of three planes.

1. To make it easier to select the handles (page 117), ensure that **Direct modification**  is switched off.
2. To make it easier to select the available planes, set the model object rendering so that the object faces are properly shown, such as using **Parts rendered** (Ctrl+4).
3. In a custom component view, select an object that has handles, such as a part, bolt group, or part cut.

Hold down the **Alt** key and use area selection (from left to right) to select multiple handles.



4. Select the handle that you want to bind to a plane.

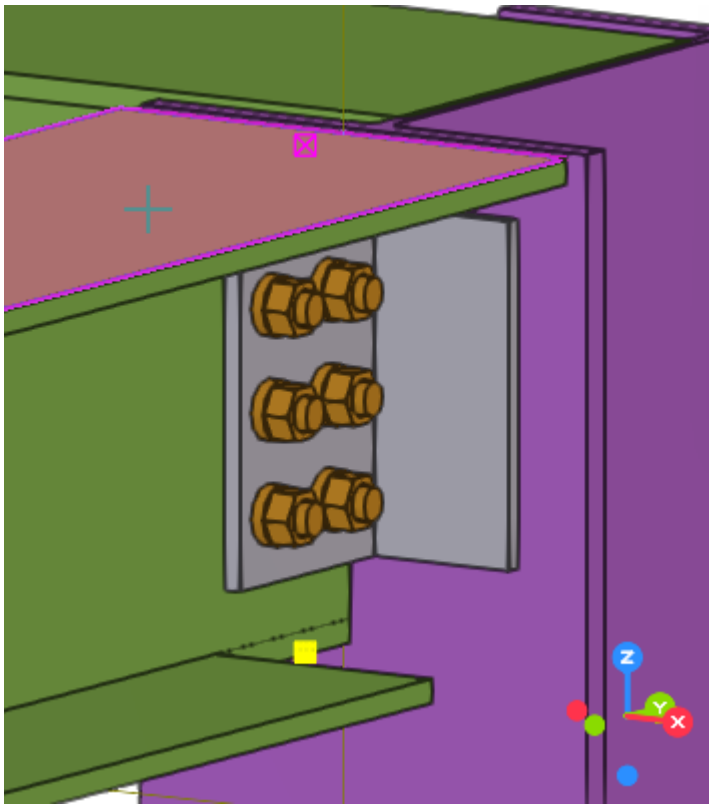


5. In the custom component editor, click the **Create distance** button.

   You can also right-click and select **Bind to plane**.

6. Move the mouse pointer in a custom component view to highlight the plane that you want to bind with the handles.

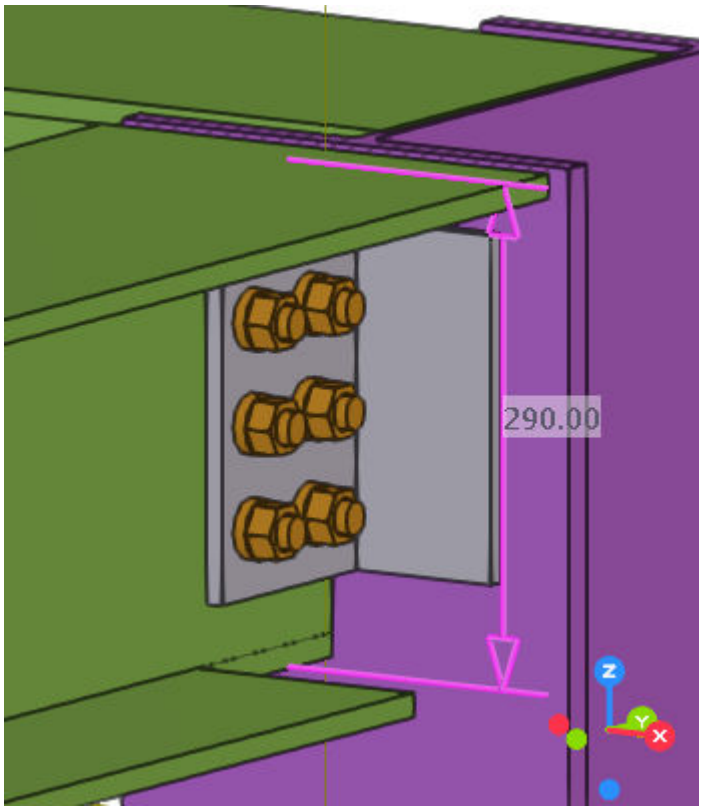
For example:



NOTE If you cannot highlight the correct plane, change the plane type on the **Custom component editor** toolbar. Boundary and component planes work for most profile types, so try to use them whenever you can.

7. Click the plane to create the binding.

Tekla Structures displays a distance symbol for the binding.



The corresponding distance variable is shown in the **Variables** dialog:

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
D1	290.00	290.00	Length	Distance	Show	D1.ANGLE.Extrema behind plane

NOTE If you have created a nested custom component and have used a component of the type plug-in as a sub-component of a nested component, or another custom component as a sub-component of a nested component, the bindings may be lost or do not work as desired when you save the nested component and use it in a model.

Bind objects automatically (create automatic distances)

When you create distances by binding objects automatically, Tekla Structures automatically binds the custom component objects or their handles to existing planes if the objects or handles are located exactly on the plane. Tekla Structures creates bindings from up to three directions to the existing planes.

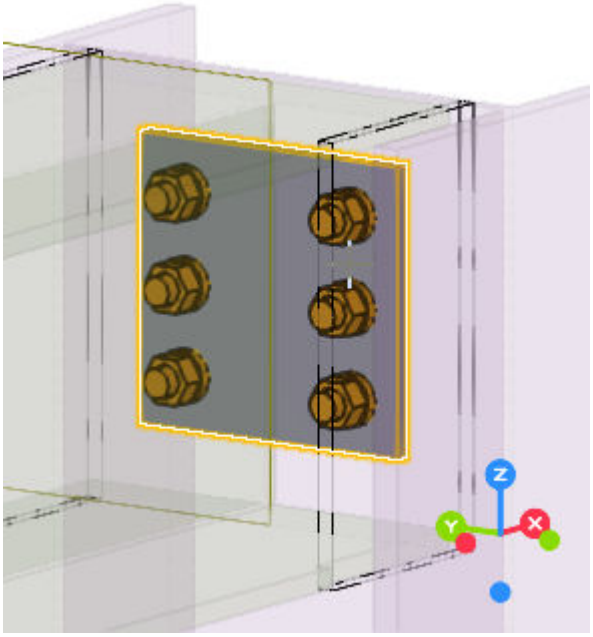
Tekla Structures selects planes in the following order:

1. construction planes

2. planes of the main and secondary parts

NOTE You cannot bind custom parts automatically, because they do not have a main part.

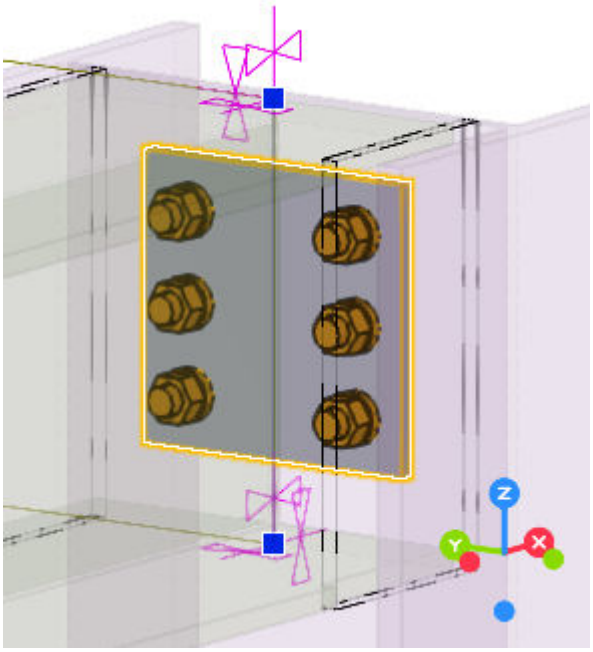
1. In the custom component editor, click the **Automatically create distances for picked components' handle points** button.
2. In a custom component view, select an object that has handles (page 117), such as a part, bolt group, or part cut.



3. Click the middle mouse button to bind the object.

   Tekla Structures binds the object from a maximum of three directions to the existing planes.

   Tekla Structures displays a distance symbol for each binding. Select the object to see the bindings.



The corresponding distance variables are shown in the **Variables** dialog:

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
D1	0.00	0.00	Length	Distance	Hide	D1.PLATE.XY plane
D2	0.00	0.00	Length	Distance	Hide	D2.PLATE.Extrema left plane
D3	0.00	0.00	Length	Distance	Hide	D3.PLATE.Extrema front plane
D4	0.00	0.00	Length	Distance	Hide	D4.PLATE.XY plane
D5	0.00	0.00	Length	Distance	Hide	D5.PLATE.Extrema right plane
D6	0.00	0.00	Length	Distance	Hide	D6.PLATE.Extrema left plane

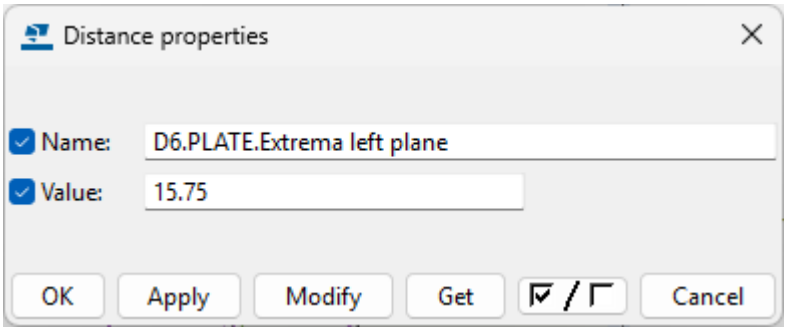
Test a binding

Test all bindings to see that they work correctly.

To be able to select distances in the model, ensure that the **Select distances**  selection switch active.

1. Double-click the distance symbol in a custom component view.

The **Distance properties** dialog opens.



2. In the **Value** box, enter a new value.
3. Click **Modify**.

   You should see the binding change in the model.

TIP Alternatively, you can test the binding in the **Variables** dialog:

a. Enter a new value in the **Formula** box.

b. Press **Enter**.

   You should see the binding change in the model.

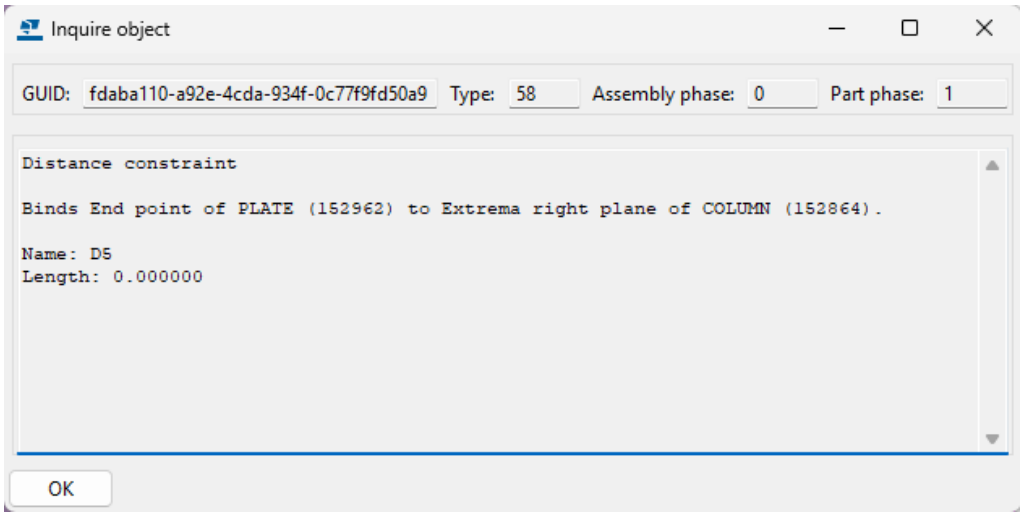
Check a binding

You can check what is bound to what by using the **Inquire object** command.

To be able to select distances in the model, ensure that the **Select distances** selection switch active.

1. On the ribbon, click **Object**.
2. Select a distance symbol in a custom component view.

The **Inquire object** dialog shows information about the binding.



Delete a binding

Bindings cannot be modified, but you can delete the existing bindings and then create new ones to rebind the objects.

To be able to select distances in the model, ensure that the **Select distances**  selection switch active.

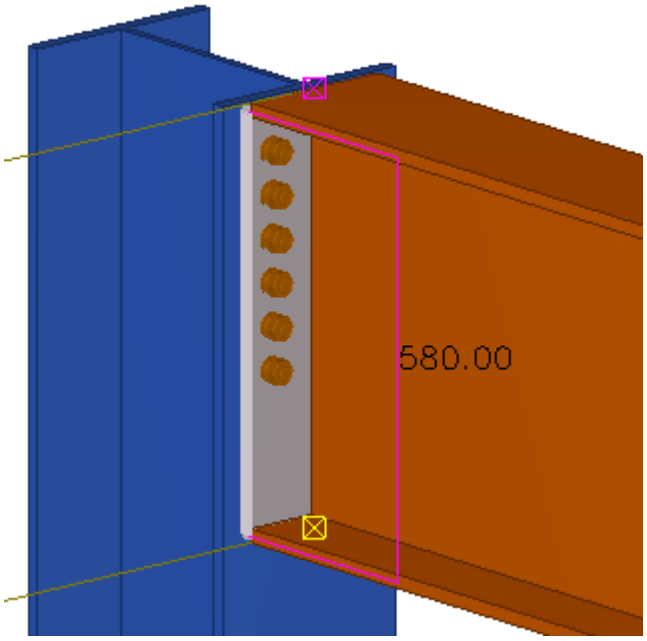
1. Select the binding in a custom component view.
2. Press **Delete**.

   You can also select the binding in the **Variables** dialog and then click the **Delete** button.

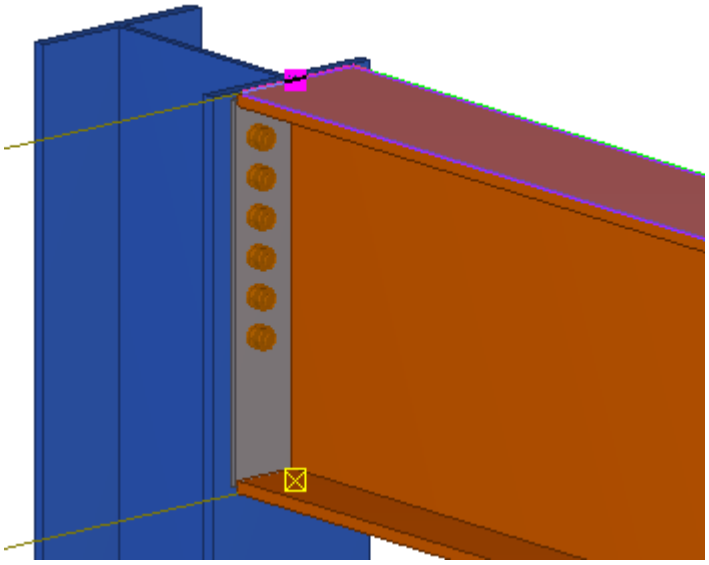
Example: Bind an end plate to a plane

This example shows how to bind the end plate top to the upper side of the beam.

1. To make it easier to select the end plate handles, ensure that **Direct modification**  is switched off.
2. In a custom component view, select the end plate to see the end plate handles.



3. Select the top handle of the end plate.
4. Right-click the top handle and select **Bind to plane**.
5. Move the pointer over the upper side of the beam flange to highlight it.

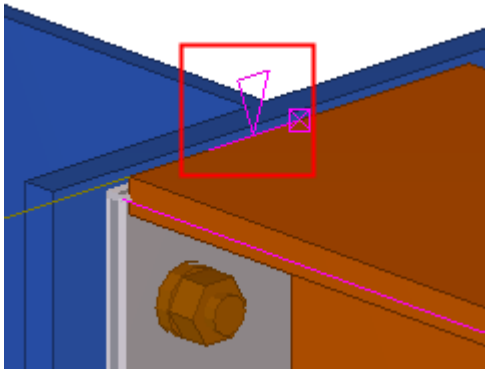


Here we are using the boundary plane type. If the part profile changes, the boundary plane is always found.

NOTE If you cannot highlight the desired plane, change the plane type on the **Custom component editor** toolbar.

6. Click the upper side of the beam flange.

A distance symbol appears in the custom component views.



7. Give a descriptive name for the binding you created:

   a. In the custom component editor, click the **Display variables** button.

      The **Variables** dialog opens.

   b. In the **Label in dialog box** box, enter `Plate top to flange top` as the name of the new binding.

   If you change the beam profile, the end plate top follows the upper side of the beam flange due to the binding.

Bind component objects using magnetic construction planes or lines

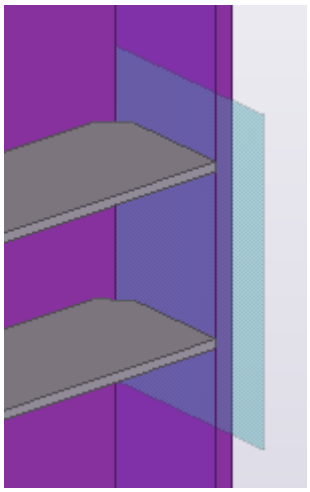
Instead of binding each component object handle to a plane separately, you can use magnetic construction planes and lines. The objects that are directly on a magnetic construction plane (or line) will move with the plane (or line), which means you only need to create one distance variable instead of 8, for example.

Bind handles using a magnetic construction plane

1. In the custom component editor, click the **Create construction plane** button.
2. Pick four points to define the shape of the construction plane.

   For example, create a plane that goes through all the handles and chamfers of the custom component.
3. Click the middle mouse button.

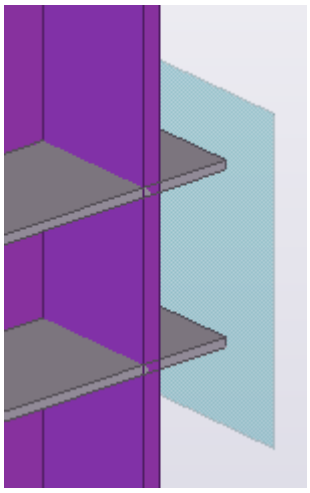
Tekla Structures creates a construction plane. For example:



4. Double-click the plane.

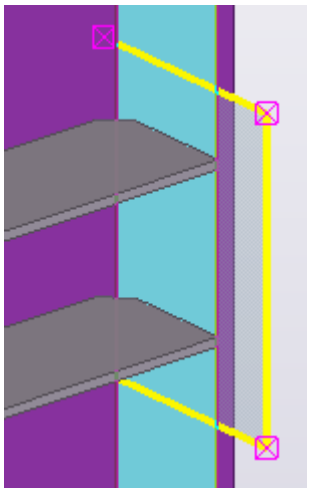
   The plane properties are shown in the property pane.
5. Enter a name for the plane.
6. In the **Magnetic** list, select **Yes**.
7. Click **Modify**.

   Now when you move the construction plane, all handles that are on the plane are moved as well:



8. Bind the construction plane to a part face:
   a. Select the construction plane, right-click and select **Bind to plane**.
   b. Select a suitable part face.

For example, the inner flange of the column:



Tekla Structures displays a distance symbol for the binding. Now if you move the part face, the handles on the magnetic construction plane will follow.

NOTE Only the objects whose handles are directly on the magnetic construction plane are affected. By default, the magnetic distance is 0.2 mm. To change this setting, use the advanced option XS_MAGNETIC_PLANE_OFFSET.

Bind handles using a magnetic construction line

1. In the custom component editor, click the **Add construction line** button.
2. Pick the start point of the construction line.
3. Pick the end point of the construction line.

   Tekla Structures creates a construction line.
4. Double-click the line.

   The line properties are shown in the property pane.
5. Enter a name for the line.
6. In the **Magnetic** list, select **Yes**.
7. Click **Modify**.

   Now when you move the construction line, all handles that are on the line are moved as well.
8. Bind the construction line to a part face:
   a. Select the construction line, right-click and select **Bind to plane**.

b. Select a suitable part face.

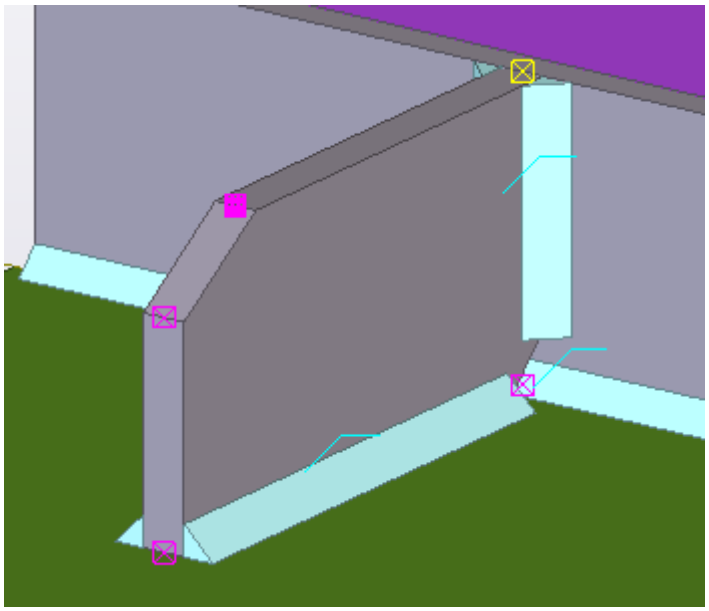
   Tekla Structures displays a distance symbol for the binding. Now if you move the part face, the handles on the magnetic construction line will follow.

Add a distance between component objects (create reference distance variables)

Use *reference distance variables* to add a distance between two points or a point and a plane. The reference distance changes as you move the objects it refers to. You can use reference distances in calculations, for example, to determine the spacing of rungs on a ladder. Reference distance variables automatically get the prefix **D** (distance), which is shown in the **Variables** dialog.

1. To make it easier to select the handles (page 117), ensure that **Direct modification** is switched off.
2. To make it easier to select the available planes, set the model object rendering so that the object faces are properly shown, such as using **Parts rendered** (Ctrl+4).
3. In a custom component view, select a handle.

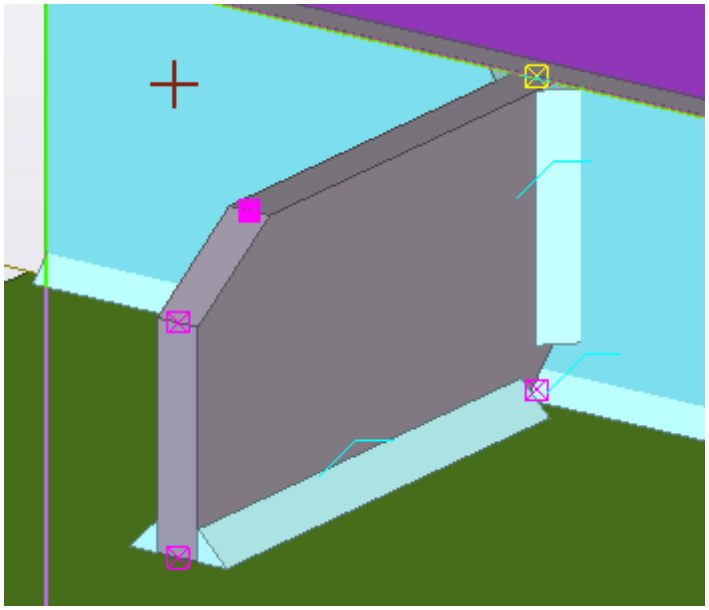
   This is the starting point for your measurement.



4. In the custom component editor, click the **Create reference distance** button.

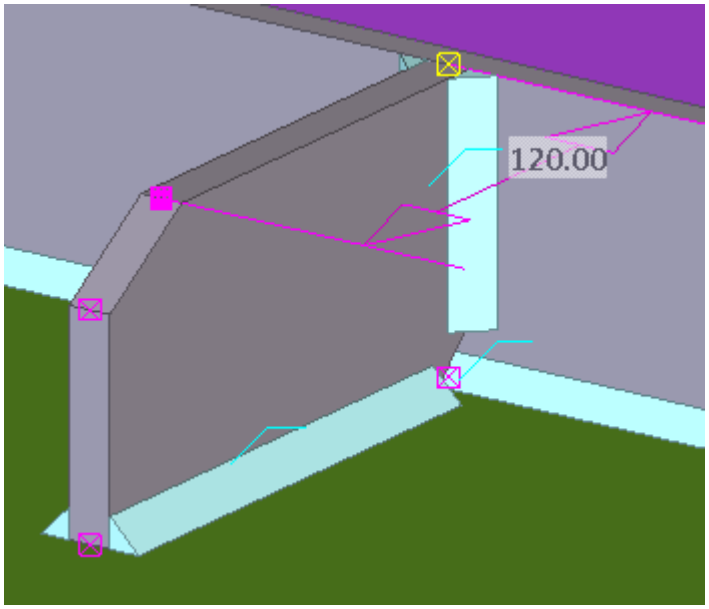
5. Move the mouse pointer in a custom component view to highlight the plane.

   This will be the end point for your measurement. If you cannot highlight the correct plane, change the plane type on the **Custom component editor** toolbar.



6. Click the plane to select it.

   Tekla Structures displays the distance.



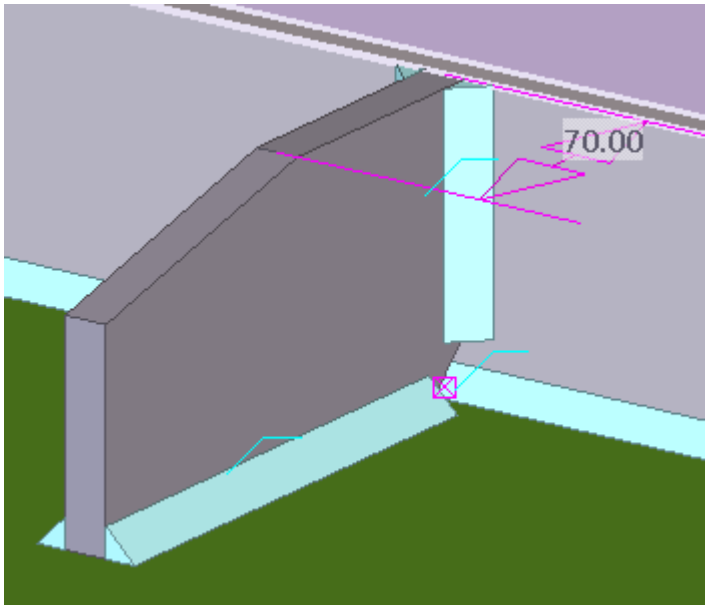
   The corresponding reference distance variable is shown in the **Variables** dialog:

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
D1	120.00	120.00	Length	Distance	Hide	D1.HAUNCH_PLATE.Web right plane

Note that the **Create reference distance** command remains active. You can click more planes if you want to measure other distances.

7. To stop measuring, press **Esc**.
8. To check that the reference distance works correctly, move the handle.

   The distance changes accordingly. For example:



See also

Add variables to a custom component (page 893)

Bind component objects to a plane (create distance variables) (page 898)

Use parametric variables to set object properties

Use *parametric variables* to set basic properties (such as name, material, profile, position number, and so on) for any object the custom component creates. Parametric variables automatically get the prefix **P** (parameter), which is shown in the **Variables** dialog.

Example: Create a parametric variable for weld size

The following example shows how to create a parametric variable that sets all welds in a custom component to a given size. After creating the variable, you are able to change the weld size directly in the custom component's dialog.

1. Select the custom component in the model, right-click and select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog opens.
3. Create a new parametric variable.
   a. Click **Add**.
   b. In the **Value type** list, change the variable's value type to **Length**.

      The type determines what kind of values can be used with this variable. In this example, select **Length**, which is suitable for lengths and distances.
   c. In the **Name** box, enter `Weldsize` as the name of the variable.

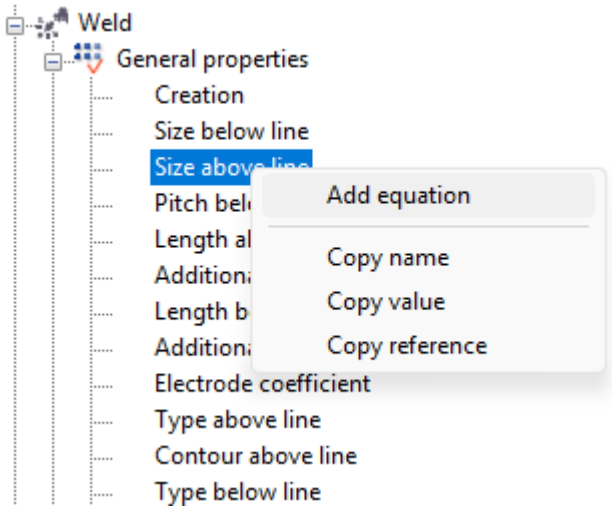
      You can also use the default name, such as **P1**. In this example, enter `Weldsize` as the name of the variable.
   d. Leave the **Formula** box empty.
   e. In the **Visibility** list, define whether the variable will be visible in the custom component's dialog.

      Hide the variable if you only use it in calculations. Show the variable if you want to be able to edit the value in the custom component's dialog. In this example, select **Show**.
   f. In the **Label in dialog box** box, enter a descriptive name for the parametric variable, for example, `Weld size`.
4. Click **Close**.

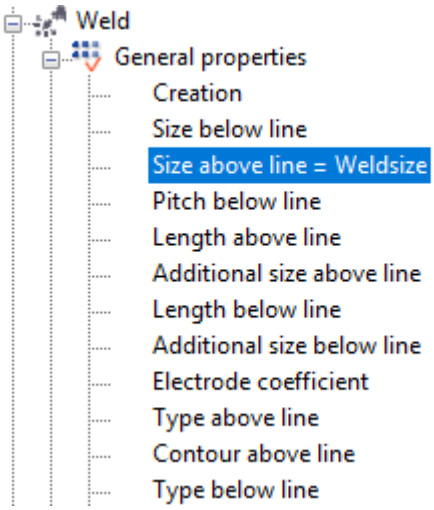
   You have now created a parametric variable with the following settings:

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
Weldsize	0.00	0.00	Length	Parameter	Show	Weld size

5. Link the parametric variable to the **Size above line** property.
   a. In the **Custom component browser**, browse for the **Size above line** property of the weld.
   b. Right-click **Size above line** and select **Add equation**.



c. Enter `Weldsize` after the equal sign, and then press **Enter**.



d. Repeat the procedure for the other welds as well, so that they will all be linked to the same property.

6. Save the custom component.
7. Close the custom component editor.
8. Select the custom component in the model, and open the custom component dialog.

The option to change the weld size is visible in the custom component's dialog.



If you change the weld size value now, the size of all welds within the custom component will change accordingly.

For more examples of the use of parametric variables, see Examples of parametric variables and variable formulas in custom components (page 934).

Copy properties and property references from another object

You can copy properties, such as names and values, from other objects and use them to determine the properties of a custom component. You can also copy property *references*. The link is dynamic, so when the property changes, the reference reflects the change. For example, you can use a beam length reference in variable formulas. Even if the length changes, the correct value is always used in calculations.

1. In the **Custom component browser**, browse for the object property you want to copy.

   To find the required component object more easily, select it in a custom component view. Tekla Structures highlights the selected object in the **Custom component browser**.

2. Right-click the property and select one of the following:

   - **Copy name**

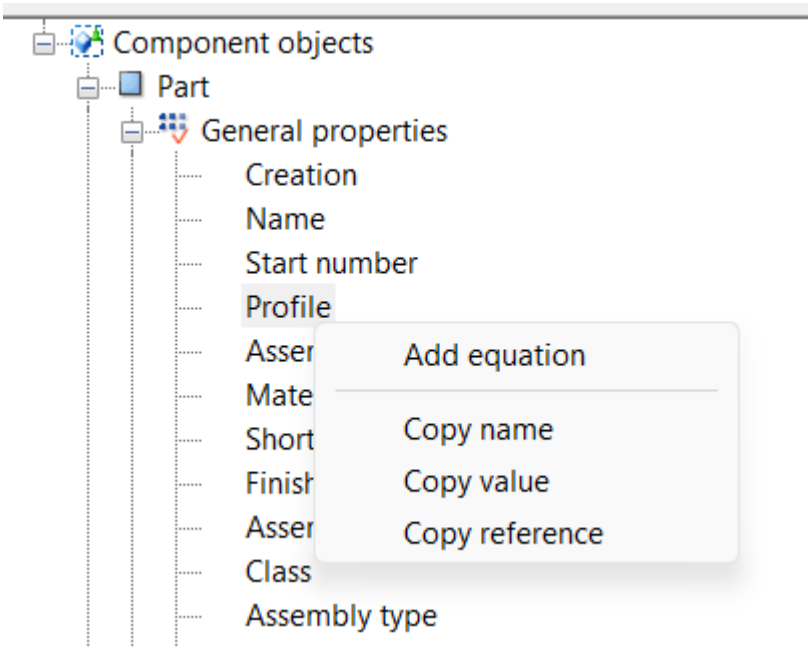
     Copies the name of the object. For example, `Material`.

   - **Copy value**

     Copies the value the object currently has. For example, `S235JR`.

   - **Copy reference**

     Copies the link to the property. For example, `fP(Material,"642dd994-5b5a-414e-8ac1-51fa026f3700").`



3. Right-click where you want to insert the object property, and then select **Paste**.

   For example, you can paste a reference to the **Formula** box in the **Variables** dialog to use it in a calculation.

See also

Edit custom components (page 887)

Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Create a variable formula

Use variable formulas to add more adaptivity to your custom components.

Variable formulas always begin with the equal sign (=).

At its simplest, a formula can be a simple dependency between two variables, stating that P2 equals half of P1 (`P2=P1/2`), for example.

To create more complex calculations, you can use functions and operators inside the formula. For example, you can add mathematical expressions, `if` statements, references to object properties, and so on.

When you create formulas, note that multiplication is faster than division, for example, `P1*0.5` is faster than `P1/2`.

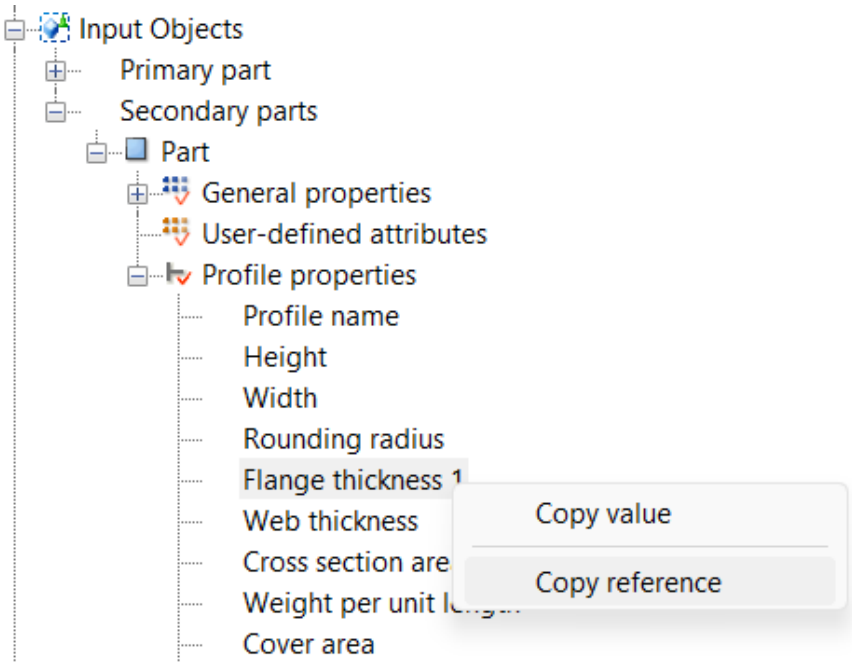
Example: Create a variable formula to set the weld size based on flange thickness

The following example shows how to create a formula that sets the weld size to half the thickness of the secondary part flange. When the component is used in a model, Tekla Structures will use the thickness of the secondary part flange to calculate the size of the weld.

1. Select the custom component in the model, right-click and select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables** button.

   The **Variables** dialog opens.
3. Create a new parametric variable.
   a. Click **Add**.
   b. In the **Name** box, enter a name for the variable.

      In this example, enter `w` as the name of the variable.
4. Copy the flange thickness reference in the **Custom component browser**.
   a. Browse for **Flange thickness 1** property in **Input Objects** --> **Secondary parts** --> **Part** --> **Profile properties**.
   b. Right-click **Flange thickness 1** and select **Copy reference**.



   Tekla Structures copies the reference to flange thickness to the clipboard.

5. To set the formula in the **Variables** dialog, modify the parametric variable as follows:

   a. In the **Formula** box, type `=`, right-click, and select **Paste**.

      Tekla Structures pastes the reference to flange thickness from the clipboard.

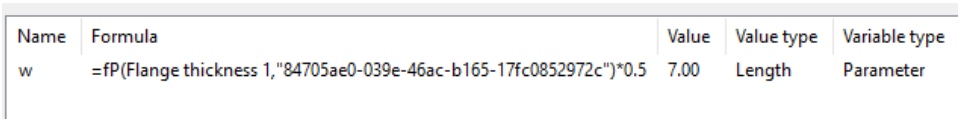
   b. After the flange thickness formula, enter `*0.5`.

      The formula should now read:

      ```
      =fP(Flange thickness 1,"<GUID>")*0.5
      ```

   c. In the **Value type** list, select **Length**.

   d. In the **Visibility** list, select **Hide**.



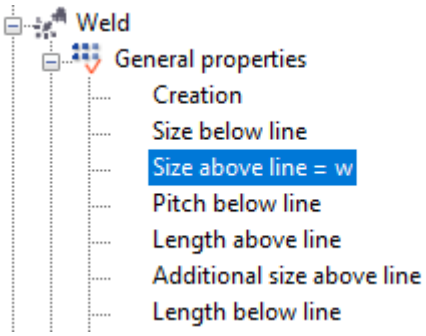
Name	Formula	Value	Value type	Variable type
w	=fP(Flange thickness 1,"84705ae0-039e-46ac-b165-17fc0852972c")*0.5	7.00	Length	Parameter

6. Link the parametric variable to the **Size above line** property.

   a. In the **Custom component browser**, browse for **Size above line** in **Component objects** --> **Weld** --> **General properties**.

   b. Right-click **Size above line** and select **Add equation**.

   c. Enter `w` after the equal sign, and then press **Enter**.



7. Save the custom component.
8. Close the custom component editor.
9. Select the custom component in the model, and open the custom component dialog.

   You can now use the thickness of the secondary part flange to calculate the size of the weld.

Functions in variable formulas

You can use functions to calculate values for parametric variables. Variable formulas always begin with the equal sign (=).

For more information, see Use parametric variables to set object properties (page 913).

Arithmetic operators

Use arithmetic operators to combine expressions that return numeric values.

Operator	Description	Notes
+	addition	Use also to create strings of parameters.
-	subtraction	
*	multiplication	Multiplication is faster than division. `=D1*0.5` is faster than `=D1/2`
/	division	

Logical and comparison operators

Use logical and comparison operators inside **if** statements. You can use **if-then-else** statements to test a condition and to set the value according to the result.

For example:

```
=if (D1>200) then 20 else 10 endif
```

You can use the following operators inside if statements:

Operator	Description	Example
==	both sides are equal	
!=	sides are not equal	
<	left side is smaller	
<=	left side is smaller or equal	
>	right side is smaller	
>=	right side is smaller or equal	
&&	logical AND both conditions must be true	`=if (D1==200 && D2<40) then 6 else 0 endif` If D1 is 200 and D2 smaller than 40, the result is 6, otherwise 0.

Operator	Description	Example
||	logical OR only one condition must be true	`=if (D1==200 || D2<40) then 6 else 0 endif` If D1 is 200 or D2 is smaller than 40, the result is 6, otherwise 0.

Reference functions

Use reference functions to refer to the property of another object, such as the plate thickness of a secondary part. Tekla Structures refers to the object on the system level, so if the object property changes, so does the reference function value.

Function	Description	Example
`fTpl("template attribute", "object GUID")`	Returns the template attribute value of an object that has a given object GUID.	`=fTpl("WEIGHT","ID50B8559A-0000-010B-3133-353432373038")` returns the weight of an object whose GUID is ID50B8559A-0000-010B-3133-353432373038.
`fP("user-defined attribute", "object GUID")`	Returns the user-defined attribute value of an object that has a given object GUID.	`=fP("comment", "ID50B8559A-0000-010B-3133-353432373038")` returns the user-defined attribute **comment** of an object whose GUID is ID50B8559A-0000-010B-3133-353432373038.
`fValueOf("parameter")`	Returns the value of the parameter.	If the equation is `=P2+"*"+P3`, the result is `P2*P3` With `=fValueOf("P2")+"*"+fValueOf("P3")`, where P2=780 and P3=480, the result is `780*480`
`fRebarCatalogValue(BarGrade, BarSize, Usage, FieldName)`	Returns the reinforcing bar catalog value of an object. `Usage` can be either 2 ("Tie") or 1 ("Main"). `FieldName` must be one of the following: • 0 NominalDiameter	`fRebarCatalogValue("A500HW", "10", 1, 2)` returns the size, usage, and weight of an object whose reinforcing bar grade is A500HW.

Function	Description	Example
	• 1 ActualDiameter • 2 Weight • 3 MinRadius • 4 Hook1Radius • 5 Hook1Angle • 6 Hook1Length • 7 HookRadius • 8 Hook2Angle • 9 Hook2Length • 10 Hook3Radius • 11 Hook3Angl • 12 Hook3Length • 13 Area	

ASCII file as a reference function

You can refer to ASCII files to get data. Tekla Structures searches for the files in the following order:

1. model folder
2. `..<model folder>\CustomComponentDialogFiles`
3. project folder (`XS_PROJECT`)
4. firm folder (`XS_FIRM`)
5. system folder (`XS_SYSTEM`)
6. profile database folder (`XS_PROFDB`)
7. `<model folder>\cr` (cold rolled components)

The parameters used are: file name, key value of row, column number, separator(s), allow empty columns. The format for reading the files is as follows:

```
fVF("filename", "key_value_of_row", column_number,
"separator(s)", FALSE)
```

- **Key value of a row**: A unique key value string that is used to select the line in the file. The search is case-sensitive. The string must not contain spaces, tabs, or any of the specified separator characters.
- **Column number**: An index starting from 1 to select the value from the chosen line. Index 1 corresponds to the key value.

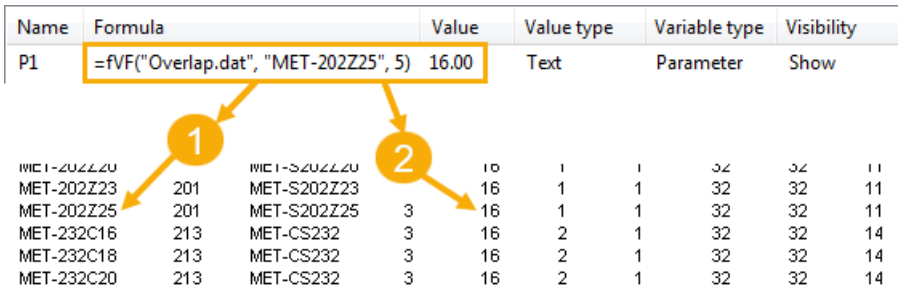
- **Separators**: Optional parameter. The separator consists of one or more characters used for data separation. The default separators are space and tab.

  The specified character sequence will be interpreted as a set of characters, each treated as an individual separator, not as a group. Use "\t" to represent a tab. For example, " \t" means that space and tab will be treated as separators, and ",.;" means that comma, dot, and semicolon will be treated as separators.
- **Allowing empty columns**: Optional parameter. The default for allowing empty columns is `FALSE`.

  If it is set to `TRUE`, empty entries from the file line will be considered when the function is resolved. If it is missing or set to `FALSE`, empty entries are removed.

Example

The `=fVF("Overlap.dat", "MET-202Z25", 5)` function is in the **Formula** box in the **Variables** dialog. The function gets the value `16.0` for the profile `MET-202Z25`, from the `Overlap.dat` file.



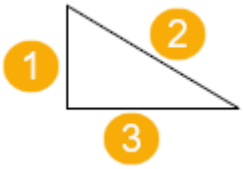
1. Key value of the row (MET-202Z25)
2. Column number (5)

For more information, see the How to read data from a text file in Custom Components support article.

Mathematical functions

Use mathematical functions to create more complex mathematical expressions.

Function	Description	Example
fabs(parameter)	Returns the absolute value of the parameter	`=fabs(D1)` returns 15 if D1 = -15
exp(power)	Returns `e` raised to the power `e` is Euler's number.	`=exp(D1)` returns 7.39 if D1 = 2

Function	Description	Example
ln(parameter)	Returns the natural logarithm of the parameter (base number `e`)	`=ln(P2)` returns 2.71 if P2 = 15
log(parameter)	Returns the logarithm of the parameter (base number 10)	`=log(D1)` returns 2 if D1=100
sqrt(parameter)	Returns the square root of the parameter	`=sqrt(D1)` returns 4 if D1 = 16
mod(dividend, divider)	Returns the modulo of the division	`=mod(D1, 5)` returns 1 if D1 = 16
pow(base number, power)	Returns the base number raised to the specified power	`=pow(D1, D2)` returns 9 if D1 = 3 and D2 = 2
hypot(side1,side2)	Returns the hypotenuse  **(1)**: side1 **(2)**: hypotenuse **(3)**: side2	`=hypot(D1, D2)` returns 5 if D1 = 3 and D2 = 4
n!(parameter)	Returns the factorial of the parameter	`=n!(P2)` returns 24 if P2 = 4 (1*2*3*4)
round(parameter, accuracy)	Returns the parameter rounded off to the given accuracy	`=round(P1, 0.1)` returns 10.600 if P1 = 10.567
PI	Returns the value of pi to 31 decimal places	`=PI` returns 3.14159265358979323846264 33832795

Statistical functions

Use statistical functions to calculate sums and averages, and to round values.

Function	Description	Example (P1 = 1.4 P2 = 2.3)
ceil()	Returns the smallest whole number greater than or equal to the parameter	`=ceil(P1)` returns 2

Function	Description	Example (P1 = 1.4 P2 = 2.3)
floor()	Returns the largest whole number less than or equal to the parameter	`=floor(P1)` returns 1
min()	Returns the smallest parameter	`=min(P1, P2)` returns 1.4
max()	Returns the largest parameter	`=max(P1, P2)` returns 2.3
sum()	Sum of the parameters	`=sum(P1, P2)` returns 3.7
sqsum()	Sum of the squared parameters: (parameter1)2 + (parameter2)2	`=sqsum(P1, P2)` returns 7.25
ave()	Average of the parameters	`=ave(P1, P2)` returns 1.85
sqave()	Average of the squared parameters	`=sqave(P1, P2)` returns 3.625

Example: Ceil and floor statistical functions

In this example, you have the following parametric variables:

- Beam length: P1 = 3500
- Post spacing: P2 = 450

P1 / P2 = 7.7778

You can use the `ceil` and `floor` statistical functions to round the value and then use the rounded value as the number of beam posts:

- `=ceil(P1/P2)` returns 8
- `=floor(P1/P2)` returns 7

Data type conversion functions

Use data type conversion functions to convert values into another data type. Conversion functions perform conversions between the three supported data types: integers, doubles, and strings.

Function	Description	Example
int()	Converts data to integer	Useful especially for calculating profile dimensions: `=int(100.0132222000)` returns 100, if decimals are set to 0 in the **Options** dialog
double()	Converts data to a double	
string()	Converts data to string	

Function	Description	Example
imp()	Converts imperial units Use this function in calculations instead of imperial units. You cannot use imperial units directly in calculations.	For the following examples, length unit is set to mm and decimals are set to 2 in the **Options** dialog. `=imp(1,1,1,2)` meaning 1 foot 1 1/2 inch returns 342.90 mm `=imp(1,1,2)` meaning 1 1/2 inches returns 38.10 mm `=imp(1,2)` meaning 1/2 inches returns 12.70 mm `=imp(1)` meaning 1 inch returns 25.40 mm `=3'/3"` is not possible, but `=imp(36)/imp(3)` is ok
vwu(value, unit)	Converts the length values and angle values. The available units are: • "ft" ("feet", "foot") • "in" ("inch", "inches") • "m" • "cm" • "mm" • "rad" • "deg"	`=vwu(4.0,"in")` returns 101.60 mm, if length unit is set to mm and decimals are set to 2 in the **Options** dialog `=vwu(2.0,"rad")` returns 114.59 degrees, if angle is set to degrees and decimals are set to 2 in the **Options** dialog Note that Tekla Structures automatically converts angles from degrees to radians by multiplying by `PI/180`. To ensure that the angles are correctly calculated, use `vwu("rad")` to convert radians to degrees. This allows Tekla Structures to convert them back to radians during calculations.

NOTE The units depend on the settings in **File menu** --> **Settings** --> **Options** --> **Units and decimals**.

String operations

Use string operations to manipulate character strings. Strings must be inside quotation marks in variable formulas.

Operation	Description	Example (P1 = "PL100*10")
match(parameter1, parameter2)	Returns 1 if parameters are equal and 0 if different. You can also use wildcards `*`, `?`, and `[ ]` with the match function.	`=match(P1, "PL100*10")` returns 1 Accept all profiles starting with PFC: `=match(P4, "PFC*")` Accept profiles starting with PFC, and height starts with 2, 3, 4 or 5: `=match(P4, "PFC[2345]*")` Accept profiles starting with PFC, heights are 200, 300, 400 or 500 and width starts with 7: `=match(P4, "PFC[2345]00?7*")`
length(parameter)	Returns the number of characters in the parameter.	`=length(P1)` returns 8
find(parameter, string)	Returns the order number (starting at zero) of the specified string and -1 if the specified string is not found from the parameter.	`=find(P1, "*")` returns 5
getat(parameter, n)	Returns the n:th (starting at zero) character from the parameter.	`=getat(P1, 1)` returns "L"
setat(parameter, n, character)	Sets the n:th (starting at zero) character to the specified character in the parameter.	`=setat(P1, 0, "B")` returns "BL100*10"
mid(string, n, x)	Returns x characters from the string starting from n:th (starting at zero) character. If you leave out the last argument (x), returns the last part of the string.	`=mid(P1,2,3)` returns "100"
reverse(string)	Reverses the given string.	`=reverse(P1)` returns "01*001LP"

Example 1

To define profile size PL100*10 with two variables P2 = 100 and P3 = 10, enter the formula as follows:

```
="PL"+P2+"*"+P3
```

Example 2

Tekla Structures handles bolt spacings as strings. To define bolt spacing, set **Value type** to **Distance list** and enter the formula as follows:

```
=P1+" "+P2
```

This results in 100 200, if P1 = 100 (**length**) and P2 = 200 (**length**).

Trigonometric functions

Use trigonometric functions to calculate angles.

NOTE When you use trigonometric functions in variable formulas, you need to include a prefix to define the unit. If you do not include a prefix, Tekla Structures uses radians as the default unit.

- d is degree. For example, `sin(d180)`
- r is radians (default). For example, `sin(r3.14)` or `sin(3.14)`

Function	Description	Example
sin()	Returns the sine value	`=sin(d45)` returns 0.71
cos()	Returns the cosine value	`=cos(d45)` returns 0.71
tan()	Returns the tangent value	`=tan(d45)` returns 1.00
asin()	Inverse function of sin(), return value in radians	`=asin(1)` returns 1.571 rad
acos()	Inverse function of cos(), return value in radians	`=acos(1)` returns 0 rad
atan()	Inverse function of tan(), return value in radians	`=atan(1)` returns 0.785 rad
sinh()	Returns the hyperbolical sine value	`=sinh(d45)` returns 0.87
cosh()	Returns the hyperbolical cosine value	`=cosh(d45)` returns 1.32
tanh()	Returns the hyperbolical tangent value	`=tanh(d45)` returns 0.66
atan2()	Returns the angle whose tangent is the quotient of the two numbers. Return value in radians	`=atan2(1,3)` returns 0.32

Market size function

Use the market size function in a custom component to select a suitable plate dimension (usually plate thickness) from the available market sizes. For example, a plate's thickness should match the web of a beam.

Function	Description	Example
fMarketSize(material, thickness, extrastep)	Returns the next available market size for the material from the `marketsize.dat` file, based on the thickness you specify. The file must be in the `..\environments\your_environment\profil` folder or the system folder. For an extra step, enter a number to define the increment to the next size (default is 0).	`=fMarketSize("S235JR", 10, 0)`

Example

In this example, you have the following data in `marketsize.dat`:

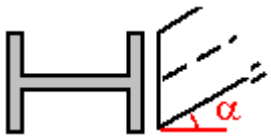
```
S235JR,6,9,12,16,19,22
SS400,1.6,2.3,3.2,4.5,6,9,12,16,19,22,25,28,32,38
DEFAULT,6,9,12,16,19,22,25,28,32,38
```

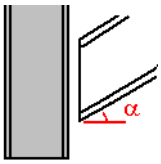
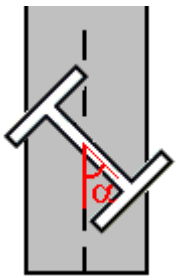
The first item in a row is a material grade followed by available plate thicknesses in millimeters. The DEFAULT line lists the thicknesses available in all other material grades.

With the above data, the function `=fMarketSize("S235JR",10,0)` would return 12, and `=fMarketSize("S235JR",10,1)` would return 16 (one size up).

Framing condition functions

Use the framing condition functions to return the skew, slope, and cant angle of the secondary beam relative to the main part (column or beam).

Function	Description	Example
fAD("skew", GUID)	Returns the skewed angle of the secondary part whose GUID is given. 	`=fAD("skew","ID50B8559A-0000-010B-3133-353432373038")` returns 45 ID50B8559A-0000-010B-3133-353432373038 is the GUID of the secondary part, which is at a 45 degree angle to the main part.

Function	Description	Example
fAD("slope", GUID)	Returns the sloped angle of the secondary part whose GUID is given. 	`=fAD("slope","ID50B8559A-0000-010B-3133-353432373038")`
fAD("cant", GUID)	Returns the cant angle of rotated secondary part whose GUID is given. 	`=fAD("cant","ID50B8559A-0000-010B-3133-353432373038")`

NOTE

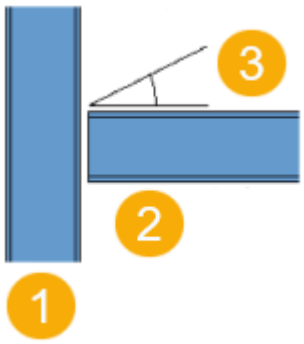
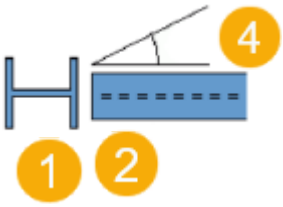
- These functions do not return positive and negative slope and skew values. It is not possible to determine up or down slope and left or right skew with these functions.
- The maximum skew angle to return is 45 degrees.
- Tekla Structures calculates the angles in 2D so that slope and skew are isolated from each other. For example, the skew angle is not taken into consideration when calculating the slope angle, which means that the slope angle value stays the same regardless of the secondary part's rotation around the primary part.

  To find out the true 3D slope with the skew included, you can use the following mathematical formula:

  ```
  TRUE_SLOPE = atan( tan(SLOPE) * cos(SKEW))
  ```

Example 1

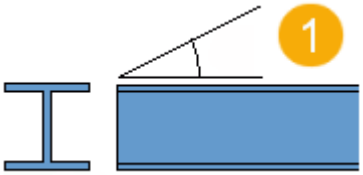
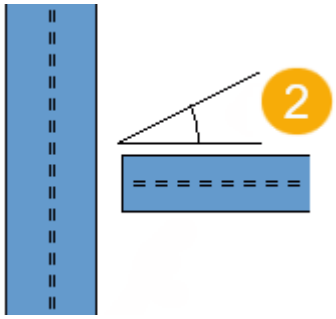
The slope and skew are relative to a beam framing into a column.

Beam to column side view	Beam to column top view
	

1. Column
2. Beam
3. **Slope**
4. **Skew**

Example 2

With two beams, the **slope** is actually the horizontal skew of the beam framing into the other beam, and the vertical slope of the beam relative to the main is actually the **skew** angle.

Beam to beam side view	Beam to beam top view
	

1. **Skew**
2. **Slope**

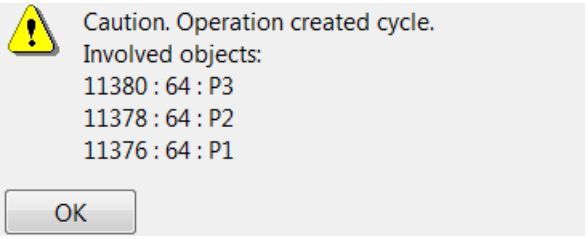
How to avoid cyclic dependencies in variable formulas

Be careful not to create cyclic dependencies between variables, or else the custom component will not work correctly. A cyclic dependency chain contains formulas that make a variable eventually dependent on itself.

In the following example, variable P1 becomes dependent on itself, through variables P2 and P3:

Name	Formula
P1	=P2
P2	=P3/4
P3	=P1*2

Cyclic dependencies may also occur when binding handles to other objects (page 898) or when using magnetic construction planes (page 908). When you create new formulas, bindings, or magnetic construction planes, Tekla Structures checks if they create cyclic dependency chains in a custom component. If that happens, a warning message "`Caution. Operation created cycle.`" is displayed.



Tekla Structures also writes the message "`Cycle detected in parametric solver`" into the session history log file and lists the objects involved in the cyclic dependency, to help you find and remove the cyclic dependency. If you do not remove it, the custom component will not work correctly.

See also

Bind component objects to a plane (create distance variables) (page 898)

Bind component objects using magnetic construction planes or lines (page 908)

Examples of parametric variables and variable formulas in custom components (page 934)

Functions in variable formulas (page 919)

7.4 Tips for creating custom components

- **Enter short, logical names for custom components.**

  Use the description field to describe the component and to explain what it does.

- **Create simple components for specific situations.**

  Simple components are easier and faster to model, and also much easier to use. Avoid creating a single, complex component which you will use for every possible purpose.

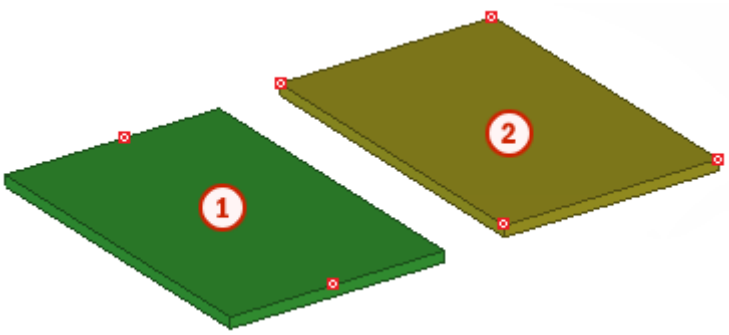
- **Consider creating a separate component model.**

  Use that model when you create and test custom components.

- **Use the simplest part you can with as few handles as possible.**

  For example, if all you need is a rectangular shape, use a rectangular plate, not a contour plate. Rectangular plates only have two handles, so you only need to create a few bindings to manipulate them. Contour plates require more bindings because they have four handles.

  Excessive bindings can cause a slower, less responsive model.



1. Rectangular plate
2. Contour plate

- **Model parts only as accurately as you need.**

  If the only part information required is a part mark in a general arrangement drawing, plus a quantity on a materials list, create a simple bar or plate. If you need to include the part in a detailed view later on, simply re-model the part more accurately at that point.

- **Model embeds as custom parts and include them in components.**

  The majority of embeds are standard embeds, which are designed and manufactured as mass production, and stored to be readily available at the factory. The other type of embeds is a bespoke embed, which is manufactured for a specific project, and needs its own workshop drawing, and which are quantified more accurately.

  To have suitable embeds in you model, you can either

  - create your own embeds as custom parts
  - create your own embeds as items

- use Tekla Warehouse to find embeds.

  If you create your own embed custom components, be careful with the sub-assembly hierarchy. It is better to have a clear sub-assembly main part and then the other parts added to the main part.

- **You can use custom connections in AutoConnections.**
- **You can use custom components in AutoDefaults.**

  Custom components are listed in the **Other components** group. Newly created custom components are not listed in the **Other components** group until you have reopened the model. Custom connections, details and seams work in AutoDefaults.

When you update custom components to a new Tekla Structures version

When you start using a new version of Tekla Structures, always check that custom components created in older versions work correctly in the new version.

When you edit custom components created with an older version of Tekla Structures, and the new version contains improvements that require an update, Tekla Structures asks whether you want to update the component. If you do not update the component, it works in the same manner as in the version where it was originally created, but you do not gain the benefits of the new improvements.

If you choose to update the component, you need to check and sometimes recreate dimensions depending on the improvements. When you delete a dimension and create a new one (even with the same name), the equations that contain the dimension also need to be modified, because the dependency created by the equation is lost when a dimension is deleted. You can recreate dimensions and modify equations in the custom component editor.

See also

Define custom components (page 866)

Edit custom components (page 887)

Add variables to a custom component (page 893)

7.5 Examples of parametric variables and variable formulas in custom components

Here you will find some examples that demonstrate how to use parametric variables and variable formulas to create intelligent custom components that adapt to changes in the model.

NOTE There are some limitations concerning the variable names.

- In some of the examples below, we reference variables by name. To be able to correctly reference a variable in your formula, variable name must be 19 characters or shorter. Variables with longer names will not work correctly when referenced.
- Variable names cannot contain mathematical operators (`+`, `-`, `*`, `/`).
- You cannot use a mathematical constant, such as `PI` or `e`, as a variable name.

The examples are independent from each other.

- Example of a variable formula: Create new component objects (page 942)

  This example shows how to create a parametric variable that adds bolts to the custom component.

- Example of a variable formula: Change the end plate material (page 943)

  This example shows how to link a parametric variable to the end plate material of a component object.

- Example of a variable formula: Replace a sub-component in a nested component (page 949)

  This example shows how to create a parametric variable that replaces sub-components with other sub-components.

- Example of a variable formula: Modify a sub-component by using a component attribute file in a nested component (page 951)

  This example shows how to create a parametric variable that modifies a sub-component on the basis of a component attribute file.

- Example of a variable formula: Define the bolt size and bolt standard (page 953)

  This example shows how to create two parametric variables that determine the bolt size and bolt standard.

- Example of a variable formula: Calculate the bolt group distance from the beam flange (page 955)

  This example shows how to create a variable formula that calculates the bolt group distance from the beam flange.

- Example of a variable formula: Calculate the number of bolt rows (page 958)

  This example shows how to create a variable formula that calculates the number of bolt rows based on the beam height. You will use `if` statements in the calculations.

- Example of a variable formula: Define the stiffener position using construction planes (page 960)

  This example shows how to use construction planes for determining the position of the stiffeners. You will position the stiffeners so that they divide the beam into three equally long sections.
- Example of a variable formula: Link parametric variables to user-defined attributes (page 964)

  This example shows how to link parametric variables to the user-defined attributes of panels. You can then use the user-defined attributes in view filters to show or hide the panels.
- Example of a variable formula: Calculate the number of handrail posts using a template attribute (page 966)

  This example shows how to create a variable formula that calculates the number of handrail posts based on the length template attribute of the beam. The handrail posts are created at both ends of the beam and one of them is copied with the **Array of objects (29)** component.
- Example of a variable formula: Create a custom stiffener detail with variables (page 970)

  This example shows how to create a custom stiffener detail with variables that control the shape and position of the stiffeners.
- Example of a variable formula: Link an Excel spreadsheet to a custom component (page 980)

  This example shows how to link a parametric variable to an Excel spreadsheet. For example, you can use Excel spreadsheets to check connections.
- Examples of a variable formula: Rebar set modifiers in custom components (page 981)

  These examples show you how to use rebar set modifiers for defining the properties and hooks of rebar set bars in custom components.

Variable properties in custom components

Use the **Variables** dialog in custom components to view, modify, and create parametric variables, and to view fixed and reference distance variables.

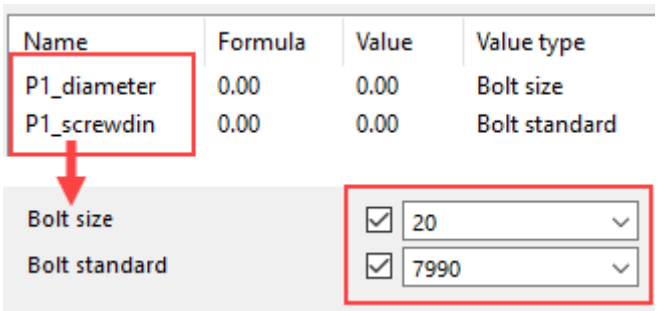
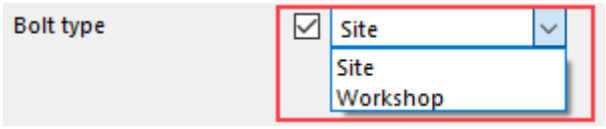
In addition to custom components, Tekla Structures uses variables with sketched cross sections and in parametric modeling. The examples below are

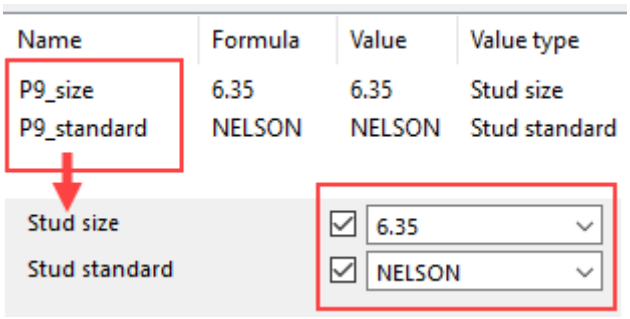
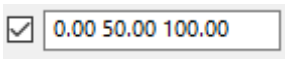
given for custom components, but the same principles apply also to sketched cross sections and parametric modeling.

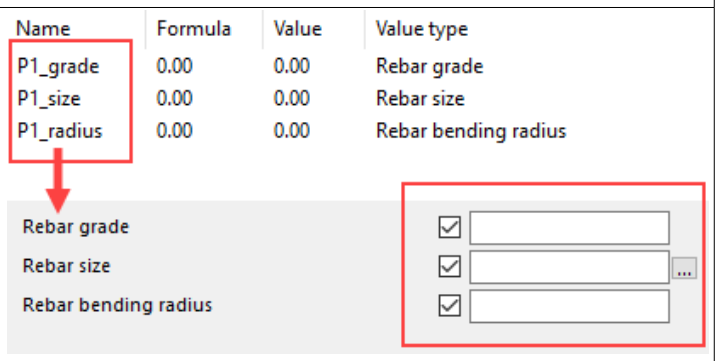
Option	Description
Category	**Component parameters** lists all variables in the component. **Model parameters** lists variables in the current model (such as bindings between a part's end point and a grid plane).
Name	A unique name of a variable. Use this name to refer to the variable in the custom component editor. To be able to correctly reference a variable, its name must be 19 characters or shorter. Variables with longer names will not work correctly when referenced.
Formula	Use this box to enter a value or a formula. For more information on formulas, see Create a variable formula (page 917). Formulas begin with =.
Value	Shows the current value of **Formula**.
Value type	Select a value type from the list. The type determines what kind of value you can enter for the variable.
Variable type	This property can be either **Distance** or **Parametric**.
Visibility	Use this setting to control the visibility of a variable. Set to **Show** to display the variable in the custom component dialog.
Label in dialog box	The name of the variable that Tekla Structures displays in the custom component dialog. The maximum length is 30 characters.

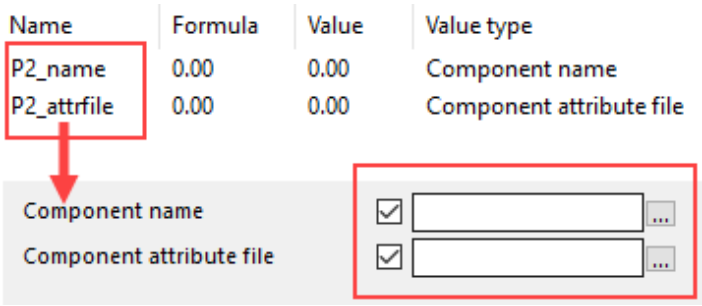
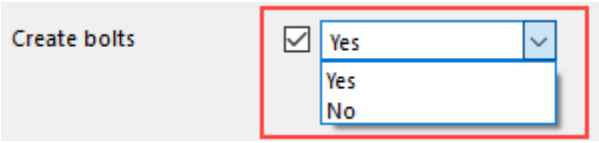
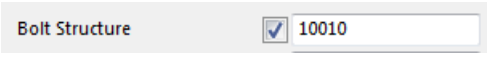
Value types

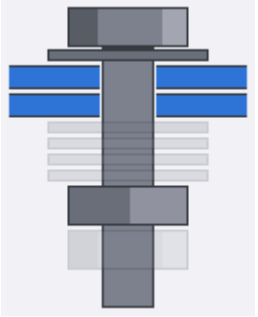
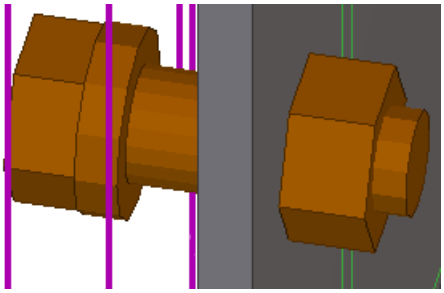
Option	Description
Number	A whole (integer) number. Use for quantity and multiplier.
Length	A decimal (floating point) number. Use for lengths and distances. Length numbers have unit (mm, inch, etc.) and they are rounded to two decimal places.

Option	Description
Text	A text (ASCII) string.
Factor	A decimal value without a unit. You can set the number of decimals for the value type in **File** --> **Settings** --> **Options** --> **Units and decimals**.
Angle	A decimal number type for storing angles, stored to one decimal place, in radians.
Material	A data type associated with the material catalog. Use to select a material grade from the **Select Material** dialog.
Profile	A data type associated with the profile catalog. Use to select a profile from the **Select Profile** dialog.
Bolt size **Bolt standard**	Data types linked to the bolt catalog. **Bolt size** works with **Bolt standard**. They have a fixed naming format: `Px_diameter` and `Px_screwdin`. Do not change the fixed name. To show values for these in the component's dialog, `x` must be the same for both, for example, `P1_diameter` and `P1_screwdin`. 
Bolt type	For determining the bolt type (site/workshop) in the custom component dialog. Linked to the **Bolt type** property of bolts in the **Custom component browser**. 
Stud size **Stud standard** **Stud length**	Data types linked to the bolt catalog. **Stud size**, **Stud standard** and **Stud length** work together. They have a fixed naming format: `Px_size`, `Px_standard` and `Px_length`. Do not change the fixed names. To show values for these in the component's dialog, `x` must be the same for all of

Option	Description
	them. For example, `P9_size`, `P9_standard`, and `P9_length`. 
Hole type	A data type for determining whether holes are special holes and for selecting the special hole type (slotted/oversized/tapped/no hole).
Plain hole type	A data type for determining whether holes are through or blind holes.
Distance list	Use with options that have several length values, such as bolt spacings. Use space as a separator between the distances. 
Distance list total	Use for calculating the total value of several listed length values, such as bolt spacings. Use space as a separator between the distances.
Weld type	A data type for selecting the weld type. 
Chamfer type	A data type for determining the shape of a chamfer. For more information, see Chamfer part corners and edges (page 420).
Welding site	A data type for determining the welding place: workshop or building site.
Rebar grade **Rebar size** **Rebar bending radius**	Data types linked to reinforcement catalog. **Rebar grade**, **Rebar size**, and **Rebar bending radius** work together. They have a fixed naming format: `Px_grade`, `Px_size`, and `Px_radius`. Do not change the fixed name. To show values for these in the component's dialog, `x` must be the same for all, for example, `P1_grade`, `P1_size`, and `P1_radius`.

Option	Description
	
Rebar hook type	Used for rebar set end detail modifiers, to specify the hook type.
Length adjustment type	Used for rebar set end detail modifiers, to specify how the bar length is extended or shortened, according to a specified end offset or leg length.
Bars affected	Used for rebar set modifiers, to specify how many bars are to be modified (1/1, 1/2, and so on).
Rebar stagger type	Used for rebar set splitters, to specify the stagger type (left/right/middle).
Rebar lap side	Used for rebar set splitters, to specify the side of the overlap (left/right/middle).
Rebar lap placement	Used for rebar set splitters, to determine whether the lapping bars are parallel to each other or on top of each other.
Rebar lap type	Used for rebar set splitters, to determine whether the reinforcing bars are kept straight at lap splices by offsetting entire bars, or placed slanted by offsetting bar ends.
Reinforcement mesh	For determining meshes in custom components. Linked to the **Catalog name** property of reinforcement meshes in the **Custom component browser**.
Cross bar location	Used for rebar meshes, to determine whether the crossing bars are located above or below the longitudinal bars.
Component name **Component attribute file**	Use **Component name** for replacing a sub-component inside a custom component with another sub-component. Linked to the **Name** property of objects in the **Custom component browser**. Use **Component attribute file** for setting the properties of a sub-component inside a custom component.

Option	Description
	Component name and **Component attribute file** work together. They have a fixed naming format: `Px_name` and `Px_attrfile`. Do not change the fixed name. To show values for these in the component's dialog, `x` must be the same for both, for example, `P2_name` and `P2_attrfile`. 
Yes/No	For determining whether or not Tekla Structures creates an object in a custom component. Linked to the **Creation** property of objects in the **Custom component browser**. 
Shape	A data type associated with the shape catalog. Use for selecting a shape from the **Shape catalog** dialog.
Bitmask	For defining bolt assembly (nuts and washers) and parts with special holes. Linked to the **Bolt structure** and **Parts with special holes** properties of bolts in the **Custom component browser**. The value is a five-digit series of ones and zeros. This relates to the checkboxes in the bolt properties. 1 means that a checkbox is selected, 0 means that a checkbox is clear. In the example below, the value of 10010 means that a bolt with a washer and a nut is created in the bolt assembly. 

Option	Description
	 

See also

Examples of parametric variables and variable formulas in custom components (page 934)

Add variables to a custom component (page 893)

Example of a variable formula: Create new component objects

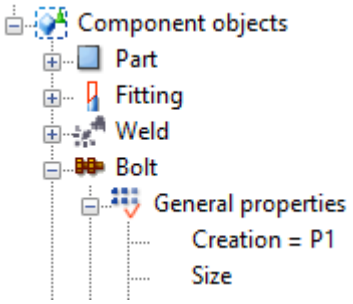
This example shows how to create a parametric variable that adds bolts to the custom component.

1. In the model, select the custom component, right-click, and then select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog opens.
3. To create a new parametric variable, click **Add**.
4. Modify the parametric variable as follows:
   a. From the **Value type** list, select **Yes/No**.
   b. In the **Label in dialog box** box, enter `Create bolts`.

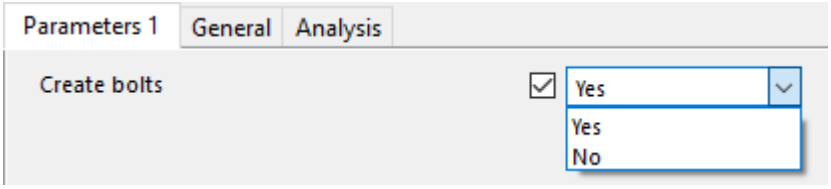
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Yes/No	Parameter	Show	Create bolts

5. Link the parametric variable to the **Creation** property of the bolts.
   a. In a custom component view, select the bolt group to highlight it in the **Custom component browser**.
   b. In the **Custom component browser**, browse for **Bolt**.
   c. Right-click **Creation** and select **Add equation**.
   d. Enter `P1` after the equal sign, and then press **Enter**.



6. Save the custom component.
7. Close the custom component editor.
8. In the model, select the custom component, and open the custom component dialog.

   The option to create bolts is visible in the custom component's dialog.



See also

Edit custom components (page 887)

Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Change the end plate material

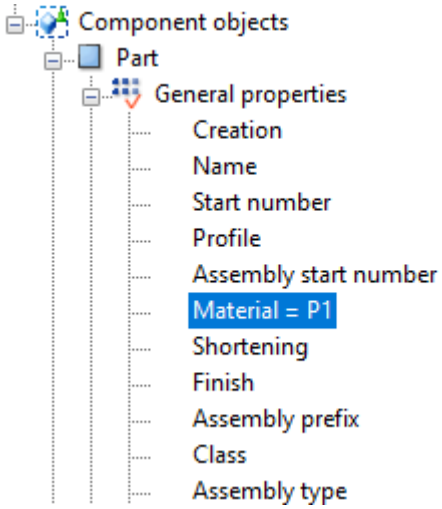
This example shows how to link a parametric variable to the end plate material of a component object so that you change the material if needed.

1. In the model, select the custom component, right-click, and then select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog opens.
3. To create a new parametric variable, click **Add**.
4. Modify the parametric variable as follows:
   a. From the **Value type** list, select **Material**.
   b. In the **Label in dialog box** box, enter `End plate material`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0.00	0.00	Material	Parameter	Show	End plate material

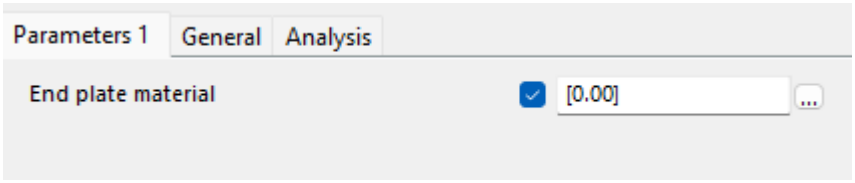
5. Link the parametric variable to the **Material** property of the end plate.
   a. In the **Custom component browser**, browse for the **Material** property of the end plate.
   b. Right-click **Material** and select **Add equation**.
   c. Enter `P1` after the equal sign, and then press **Enter**.



6. Save the custom component.
7. Close the custom component editor.

8. Select the custom component in the model, and open the custom component dialog.

   The option to change the end plate material is in the custom component's dialog.



9. To change the material, click the **...** button next to the name box.
10. Select the material from material catalog, and in the custom component dialog click **Modify**.

    Tekla Structures changes the end plate material in the custom component.

See also

Edit custom components (page 887)

Add variables to a custom component (page 893)

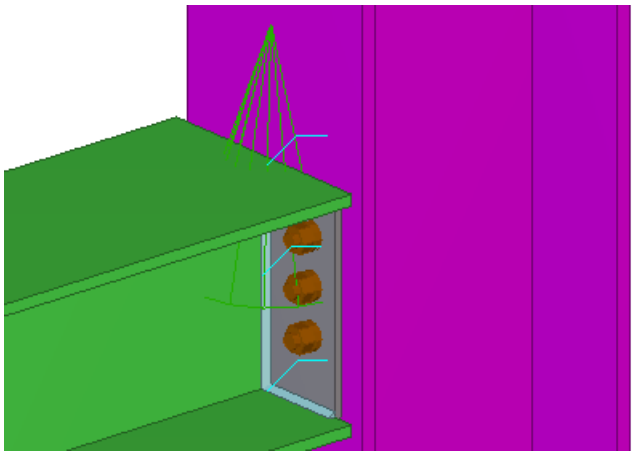
Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

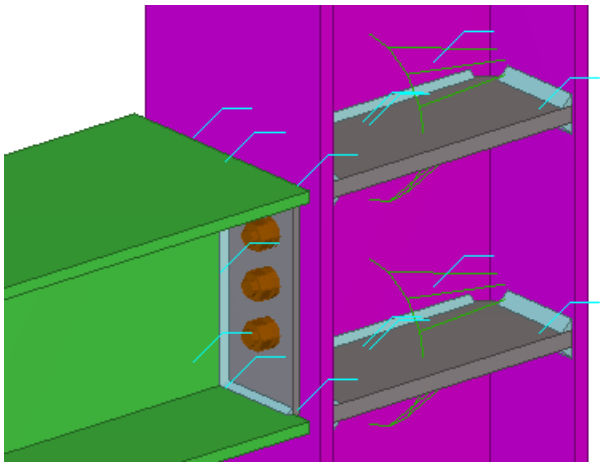
Example of a variable formula: Create a nested custom connection with stiffeners

This example shows how to create a nested custom connection that consists of an end plate, a bolt group, welds, and two **Stiffeners (1003)** components. The stiffeners are optional, which means that you can choose whether or not to create them when using the component in a model.

1. Add an **End plate (144)** component to the model.



2. Explode the end plate component.
3. Add two **Stiffeners (1003)** components.



4. Create a nested custom component that contains the stiffeners and the end plate objects.

   a. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

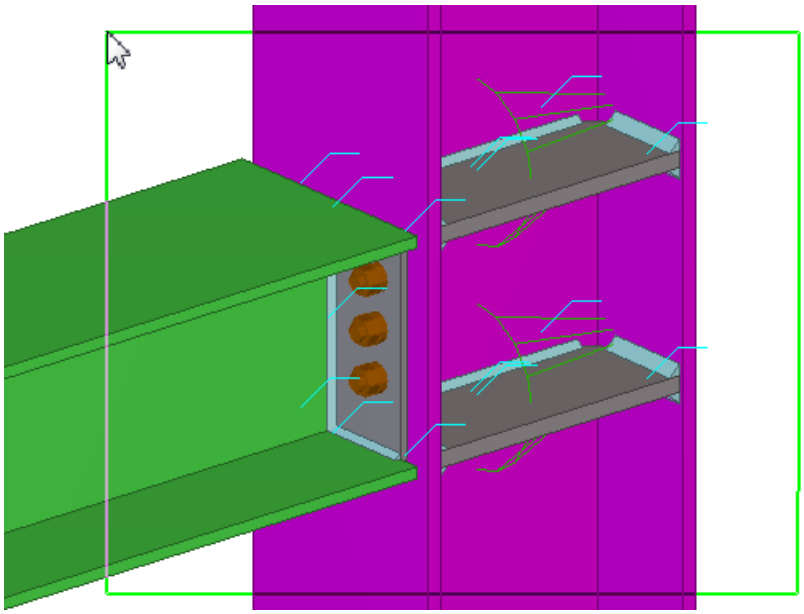
   b. Click the **Access advanced features** button and select **Define custom component...**.

   c. From the **Type** list, select **Connection**.

   d. In the **Name** box, enter `End plate with stiffeners`.

   e. Click **Next >**.

   f. Make an area selection (from right to left) to include the following objects in the nested component: the column, the beam, the stiffener components, and all the end plate objects.

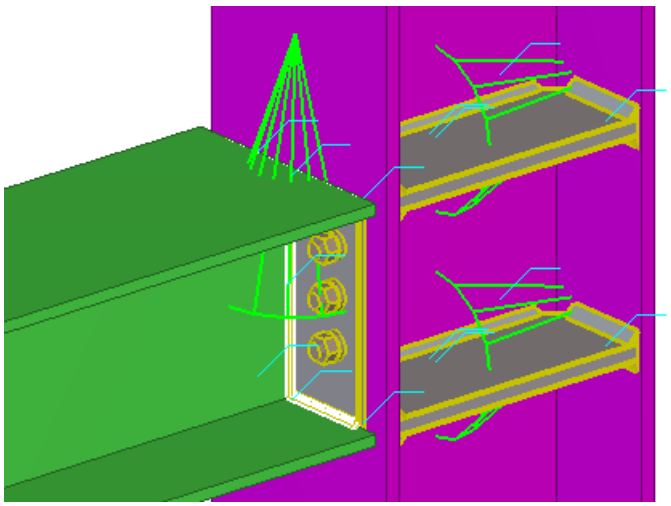


g. Click **Next >**.

h. Choose the column as the main part of the nested component, and then click **Next >**.

i. Choose the beam as the secondary part of the nested component, and then click **Finish**.

Tekla Structures creates the nested custom component.



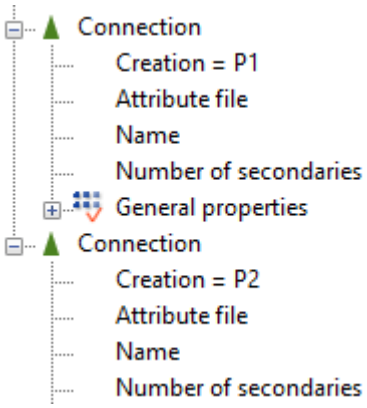
5. In the model, select the nested custom component you just created.
6. Right-click and select **Edit custom component**.
7. On the **Custom component editor** toolbar, click the **Display variables** button.

The **Variables** dialog opens.

8. To create two new parametric variables, click **Add** twice.
9. Modify the first parametric variable **P1** as follows:
   a. In the **Value type** list, select **Yes/No**.
   b. In the **Label in dialog box** box, enter `Create stiffener 1`.
10. Modify the second parametric variable **P2** as follows:
   a. From the **Value type** list, select **Yes/No**.
   b. In the **Label in dialog box** box, enter `Create stiffener 2`.

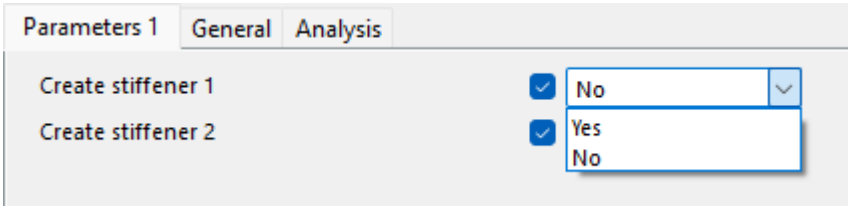
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Yes/No	Parameter	Show	Create stiffener 1
P2	0	0	Yes/No	Parameter	Show	Create stiffener 2

11. Link the parametric variables to the **Creation** property of the two stiffeners:
   a. In the **Custom component browser**, browse for the uppermost **Connection**.
   b. Right-click **Creation** and select **Add equation**.
   c. Enter `P1` after the equal sign, and then press **Enter**.
   d. Browse for the second **Connection**.
   e. Right-click **Creation** and select **Add equation**.
   f. Enter `P2` after the equals sign, and then press **Enter**.



12. Save the custom component.
13. Close the custom component editor.
14. Select the custom component in the model, and open the custom component dialog.

The option to create stiffeners is visible in the custom component's dialog.



See also

Edit custom components (page 887)

Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Replace a sub-component in a nested component

This example shows how to create a parametric variable that replaces a sub-component with another one in a nested custom component.

As a prerequisite, you need to have a nested custom component that includes a sub-component (in this case `My connection`). Replace the sub-component with another sub-component (`My connection 2`).

1. In the model, select the custom component, right-click, and then select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog opens.
3. To create a new parametric variable, click **Add**.
4. Modify the parametric variable as follows:
   a. From the **Value type** list, select **Component name**.

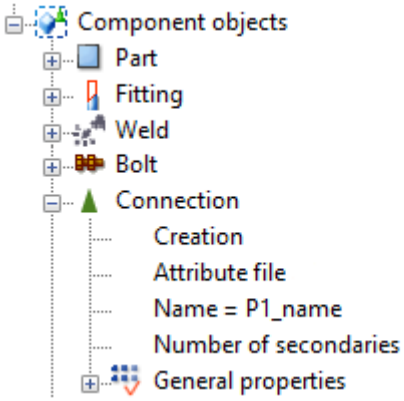
      Tekla Structures automatically adds the suffix `_name` to the variable name. Do not delete the suffix.

   b. In the **Formula** box, enter the name of sub-component you want to replace, in this case `My connection`.

   c. In the **Label in dialog box** box, enter a suitable name for the label which will be visible in the custom component dialog, for example, `New connection`.

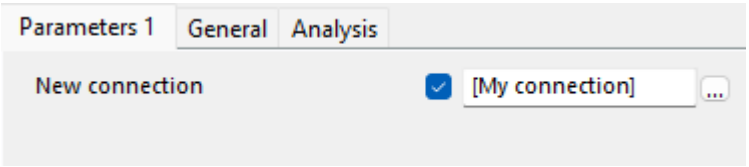
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1_name	My connection	My connection	Component name	Parameter	Show	New connection

5. Link the parametric variable to the **Name** property of the sub-component:
   a. In the **Custom component browser**, browse for the **Name** property of the sub-component.
   b. Right-click **Name** and select **Add equation**.
   c. Enter `P1_name` after the equal sign, and press **Enter**.



6. Save the custom component.
7. Close the custom component editor.
8. Select the custom component in the model, and open the custom component dialog.

   The option to change the sub-component is visible in the custom component's dialog.



9. To change the sub-component, click the **...** button next to the name box.
10. In the **Applications & components** catalog, select the `My connection 2` sub-component. It will replace the `My connection` in the custom component.
11. In the custom component dialog, click **Modify**.

    Tekla Structures changes the sub-component in the nested custom component.

See also

Edit custom components (page 887)

Add variables to a custom component (page 893)

Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Modify a sub-component by using a component attribute file in a nested component (page 951)

Example of a variable formula: Modify a sub-component by using a component attribute file in a nested component

This example shows how to create a parametric variable that modifies a sub-component in a nested custom component on the basis of a component attribute file.

In this example you will use the value types **Component name** and **Component attribute file**, which work together. Use **Component name** for replacing a sub-component inside a custom component with another sub-component, and **Component attribute file** for selecting a different property file for the sub-component.

Component name and **Component attribute file** follow a fixed naming format: `Px_name` and `Px_attrfile`. Do not change the fixed name.

To show values for these in the custom component's dialog, `x` must be the same for both, for example, `P2_name` and `P2_attrfile`.

1. In the model, select the custom component, right-click, and then select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog opens.
3. To create two new parametric variables, click **Add** twice.
4. Modify the first parametric variable as follows:
   a. From the **Value type** list, select **Component name**.

      Tekla Structures automatically adds the suffix `_name` to the name of the variables. Do not delete the suffix.

   b. In the **Label in dialog box** box, enter `New component`.
5. Modify the second parametric variable as follows:
   a. From the **Value type** list, select **Component attribute file**.

      Tekla Structures automatically adds the suffix `_attrfile` to the name of the variable. Do not delete the suffix.

   b. In the **Name** box, change the prefix of the second variable so that the prefixes for the two variables are the same.

In this example, the prefix is P1.

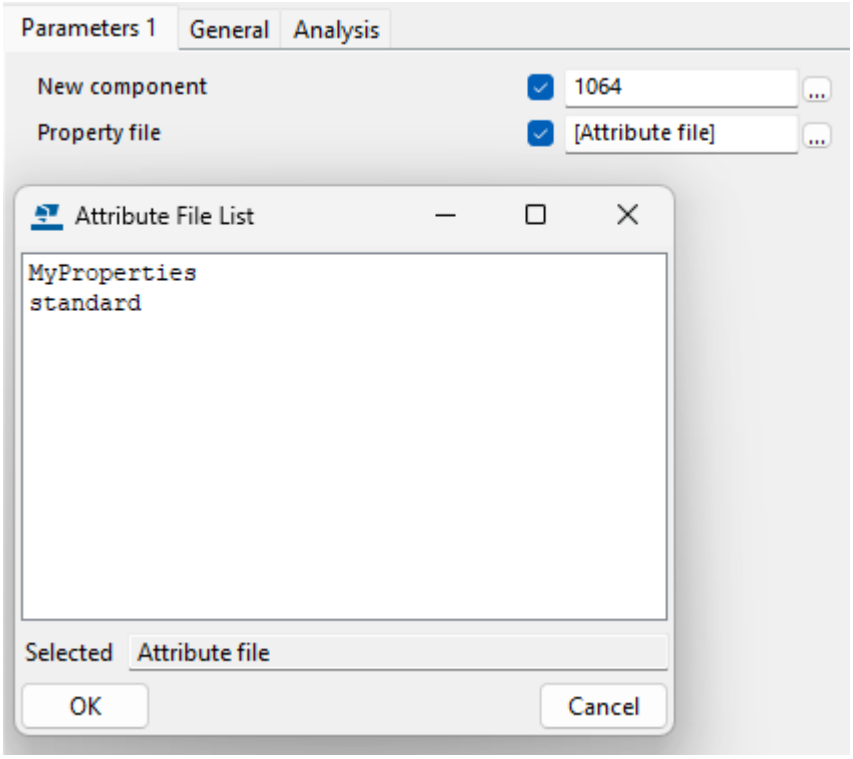
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1_name	Component	Component	Component name	Parameter	Show	New component
P1_attrfile	Attribute file	Attribute file	Component attribute file	Parameter	Show	Property file

NOTE The component name and component attribute file variables must always have the same prefix, otherwise they do not work.

c. In the **Label in dialog box** box, enter `Property file`.

6. Link the parametric variables to the **Name** and **Attribute file** properties of the nested component:
   a. In the **Custom component browser**, browse for the **Name** and **Attribute file** properties of the bolt group.
   b. Right-click **Name** and select **Add equation**.
   c. Enter `P1_name` after the equal sign, and then press **Enter**.
   d. Right-click **Attribute file** and select **Add equation**.
   e. Enter `P1_attrfile` after the equals sign, and then press **Enter**.
7. Save the custom component.
8. Close the custom component editor.
9. In the model, select the custom component, and open the custom component dialog.
10. To change the sub-component, click the **...** button next to the name box.
11. In the **Applications & components** catalog, select a suitable sub-component.
12. To change the property file, click the **...** button next to the name box.
13. In the **Attribute file list** dialog, select a different attribute file.

You can now change the sub-component and its property file directly from the custom component's dialog.



14. In the custom component dialog, click **Modify**.

    Tekla Structures changes the sub-component and its attribute file in the nested custom component.

See also

Edit custom components (page 887)

Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Replace a sub-component in a nested component (page 949)

Example of a variable formula: Define the bolt size and bolt standard

This example shows how to create two parametric variables that define the bolt size and bolt standard.

In this example you will use the value types **Bolt size** and **Bolt standard**, which work together, and are linked to the bolt catalog. They have a fixed naming format: `Px_diameter` and `Px_screwdin`. Do not change the fixed name.

To show values for these in the custom component's dialog, `x` must be the same for both, for example, `P1_diameter` and `P1_screwdin`.

1. In the model, select the custom component, right-click, and then select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog opens.
3. To create two new parametric variables, click **Add** twice.
4. Modify the first parametric variable as follows:
   a. From the **Value type** list, select **Bolt size**.

      Tekla Structures automatically adds the suffix `_diameter` to the name of the variables. Do not delete the suffix.
   b. In the **Label in dialog box** box, enter `Bolt size`.
5. Modify the second parametric variable as follows:
   a. From the **Value type** list, select **Bolt standard**.

      Tekla Structures automatically adds the suffix `_screwdin` to the name of the variable. Do not delete the suffix.
   b. In the **Name** box, change the prefix of the second variable so that the prefixes for the two variables are the same.

      In this example, the prefix is P1.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1_diameter	0.00	0.00	Bolt size	Parameter	Show	Bolt size
P1_screwdin	0	0	Bolt standard	Parameter	Show	Bolt standard

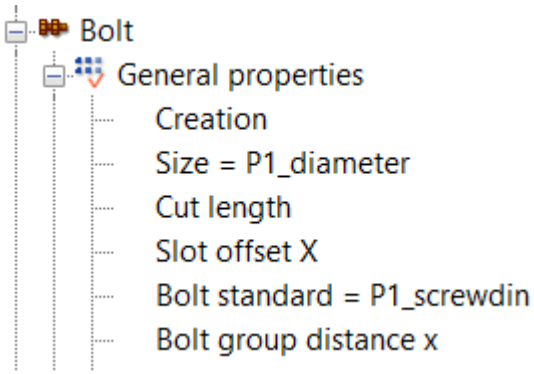
   NOTE The bolt size and bolt standard variables must always have the same prefix, otherwise they do not work.

   c. In the **Label in dialog box** box, enter `Bolt standard`.
6. Link the parametric variables to the **Size** and **Bolt standard** properties of the bolt group:
   a. In the **Custom component browser**, browse for the **Size** and **Bolt standard** properties of the bolt group.
   b. Right-click **Size** and select **Add equation**.

c. Enter `P1_diameter` after the equals sign, and then press **Enter**.

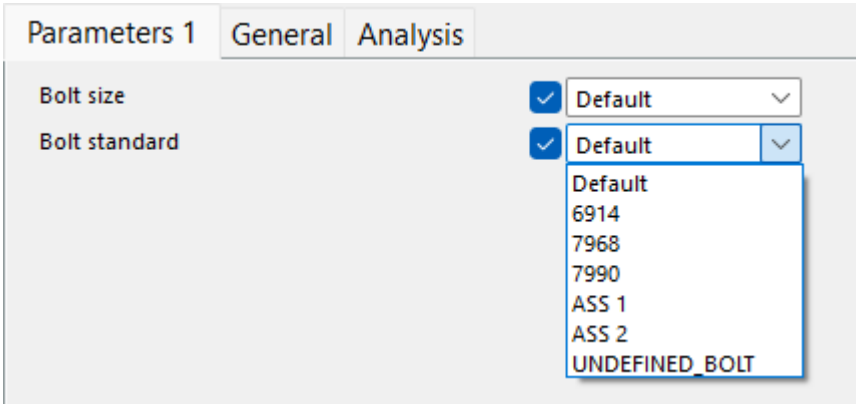
d. Right-click **Bolt standard** and select **Add equation**.

e. Enter `P1_screwdin` after the equals sign, and then press **Enter**.



7. Save the custom component.
8. Close the custom component editor.
9. Select the custom component in the model, and open the custom component dialog.

   The option to define the bolt size and bolt standard is visible in the custom component's dialog.



See also

Edit custom components (page 887)

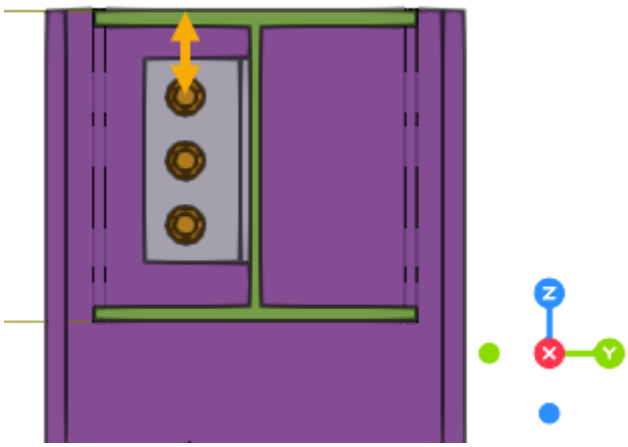
Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Calculate the bolt group distance from the beam flange

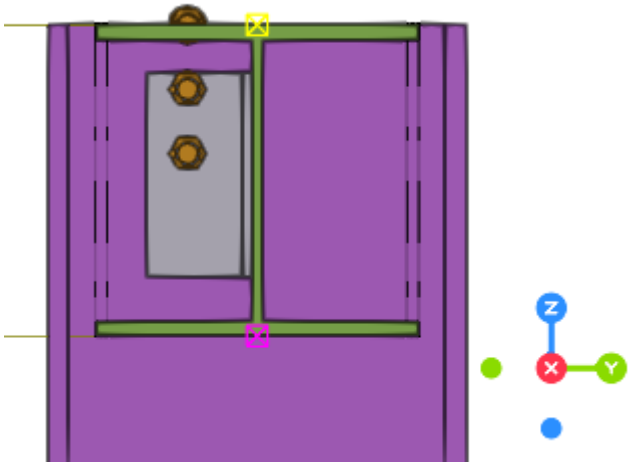
This example shows how to create a variable formula that calculates the distance from the secondary beam top flange to the first bolt in the bolt group.



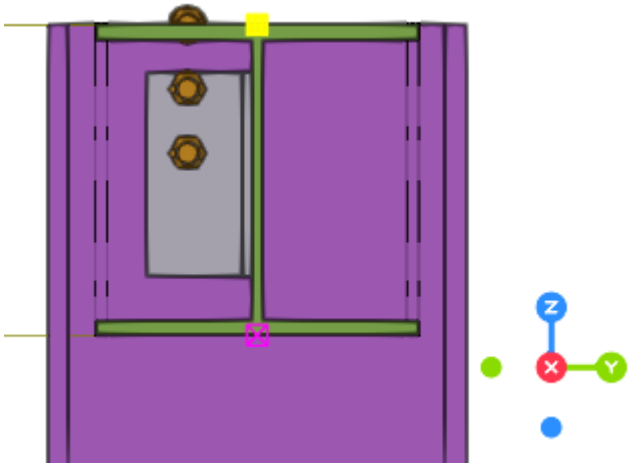
1. Modify the bolt group properties as follows:
   a. In a custom component view, double-click the bolt group.

      The **Bolt** properties open in the property pane.
   b. Clear all values that are in the **Offset from** section.
   c. Click **Modify** in the property pane.

      The bolt group moves to the same level with the start point handle of the bolt group.

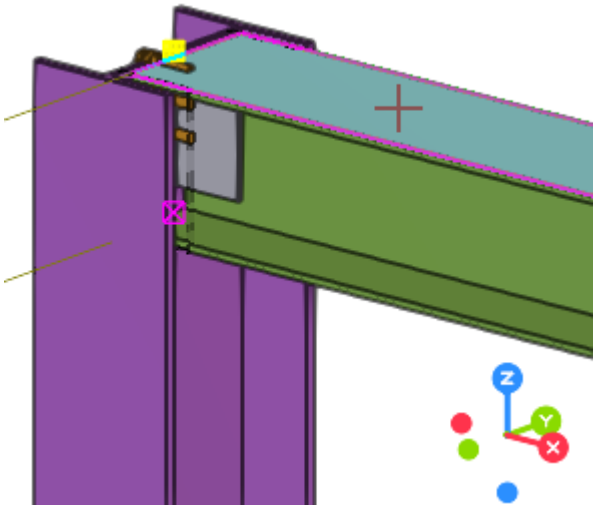


2. Bind the bolt group to the secondary beam top flange:
   a. In a custom component view, select the bolt group.
   b. Select the yellow top handle.



c. Right-click the handle and select **Bind to plane**.

d. Select the top flange of the secondary beam.



A new distance variable is created.

3. On the **Custom component editor** toolbar, click the **Display variables** button.

   The **Variables** dialog opens, and it includes the newly created distance variable.

4. To add a new parametric variable, click **Add**.

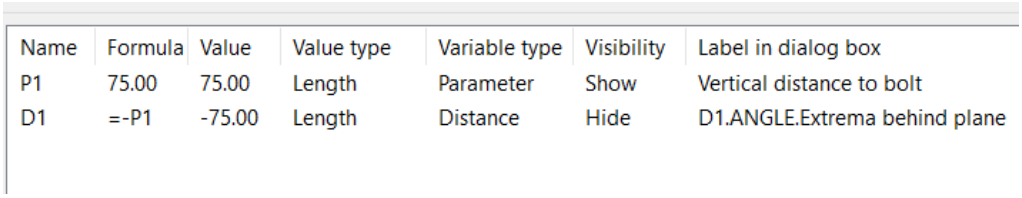
5. Modify the variable **P1** as follows:

   a. In the **Formula** box, enter a distance value.

   b. In the **Label in dialog box** box, enter a suitable label, for example, `Vertical distance to bolt`.

6. Modify the distance variable **D1** as follows:

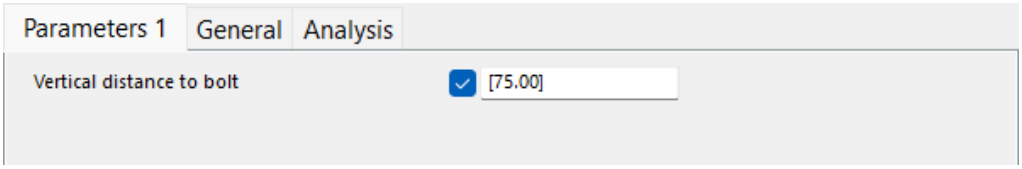
a. In the **Formula** box, enter `=-P1`.



Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	75.00	75.00	Length	Parameter	Show	Vertical distance to bolt
D1	=-P1	-75.00	Length	Distance	Hide	D1.ANGLE.Extrema behind plane

7. Save the custom component.
8. Close the custom component editor.
9. Select the custom component in the model, and open the custom component dialog.

   You can now determine the bolt group distance from the secondary beam flange by modifying the value in the custom component's dialog.



See also

Edit custom components (page 887)

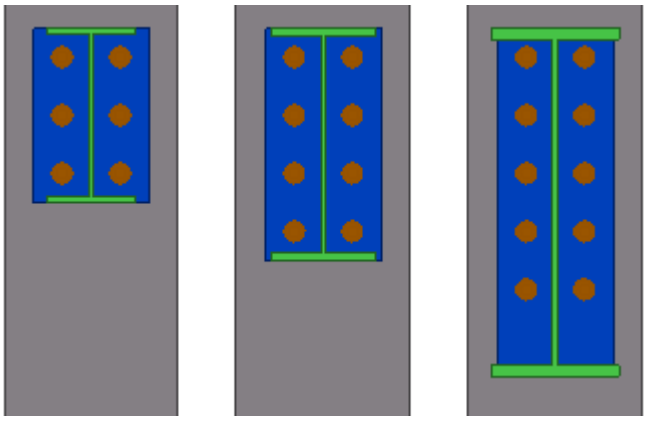
Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Calculate the number of bolt rows

This example shows how to create a variable formula that calculates the number of bolt rows based on the beam height. You will use `if` statements in the calculations.



1. In the model, select the custom component, right-click, and then select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables** button.

   The **Variables** dialog opens.
3. To create two new parametric variables, click **Add** twice.
4. Modify the variable **P1** as follows:
   a. In the **Value type** list, select **Number**.
5. In the **Custom component browser**, browse for the height property of the beam.
6. Right-click **Height** and select **Copy reference**.

   Tekla Structures copies the reference to beam height to the clipboard.
7. In the **Variables** dialog, modify the variable **P1** as follows:
   a. In the **Formula** box, enter the following `if` statement for the parametric variable:

```
=if (fP(Height,"b9fff400-5cdf-45b8-8376-93b6c4690679")< 301) then 2 else
(if (fP(Height,"b9fff400-5cdf-45b8-8376-93b6c4690679")>501) then 4 else 3
endif) endif
```

   In the formula, `fP(Height,"b9fff400-5cdf-45b8-8376-93b6c4690679")` is the beam height reference, including the beam GUID, copied from the **Custom component browser**. The variable gets its value in the following way:
   - If the beam height is less than 301 mm, the value is 2.
   - If the beam height is greater than 501 mm, the value is 4.
   - If the beam height is between 300 and 500 mm, the value is 3.
8. Modify the variable **P2** as follows:
   a. From the **Value type** list, select **Distance list**.

b. In the **Formula** box, enter `=P1+"*"+100`

In the formula, `100` is the bolt spacing and the `P1` value is the number of bolt rows.

Name	Formula	Value	Value type
P1	=if (fP(Height,"b9fff40...	2	Number
P2	=P1+"*"+100	2*100.00	Distance list

9. In the **Custom component browser**, browse for the **Bolt group distance x** property.
   a. Right-click **Bolt group distance x** and select **Add equation**.
   b. Enter `P2` after the equal sign, and then press **Enter**.
10. Save the custom component.
11. Close the custom component editor.
12. Select the custom component in the model, and open the custom component dialog.

    When you change the beam height, the number of bolt rows changes as well.

See also

Edit custom components (page 887)

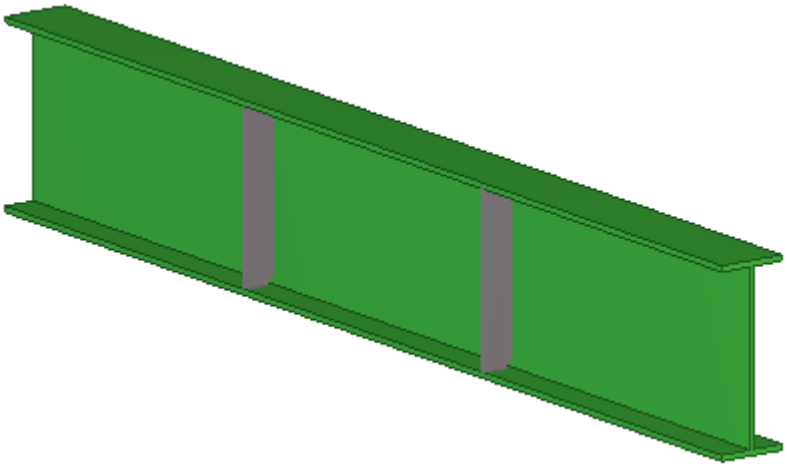
Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Define the stiffener position using construction planes

This example shows how to use construction planes for defining the position of the stiffeners. You will position two stiffeners so that they divide the beam into three equally long sections.



1. To make it easier to select the handles (page 117), ensure that **Direct modification** is switched off.
2. On the **Custom component editor** toolbar, click the **Display variables** button.

   The **Variables** dialog opens.
3. To create two new parametric variables, click **Add** twice.
4. Inquire the GUID of the beam:
   a. Right-click the beam in a custom component view and select **Inquire** .
   b. In the **Inquire object** dialog, copy the beam's GUID.

   GUID: 88b06118-6035-4278-9a66-26f02bcfb45f
5. In the **Variables** dialog, modify the first parametric variable **P1** as follows:
   a. In the **Formula** box, enter `=fTpl("LENGTH","b9fff400-5cdf-45b8-8376-93b6c4690679")` .

      In the formula, `b9fff400-5cdf-45b8-8376-93b6c4690679` is the GUID of the beam.

      Reference functions consist of the following syntax: `fTpl("template attribute", "object GUID")`. You need to always check the GUID of the object that you use with the reference function.

      The value of the variable is now the same as the beam length. If you change the beam length, also the value changes.
   b. In the **Label in dialog box** box, enter `Beam length`.
6. Modify the second parametric variable **P2** as follows:

a. In the **Formula** box, enter `=P1/3`.

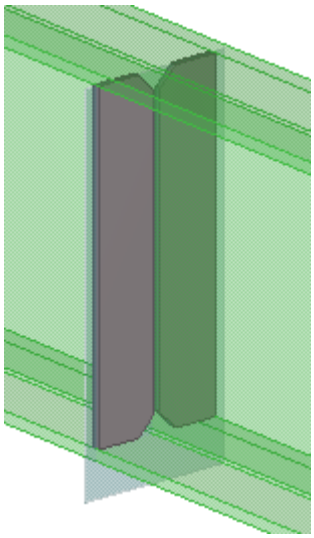
b. In the **Label in dialog box** box, enter `3rd point`.

7. Create a construction plane:

   a. On the **Custom component editor** toolbar, click the **Create construction plane** button.

   b. Pick the required points to define the shape of the construction plane.

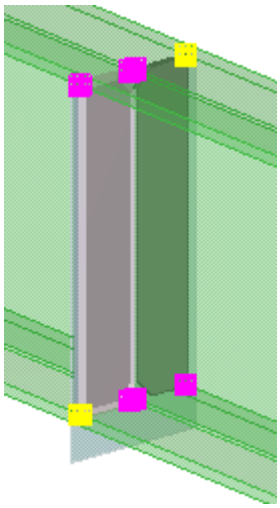
   c. Click the middle mouse button to create a construction plane at the center of a stiffener at one end.



8. Bind the stiffener to the construction plane:

   a. Select the stiffener.

   b. Hold down **Alt** and use area selection (from left to right) to select all stiffener handles.



c. Right-click and select **Bind to plane**.

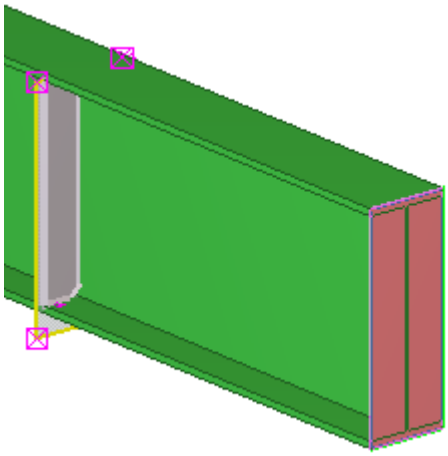
d. Bind the stiffener handles to the construction plane.

9. Bind the construction plane to the beam end:

   a. Select the construction plane.

   b. Right-click and select **Bind to plane**.

   c. Bind the construction plane to the beam end.



10. Repeat the above steps for the stiffener at the other end:

    a. Create a construction plane.

    b. Bind the stiffener to the construction plane.

    c. Bind the construction plane to the beam end.

    You have now created two distance variables that bind the construction planes to the beam end.

11. Modify the distance variables as follows:

    a. In the **Formula** box, enter `=P2`.

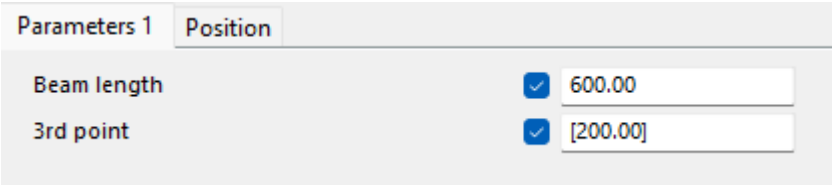
       Do this for both of the distance variablesthat bind the construction planes to the beam end.

12. Save the custom component.

13. Close the custom component editor.

14. Select the custom component in the model, and open the custom component dialog.

If you now change the beam length, the position of the stiffeners changes so that the stiffeners divide the beam into three equally long sections.



See also

Edit custom components (page 887)

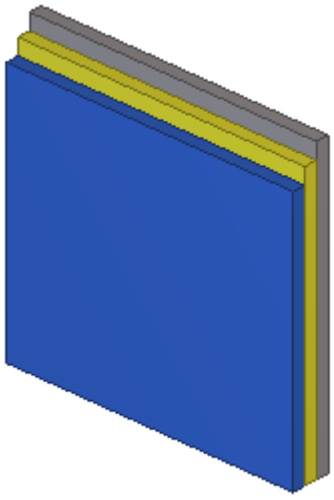
Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Link parametric variables to user-defined attributes

This example shows how to link parametric variables to the user-defined attributes of panels in a custom component. You can then use the user-defined attributes in view filters to show or hide the panels in the model.



1. In the model, select the custom component, right-click, and then select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables** button.

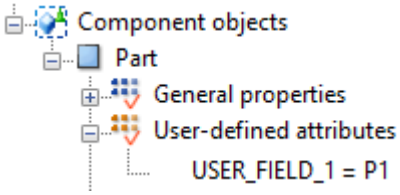
   The **Variables** dialog opens.

3. To create a new parametric variable, click **Add**.
4. Modify the variable as follows:
   a. From the **Value type** list, select **Text**.
   b. In the **Formula** box, enter `Type1`.
   c. In the **Label in dialog box** box, enter `Panel1`.
5. In the **Custom component browser**, browse for the user-defined attributes of the first panel.

   You will link the **P1** variable to the **USER_FIELD_1** property. However, the property is not visible in the **Custom component browser**.
6. Make the **USER_FIELD_1** property visible in the **Custom component browser**:
   a. Double-click the first panel in custom component view.

      The panel properties open in the property pane.
   b. Under **More**, click the **User-defined attributes** buttton to open the dialog for user-defined attributes.
   c. Go to the **Parameters** tab.
   d. Enter text in the **User field 1** box.
   e. Click **Modify**.
7. In the **Custom component browser**, click **Refresh**.

   USER_FIELD_1 appears under **User-defined attributes** in the **Custom component browser**.
8. Link **P1** to **USER_FIELD_1**.
   a. Right-click **USER_FIELD_1** and select **Add equation**.
   b. Enter `P1` after the equal sign, and then press **Enter**.



9. Create two new parametric variables and link them to the user-defined attributes of the other two panels.
10. Save the custom component.
11. Close the custom component editor.

    You can now create a view filter in the model to hide or show panels using the **User field 1** property and the **Formula** values you entered for the parametric variables in the filter.

See also

Create new filters (page 172)

Add variables to a custom component (page 893)

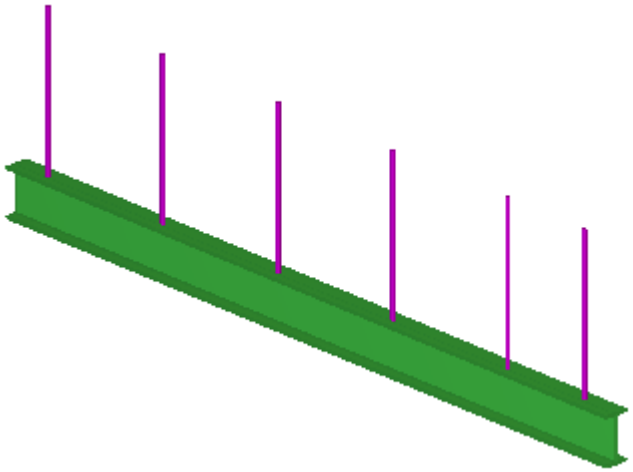
Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Calculate the number of handrail posts using a template attribute

This example shows how to create a variable formula that calculates the number of handrail posts in a custom component based on the length template attribute of the beam.

The handrail posts have been created at both ends of the beam, and one of them has been copied with the **Array of objects (29)** component.



1. In the model, select the custom component, right-click, and then select **Edit custom component**.
2. On the **Custom component editor** toolbar, click the **Display variables** button.

   The **Variables** dialog opens.
3. To create three new parametric variables, click **Add** three times.
4. Modify variable **P1** as follows:
   a. In the **Formula** box, enter `250`.
   b. In the **Label in dialog box** box, enter `End distance`.
5. Modify variable **P2** as follows:

a. In the **Formula** box, enter `900`.

b. In the **Label in dialog box** box, enter `Spacing`.

6. Modify variable **P3** as follows:

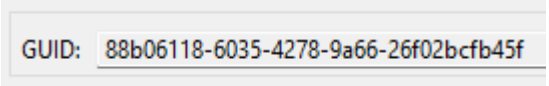
   a. In the **Value type** box, select **Number**.

   b. In the **Label in dialog box** box, enter `Number of posts`.

7. Inquire the GUID of the beam:

   a. Right-click the beam in a custom component view and select **Inquire** .

   b. In the **Inquire object** dialog, copy the beam's GUID.

8. In the **Formula** box of the **P3** variable, enter `=fTpl("LENGTH","04c3ff6a-8ee4-4a1c-8592-ba0ffd5556ea")-(P1*2))/P2`.

   In the formula, `04c3ff6a-8ee4-4a1c-8592-ba0ffd5556ea` is the GUID of the beam.

   Reference functions consist of the following syntax: `fTpl("template attribute", "object GUID")`. You need to always check the GUID of the object that you use with the reference function.

   The number of posts is calculated as follows: First the end distances are subtracted from the beam length, and then the result is divided by the post spacing.

9. Link the parametric variables **P2** and **P3** to the **dist_between_elem** and **number_of_arrays** properties of the **Array of objects (29)** component.

   a. In the **Custom component browser**, browse for the **dist_between_elem** property.

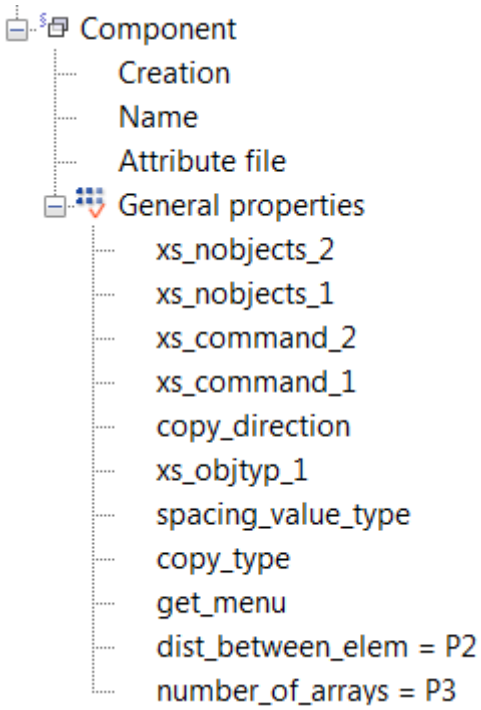
   b. Right-click and select **Add equation**.

   c. Enter `P2` after the equal sign, and then press **Enter**.

   d. Browse for the **number_of_arrays** property.

   e. Right-click and select **Add equation**.

   f. Enter `P3` after the equal sign, and then press **Enter**.



10. Bind the first handrail post to the beam end.
    a. In a custom component view, select the post.
    b. To select the handles of the first post, hold down the **Alt** key and use area selection (from left to right) to select multiple handles.
    c. Right-click and select **Bind to plane**.
    d. In a custom component view, move the mouse pointer to highlight the plane you want to bind to the handles.



11. Bind the last handrail post to the other beam end in a similar way.
12. Modify the distance variables as follows:
    a. In the **Formula** box, enter `=P1`.
    b. In the **Visibility** list, select **Hide**.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	250.00	250.00	Length	Parameter	Show	End distance
P2	900.00	900.00	Length	Parameter	Show	Spacing
P3	=(fTpl("LENGTH","c767511a-3...	5	Number	Parameter	Show	Number of posts
D1	=P1	250.00	Length	Distance	Hide	D1.COLUMN.BEAM
D2	=P1	250.00	Length	Distance	Hide	D2.COLUMN.BEAM
D3	=P1	250.00	Length	Distance	Hide	D3.COLUMN.BEAM
D4	=P1	250.00	Length	Distance	Hide	D4.COLUMN.BEAM

13. Save the custom component.
14. Close the custom component editor.
15. Select the custom component in the model, and open the custom component dialog.

    You can now change the spacing and the end distance of the handrail posts in the custom component's dialog. Tekla Structures calculates the

number of posts based on the spacing, end distance, and length of the beam.



See also

Edit custom components (page 887)

Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

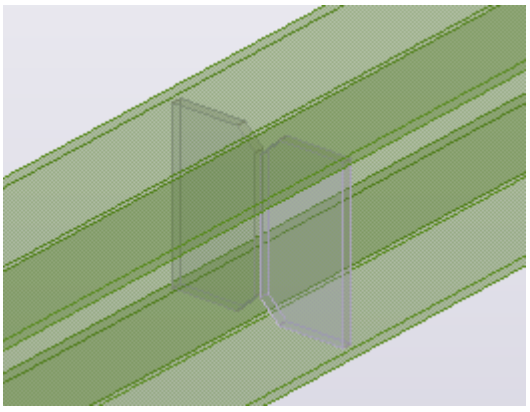
Examples of parametric variables and variable formulas in custom components (page 934)

Example of a variable formula: Create a custom stiffener detail with variables

This example shows how to create a custom stiffener detail with variables that control the shape and position of the stiffeners.

Create a basic custom stiffener detail

1. Create a beam with two stiffeners.



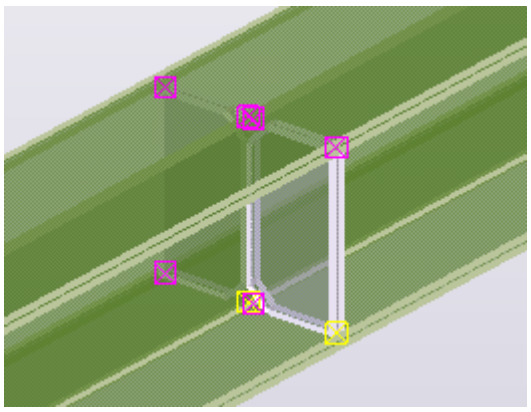
TIP To create the stiffeners, you can use the **Stiffeners (1003)** component and then explode the component.

2. Click the **Applications & components** button in the side pane to open the **Applications & components** catalog.

3. Click the **Access advanced features** button and select **Define custom component...**.

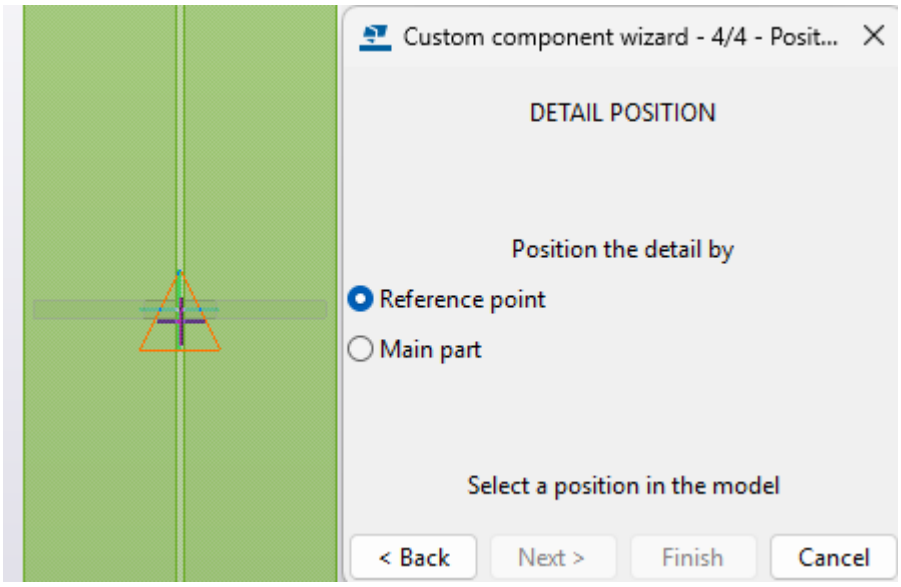
   The **Custom component wizard** dialog opens.

4. In the **Type** list, select **Detail**.
5. In the **Name** box, type `Stiffeners`.
6. Click **Next >**.
7. Select the stiffeners and the beam as the objects that form the custom component.

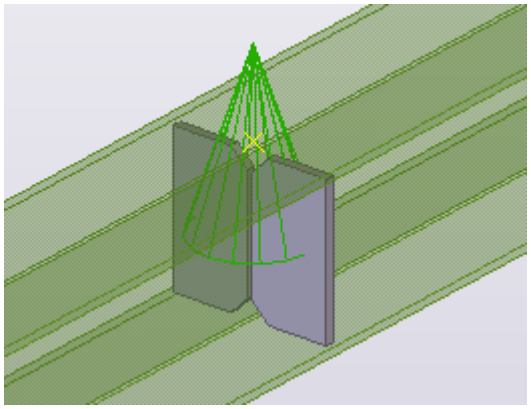
8. Click **Next >**.
9. Select the beam as the main part.
10. Click **Next >**.
11. Select the middle point of the beam as the reference point.

TIP Switch to the plane view (page 48) to select the middle point more easily.



12. Click **Finish** to finish creating the stiffener detail.

    Tekla Structures displays a component symbol for the new custom component and adds the stiffener detail to the component catalog.

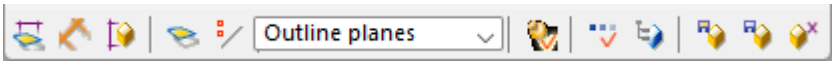


Create bindings to control the stiffener shape

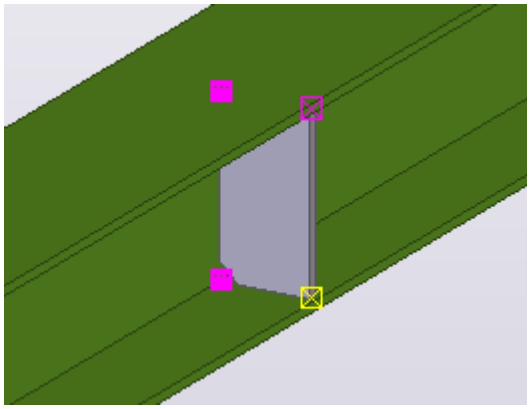
Bind the custom component handles to a plane to control the shape of the stiffeners.

1. To make it easier to select the handles (page 117), ensure that **Direct modification** is switched off.
2. To make it easier to select the available planes, set the model object rendering so that the object faces are properly shown, such as using **Parts rendered** (Ctrl+4).
3. In the model, select the custom component.
4. Right-click and select **Edit custom component**.

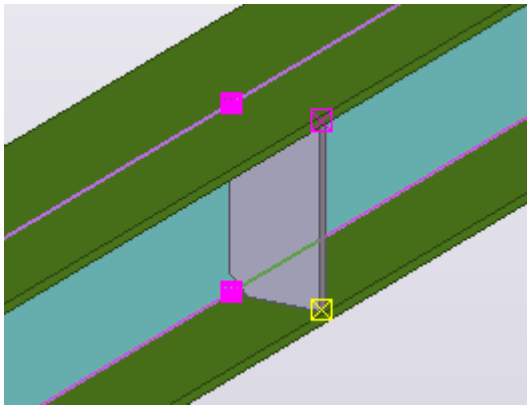
5. On the **Custom component editor** toolbar, select **Outline planes** from the list.



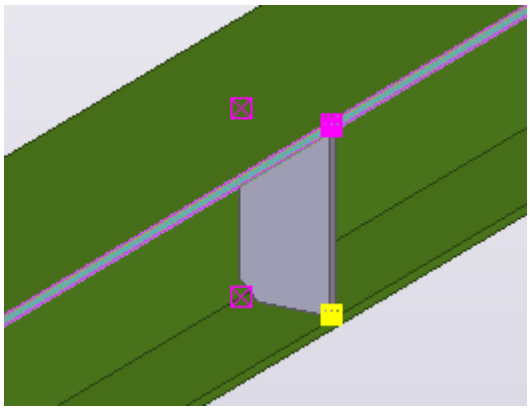
6. In a custom component view, select the stiffener on the right.
7. To bind the two inside handles of the stiffener to the beam web:
   a. Select the two handles next to the beam web.



   b. Right-click and select **Bind to plane**.
   c. Move the mouse pointer over the face of the web to highlight it.

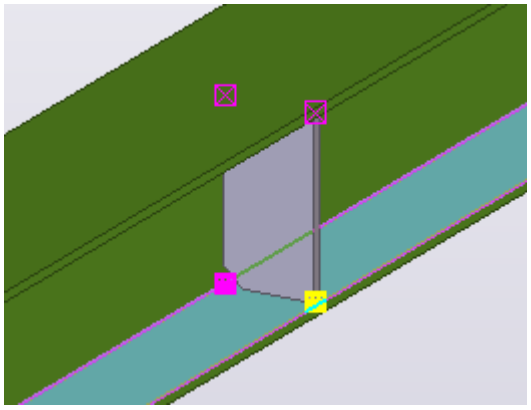


   d. Click the web to bind the handles.
8. To bind the two outside handles of the stiffener to the face of the top flange:
   a. Select the two handles.
   b. Right-click and select **Bind to plane**.
   c. Move the mouse pointer over the face of the top flange to highlight it.

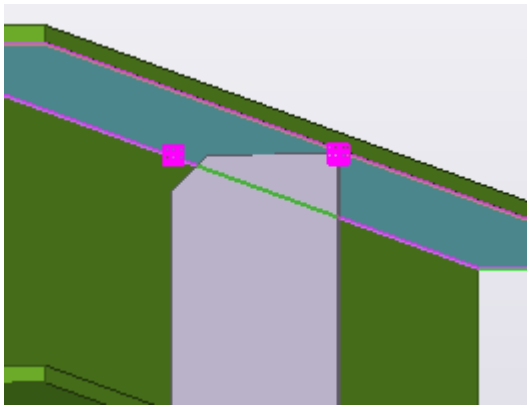


d. Click the top flange to bind the handles.

9. Bind the two bottom handles of the stiffener to the inside face of the bottom flange in a similar way as above.



10. Bind the two top handles of the stiffener to the inside face of the top flange in a similar way as above.



11. Select the stiffener on the left and create distance variables for the stiffener in the same way as for the right stiffener.

12. On the **Custom component editor** toolbar, click the **Display variables**  button.

    The **Variables** dialog opens.

13. To create a new parametric variable **P1**, click **Add**.
14. Modify the variable **P1** as follows:
    a. In the **Formula** box, enter `10`.
    b. In the **Label in dialog box** box, enter `Stiffener set back`.
15. For all variables that got values during the binding of the handles, enter `=P1` in the **Formula** box.

    For example:

Name	Formula	Value	Value type
D1	0.00	0.00	Length
D2	0.00	0.00	Length
D3	10.00	10.00	Length
D4	10.00	10.00	Length

    The variable **P1** now controls the distances of these variables.

16. In the **Visibility** list, set the variable **P1** to **Show** and the other variables to **Hide**.

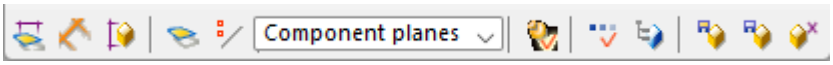
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
D1	0.00	0.00	Length	Distance	Hide	D1.PLATE.Web right plane
D2	0.00	0.00	Length	Distance	Hide	D2.PLATE.Web right plane
D3	=P1	10.00	Length	Distance	Hide	D3.PLATE.Upper flange right plane
D4	=P1	10.00	Length	Distance	Hide	D4.PLATE.Upper flange right plane
D5	0.00	0.00	Length	Distance	Hide	D5.PLATE.Lower flange top right plane
D6	0.00	0.00	Length	Distance	Hide	D6.PLATE.Lower flange top right plane
D7	0.00	0.00	Length	Distance	Hide	D7.PLATE.Upper flange bottom right plane
D8	0.00	0.00	Length	Distance	Hide	D8.PLATE.Upper flange bottom right plane
D9	0.00	0.00	Length	Distance	Hide	D9.PLATE.Web left plane
D10	0.00	0.00	Length	Distance	Hide	D10.PLATE.Web left plane
D11	=P1	10.00	Length	Distance	Hide	D11.PLATE.Upper flange left plane
D12	=P1	10.00	Length	Distance	Hide	D12.PLATE.Upper flange left plane
D13	0.00	0.00	Length	Distance	Hide	D13.PLATE.Lower flange top left plane
D14	0.00	0.00	Length	Distance	Hide	D14.PLATE.Lower flange top left plane
D15	0.00	0.00	Length	Distance	Hide	D15.PLATE.Upper flange bottom left plane
D16	0.00	0.00	Length	Distance	Hide	D16.PLATE.Upper flange bottom left plane
P1	10.00	10.00	Length	Parameter	Show	Stiffener set back

    You have now created distance variables that control the stiffener shape.

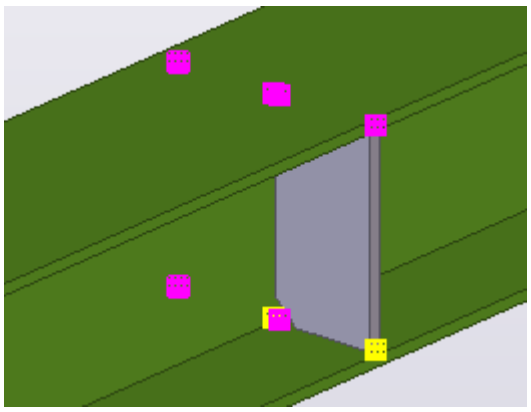
Create bindings to control the stiffener position

Bind the custom component handles to a plane to control the position of the stiffeners.

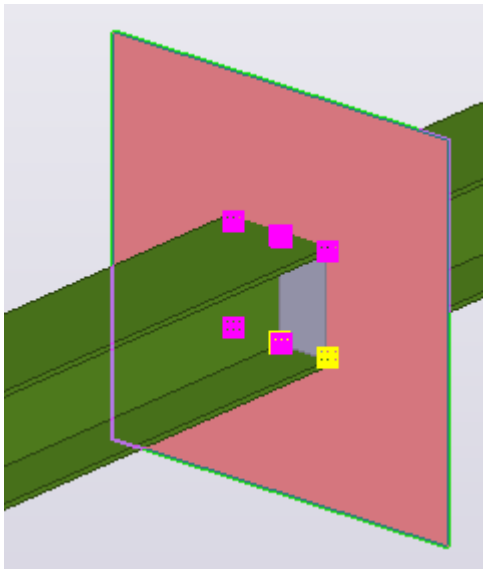
1. On the **Custom component editor** toolbar, select **Component planes** from the list.



2. Select all the handles of both stiffeners.



3. Right-click and select **Bind to plane**.
4. Bind the handles to the vertical component plane.



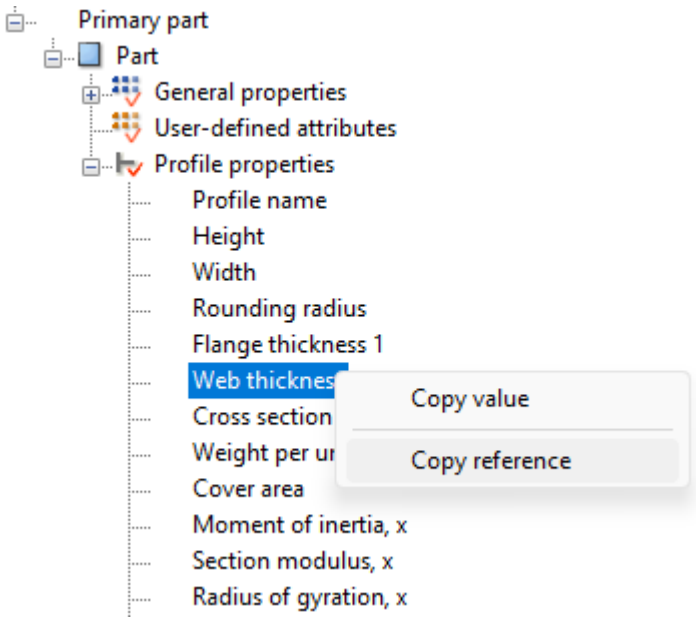
You have now created distance variables that control the position of the stiffeners.

Create variables to control the stiffener thickness

Control the stiffener thickness so that it is one and a half times the web thickness rounded up to the nearest available plate thickness. The available thickness values are 10, 12, and 16 mm.

1. On the **Custom component editor** toolbar, click the **Display variables** button.

   The **Variables** dialog opens.
2. To create a new parametric variable **P2**, click **Add**.
3. Modify the variable **P2** as follows:
   a. In the **Formula** box, enter `=1.5*`.
   b. In the **Visibility** list, select **Hide**.
   c. In the **Label in dialog box** box, enter `Plate calculation`.
4. Select the beam in a custom component view to highlight the beam (main part) in the **Custom component browser**.
5. In the **Custom component browser**, select **Web thickness** of the main part.
6. Right-click **Web thickness** and select **Copy reference**.



   Tekla Structures copies the reference to web thickness to the clipboard.
7. In the **Formula** box, paste the reference value after `=1.5*`.

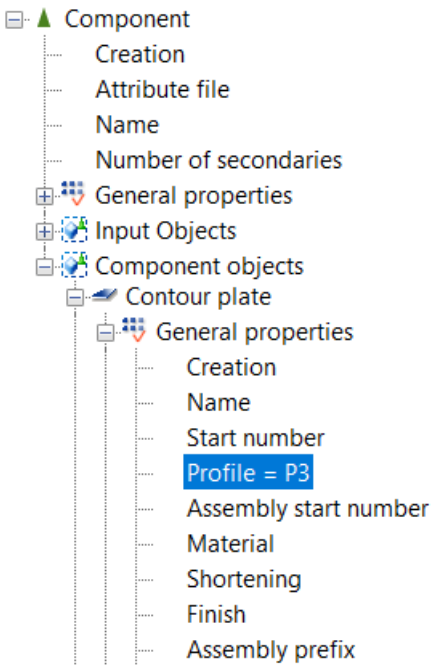
   ```
   =1.5*fP(Web thickness,"9582798d-
   e45b-40fb-9528-987872e8cdd2")
   ```

   In the formula, `fP(Web thickness,"9582798d-e45b-40fb-9528-987872e8cdd2")` is the web thickness reference, including the beam GUID, copied from the **Custom component browser**.

NOTE A reference function refers to the property of an object, such as the web thickness of a part. If the object property changes, so does the reference function value.

8. To create a new parametric variable **P3**, click **Add**.
9. Modify the variable **P3** as follows:
   a. In the **Value type** list, select **Number**.
   b. In the **Formula** box, enter `=if (P2 < 12 && P2 > 10) then 12 else if (P2 > 12) then 16 else 10 endif endif`.

      This means that if **P2** is less than 12 and greater than 10, the thickness is 12. If **P2** is greater than 12, the thickness is 16. If none of these conditions are met, the thickness is 10.
10. In the **Custom component browser**, link the variable **P3** to the **Profile** property of the first contour plate.



11. Repeat step 10 for the second contour plate.

    You have now created and linked all required variables that control the stiffener thickness according to the web thickness.

Create variables to control the creation of stiffener plates

Create five variables to control which stiffener plates are created and what is the class of the plates.

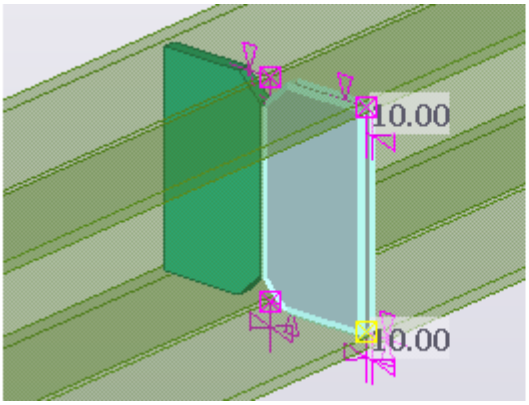
1. On the **Custom component editor** toolbar, click the **Display variables**  button.

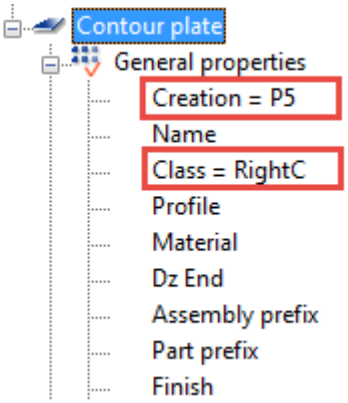
   The **Variables** dialog opens.
2. To create a new parametric variable **P4**, click **Add**.
3. Modify the variable **P4** as follows:
   a. In the **Formula** box, enter `2`.
   b. In the **Value type** list, select **Number**.
   c. In the **Visibility** list, select **Show**.
   d. In the **Label in dialog box** box, enter `Plates created`.
4. To create a new parametric variable **P5**, click **Add**.
5. Modify the variable **P5** as follows:
   a. In the **Formula** box, enter `=if P4==0 then 0 else 1 endif`.
   b. In the **Value type** list, select **Yes/No**.
   c. In the **Visibility** list, select **Hide**.
   d. In the **Label in dialog box** box, enter `Do not create right`.
6. To create a new parametric variable **P6**, click **Add**.
7. Modify the variable **P6** as follows:
   a. In the **Formula** box, enter `=if P4==1 then 0 else 1 endif`.
   b. In the **Value type** list, select **Yes/No**.
   c. In the **Visibility** list, select **Hide**.
   d. In the **Label in dialog box** box, enter `Do not create left`.
8. To create a new parametric variable **P7**, click **Add**.
9. Modify the variable **P7** as follows:
   a. Rename **P7** as `LeftC`.
   b. In the **Formula** box, enter `4`.
   c. In the **Value type** list, select **Number**.
   d. In the **Visibility** list, select **Show**.
   e. In the **Label in dialog box** box, enter `Left plate class`.
10. To create a new parametric variable**P8**, click **Add**.
11. Modify the variable **P8** as follows:
    a. Rename **P8** as `RightC`.
    b. In the **Formula** box, enter `5`.
    c. In the **Value type** list, select **Number**.

d. In the **Visibility** list, select **Show**.

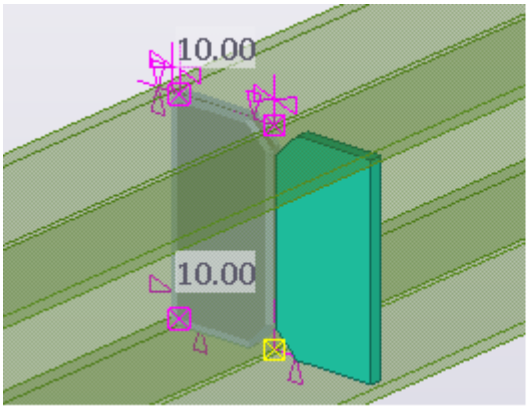
e. In the **Label in dialog box** box, enter `Right plate class`.

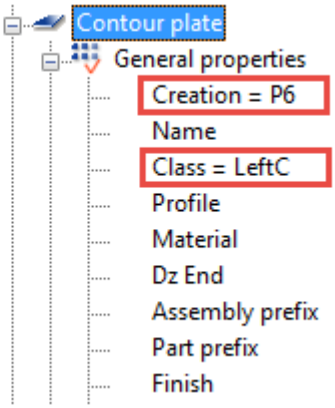
12. In the **Custom component browser**, link the variables **P5** and `RightC` to the right stiffener plate.





13. Link the variables **P6** and `LeftC` to the left stiffener plate.





You can now continue to customize the dialog of the stiffener detail to make it easier to adjust the settings later on. Next, do the following:

1. Add a list with images. (page 1012)
2. Arrange text boxes and labels. (page 1017)
3. Dim unavailable options. (page 1019)

Example of a variable formula: Link an Excel spreadsheet to a custom component

This example shows how to link a parametric variable to an Excel spreadsheet. For example, you can use Excel spreadsheets to check connections.

1. Create an Excel spreadsheet.

   The name of the spreadsheet file must be `component_"component_name".xls`. For example,

`component_stiffener.xls` for a custom component whose name is stiffener.

2. Save the Excel spreadsheet in the model folder: `..\<model>\exceldesign\component_stiffener.xls`.

   Alternatively, you can save the spreadsheet in the folder defined with the `XS_EXTERNAL_EXCEL_DESIGN_PATH` advanced option.

3. Select the custom component in the model, right-click and select **Edit custom component**.
4. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog opens.

5. To create a new parametric variable, click **Add**.
6. Modify the variable as follows:
   a. From the **Value type** list, select **Yes/No**.
   b. In the **Name** box, enter `use_externaldesign`.
   c. In the **Label in dialog box** box, enter `Use external design`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
use_externaldesign	0	0	Yes/No	Parameter	Show	Use external design

7. Save the custom component.
8. Close the custom component editor.
9. Select the custom component in the model, and open the custom component dialog.

   The option to use external design is visible in the custom component's dialog.

See also

Excel spreadsheets in connection design (page 852)

Edit custom components (page 887)

Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

Examples of a variable formula: Rebar set modifiers in custom components

You can use rebar set modifiers in custom components. You can define the rebar set bar and modifier properties using parametric variables.

For each modifier property that you want to parametrize, the corresponding **Apply** property is also needed. By using the **Apply** property you can override an existing property value with an empty value. Clearing an existing value is not possible without the **Apply** property.

Example: Define the class and size of rebar set bars using a property modifier

This example shows how to use a rebar set property modifier for defining the class and size of certain rebar set bars in a custom component. You will define the class and size using parametric variables and modifier-specific **Apply** properties.

The parametric variable for class will be defined so that if the class is set to 0, the class value is not applied, but the original class of the rebar set is used.

1. In the model, select a previously created custom component (page 866) that contains a rebar set and a property modifier.

   NOTE Custom parts do not have a component symbol in the model.

   To select custom components, ensure that the **Select components**  selection switch is active.

2. Right-click and select **Edit custom component**.
3. On the **Custom component editor** toolbar, click the **Display variables**  button.

   The **Variables** dialog opens.
4. In the **Variables** dialog, create and define parametric variables as follows:
   a. To create three new parametric variables, click **Add** three times.

      The variable names will be **P1**, **P2**, and **P3**.
   b. Modify the variable **P1** for class number input as follows:
      - In the **Value type** list, select **Number**.
      - In the **Label in dialog box** box, enter `Class`.
   c. Modify the variable **P2** for **Apply** property control as follows:

- In the **Formula** box, enter `=if (P1==0) then 0 else 1 endif`.

  This means that if **P1** (**Class**) is set to 0, the class property is not applied when you use the custom component. If **P1** is set to any other value, then the class property is applied.
- In the **Value type** list, select **Yes/No**.
- In the **Visibility** list, select **Hide**.

  This means that the variable **P2** will not be visible in the custom component dialog.
- In the **Label in dialog box** box, enter `Apply class`.

d. Modify the variable **P3** for bar size input as follows:

- In the **Name** box, change the name to `P3_size`.
- In the **Value type** list, select **Rebar size**.
- In the **Label in dialog box** box, enter `Bar size`.
- In the **Formula** box, replace zero with a valid bar size value.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Number	Parameter	Show	Class
P2	=if (P1==0) then 0 else 1 endif	0	Yes/No	Parameter	Hide	Apply class
P3_size	12	12	Rebar size	Parameter	Show	Bar size

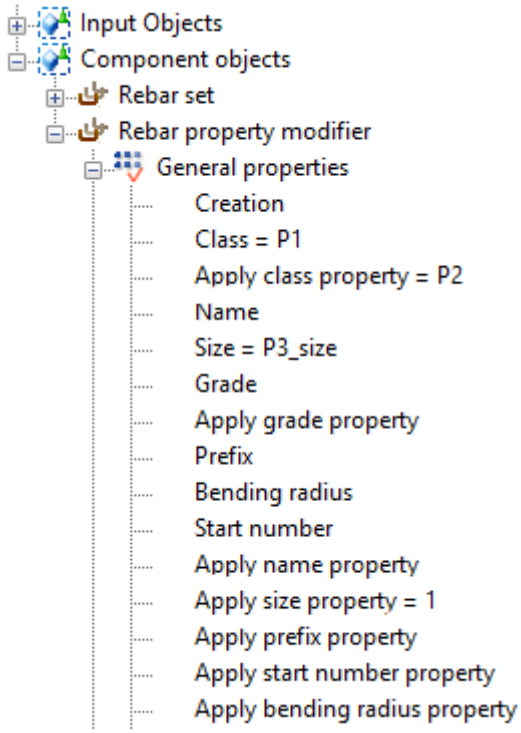
5. In the **Custom component browser**, link the parametric variables to the property modifier properties:

   a. Browse for **Component objects** --> **Rebar property modifier** --> **General properties**.

   b. Right-click **Class**, select **Add equation**, enter `P1` after the equal sign (=), and then press **Enter**.

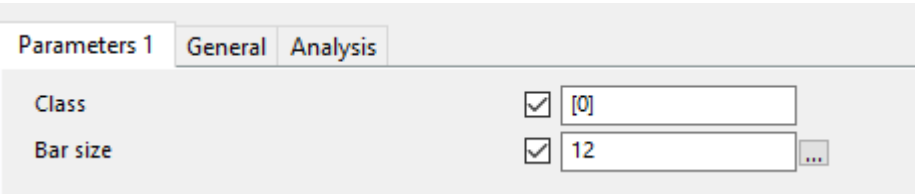
   Similarly, link the other variables and properties as follows:

   - **Apply class property** = `P2`
   - **Size** = `P3_size`
   - **Apply size property** = `1`



6. Save and close the modified custom component.

Now you have the following properties available in the custom component's dialog and you can modify the class and size of those rebar set bars that are affected by the property modifier:



You can use the component in locations similar to where the component was originally created. This component is not adaptive and Tekla Structures does not adjust the component dimensions to suit any changes in the model. To make the custom component adaptive, you need to modify it in the custom component editor.

Example: Create and modify rebar hooks using an end detail modifier

This example shows how to use a rebar set end detail modifier for creating hooks at certain rebar set bar ends in a custom component. You will define the hook properties using parametric variables and modifier-specific **Apply** properties.

1. In the model, select a previously created custom component that contains a rebar set and an end detail modifier.

NOTE Custom parts do not have a component symbol in the model.

To select custom components, ensure that the **Select components**  selection switch is active.

2. Right-click and select **Edit custom component**.
3. On the **Custom component editor** toolbar, click the **Display variables** button .

   The **Variables** dialog opens.
4. In the **Variables** dialog, create and define parametric variables as follows:
   a. To create four new parametric variables, click **Add** four times.

      The variable names will be **P1**, **P2**, **P3**, and **P4**.
   b. Modify the variable **P1** for hook type input as follows:
      - In the **Value type** list, select **Rebar hook type**.
      - In the **Label in dialog box** box, enter `Hook type`.
      - In the **Formula** box, enter `4` for a custom hook.

        The different hook types are identified with numbers: 1 = 90 degree hook, 2 = 135 degree hook, 3 = 180 degree hook, 4 = custom hook.
   c. Modify the variable **P2** for hook angle input as follows:
      - In the **Value type** list, select **Number**.

        Note that although **Angle** is available as a value type, the **Number** option must be used for the hook angle.
      - In the **Label in dialog box** box, enter `Custom hook angle`.
   d. Modify the variable **P3** for hook length input as follows:
      - In the **Value type** list, select **Number**.
      - In the **Label in dialog box** box, enter `Custom hook length`.
   e. Modify the variable **P4** for hook radius input as follows:
      - In the **Value type** list, select **Number**.
      - In the **Label in dialog box** box, enter `Custom hook radius`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	4	4	Rebar hook type	Parameter	Show	Hook type
P2	0	0	Number	Parameter	Show	Custom hook angle
P3	0	0	Number	Parameter	Show	Custom hook length
P4	0	0	Number	Parameter	Show	Custom hook radius

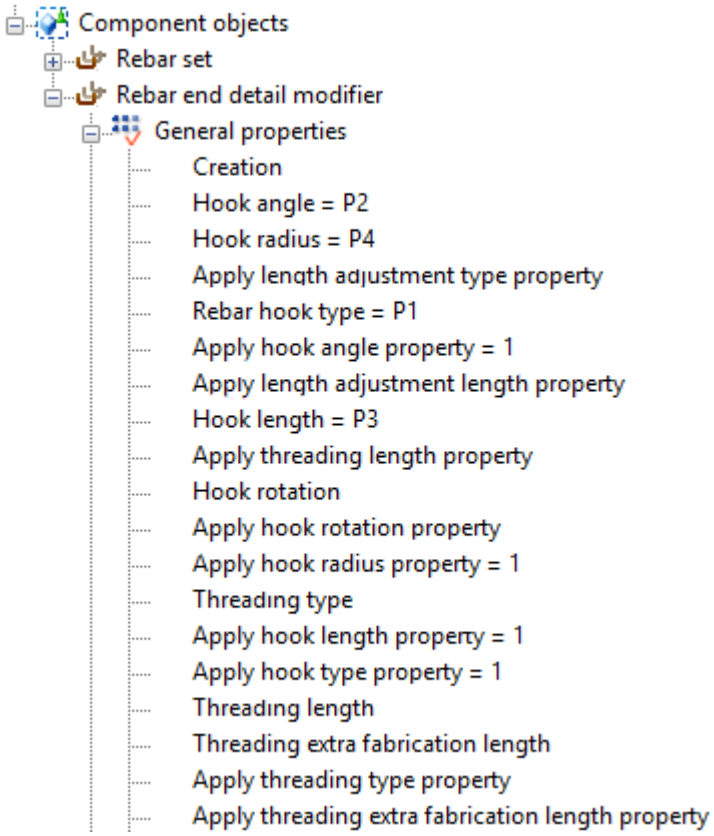
5. In the **Custom component browser**, link the parametric variables to the end detail modifier properties:

   a. Browse for **Component objects** --> **Rebar end detail modifier** --> **General properties**.

   b. Right-click **Hook angle**, select **Add equation**, enter `P2` after the equal sign (=), and then press **Enter**.

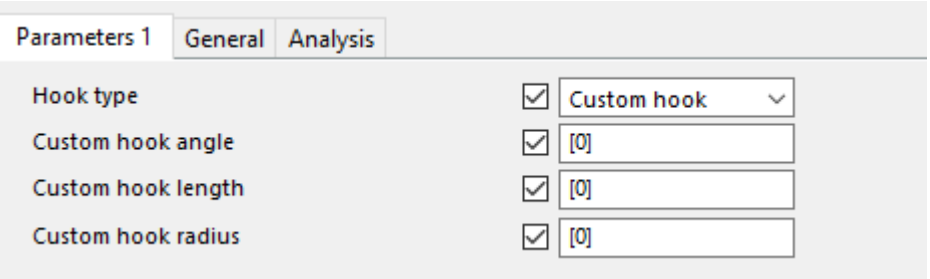
      Similarly, link the other variables and properties as follows:

      - **Hook radius** = `P4`
      - **Rebar hook type** = `P1`
      - **Apply hook angle property** = `1`
      - **Hook length** = `P3`
      - **Apply hook radius property** = `1`
      - **Apply hook length property** = `1`
      - **Apply hook type property** = `1`



6. Save and close the modified custom component.

Now you have the following properties available in the custom component's dialog and you can modify the hooks of those rebar set bars that are affected by the end detail modifier:



You can use the component in locations similar to where the component was originally created. This component is not adaptive and Tekla Structures does not adjust the component dimensions to suit any changes in the model. To make the custom component adaptive, you need to modify it in the custom component editor.

See also

Edit custom components (page 887)

Add variables to a custom component (page 893)

Variable properties in custom components (page 936)

Examples of parametric variables and variable formulas in custom components (page 934)

7.6 Import and export custom components

You can import and export custom components as `.uel` files between models.

NOTE You can also share and store your custom components in Tekla Warehouse, and also download custom components made by other users.

Before sharing custom components with others, note the following:

- Provide essential information with the custom components. If you distribute your component to other users, remember to list the profiles it works with.
- Use fixed profiles (page 332) whenever possible. If your custom component contains user-defined cross sections, remember to include them when you copy the custom component to a new location.

Export a custom component

You can export custom components in a `.uel` file.

1. Click the **Applications & components**  button in the side pane to open the **Applications & components** catalog.
2. In the catalog, select the custom components you want to export.
3. Right-click the selection, and then select **Publish**.
4. Browse for the folder where you want to save the file.
5. Enter a name for the export file.

   The file name extension is `.uel`. Do not change the file name after exporting the custom component. If the file name differs from the name in the **Applications & components** catalog, it may be difficult to find the right component later on.

   The data files that are used by the custom component are included in the exported `.uel` file. The data files need to be located in the model folder or in the `CustomComponentDialogFiles` subfolder to be exported. Only files specified directly in `fVF` functions are exported. For example: `=fVF("myData.dat", ...)` directly specifies the file, but `=fVF(P1, ...)` does not.
6. Click **Save** to export the custom components.

TIP If you want to export custom components as separate files, select the custom components in the **Applications & components** catalog, right-click, and then select **Publish separately**.

Import a custom component

You can import previously made custom components to another model.

1. Click the **Applications & components**  button in the side pane to open the **Applications & components** catalog.
2. Click the **Access advanced features**  button , and then select **Import**.
3. Browse for the folder that contains the export file.

   The location depends on where you saved the file when you exported the custom component.
4. Select the export file.
5. Click **Open** to import the custom components.

TIP You can import custom components to a new model automatically by using the `XS_UEL_IMPORT_FOLDER` advanced option. Export all custom components to certain folders and enter these folders

as the value for the `XS_UEL_IMPORT_FOLDER` advanced option to easily import the custom components to new models.

7.7 Frequently asked questions about custom components

Find answers to some common questions about custom components.

General

Question 1

Are there "best methods" how to create custom components?

Not really. However, multiplication is faster than division. For example, `P1*0.5` is faster than `P1/2`.

Question 2

Can I change the settings of the custom component views?

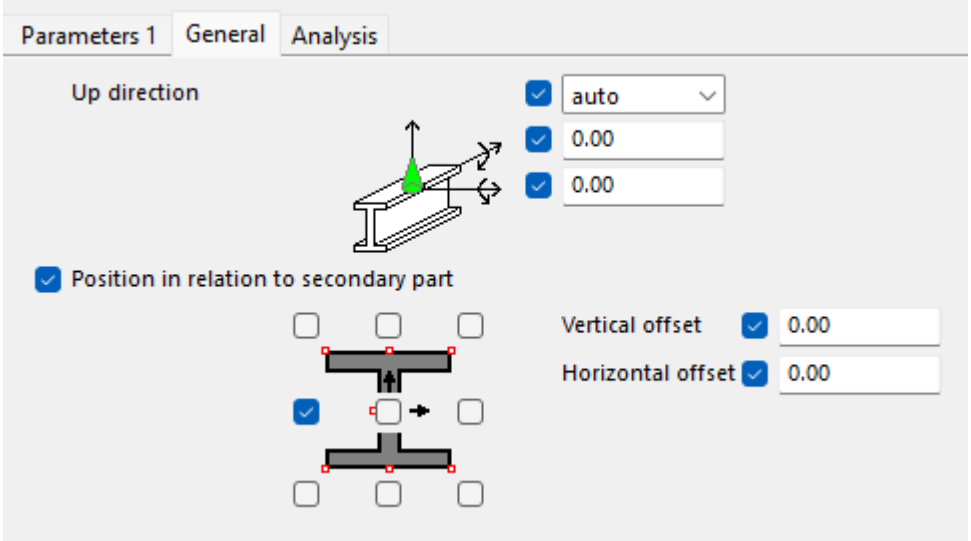
Yes. Double-click anywhere on the model background to open the **View properties** dialog and change the settings as required. Then save the settings with the name that corresponds with the custom component editor view that you want to modify:

- custom_object_editor_front_view
- custom_object_editor_end_view
- custom_object_editor_top_view
- custom_object_editor_perspective_view

Question 3

*I am unable to create two components between the same parts. I have selected the **Allow multiple instances of connection between same parts** checkbox in the custom component settings. Why does this not work with my custom component?*

You also need to change **Position in relation to secondary part** in the custom component dialog to be different from the center option. Then you can create two components between the same parts.

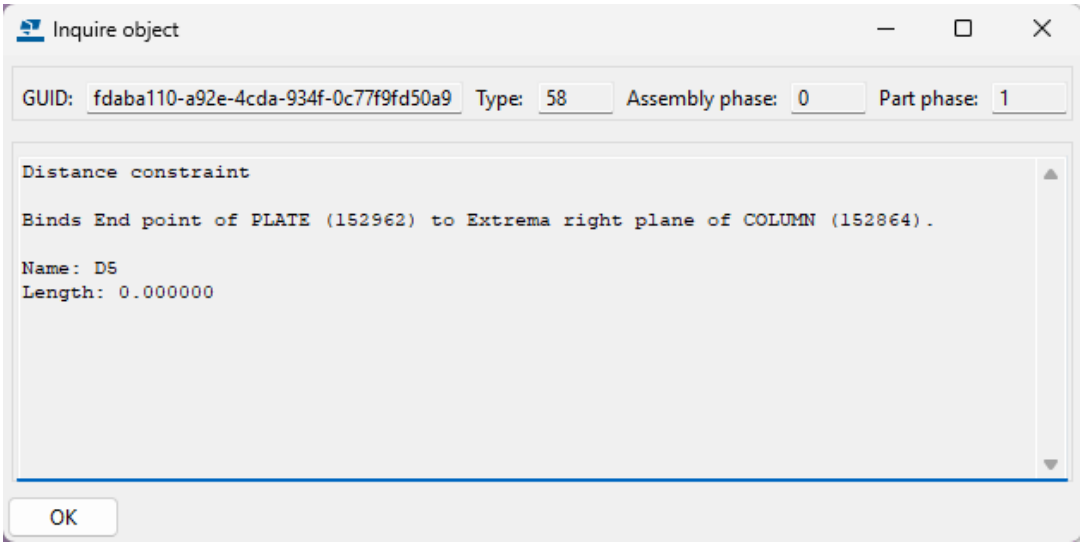


Creating distance variables (binding)

Question 1

Is there any way to know what is bound to what?

On the ribbon, click **Object** and select a distance symbol in a custom component view to see information on the binding.



Question 2

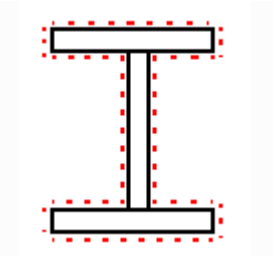
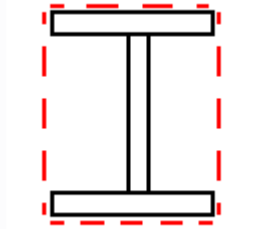
Is it possible to change an existing binding?

No, you must delete the binding and create a new one.

Question 3

Is there any difference between outline and boundary planes?

Yes. Outline planes follow the outline of the part and boundary planes form the outer shape of the part.

Outline planes	You can select the outer and inner surfaces of a profile. 
Boundary planes	You can select the edges of a bounding box that surrounds the profile. 

TIP Boundary planes work for most profile types, so try to use them rather than outline planes whenever you can.

Question 4

How do I bind to construction planes?

You can bind to construction planes like to any other planes:

1. Select the handles and right-click.
2. Select **Bind to plane**.
3. Select the construction plane.

NOTE Construction planes are not highlighted.

Question 5

*What does the **Create distance** command do on the custom component editor toolbar?*

With the command you select the plane first and then the handle or chamfer to bind.

Question 6

*What does the **Automatically create distances for picked components' handle points** command do on the custom component editor toolbar?*

The command automatically binds the selected part to their handles' contact planes. Tekla Structures selects planes in the following order:

1. construction planes
2. planes of main and secondary parts

Editing variables

Question 1

Are there any reserved variable names?

Yes, the following parametric variables must have the same prefix:

- **Bolt size** and **Bolt standard**
- **Stud size**, **Stud standard** and **Stud length**
- **Rebar size**, **Rebar grade** and **Rebar bending radius**
- **Component name** and **Component attribute file**

The suffixes are automatically added when you change **Value type** in the **Variables** dialog box:

Name	Formula	Value	Value type	Variable type
P1_diameter	12.00	12.00	Bolt size	Parameter
P2_screwdin	7990	7990	Bolt standard	Parameter

You need to change the prefix to suit the related variable:

Name	Formula	Value	Value type	Variable type
P1_diameter	12.00	12.00	Bolt size	Parameter
P1_screwdin	7990	7990	Bolt standard	Parameter

If the prefixes do not match, the variables do not work correctly.

Question 2

Is it possible to round a value to the nearest ten, for example?

Use the `round()` function to round the values.

For example, `round(17.5, 20)` rounds 17.5 up to 20.

`round(17.1, 15)` rounds 17.1 down to 15.

Question 3

*Is there a limit to the length of the **Label in dialog box**?*

Yes, the limit is 30 characters. You can add more characters in an input file, if required.

Question 4

I want to have two separate values for the plate width and plate thickness. How can I use these values to make a profile name, for example `PLT200*100`? How do I stop the values from having decimals?

Use the `int()` function to convert decimal numbers to integers.

In the example below, **P2** is the plate width and **P3** is the plate thickness. Use the following formula to make the profile:

```
="PLT"+int(P2)+"*"+int(P3)"
```

Name	Formula	Value	Value type	Variable type
P2	200.00	200.00	Length	Parameter
P3	10.00	10.00	Length	Parameter
P4	400.00	400.00	Length	Parameter
P5	10.00	10.00	Length	Parameter
P6	="PLT"+int(P2)+"*"+int(P3)	PLT200*10	Profile	Parameter
P7	="PLT"+int(P4)+"*"+int(P5)	PLT400*10	Profile	Parameter
P8	100.00	100.00	Length	Parameter

Question 5

Is it possible to use formulas in a distance list?

Yes, below is as an example of using a formula in a distance list.

Name	Formula	Value	Value type
P7	22.00	22.00	Length
P8	90.00	90.00	Length
P9	=if (P8<100) then P7 else (P7 + " " + P10) endif	22.00	Distance list
P10	100.00	100.00	Length

Name	Formula	Value	Value type
P7	22.00	22.00	Length
P8	120.00	120.00	Length
P9	=if (P8<100) then P7 else (P7 + " " + P10) endif	22.00 100.00	Distance list
P10	100.00	100.00	Length

Distance lists are treated as text in Tekla Structures so you must use quotation marks. For example, `P1 + " " + P5`

Custom component browser

Question 1

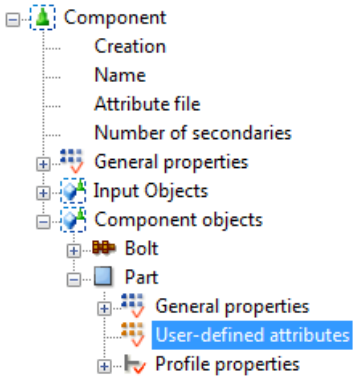
*Why is the **Custom component browser** not updated when I add or delete parts in the custom component editor?*

You need to update the **Custom component browser** by clicking **Refresh**.

Question 2

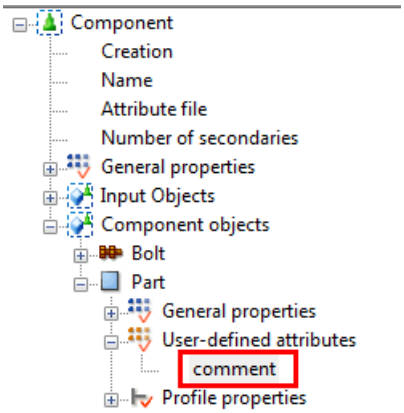
Is there any way to show user-defined attributes in the custom component browser?

Yes, you need to change the required value from the default setting. The default values are not shown in the **Custom component browser**.



To show the required attribute in the custom component browser:

1. Go to the property pane and open the open the dialog for user-defined attributes.
2. Change the value of the required user-define attribute.
3. Click **Refresh** in the **Custom component browser** to show the attribute.



Editing an input file

Question 1

I added an image to a custom component dialog by editing an input (`*.inp`) file. If I make changes in the custom component editor, is the input file overwritten?

Yes, but Tekla Structures automatically creates a backup of the input file.

Question 2

Why can I not use Tab 4 when I edit an input file?

Tab 4 is reserved for the **General** tab.

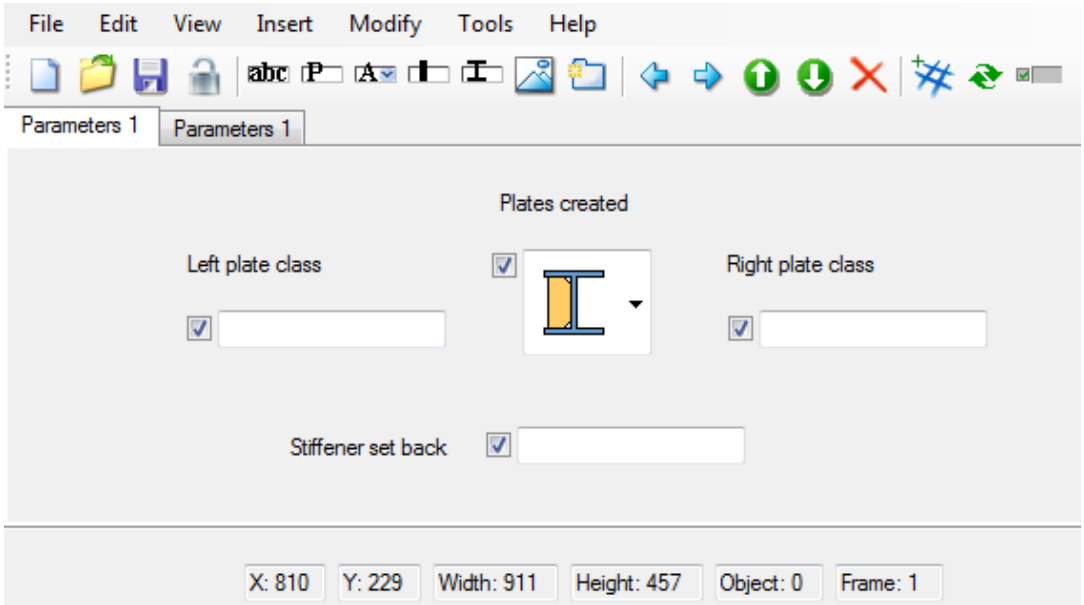
7.8 Customize the custom component dialogs

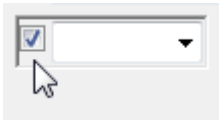
Tekla Structures automatically creates a dialog for each custom component you define. Each custom component has an input (`.inp`) file that defines the contents of the custom component's dialog. You can customize the dialog by using the **Custom component dialog editor** tool.

Alternatively, the dialog input (`.inp`) files can be manually customized in a text editor. For more information, see Customize the dialogs of custom components by using a text editor (page 999).

Modify a custom component dialog

To open the **Custom component dialog editor**, select a custom component in the model, right-click, and select **Edit custom component dialog**.



To	Do this
View and edit object properties	1. Select a dialog element. For example, a text box. 2. Click **Modify** --> **Properties**. Now you can view and edit the current properties of the dialog element. For example, you can check that you have the correct text box under each label in the dialog. Alternatively, you can double-click the dialog element. If the dialog element will not open for viewing and editing, try double-clicking the space right underneath the checkbox: 
Add a dialog element	Click **Insert** and select a suitable element from the list. The options are: • **Tab Page**: add a new tab • **Label**: add a label for a text box or list • **Parameter**: add a text box • **Attribute**: add a list • **Part**: add some basic part properties • **Profile**: add some basic profile properties • **Picture**: add an illustrative image of the custom component
Add an image	1. Click **Insert** --> **Picture** to show the contents of the **Image Folder** set in **Tools** --> **Options**. 2. Select an image. The image must be in the bitmap (`.bmp`) format. 3. Click **Open**. 4. Drag the image to the desired location.
Add a tab	1. Click **Insert** --> **Tab Page**. 2. Double-click the new tab. 3. Enter a new name, and then press **Enter**. Note that each tab may contain up to 25 fields. If you have more than 25 fields visible, Tekla Structures automatically creates another tab.

To	Do this
Show or hide the pixel grid	Click . Tekla Structures displays a pixel grid that makes it easier to align elements in the dialog.
Move a dialog element	Drag the dialog element to a new location. You can also use the keyboard shortcuts **Ctrl+X** (cut), **Ctrl+C** (copy), and **Ctrl+V** (paste). For example, to move a dialog element to another tab: select the dialog element, press **Ctrl+X**, go to another tab, and press **Ctrl+V**.
Select multiple dialog elements	Hold down the **Ctrl** key and click the dialog elements, or use area selection.
Rename a tab or text box label	1. Double-click the tab or text box label. 2. Type a new name. 3. Press **Enter**.
Remove a dialog element	1. Select the dialog element you want to remove. 2. Press **Delete**.
Remove a tab	1. Select the tab. 2. Right-click and select **Delete**.
Add images to a list	1. Select the list element. 2. Click **Modify** --> **Properties**. 3. Click **Edit Values**. 4. Click **Browse Add**. 5. Select the image you want to use and click **Open**. 6. Repeat steps 4–5 for any other images you want to use. 7. Click **OK** to save the changes.
Save the changes	Click **File** --> **Save**.

Custom component input files

Each custom component has an input (`.inp`) file that defines the contents of the custom component's dialog.

When you create a new custom component, Tekla Structures automatically creates an input file for the component. The input file is located in the `\CustomComponentDialogFiles` folder under the model folder. The input

file has the same name as the custom component, and the file name extension is `.inp`.

When you modify a custom component (page 887), you will lose any changes you have made to the input file. However, when you modify the custom component, Tekla Structures automatically creates a backup copy of the input file. The backup copy has the file name extension `.inp_bak`, and it is located in the `\CustomComponentDialogFiles` folder under the model folder. Tekla Structures displays a notification when the backup file is created.

Lock or unlock the custom component input file

You can lock the custom component's input (`.inp`) file to prevent accidental modifications. If the file is unlocked, and someone else updates the custom component in the custom component editor, all your modifications to the dialog will be lost.

1. In the model, select the custom component whose input file you want to lock or unlock.
2. Right-click and select **Edit custom component dialog**.
3. In the **Custom component dialog editor**, click the **Lock/Unlock** button .

If someone modifies the custom component in the custom component editor when the `.inp` file is locked, the `.inp` file will not be updated. You can still modify the dialog in the **Custom component dialog editor** even if the `.inp` file is locked.

Custom component dialog editor settings

In the **Custom component dialog editor**, click **Tools** --> **Options** to view and modify basic settings in the custom component dialog editor. Click **Tools** --> **Change Language** to change the language of the custom component dialog editor.

Option	Description
Image folder	The location of the image folder. To restore the default folder setting, click **Default**.
Project folder	The location of the project folder. When you create a completely new input file by clicking **File** --> **New** and then save it, the file is saved in the project folder. Note that existing input files are saved under the model folder.
Parameter width	The default width for text boxes.
Attribute width	The default width for lists.

Option	Description
Grid spacing X **Grid spacing Y**	The spacing of the pixel grid in the X and Y directions. The default value is `5`.
Snap to grid	Select to show or hide the pixel grid.

Option	Description
Language	Select a language from the list. Close and reopen the dialog editor for the change to take effect. You have the following options: • **Auto**: the dialog editor follows the language of the Tekla Structures user interface • **English** • **Dutch** • **French** • **German** • **Italian** • **Spanish** • **Japanese** • **Chinese Simplified** • **Chinese Traditional** • **Czech** • **Portuguese Brazilian** • **Hungarian** • **Polish** • **Russian** • **Korean**

Customize the custom component dialogs by using a text editor

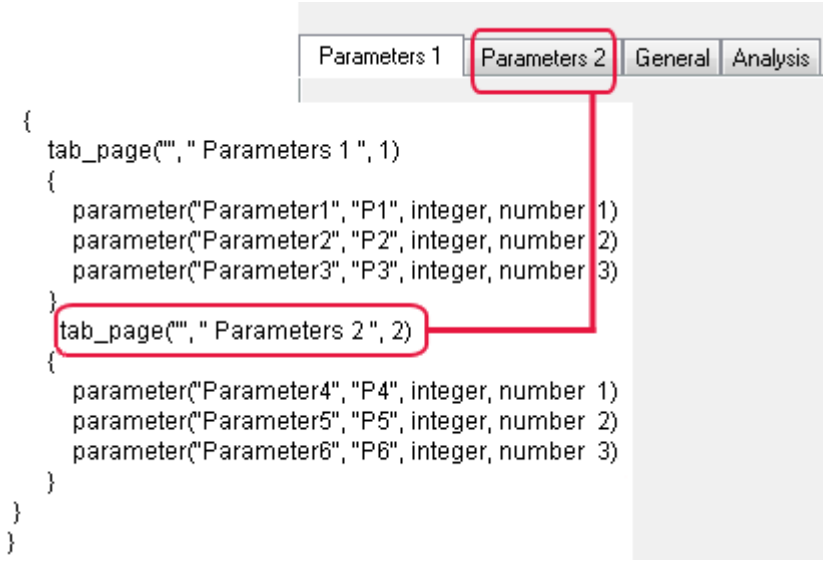
Each custom component has an input (`.inp`) file that defines the contents of the custom component's dialog. The dialog input files can be manually customized in a text editor. Be careful when modifying an input file, as errors may cause the dialog to disappear.

Note that the **General** tab is reserved for predefined general properties. You cannot rename the **General** tab or add more parameters to it.

Alternatively, you can use the **Custom component dialog editor** tool to customize the dialogs. For more information, see Customize the custom component dialogs (page 995).

Add new tabs

1. Open the `.inp` file in a text editor.
2. Add a new tab definition, as shown below:



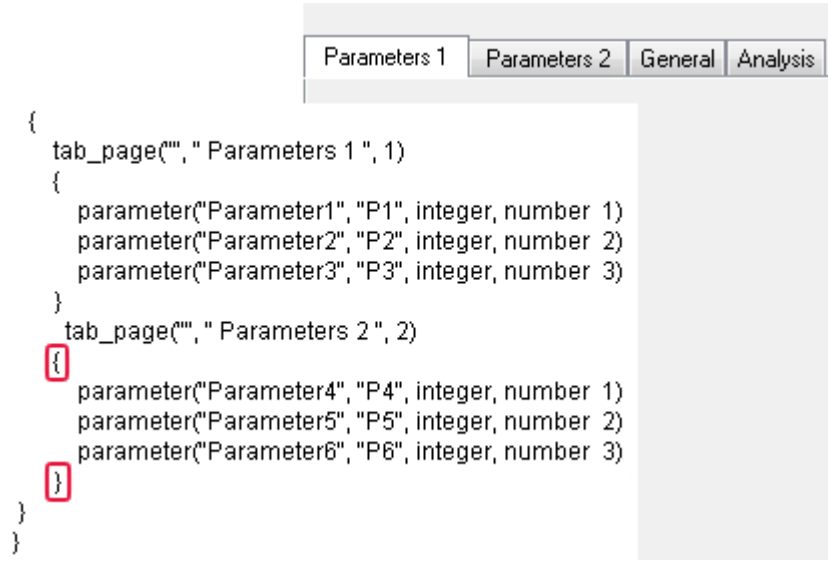
   The tab definitions are of format `tab_page("", " tab_name ", tab_number).`

3. Save the `.inp` file.

NOTE The fourth tab (**General**) is reserved for the predefined general properties, so you cannot add your own parameters to it.

Add text boxes

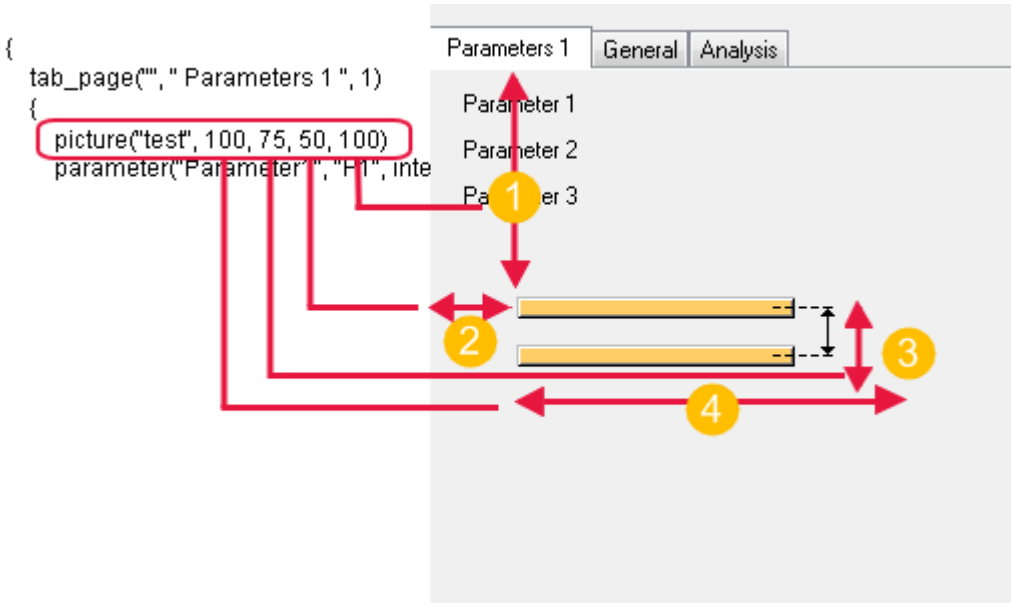
1. Open the `.inp` file in a text editor.
2. Add `parameter` elements and enclose them in curly brackets, as shown below:



3. Save the `.inp` file.

Add images

1. Create an image and save it in bitmap (`.bmp`) format in the `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps` folder.
2. Open the `.inp` file in a text editor.
3. Add an image definition, as shown below:



(1) y = 100

(2) x = 50

(3) height = 75

(4) width = 100

The image definitions are of format `picture("image_name", width, height, x_coordinate, y_coordinate).`

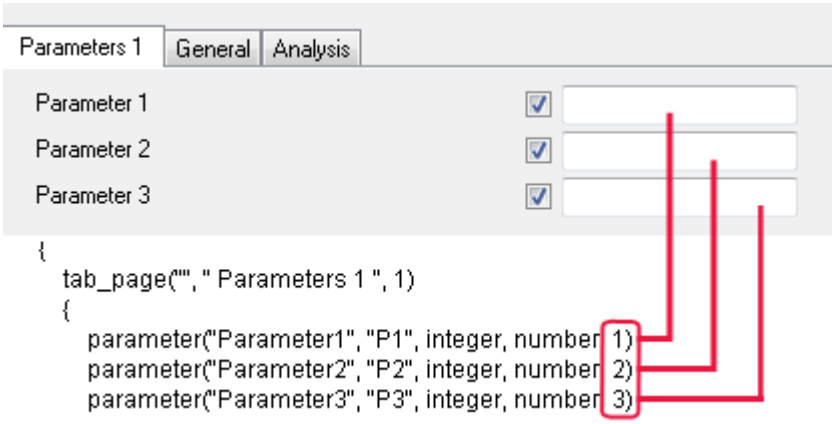
4. Save the `.inp` file.

Change the order of boxes

By default, variables appear in a custom component dialog in the same order as they were created.

1. Open the `.inp` file in a text editor.
2. Change the last number in the parameter definition.

   The boxes are listed from top to bottom, as shown below:



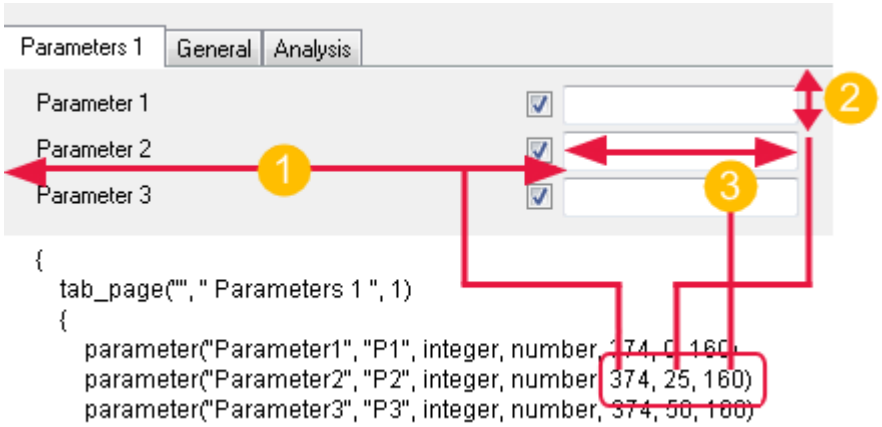
3. Save the `.inp` file.

Change the location of boxes

You can define an exact location for each text box.

1. Open the `.inp` file in a text editor.
2. Define the exact location of the box using three values: the x coordinate, y coordinate, and width of the box.

For example:



(1) x = 374

(2) y = 25

(3) width = 160

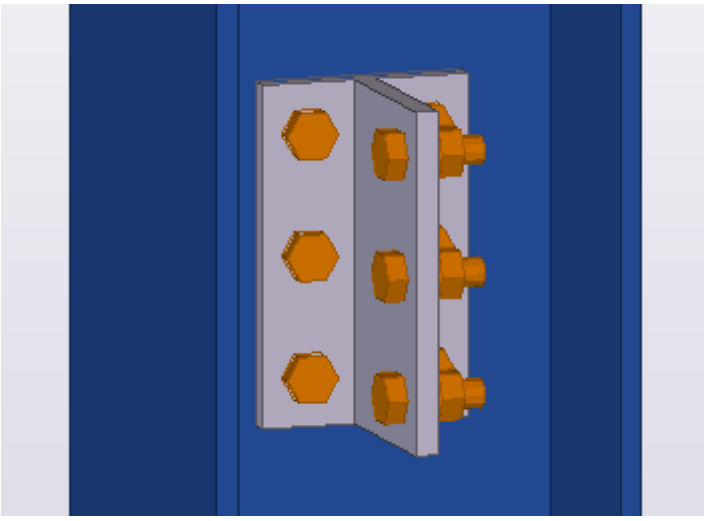
3. Save the `.inp` file.

Example: Add a group of checkboxes in the custom component dialog

This example shows how to add a checkbox for each bolt group in a custom component by modifying the `.inp` file. When the component is used in a model, you can select which bolts to create by selecting the desired checkboxes.

1. Define a custom component that contains bolts.

   For example, create a custom tee connection that creates one bolt group and three single bolts:



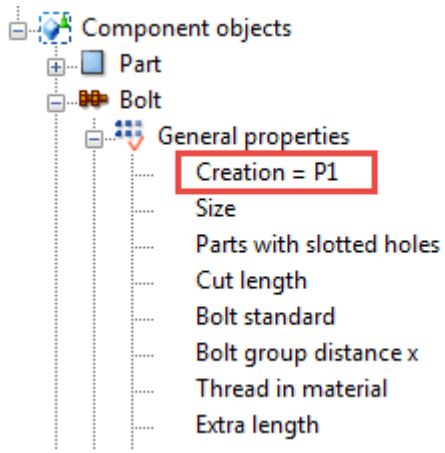
2. Create parametric variables that control the creation of bolts.

With checkbox groups, the **Value type** of these variables must be **Yes/No**. For example, create three variables **P1**, **P2**, and **P3**, one for each single bolt in the custom tee connection.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Yes/No	Parameter	Show	Parameter1
P2	0	0	Yes/No	Parameter	Show	Parameter2
P3	0	0	Yes/No	Parameter	Show	Parameter3

3. Link the variables to the **Creation** property of the bolts.

   For example, link the variable **P1** to the **Creation** property of the first bolt, the variable **P2** to the **Creation** property of the second bolt, and so on.



4. Save the custom component.
5. In the model, click **File** --> **Open model folder** to open the current model folder.
6. Go to the `\CustomComponentDialogFiles` folder.
7. Open the `.inp` file in a text editor.
8. Add an image definition.

For example:

```
page("TeklaStructures","")
{
    detail(1, "tee")
    {
        tab_page("", " Parameters 1 ", 1)
        {
    picture("CustomTee",100,100,200,100) /*Bolts*/
        }
    }
}
```

If you use a custom image, save it in bitmap (`.bmp`) format in the `..\TeklaStructures\<version>\Bitmaps` folder.

9. Add a `toggle_group` element to define the toggle group origin, that is, the position of the group of checkboxes in the custom component dialog.

   Use the x and y coordinate values to define the position. For example:

```
page("TeklaStructures","")
{
    detail(1, "tee")
    {
        tab_page("", " Parameters 1 ", 1)
        {
    picture("CustomTee",100,100,200,100) /*Bolts*/
    toggle_group(200,320,
        )
        }
    }
}
```

   (1) x = 200

   (2) y = 320

10. Within the `toggle_group` element, add a line for each checkbox you want to add.

Use the same parametric variables that you created in step 2.

```
page("TeklaStructures","")
{
   detail(1, "tee")
   {
       tab_page("", " Parameters 1 ", 1)
       {
   picture("CustomTee",100,100,200,100) /*Bolts*/
   toggle_group(200,320,

     "P1", 160, -165, "0",
     "P2", 160, -135, "0",
     "P3", 160, -105, "0")
       }
   }
}
```

The two numeric values after the variable name are offsets from the toggle group origin. For example, the first definition `"P1", 160, -165, "0"` means that the checkbox for the variable **P1** is located 160 steps right and 165 steps up from the toggle group origin.

Direction	Negative values	Positive values
X	left	right
Y	up	down

```
page("TeklaStructures","")
{
   detail(1, "tee")
   {
       tab_page("", " Parameters 1 ", 1)
       {
   picture("CustomTee",100,100,200,100) /*Bolts*/
   toggle_group(200,320,

     "P1", 160, -165, "0",
     "P2", 160, -135, "0",
     "P3", 160, -105, "0")
       }
   }
}
```

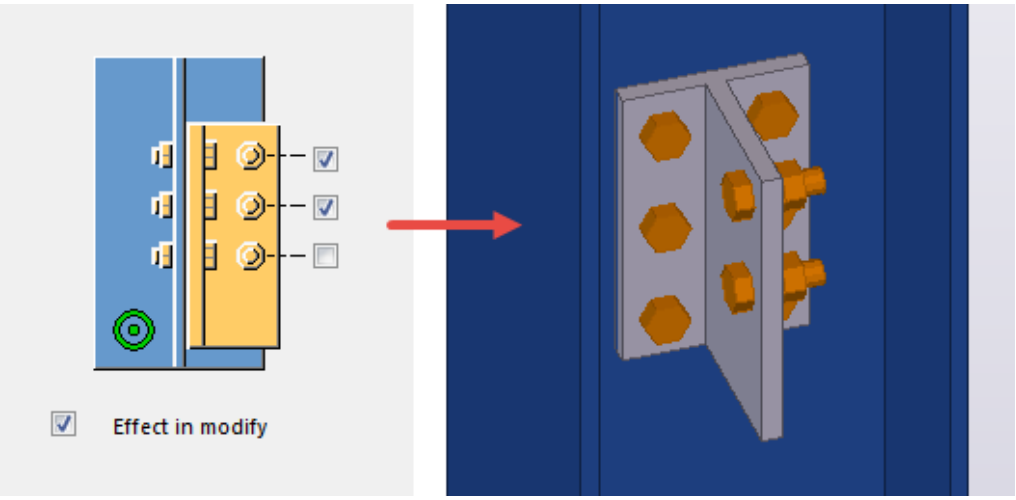


(1) offset in X direction

(2) offset in Y direction

11. Save the `.inp` file.
12. Close and reopen the model for the change to take effect.

Now when you select and clear checkboxes in the dialog, the number of bolts changes accordingly in the model. For example:



NOTE Tekla Structures automatically adds the **Effect in modify** label and checkbox for each toggle group you create.

Default properties of custom component dialogs

Each custom component has a dialog that you can modify. By default, the dialog has a **General** tab for custom connections, details, and seams, and a **Position** tab for custom parts.

For more information, see Customize the custom component dialogs (page 995).

To view the current properties, double-click the custom component in the model.

Default properties of custom connections, details, and seams

By default, the dialog of a custom connection, detail, or seam has the following options:

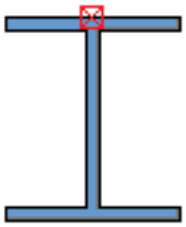
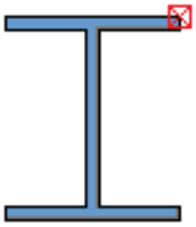
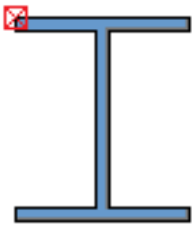
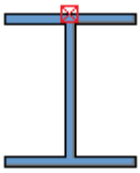
Option	Description	Note
Up direction	Indicates how the component is rotated around the secondary part, relative to the current work plane. If there are no secondary parts, Tekla Structures rotates the connection around the main part.	

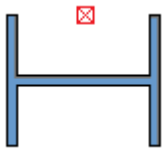
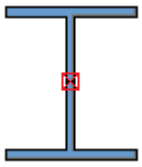
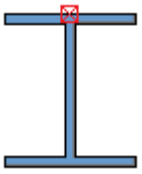
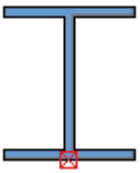
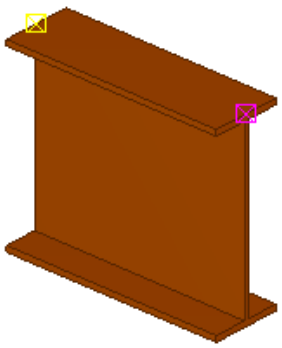
Option	Description	Note
Position in relation to primary part	The creation point of the component, relative to the main part.	Only available for details.
Position in relation to secondary part	Tekla Structures automatically places the component according to the selected option.	By default, only available for seams. If you want to use this property in connections, select the **Allow multiple instances of connection between same parts** checkbox on the **Advanced** tab when you create the component.
Place to picked positions	Select this option to place the seam at the points you pick.	Only available for seams.
Detail type	Determines on which side of the main part the component is located. The options are: • **Intermediate detail** Tekla Structures creates all components on the same side of the main part. • **End detail** Tekla Structures creates all components on the side that is closest to the details. Only affects asymmetric components.	Only available for details.
Locked	Select **Yes** to prevent other users from modifying the properties.	
Class	The class of the parts that the custom component creates.	
Connection code	Identifies the component. You can display this connection code in connection marks in drawings.	

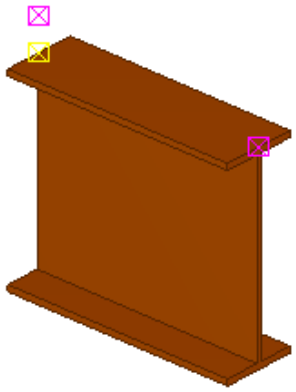
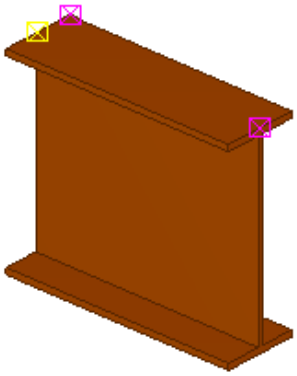
Option	Description	Note
AutoDefaults rule group	The rule group used for setting the connection properties.	
AutoConnection rule group	The rule group Tekla Structures uses to select the connection.	

Default properties of custom parts

By default, the dialog of a custom part has the following options:

Option	Description	Example
On plane	Changes part location on the work plane.	**Middle** 
		Right 
		left 
Rotation	Rotates the part in steps of 90 degrees.	**Top** and **Below** 

Option	Description	Example
		Front and **Back** 
At depth	Changes part location perpendicular to the work plane.	**Middle** 
		Front 
		Behind 
Show third handle	Sets the third handle of a nested custom part visible in the desired direction. You can bind the third handle in the desired direction and thus force the part to follow the rotation of another part.	**No** 

Option	Description	Example
		Above 
		On the left 

See also

Edit custom components (page 887)

Customize the custom component dialogs (page 995)

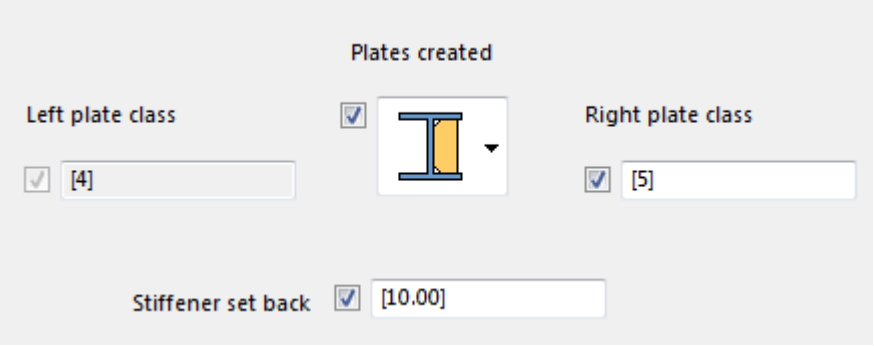
Example: Customize the dialog of a custom stiffener detail (page 1011)

Example: Customize the dialog of a custom stiffener detail

This example shows how to customize the dialog of a custom stiffener detail to make it easier to adjust the settings later on.

In the beginning, the custom component dialog box looks like the following:





Do this

1. Create a custom stiffener detail (page 970)with all the needed variables that control the creation of stiffener plates.
2. Add a list with images. (page 1012)
3. Arrange text boxes and labels. (page 1017)
4. Dim unavailable options. (page 1019)

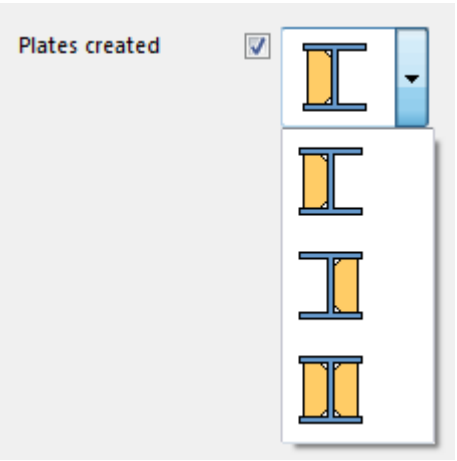
Example: Add a list with images in a custom component dialog

This example shows how to add an illustrative list in the custom stiffener detail dialog. You can do this either in the **Custom component dialog editor** tool or by editing the input (`.inp`) file manually.

In the beginning of this example the dialog has the text box shown below, and the you need to know the values (0 is left, 1 is right, and 2 is both plates) that control the creation of stiffener plates.



Replace the text box with a list that is easier to use:



Add a list by using the dialog editor

1. Create a custom stiffener detail (page 970) with all the needed variables that control which stiffener plates are created.

   In this example, the variable is called **Plates created**.

2. Open the stiffener dialog for editing.

   a. In the model, select the custom stiffener detail.

   b. Right-click and select **Edit custom component dialog**.

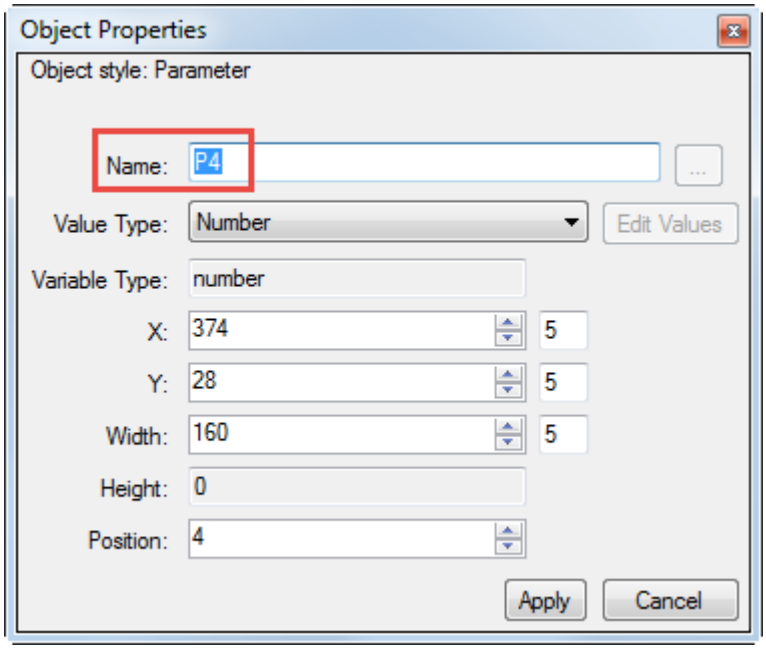
3. Check the name of the parametric variable that controls the plate creation.

   a. In the **Custom component dialog editor**, double-click the **Plates created** box.

   The **Object Properties** dialog opens.

   b. Check the name of the parametric variable.

   In this example, the name is **P4**.



   c. Click **Cancel** to close the dialog.

4. Select the **Plates created** text box and click **Delete**.

5. Click **Insert** --> **Attribute** to add a new attribute list.

6. Drag the attribute list to a suitable location, next to the **Plates created** label.

7. Select the attribute list and then click **Modify** --> **Properties** to edit its properties.

8. Enter **P4** as the **Name** of the attribute.

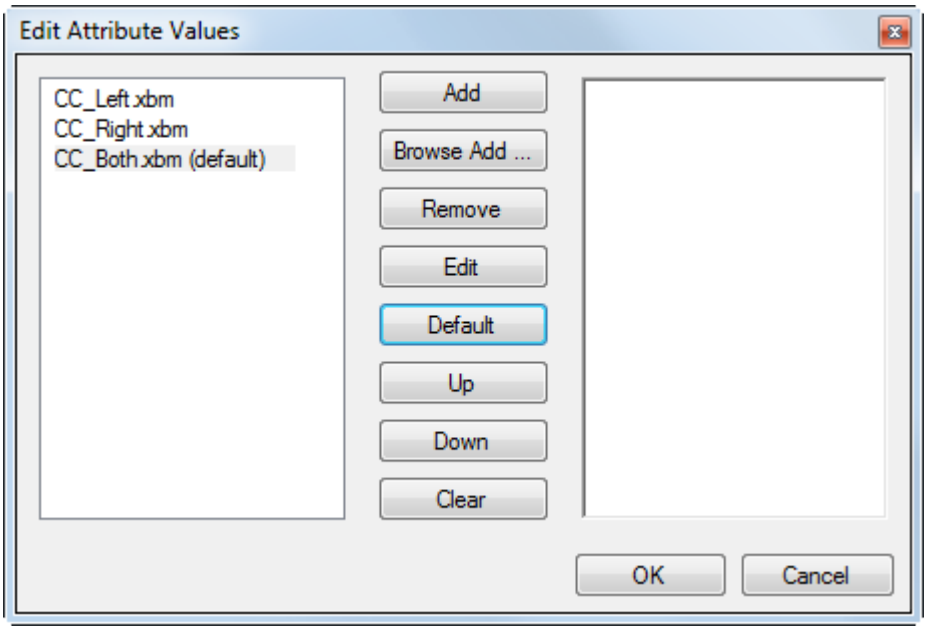
   Now the attribute list is linked to the parametric variable that controls the plate creation.

9. Click **Edit Values** to add the list items.
10. In the **Edit Attribute Values** dialog, add an image for the left plate.
    a. Click **Browse Add**.
    b. Browse for a suitable image.

       If you create new images, make sure they are in the bitmap (`.bmp`) format. Save the images in the `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps` folder.

       The maximum image size is 245x245 pixels.

    c. Click **Open**.
11. Repeat step 10 to add an image for the right plate, and then for both plates.
12. In the **Edit Attribute Values** dialog, select the image of both plates and then click **Default** to make the attribute the default value.



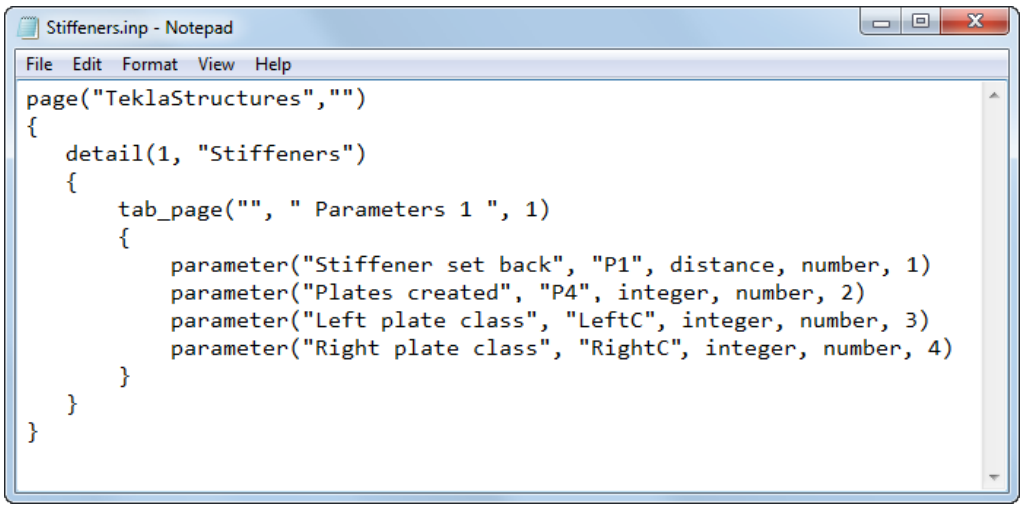
13. Click **OK**.
14. Click **Apply** in the **Object Properties** dialog, and then click **Cancel** to close the dialog.
15. In the dialog editor, click **File** --> **Save** to save the changes.
16. Close and reopen the model for the change to take effect.

Add a list by editing the .inp file

1. Create a custom stiffener detail (page 970) with all the needed variables that control which stiffener plates are created.

   In this example, the variable is called **Plates created**.

2. In the model, click **File** --> **Open model folder** to open the current model folder.
3. Go to the `\CustomComponentDialogFiles` folder.
4. Open the `.inp` file in a text editor.



5. Remove the following line:

   ```
   parameter("Plates created", "P4", integer, number, 2)
   ```

6. Add a new **Plates created** attribute with the following settings:

```
page("TeklaStructures","")
{
   detail(1, "Stiffeners")
   {
       tab_page("", " Parameters 1 ", 1)
       {
           parameter("Stiffener set back", "P1", distance, number, 1)
           parameter("Left plate class", "LeftC", integer, number, 3)
           parameter("Right plate class", "RightC", integer, number, 4)
           attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
       }
   }
}
```

7. Add a new P4 attribute with the following settings:

```
page("TeklaStructures","")
{
   detail(1, "Stiffeners")
   {
       tab_page("", " Parameters 1 ", 1)
       {
           parameter("Stiffener set back", "P1", distance, number, 1)
           parameter("Left plate class", "LeftC", integer, number, 3)
           parameter("Right plate class", "RightC", integer, number, 4)
           attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
           attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
        {
           value ("Left", 0)
           value ("Right", 0)
           value ("Both", 1)
        }
       }
   }
}
```

The list now contains three options, and **Both** is the default value. The list options are linked to the variable P4 that controls the creation of the stiffener plates.

8. Edit the line numbers so that there are no empty rows between the variables in the dialog.

```
page("TeklaStructures","")
{
   detail(1, "Stiffeners")
   {
       tab_page("", " Parameters 1 ", 1)
       {
           parameter("Stiffener set back", "P1", distance, number, 1)
           parameter("Left plate class", "LeftC", integer, number, 2)
           parameter("Right plate class", "RightC", integer, number, 3)
           attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
           attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
        {
           value ("Left", 0)
           value ("Right", 0)
           value ("Both", 1)
        }
       }
   }
}
```

9. Browse for the images you want to use in the dialog.

   If you create new images, make sure they are in the bitmap (`.bmp`) format. Save the images in the `..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps` folder.

   The maximum image size is 245x245 pixels.

10. Replace the option texts with the actual file names of the images, but with the file name extension `.xbm`.

```
page("TeklaStructures","")
{
   detail(1, "Stiffeners")
   {
       tab_page("", " Parameters 1 ", 1)
       {
           parameter("Stiffener set back", "P1", distance, number, 1)
           parameter("Left plate class", "LeftC", integer, number, 2)
           parameter("Right plate class", "RightC", integer, number, 3)
           attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
           attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
        {
           value ("CC_Left.xbm", 0)
           value ("CC_Right.xbm", 0)
           value ("CC_Both.xbm", 1)
        }
       }
   }
}
```

11. Save the `.inp` file.
12. Close and reopen the model for the change to take effect.

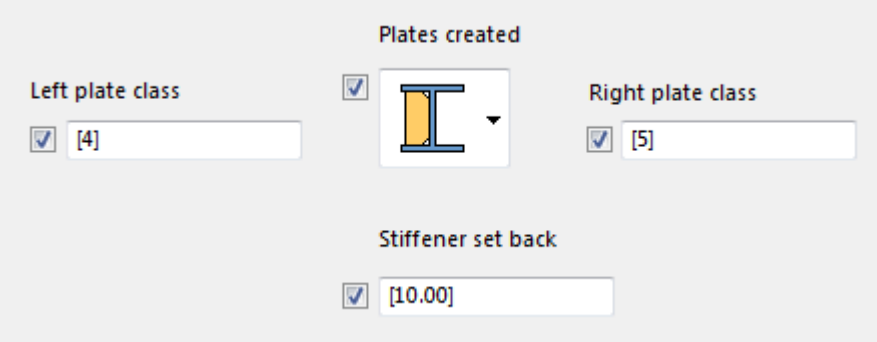
Example: Arrange text boxes and labels in a custom component dialog

This example shows how to arrange the text boxes and labels around a list in the custom component dialog. You can do this either in the **Custom component dialog editor** tool or by editing the input (`.inp`) file manually.

In the beginning of this example the dialog looks like the following:



Arrange the dialog elements more nicely, in the following manner:



Arrange the elements by using the dialog editor

1. Create a custom stiffener detail (page 970) with all the needed variables that control the creation of stiffener plates.
2. Open the stiffener dialog for editing.
   a. In the model, select the custom stiffener detail.
   b. Right-click and select **Edit custom component dialog**.
3. Drag the **Plates created** label above the list with images.
4. Drag the **Left plate class** label and the corresponding text box to the left side of the list.
5. Drag the **Right plate class** label and the corresponding text box to the right side of the list.
6. Drag the **Stiffener set back** label and the corresponding text box underneath the list.
7. In the dialog editor, click **File** --> **Save** to save the changes.
8. Close and reopen the model for the change to take effect.

Arrange the elements by editing the .inp file

1. Create a custom stiffener detail (page 970) with all the needed parametric variables that control the creation of stiffener plates.
2. In the model, click **File** --> **Open model folder** to open the current model folder.
3. Go to the `\CustomComponentDialogFiles` folder.
4. Open the `.inp` file in a text editor.
5. Edit the file as follows:

```
page("TeklaStructures", "")
{
    detail(1, "Stiffeners")
    {
        tab_page("", " Parameters 1 ", 1)
        {
            attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
            attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
            {
                value("CC_Left.xbm", 0)
                value("CC_Right.xbm", 0)
                value("CC_Both.xbm", 1)
            }
            attribute("", "Left plate class", label, "%s", none, none, "0", "0", 125, 157)
            attribute("", "Right plate class", label, "%s", none, none, "0", "0", 497, 160)
            parameter("", "LeftC", integer, number, 146, 192, 160)
            parameter("", "RightC", integer, number, 522, 194, 160)
            parameter("", "P1", distance, number, 357, 289, 160)
            attribute("", "Stiffener set back", label, "%s", none, none, "0", "0", 330, 255)
        }
    }
}
```

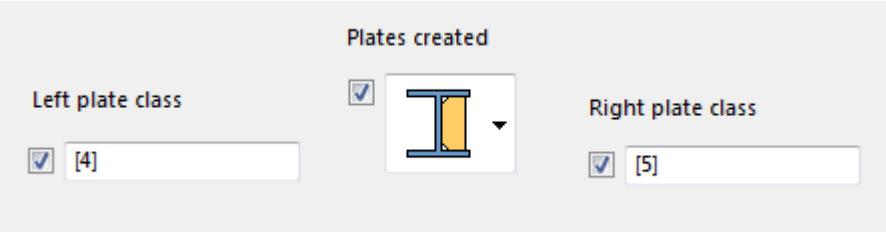
6. Save the `.inp` file.

7. Close and reopen the model for the change to take effect.

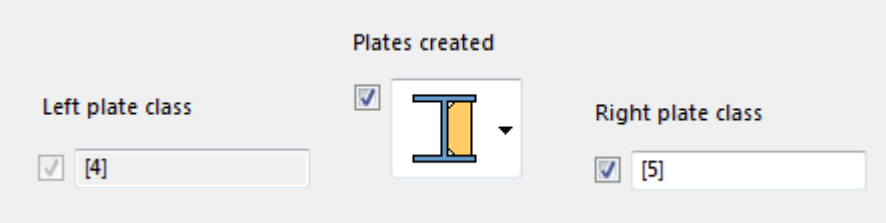
Example: Dim unavailable options in a custom component dialog

This example shows how to dim the unavailable options in the custom stiffener detail dialog based on conditions. You can do this either in the **Custom component dialog editor** tool or by editing the input (`.inp`) file manually.

In the beginning of this example all the options are available:



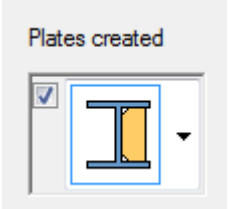
Define that the **Left plate class** text box is unavailable if only the right plate is created, and vice versa.



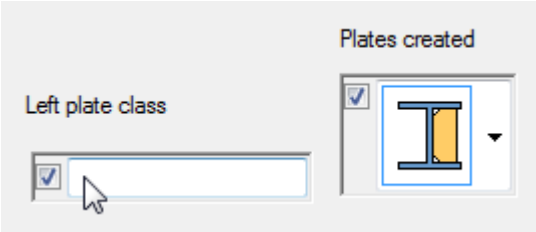
Dim unavailable options by using the dialog editor

1. Create a custom stiffener detail (page 970) with all the needed parametric variables that control the creation of stiffener plates.
2. Open the stiffener dialog for editing.
   a. In the model, select the custom stiffener detail.
   b. Right-click and select **Edit custom component dialog**.
3. Define that the **Left plate class** text box must be dimmed if only the right stiffener plate is created.
   a. In the **Plates created** list, select the image for the right plate class.

      Note that a blue selection border must be displayed for the image:



b. Hold down the **Ctrl** key and click the **Left plate class** text box.



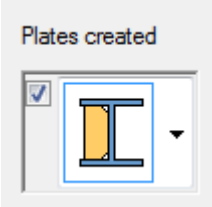
c. Click the **Toggle visibility** button.

The **Left plate class** text box is now dimmed:

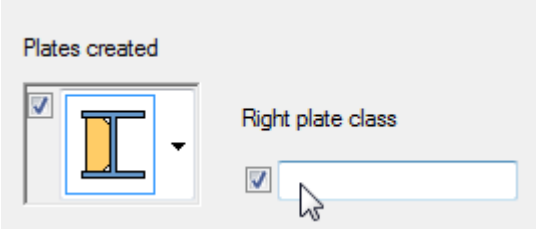


4. Unselect the **Left plate class** text box by clicking the **Right plate class** text box.
5. Define that the **Right plate class** text box must be dimmed if only the left stiffener plate is created.
   a. In the **Plates created** list, select the image for the left plate class.

   Note that a blue selection border must be displayed for the image:



   b. Hold down the **Ctrl** key and select the **Right plate class** text box.



   c. Click the **Toggle visibility** button.

The **Right plate class** text box is now dimmed:



6. In the dialog editor, click **File** --> **Save** to save the changes.
7. Close and reopen the model for the change to take effect.

Dim unavailable options by editing the .inp file

1. Create a custom stiffener detail (page 970) with all the needed parametric variables that control the creation of stiffener plates.
2. In the model, click **File** --> **Open model folder** to open the current model folder.
3. Go to the `\CustomComponentDialogFiles` folder.
4. Open the `.inp` file in a text editor.
5. Add the following line to the end of the attribute P4 line:

   `"toggle_field:LeftC=0;RightC=1"`

```
page("TeklaStructures", "")
{
    detail(1, "Stiffeners")
    {
        tab_page("", " Parameters 1 ", 1)
        {
            attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
            attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90, "toggle_field:LeftC=0;RightC=1")
            {
                value("CC_Left.xbm", 0)
                value("CC_Right.xbm", 0)
                value("CC_Both.xbm", 1)
            }
            attribute("", "Left plate class", label, "%s", none, none, "0", "0", 125, 157)
            attribute("", "Right plate class", label, "%s", none, none, "0", "0", 497, 160)
            parameter("", "LeftC", integer, number, 146, 192, 160)
            parameter("", "RightC", integer, number, 522, 194, 160)
            parameter("", "P1", distance, number, 357, 289, 160)
            attribute("", "Stiffener set back", label, "%s", none, none, "0", "0", 330, 255)
        }
    }
}
```

   The logic is the following:

   Selecting the image **CC_left** returns 0, **CC_right** returns 1, and **CC_both** returns 2.

   `toggle_field:RightC=1`

   When 0 (left) is selected, **RightC** is dimmed.

   `toggle_field:LeftC=0`

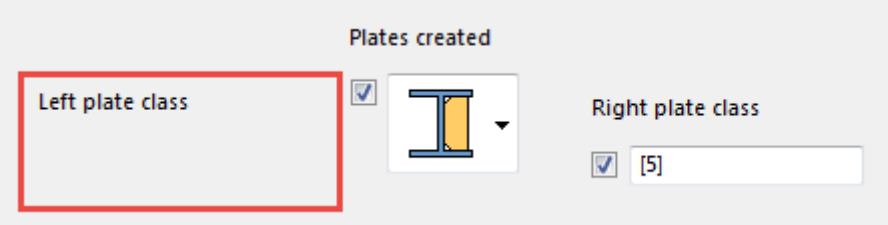
   When 1 (right) is selected, **LeftC** is dimmed.
6. Save the `.inp` file.

7. Close and reopen the model for the change to take effect.

TIP If you want to hide unavailable options instead of dimming them in the stiffener dialog, add an exclamation mark in the conditions:

```
"toggle_field:!LeftC=0;!RightC=1"
```

The option is now completely hidden when unavailable:



8 Predefined parametric profiles available in Tekla Structures

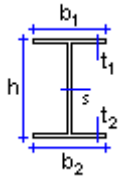
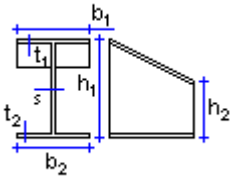
The below predefined parametric profiles are available in Tekla Structures.

The profiles are listed in the same order as they appear in the profile catalog in the default environment.

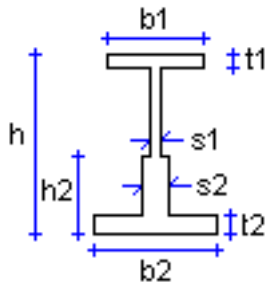
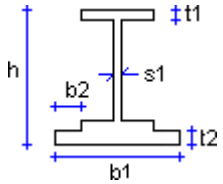
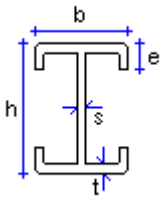
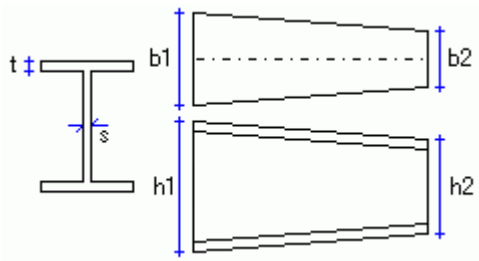
To change how the profiles are grouped in the profile catalog, modify the profile catalog rules.

You can download additional predefined profiles from Tekla Warehouse.

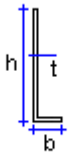
8.1 I profiles

	HIh-s-t*b (symmetric) HIh-s-t1*b1-t2*b2
	HIh1-h2-s-t*b HIh1-h2-s-t1*b1-t2*b2

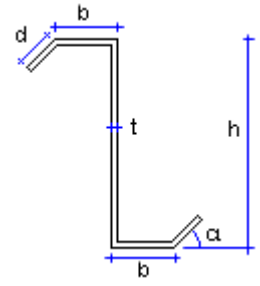
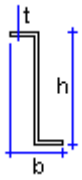
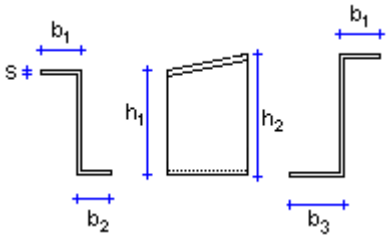
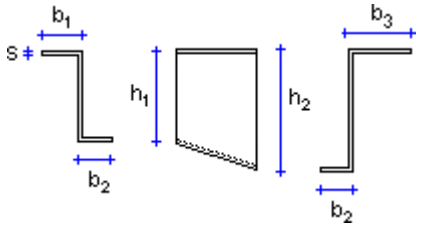
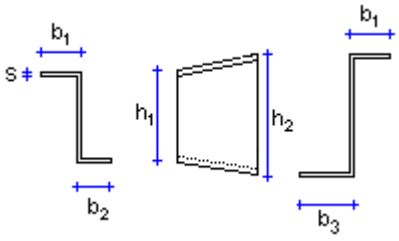
8.2 I beams (steel)

	I_BLT_Ah-b1-s1-t1*h2-b2-s2-t2
	I_BLT_B h*b1*t1*s-b2*t2
	I_HEMh*b*c*s*t
	I_VAR_Ah1-ht*b1-bt*s*t

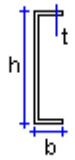
8.3 L profiles

	Lh*b*t

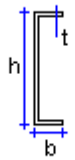
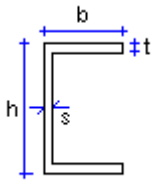
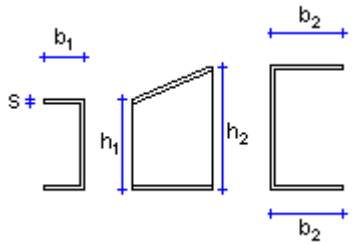
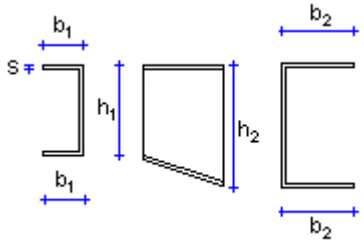
8.4 Z profiles

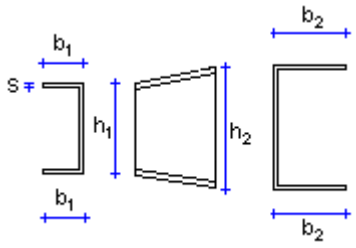
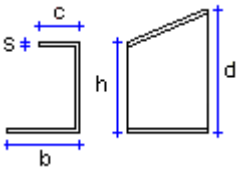
	BENTZ h*b*d*t[-a]
	Z h*b*t
	Z_VAR_A h1*b1*b2-s-h2*b3
	Z_VAR_B h1*b1*b2-s-h2*b3
	Z_VAR_C h1*b1*b2-s-h2*b3

8.5 U profiles

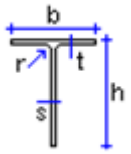
	Uh*b*t

8.6 C profiles

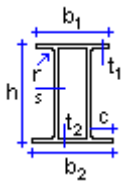
	Ch*b*t
	C_BUILTh*b*s*t
	C_VAR_Ah1*b1-s-h2*b2
	C_VAR_Bh1*b1-s-h2*b2

	C_VAR_Ch1*b1-s-h2*b2
	C_VAR_Dh-b-d-c-s

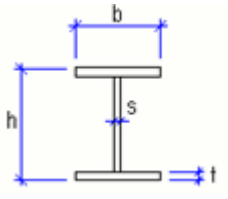
8.7 T profiles

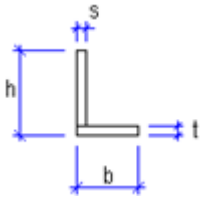
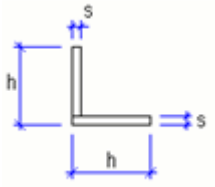
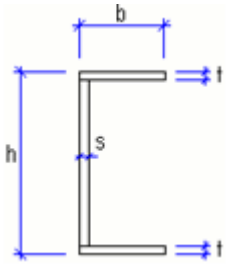
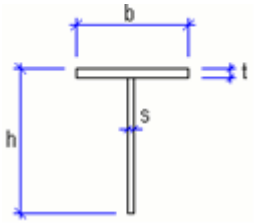
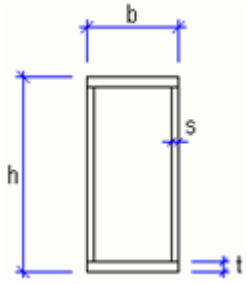
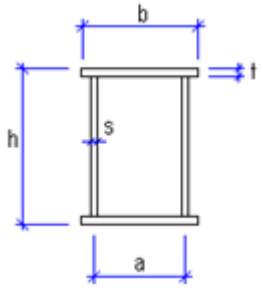
	Th-s-t-b

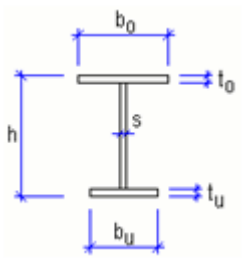
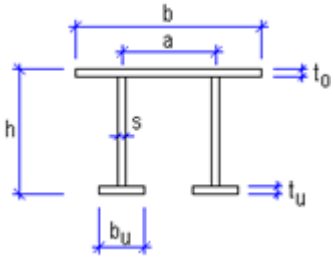
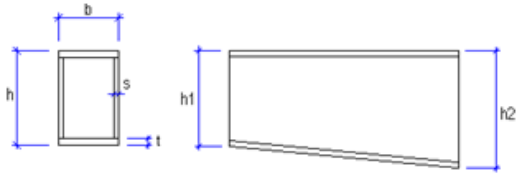
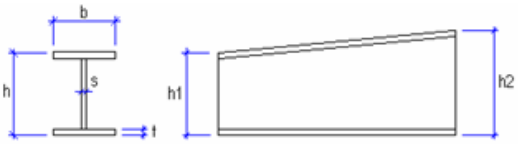
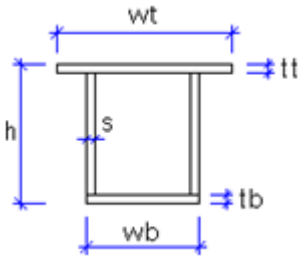
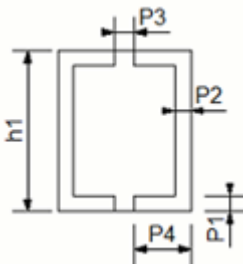
8.8 Welded box profiles

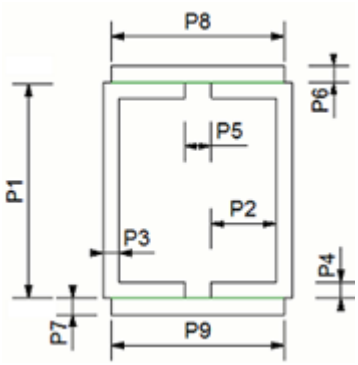
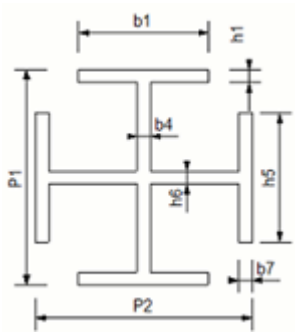
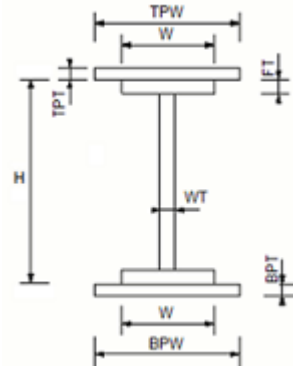
	HK h-s-t*b-c HKh-s-t1*b1-t2*b2-c

8.9 Welded beam profiles

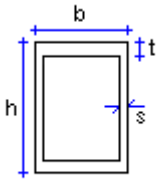
	B_WLD_A h*b*s*t

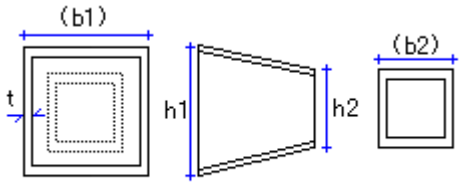
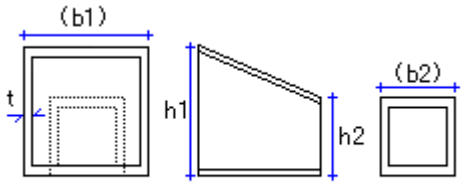
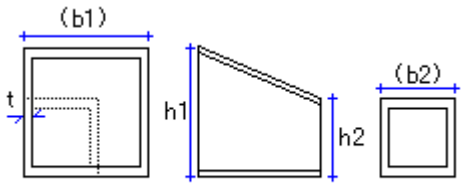
	B_WLD_B h*b*s*t
	B_WLD_C h*s
	B_WLD_D h*b*s*t
	B_WLD_E h*b*s*t
	B_WLD_F h*b*s*[t]
	B_WLD_G h*b*s*t*a

	B_WLD_H h*bo*bu*s*to*tu
	B_WLD_I h*bo*s*to*bu*tu*a
	B_WLD_J h1*h2*b*s*t
	B_WLD_K h1*h2*b*s*t
	B_WLD_L h*wt*wb*s*tt*tb
	B_WLD_M h1*p1*p2*p3*p4

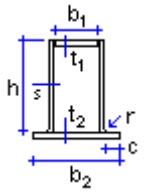
	B_WLD_N p1*p2*p3*p4*p5*p6*p7*p8*p9
	B_WLD_O b1*h1*b4*h5*b7*h6*P1*P2
	B_WLD_P W*H*FT*WT*TPT*TPW*BPT*BPW

8.10 Box profiles

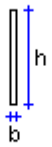
	B_BUILTh*b*s*t

	B_VAR_Ah1-h2*t
	B_VAR_Bh1-h2*t
	B_VAR_Ch1-h2*t

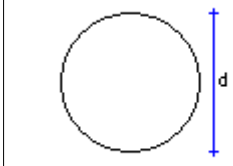
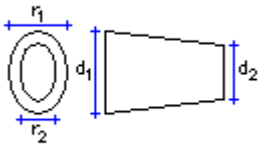
8.11 WQ profiles

	HQh-s-t1*t2*b2 HQh*s-t1*b1-t2*b2-c

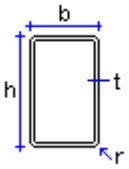
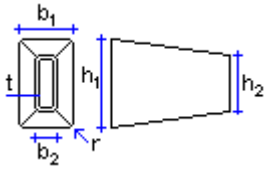
8.12 Rectangular sections

	PLh*b h=height b=thickness (smaller=b)

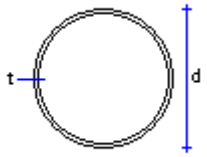
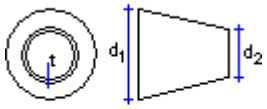
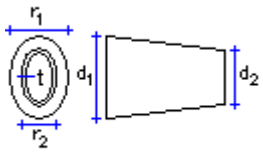
8.13 Circular sections

	Dd
	ELDd1*r1*d2*r2

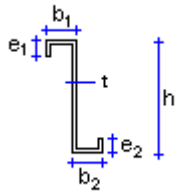
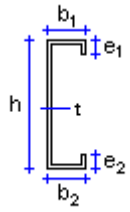
8.14 Rectangular hollow sections

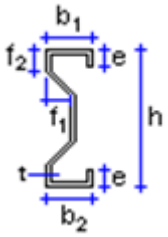
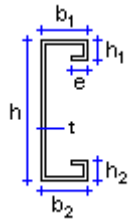
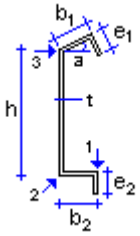
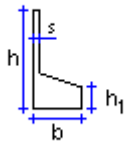
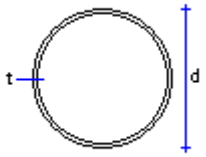
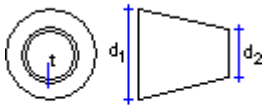
	Ph*t (symmetric) Ph*b*t
	Ph1*b1-h2*b2*t

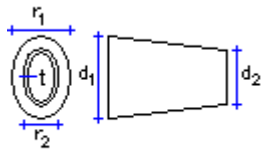
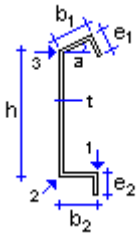
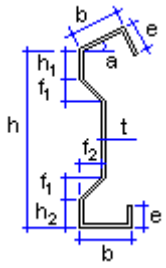
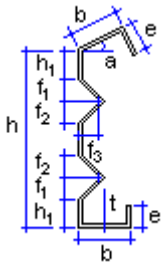
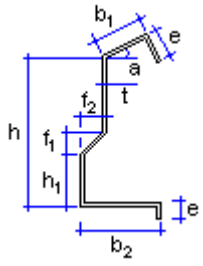
8.15 Circular hollow sections

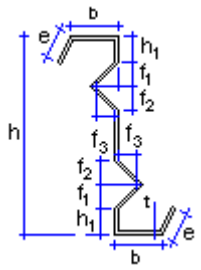
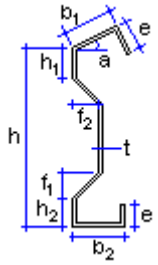
	PDd
	PDd1*d2*t
	EPDd1*r1*d2*r2*t

8.16 Cold rolled profiles

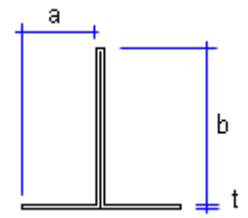
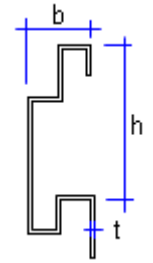
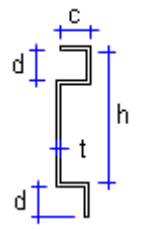
	ZZh-t-e-b (symmetric) ZZh-t-e1-b1-e2-b2
	CCh-t-e-b (symmetric) CCh-t-e1-b1-e2-b2

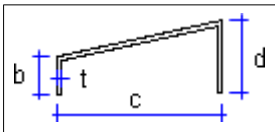
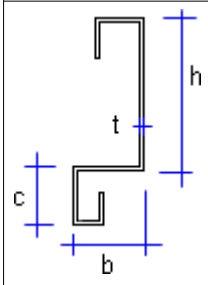
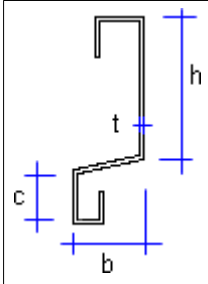
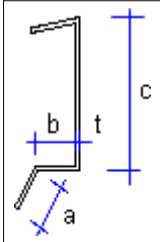
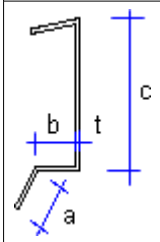
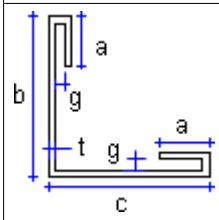
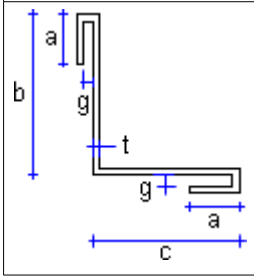
	CW h-t-e-b-f-h1 (symmetric) CW h-t-e1*b1-f1-f2-e2*b2
	CUh-t-h1-b-e (symmetric) CUh-t-h1-b1-h2-b2-e
	EBh-t-e-b-a EBh-t-e1-b1-e2-b2-a Reference points: 1=right 2=left 3=top
	BFh-s-b-h1
	SPDd*t
	SPDd2*d2*t

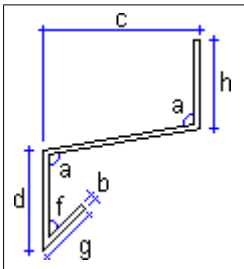
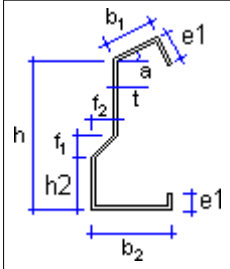
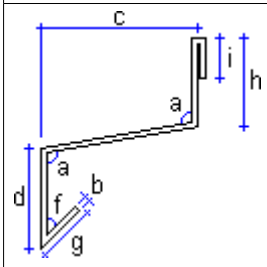
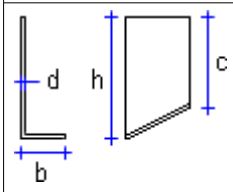
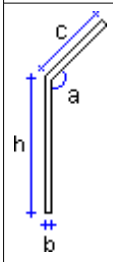
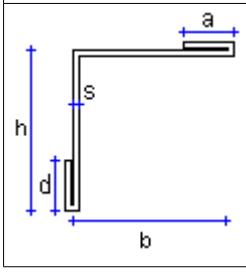
	ESPD d1-d2*t
	ECh-t-e-b-a ECh-t-e1-b1-e2-b2-a
	EDh-t-b-e-h1-h2-f1-f2-a
	EEh-t-e-b-f1-f3-h1-f2-a
	EFh-t-e-b1-b2-f1-f2/h1-a

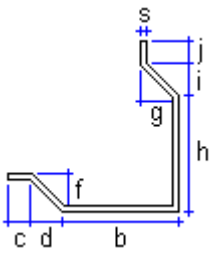
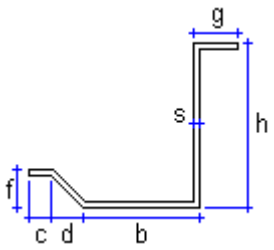
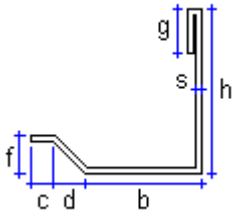
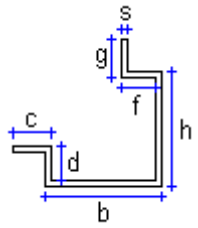
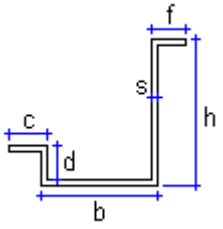
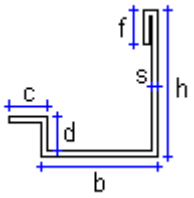
	EZh-t-e-b-f1-f3-h1-f2-a
	EWh-t-e-b1-b2-f1-f2-h2-h1-a

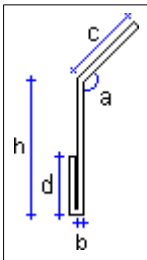
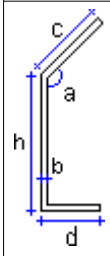
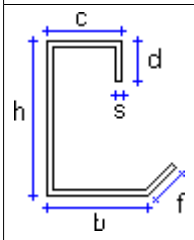
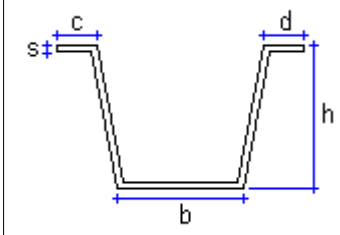
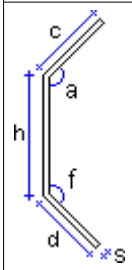
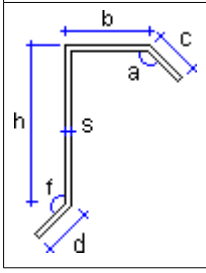
8.17 Folded plates

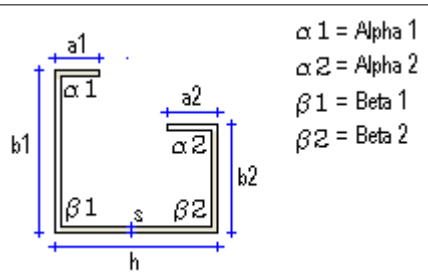
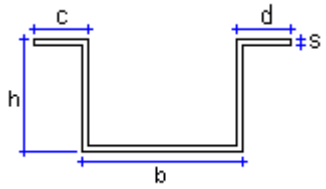
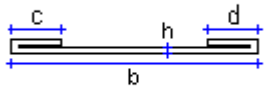
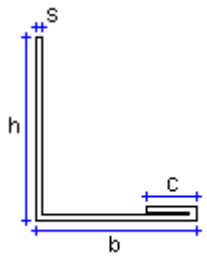
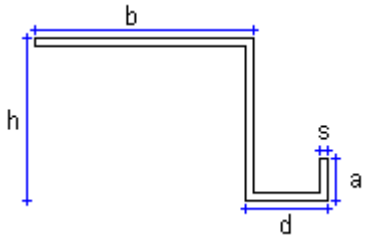
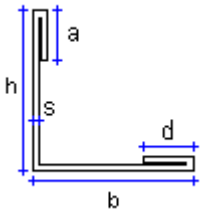
	FFLAa-b-t
	FPANBh-b-t FPANB_-b-t FPANBAh-b-t FPANBA_h-b-t
	FPANBBh-c-d-t

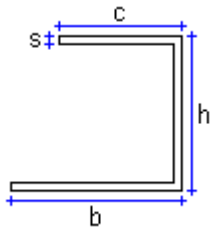
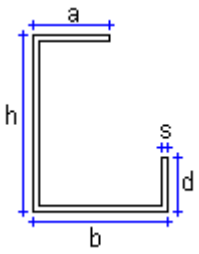
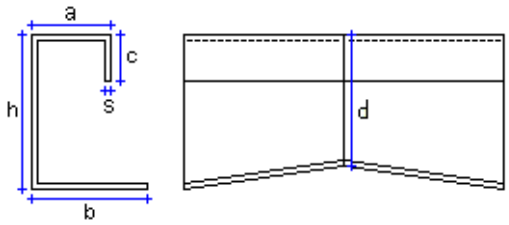
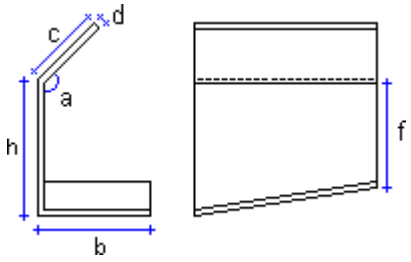
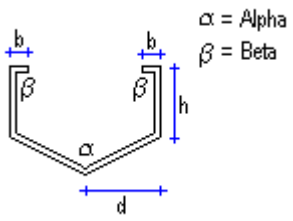
	FPANCVb-c-d-t
	FPANGh-b-c-t
	FPANGAh-b-c-t
	FPANJa-b-c-t
	FPANJa-b-c-t
	FPAN a-b-c-t-g
	FPANVVa-b-c-t-g

	FP_Ah-b-c-d-g
	FP_AAh*b2*t*a
	FP_Bh-b-c-d-g-i
	FP_BBh-b-d
	FP_Cb-h-c
	FP_CCh-b-a-d-s

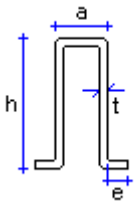
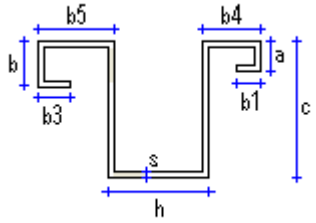
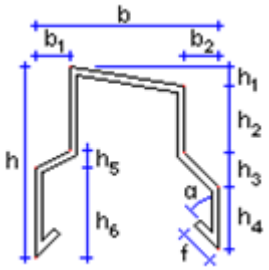
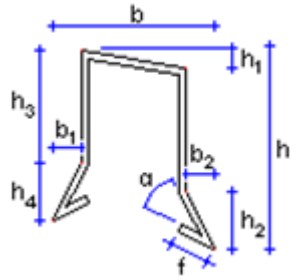
	FP_Db-h-c-d-f-g-i-j-s
	FP_Eb-h-c-d-f-g-s
	FP_Fb-h-c-d-f-g-s
	FP_Gb-h-c-d-f-g-s
	FP_Hb-h-c-d-f-s
	FP_Ib-h-c-d-f-s

	FP_Jb-h-c-d-a
	FP_Kb-h-c-d
	FP_Lb-h-c-d-f-s
	FP_Mb-h-c-d-s
	FP_Nb-h-c-d
	FP_Ob-h-c-d-s

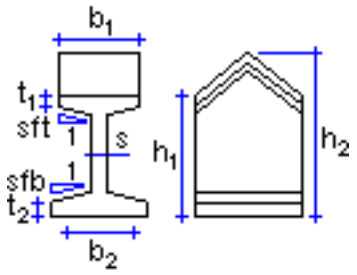
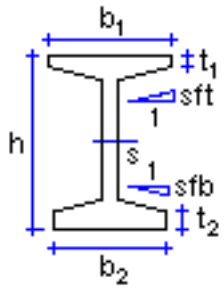
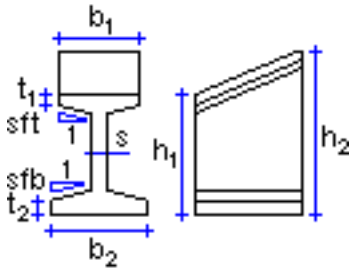
	FP_Pa1*a2*h-b1*b2-Alpha1-Alpha2-Beta1-Beta2-s
	FP_Qb-h-c-d-s
	FP_Rb-h-c-d
	FP_Sb-h-c-s
	FP_Tb-h-a-d-s
	FP_Ub-h-a-d-s

	FP_Vb-h-s-c
	FP_Wb-h-a-d-s
	FP_WWh-b-a-c-s
	FP_Yh-b-c-d
	FP_Zd-h-b-s-a-f

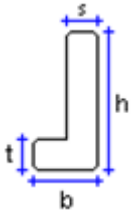
8.18 Hat profiles

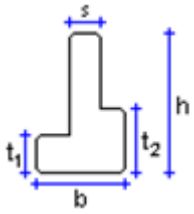
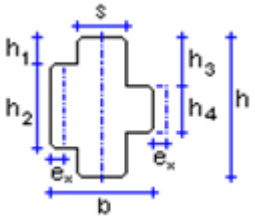
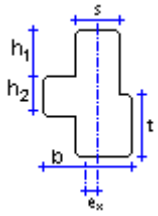
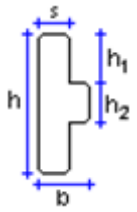
	HAT h*a*c*t
	HATCa-b-c-b1-h-b3-b4-b5-s
	HATAb1*h1*h2*h3*h4*h5*h6*b2*t*f*a*h*b
	HATBb*b1*b2*h*h1*h2*h3*h4*t*f*a

8.19 I beams (concrete)

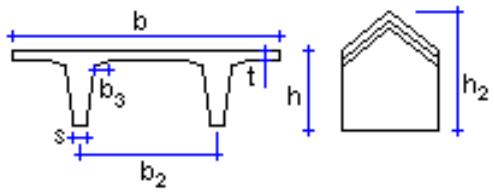
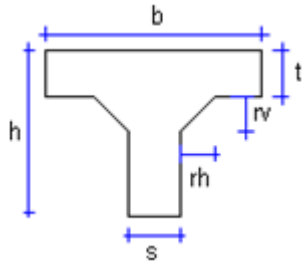
	HIIh1*b1*t1-h2-s-b2*t2[-sft[-sfb]]
	IIh*b1*t1-s-b2*t2[-sft[-sfb]]
	SIIh1*b1*t1-h2-s-b2*t2[-sft[-sfb]]

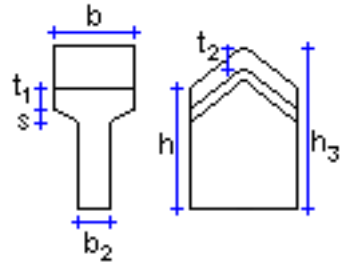
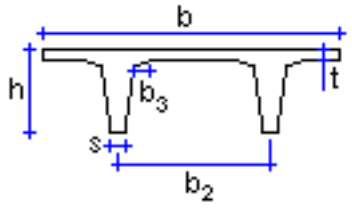
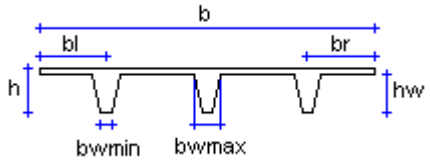
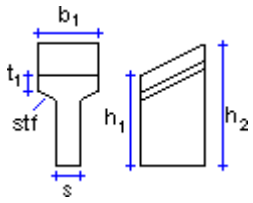
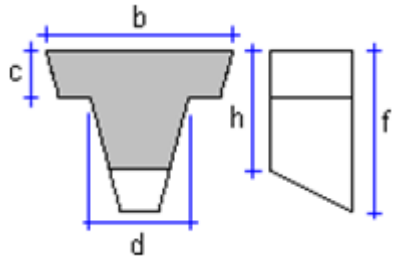
8.20 Ledger beams (concrete)

	RCLs*h-b*t

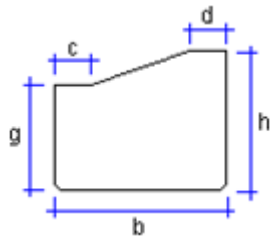
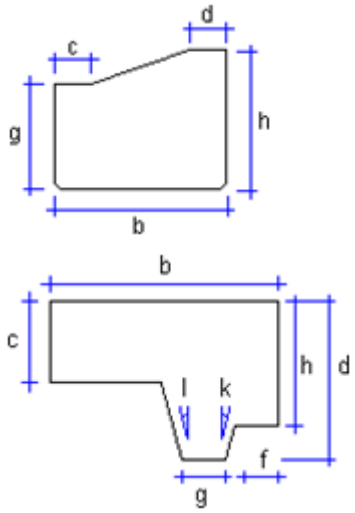
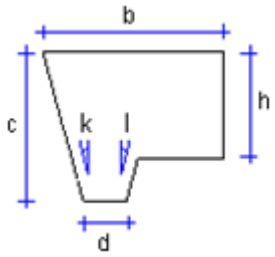
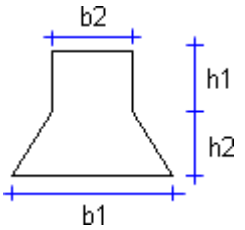
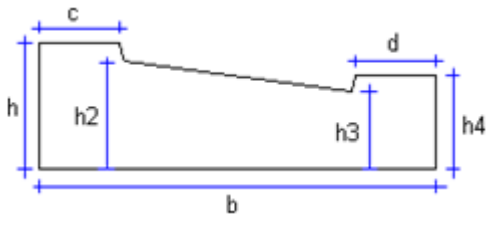
	RCDLs*h-b*t RCDLs*h-b*t1*t2
	RCDXs*h-b*h2*h1 RCDXs*h-b*h4*h3*h2*h1 RCDXs*h-b*h4*h3*h2*h1-ex
	RCXXs*h-b*t*h1-h2-ex
	RCXs*h-b*h2*h1

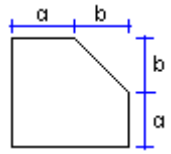
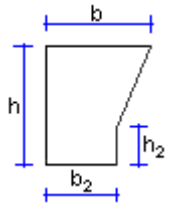
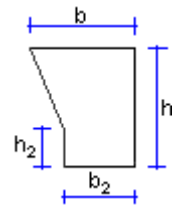
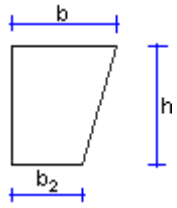
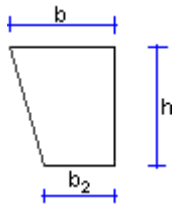
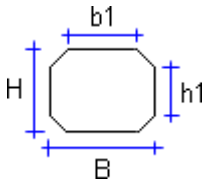
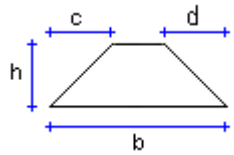
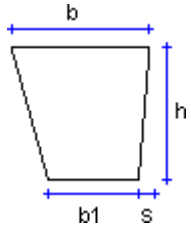
8.21 T profiles (concrete)

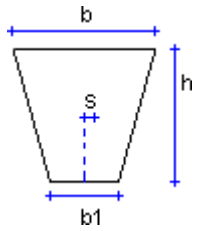
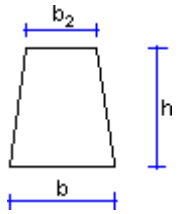
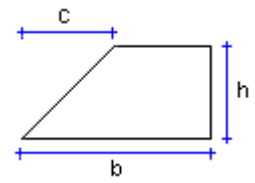
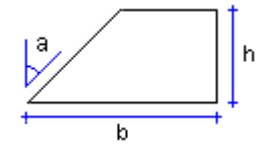
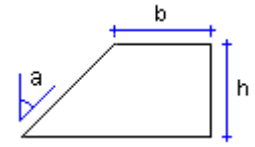
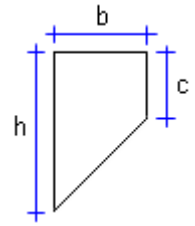
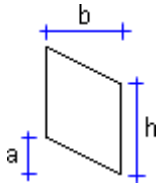
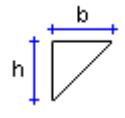
	HTTh*b-s-t-b2-h2
	TCh-b-t-s

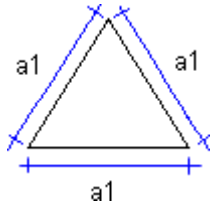
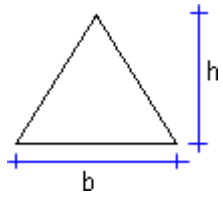
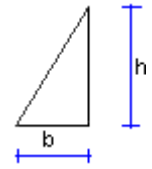
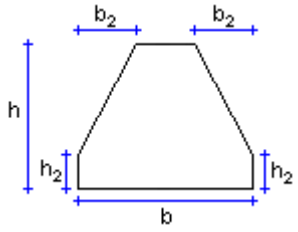
	TRIh*b-b2*t1-h3-t2
	TTh*b-s-t-b2
	TTTh*b-bl-br-hw-bwmin-bwmax
	T_VAR_Ah1*h2*s*b1*t1-sft
	T_VAR_Bh-b-c-d

8.22 Irregular beams (concrete)

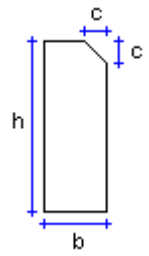
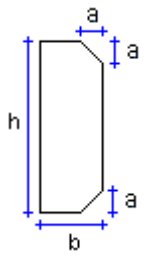
	IRR_Ab-h-g-c-d
	IRR_Bh-b-c-d-f-g
	IRR_Ch-b-c-d
	IRR_Db1*b2-h1*h2
	IRR_Eh-b-c-d-h2-h3-h4

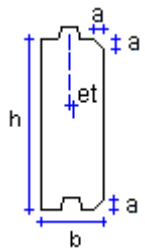
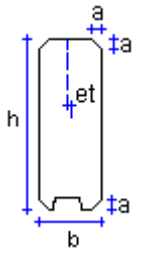
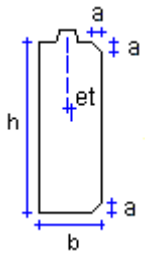
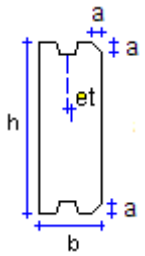
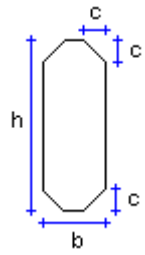
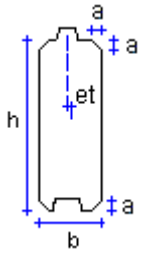
	IRR_Fa*b
	IRR_Gh*b*h2*b2
	IRR_Hh*b*h2*b2
	IRR_Ih*b*b2
	IRR_Jh*b*b2
	OCTB*b1-H*h1
	REC_Ah-b
	REC_Bh-b-b1

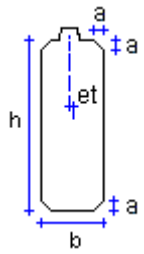
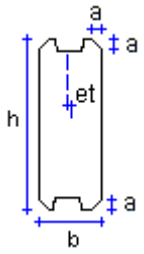
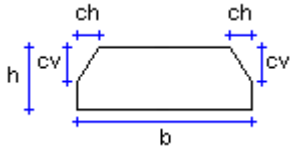
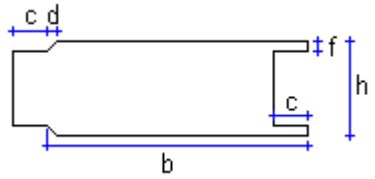
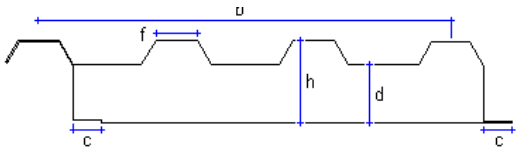
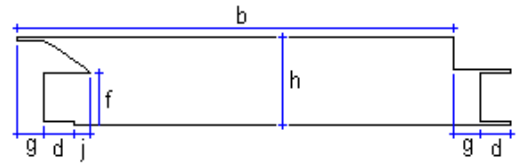
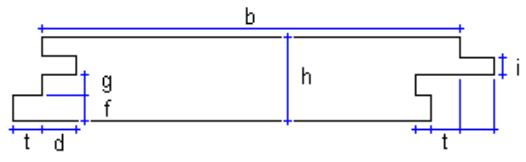
	REC_Ch-b-b1
	REC_Dh-b-b2
	REC_Eh-b
	REC_Fh-b
	REC_Gh-b
	REC_Hh-b
	REC_I a-b*h
	TRI_Ah-b

	TRI_Ba1
	TRI_Cb-h
	TRI_Dh*b
	TRI_Eb*h*h2*b2

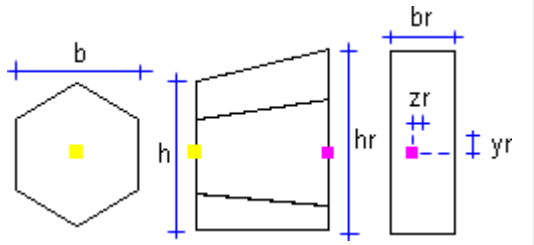
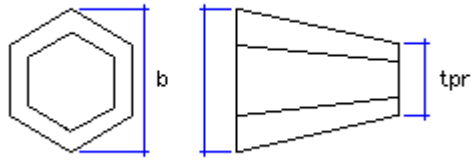
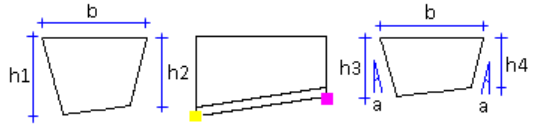
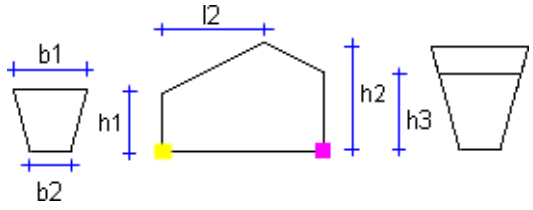
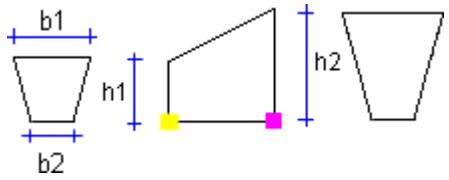
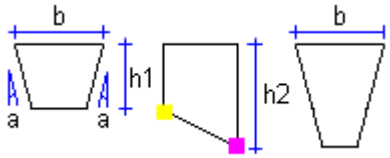
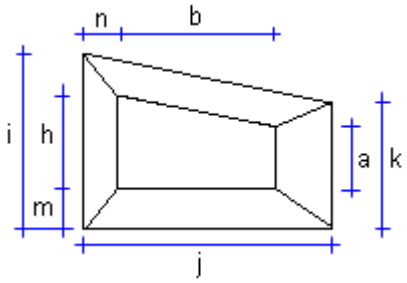
8.23 Panels

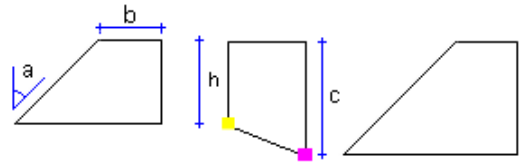
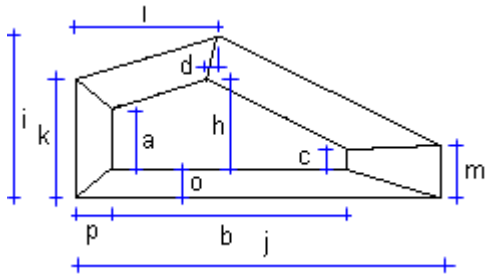
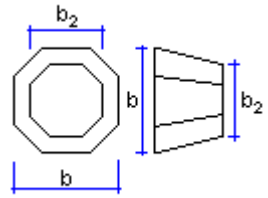
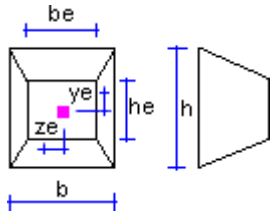
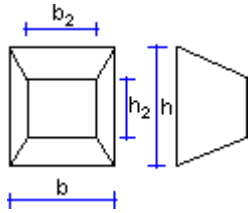
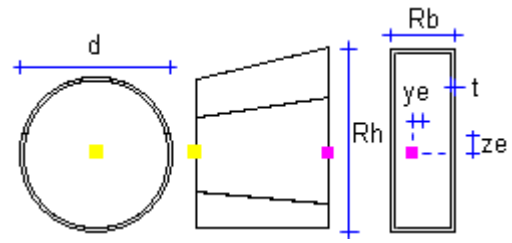
	PNL_Ah*b
	PNL_Bh*b

	PNL_Ch*b-a-ht*bt
	PNL_Dh*b-a-ht*bt
	PNL_Eh*b-a-ht*bt
	PNL_Fh*b-a-ht*bt
	PNL_Gh*b
	PNL_Hh*b-a-ht

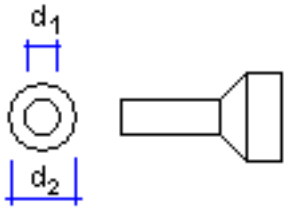
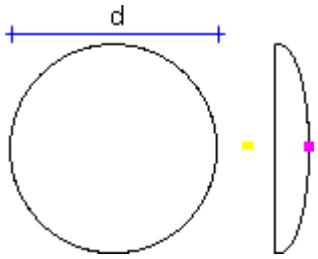
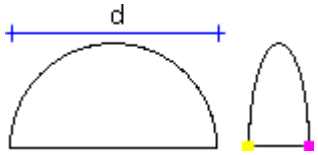
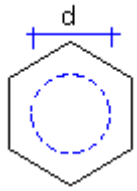
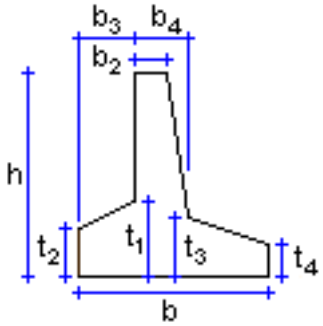
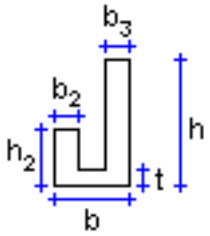
	PNL_Ih*b-a-ht*bt
	PNL_Jh*b-a-ht*bt
	PNL_Kh*b
	PNL_Lh-b-c-f
	PNL_Mh-b-c-f-d
	PNL_Nh-b-d-f-g-j
	PNL_Oh-b-d-f-g-i-t

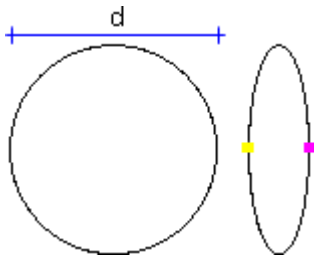
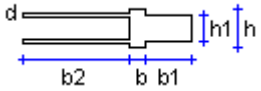
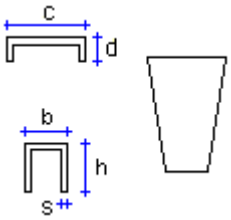
8.24 Variable cross sections

	HEXRECTh-b-br-hr
	HXGONb
	OBLINCLh1-h2-h3-h4-b
	OBLRIDh1*b1*b2-h2-h3-l2-l1
	OBLVAR_Ah1*b1*b2-h2
	OBLVAR_Bh1-h2-b
	OBLVAR_Ch-b-a-i-j-k-m-n

	OBLVAR_Dh-c-b
	OBLVAR_Eh-b-a-c-d-i-j-k-l-m-p-o
	OCTAGONb-b2
	PRMDASh*b-he*be PL_Vh*b-he*be
	PRMDh*b-h2*b2
	ROUNDRECTd-Rb*Rh-t*ye-ze

8.25 Others

	BLKSd1-d2
	CAPd
	HEMISPHERd
	NUT_Md
	RCRWh*b-b2*b3-b4-t1*t2-t3*t4
	SKh*b-h2-t-b2-b3

	SPHEREd
	STBb-h-h1-b1-b2-d
	STEPh-b*h1-b1-s

9 Modeling settings

This section provides more information about some of the settings you can modify in Tekla Structures.

Click the links below to find out more:

- Numbering settings (page 1057)
- Reinforcement settings (page 1061)

9.1 Numbering settings

This section provides more information about specific numbering settings.

Click the links below to find out more:

- General numbering settings (page 1057)
- Weld numbering settings (page 1059)
- Control number settings (page 1060)

General numbering settings

Use the **Numbering Setup** dialog to view and modify some general numbering settings.

Setting	Description
Renumber all	All parts get a new number. All information on previous numbers is lost.
Re-use old numbers	Tekla Structures reuses the numbers of parts that have been deleted. These numbers may be used to number new or modified parts.

Setting	Description
Check for standard parts	If a separate standard-part model (page 799) has been set up, Tekla Structures compares the parts in the current model to those in the standard-part model. If the part to be numbered is identical to a part in the standard-part model, Tekla Structures uses the same part number as in the standard-part model.
Compare to old	The part gets the same number as a previously numbered similar part.
Take new number	The part gets a new number even if a similar numbered part already exists.
Keep number if possible	Modified parts maintain their previous numbers if possible. Even if a part or assembly becomes identical with another part or assembly, the original position number is maintained. For example, you might have two different assemblies, B/1 and B/2, in the model. Later on you modify B/2 so that it becomes identical with B/1. If the **Keep number if possible** option is used, B/2 will maintain its original position number when you renumber the model.
Synchronize with master model (save-numbering-save)	Use this setting when working in multi-user mode. Tekla Structures locks the master model and performs a save, numbering, and save sequence, so that all other users can continue working during the operation.
Automatic cloning	If the main part of a drawing is modified and therefore gets a new assembly position, the existing drawing is automatically assigned to another part of the position. If the modified part moves to an assembly position that does not have a drawing, the original drawing is automatically cloned to reflect the changes in the modified part.
Holes	The location, size, and number of holes affects numbering.
Part name	The part name affects numbering.
Beam orientation	The orientation of beams affects numbering of assemblies.
Column orientation	The orientation of columns affects numbering of assemblies.
Assembly name	The assembly name affects numbering.
Assembly phase	Only enabled when XS_ENABLE_PHASE_OPTION_IN_NUMBERING is set to `TRUE`. The assembly phase affects numbering.
Reinforcing bars	Reinforcing bars affect numbering.

Setting	Description
Embedded objects	Sub-assemblies affect the numbering of cast units.
Surface treatment	Surface treatments affect the numbering of assemblies.
Welds	Welds affect the numbering of assemblies.
Tolerance	Objects get the same number if their dimensions differ less than the value entered in this box.
Assembly position sort order	See Number assemblies, cast units, and rebar assemblies (page 769).
Family numbering	See Assign family numbers (page 761).
Reinforcement sequence numbering	See Reinforcement sequence numbering (page 762).

See also

Adjust the numbering settings (page 760)

Numbering settings during a project (page 798)

Numbering examples (page 793)

Weld numbering settings

Use the **Weld Numbering** dialog to view and modify the weld numbering settings. The weld number is displayed in drawings and weld reports.

Option	Description
Start number	The number from which the numbering starts. Tekla Structures automatically suggests the following free number as the start number.
Apply for	Defines which objects are affected by the change. **All welds** changes the number of all welds in the model. **Selected welds** changes the number of the selected welds without affecting others.
Renumber also welds that have a number	Tekla Structures replaces existing weld numbers.
Re-use numbers of deleted welds	If some welds have been removed, Tekla Structures uses their numbers when numbering other welds.

See also

Number welds (page 770)

Control number settings

Use the **Create control numbers (S9)** dialog to view and modify the control number settings.

Option	Description
Numbering	Defines which parts get control numbers. **All** creates consecutive numbers for all parts. **By numbering series** creates control numbers for parts in a specific numbering series.
Assembly/Cast unit numbering series	Defines the prefix and start number of the numbering series for which to create control numbers. Needed only with the **By numbering series** option.
Start number of control numbers	The number from which the numbering starts.
Step value	Defines the interval between two control numbers.
Renumber	Defines how to treat parts that already have control numbers. **Yes** replaces the existing control numbers. **No** keeps the existing control numbers.
First direction	Defines in what order to assign control numbers.
Second direction	
Third direction	
Write UDA to	Defines where to save the control numbers. **Assembly** saves the control numbers to the user-defined attributes of assemblies or cast units.

Option	Description
	Main part saves the control numbers to the user-defined attributes of assembly or cast unit main parts. The control number appears on the **Parameters** tab.

See also

Control numbers (page 783)

9.2 Reinforcement settings

This section provides more information about the various reinforcement settings you can modify in Tekla Structures.

Click the links below to find out more:

Reinforcing bar and bar group properties (page 1061)

Reinforcement mesh properties (page 1064)

Rebar set properties (page 1068)

Reinforcement strand properties (page 1088)

Reinforcing bar and bar group properties

Use the **Single rebar** and **Rebar group** properties to view and modify the properties of reinforcing bars and reinforcing bar groups. The file name extension of the property file is:

- `.rbr` for bars (page 520)
- `.rbg` for groups (page 522)
- `.rci` for circular groups (page 531)
- `.rcu` for curved groups (page 529)

General, Hooks, Cover thickness, IFC export, More

The following properties are available for single reinforcing bars and reinforcing bar groups:

Option	Description	
Name	User-definable name of the bar. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.	
Grade	Steel grade of the bar.	Size-grade-radius combinations are predefined in the rebar catalog. Click the **...** button to open the **Select rebar** dialog. The dialog shows the available bar sizes for the chosen grade. You can also select whether the bar is a main bar, or a stirrup or tie.
Size	Diameter of the bar. Depending on the environment, the nominal diameter of the bar, or a mark that defines the diameter.	
Bending radius	Internal radius of the bends in the bar. You can enter a separate value for each bar bend. Separate the values with spaces. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine.	
Class	Use to group reinforcement. For example, you can display bars of different classes in different colors.	
Numbering	Mark series of the reinforcing bar.	
Hook type	Shape of the hook.	The rebar catalog (`rebar_database.inp`) contains the predefined minimum bending radius and minimum hook length for all standard hooks. See Add hooks to reinforcing bars (page 585).
Angle	Angle of a custom hook.	
Radius	Internal bending radius of a standard hook or custom hook.	
Length	Length of the straight part of a standard or custom hook.	
Cover thickness on plane	Distances from the part surfaces to the bar on the same plane as the bar.	See Define the reinforcement cover thickness (page 588).

Option	Description	
Cover thickness from plane	Distance from the part surface to the bar, or to the bar end, perpendicular to the bar plane.	
Start	Concrete cover thickness or leg length at the first end of the bar.	
End	Concrete cover thickness or leg length at the second end of the bar.	
IFC entity	For IFC export, select the IFC entity type and subtype of the bar or bar group. The available subtypes depend on the selected IFC entity. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.	
Subtype (IFC4)		
User-defined type (IFC4)		
UDAs	You can create user-defined attributes to add information about reinforcement. Attributes can consist of numbers, text, or lists. You can use the values of user-defined attributes in reports and drawings. You can also change the name of the fields, and add new ones, by editing the `objects.inp` file.	

Special, Distribution

The following properties are available for:

- reinforcing bar groups, including tapered and spiral (page 533) groups
- curved reinforcing bar groups
- circular reinforcing bar groups

Option	Description	
Rebar group type	What is the type of the group.	See Create a tapered or spiral reinforcing bar group (page 533).
Number of cross sections		
Creation method	How the bars are spaced.	See Distribute bars in a reinforcing bar group (page 576).
Number of reinforcing bars		
Number of rounds (for spiral bars)		

Option	Description	
Target spacing value		
Exact spacing value		
Exact spacing values		
Exclude	Which bars are omitted from the group. (Not available for spiral bars.)	See Delete bars from a reinforcing bar group (page 578).

Reinforcement mesh properties

Use the **Rebar mesh** properties to view and modify the properties of reinforcement meshes. The file name extension of a reinforcement mesh property file is `.rbm`.

Option	Description
Name	User-definable name of the mesh. Tekla Structures uses mesh names in reports and drawing lists.
Class	Use to group reinforcement. For example, you can display bars of different classes in different colors.
Numbering	Mark series of the mesh.
Mesh type	Shape of the mesh. Select **Polygon**, **Rectangle**, or **Bent**.
Cross bar location	Define whether the crossing bars are located above or below the longitudinal bars.
Cut by father part cuts	Define whether the polygon or part cuts in the part also cut the mesh.
Mesh	Identifier of the mesh. For standard meshes, the mesh name used in the mesh catalog. To create a **Standard** mesh, click the **...** button and select a mesh from the mesh catalog.

Option	Description
	The properties of standard meshes are defined in the `mesh_database.inp` file. To create a custom mesh (page 540), select the **Custom mesh** option and define the properties (page 1065).
Grade	Steel grade of the bars in the mesh. Available for custom meshes.
Bending radius	Internal radius of the bends in the bar. Available for bent meshes.
Hooks	See Add hooks to reinforcing bars (page 585). Available for bent meshes.
Cover thickness on plane	Distance from the part surface to the main bars on the same plane as the bars.
Cover thickness from plane	Distance from the part surface to the bar, or bar end, perpendicular to the bar plane.
Start	Thickness of concrete cover or leg length from the mesh starting point. Available for rectangular and bent meshes.
End	Thickness of concrete cover or leg length at the end point of the bar. Available for bent meshes.
IFC entity	For IFC export, select the IFC entity type and subtype of the mesh. The available subtypes depend on the selected IFC entity. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
Subtype (IFC4)	
User-defined type (IFC4)	
UDAs	You can create user-defined attributes to add information about reinforcement. Attributes can consist of numbers, text, or lists. You can use the values of user-defined attributes in reports and drawings. You can also change the name of the fields, and add new ones, by editing the `objects.inp` file.

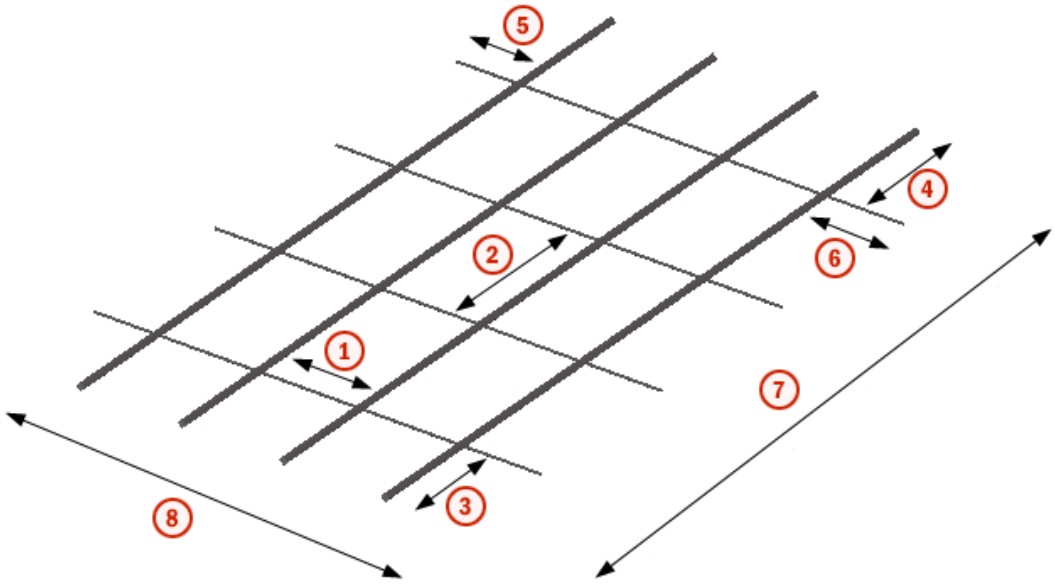
See also

Create a reinforcement mesh (page 536)

Custom reinforcement mesh properties

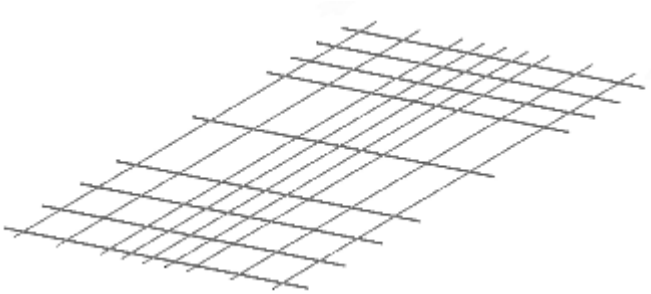
Use the **Rebar mesh** properties to view and modify the properties of customized reinforcement meshes. The file name extension of a reinforcement mesh properties file is `.rbm`.

You can define the following properties for the customized reinforcement meshes (page 540):



1. Longitudinal distance
2. Cross distance
3. Longitudinal left overhang
4. Longitudinal right overhang
5. Cross left overhang
6. Cross right overhang
7. Length
8. Width

Option	Description
Spacing method	Define how the mesh bars are distributed. • **Same distance for all**: Use to create meshes with evenly-spaced bars. Tekla Structures distributes as many bars as possible for the length of **Length** or **Width**, using the **Distances** and **Left overhang** values. The **Right overhang** is calculated automatically, and it cannot be zero.

Option	Description
	• **Multiple varying distances**: Use to create meshes with unevenly-spaced bars. Tekla Structures calculates the **Width** and **Length** based on the **Distances**, the **Left overhang** and the **Right overhang** values. If you do not change any of the values, the spacing method changes back to **Same distance for all**.
Distances	Spacing values of longitudinal or crossing bars. If you select the **Multiple varying distances** spacing method, enter all spacing values, separated by spaces. You can use multiplication to repeat spacing values. For example: `2*150 200 3*400 200 2*150` You can create meshes with unevenly-spaced bars. You can also define a different bar size or multiple different bar sizes for the longitudinal bars and the crossing bars. Multiple bar sizes enable pattern creation. For example, if you enter bar diameters 20 2*6 in the longitudinal direction, Tekla Structures creates a pattern with one size 20 bar and two size 6 bars. This pattern can be repeated in the mesh along the longitudinal direction. 
Left overhang **Right overhang**	Extensions of crossing bars over the outermost longitudinal bars. Extensions of longitudinal bars over the outermost crossing bars.
Diameters	Diameter or size of the longitudinal or crossing bars. You can define multiple diameters for the bars in both directions. Enter all the diameter values, separated by spaces. You can use multiplication to repeat diameter values. For example, `12 2*6` in longitudinal direction and `6 20 2*12` in crossing direction.
Width	Length of crossing bars.
Length	Length of longitudinal bars.

Option	Description
Grade	Steel grade of the bars in the mesh.

NOTE If a reinforcement mesh contains bars that are not defined in the rebar catalog (`rebar_database.inp`), the bars are shown as lines in the model and are given zero weight, to make it easier to spot the missing rebar definition.

See also

Create a reinforcement mesh (page 536)

Reinforcement mesh properties (page 1064)

Rebar set properties

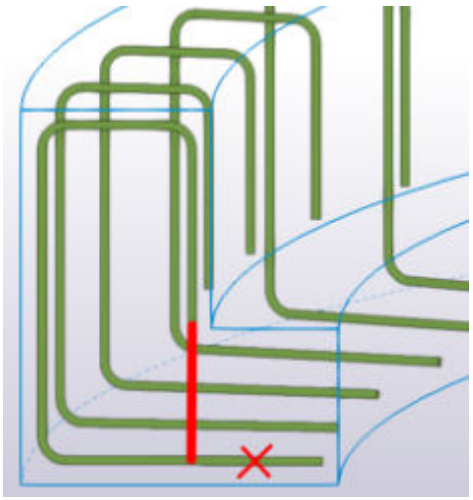
Use the property pane or the contextual toolbar to view and modify the properties of the rebar sets. The file name extension of the property file is `.rst`.

General

Option	Description	
Name	User-definable name of the bars. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.	
Grade	Steel grade of the bars.	Grade-size-radius combinations are predefined in the rebar catalog. Click the **...** button in the property pane to open the **Select rebar** dialog box. The dialog box shows the available bar sizes for the chosen grade. You can also select whether the bars are main bars, or stirrups or ties.
Size	Diameter of the bars. Depending on the environment, the nominal diameter of the bars, or a mark that defines the diameter.	
Bending radius	Internal radius of the bends in the bars. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the	

Option	Description	
	size of the mandrels on the bar-bending machine. Automatic values are shown in square brackets, for example [120.00].	
Class	Used to group reinforcement. For example, you can display bars of different classes in different colors.	
Numbering	Numbering series of the bars.	

Special

Option	Description
Close shape	Select whether disconnected bar legs should be connected, where possible, to form a continuous closed or open bar shape. 

Primary guideline

Option	Description
Follow edges	Select whether the rebar set guideline attempts to follow the leg face edges that are located between the guideline end points.
Chamfer type	To adjust how rebar set bars follow a sharp or curved part corner, define the chamfer type of the guideline at the part corner: select **None** for a sharp corner or **Arc point** for a curved corner.

Distribution

Option	Description
Start offset	The offsets at the start and end of the guideline. By default, Tekla Structures calculates the offset values according to the concrete cover settings and bar diameter. Automatic values are shown in square brackets, for example [32.00]. Note that the automatic offset values may change if the outermost rebar set bars are split by splitters and the split bars would end up in the concrete cover area.
End offset	
Creation method	How the bars are spaced.
Number of reinforcing bars	
Target spacing value	
Exact spacing value	
Exact spacing values	
Exclude	Which bars are omitted from the rebar set.

Bar placement

Option	Description
Automatic layering	Select whether or not the bars in the rebar set are placed and layered automatically. The options are: • **On**: This is the default option. The concrete cover values are automatically inherited from the parent concrete part, and the bars are placed on different layers according to their creation order. With this option, the bars automatically react to changes in the surrounding objects. For example, a layer of bars may move when another layer of bars is created, moved, or deleted at the same concrete surface. • **Off**: This option is selected when the rebar set is frozen (page 571). The bars do not automatically react to changes in the surrounding objects, but you can manually define the bar placement using **Offsets**, for example.

Option	Description
Layer number	Available when **Automatic layering** is on (the rebar set is **not** frozen). Defines the order of the bar layers. The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers. After you click **Modify** in the **Rebar set** property pane, **Layer number** shows **Auto**, but the entered value still has an effect. If you do not define the layer numbers, Tekla Structures arranges the bar layers according to their creation order. The bar layer that is created first is closest to the concrete surface. You can also adjust the layer order (page 546) by using these options on the contextual toolbar:  Note that if you copy properties from one rebar set to another, the layer number is not copied. Note that even if **Automatic layering** is off (the rebar set is frozen), the layer number may change, for example, if a neighboring rebar set is deleted. Locking a frozen rebar set prevents the layer number from changing.
Offsets	Available when **Automatic layering** is off (the rebar set is frozen (page 571)). The offset of the bar legs from the leg faces or leg surfaces on the bar plane (**At depth**), and the offsets at the start and end of the bars.
Adaptivity	Select how the rebar set adapts (page 438) to changes in the parent concrete part it is attached to. The options are: • **Default**: The model's **Default adaptivity** option selected in **File** --> **Settings** --> **Options** --> **General** is used for the rebar set. • **Off** • **Relative** • **Fixed** If you freeze (page 571) the rebar set, **Adaptivity** is automatically set to **Off**, but you can manually select another option.

Option	Description
	If you unfreeze a frozen rebar set, **Adaptivity** is set to **Default**.

Advanced: Rounding

Option	Description
Straight bars	Define whether the lengths of straight bars, first and last legs, and intermediate legs are rounded, and whether the bar lengths are rounded up, down, or to the nearest suitable number according to the rounding accuracy.
First and last legs	
Intermediate legs	
Rounding up at splitters	At splitter locations, define how much the bar lengths can be rounded up.

Advanced: Step tapering

Option	Description
Type	Define whether the bars are step tapered, and how the tapering steps are created. The options are **None**, **Distance**, and **Number of bars**.
	If you select the **Number of bars** option, enter the number of bars in one tapering step.
Straight bars	If you select the **Distance** option, enter the tapering step values for straight bars, first and last legs, and intermediate legs.
First and last legs	
Intermediate legs	

Advanced: Minimum lengths to be created

Option	Description
Minimum bar length	Use to prevent Tekla Structures from creating reinforcing bars that are too short. This setting is primarily for straight bars. Enter the minimum bar length as **Distance** or as **Coefficient of bar diameter**.
Minimum straight start/end leg length	Use for bent reinforcing bars. Enter the minimum leg length as **Distance** or as **Coefficient of bar diameter**.

IFC export

For IFC export, select an option in **IFC entity** and in **Subtype (IFC4)**. The available subtypes depend on the selected IFC entity.

You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.

More

Click the **User-defined attributes** button to open the user-defined attributes of the rebar sets. The file name extension of the user-defined attribute file is `.rst.more`.

You can use user-defined attributes to define or override settings, such as bar layer prefix and number, or grouping settings.

The **Smoothing factor** attribute on the **Rebar set** tab is useful when leg surfaces in complex geometry parts do not produce the desired bar geometry. There might be too short consecutive, zigzagging bar segments, or in some cases bars might not follow the concrete surface closely enough. **Smoothing factor** is a percentage of the overall leg length, so a value between 0 and 100. If a leg segment is shorter than this percentage of the overall leg length, it is eliminated. The default value is 0.1. To eliminate the leg segments causing a zigzag effect, increase the value. To make the bars follow the concrete surface more closely, decrease the value.

See also

Create a rebar set (page 494)

Modify a rebar set (page 545)

Secondary guideline properties (page 1073)

Leg face properties (page 1075)

Leg surface properties (page 1076)

Property modifier properties (page 1078)

End detail modifier properties (page 1081)

Splitter properties (page 1085)

Secondary guideline properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set secondary guidelines.

General

To make a secondary guideline follow the leg face edges that are located between the guideline end points, select **Yes** from the **Follow edges** list in the property pane. Alternatively, select the secondary guideline and click  on the contextual toolbar.

To adjust how rebar set bars follow a sharp or curved part corner, define the **Chamfer type** of the guideline at the part corner: in the property pane, select **None** for a sharp corner or **Arc point** for a curved corner. Alternatively, you can adjust the chamfer type of the selected guideline using the contextual

toolbar: click the chamfer type button and then select **None** or **Arc point** to switch between the chamfer types.

Spacing properties

If you want a secondary guideline to have offsets similar to the primary guideline, select **Yes** from the **Inherit offsets from primary** list in the property pane. Alternatively, select the secondary guideline and click on the contextual toolbar.

If you want a secondary guideline to have spacing properties similar to the primary guideline, select **Yes** from the **Inherit spacings from primary** list in the property pane. Alternatively, select the secondary guideline and click on the contextual toolbar. The spacing values and ranges are automatically scaled by the ratio of the secondary guideline length to the primary guideline length.

If you want to define the secondary guideline spacing properties independently from the primary guideline, select **No** from the **Inherit offsets from primary** and **Inherit spacings from primary** lists, and then modify the following spacing properties as needed:

Option	Description
Start offset	The offsets at the start and end of the guideline. By default, Tekla Structures calculates the offset values according to the concrete cover settings and bar diameter. Automatic values are shown in square brackets, for example [32.00]. Note that the automatic offset values may change if the outermost rebar set bars are split by splitters and the split bars would end up in the concrete cover area.
End offset	
Creation method	How the bars are spaced.
Number of reinforcing bars	
Target spacing value	
Exact spacing value	
Exact spacing values	
Exclude	Which bars are omitted from the rebar set.

See also

Modify a rebar set locally using modifiers (page 557)

Distribute bars in a rebar set (page 565)

Rebar set properties (page 1068)

Leg face properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set leg faces.

Bar leg placement

Option	Description
Automatic layering	Select whether or not the rebar set bars at the leg face are placed and layered automatically. The options are: • **On**: This is the default option. The concrete cover values are automatically inherited from the parent concrete part, and the bars are placed on different layers according to their creation order. With this option, the bars automatically react to changes in the surrounding objects. For example, a layer of bars may move when another layer of bars is created, moved, or deleted at the same concrete surface. • **Off**: This option is selected when the leg face or rebar set is frozen (page 571). The bars do not automatically react to changes in the surrounding objects, but you can manually define the bar leg placement using **Offset**, for example.
Layer number	Available when **Automatic layering** is on. Defines the order of the bar layers. The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative values. By default, Tekla Structures arranges the bar layers according to their creation order. The bar layer that is created first is closest to the concrete surface. You can also adjust the layer order by using these options on the contextual toolbar:  Note that if you copy properties from one leg face to another, the layer number is not copied.
Flip bar side	Shows if the bars are flipped over to the other side of the leg face (**Yes**) or not (**No**). The default value is **No**.

Option	Description
Offset	The options are: • **From concrete cover**: Distance from the concrete cover to the bars when **Automatic layering** is on. A negative value moves the bars into the concrete cover area, or even outside the concrete. • **From previous layer**: Distance between two bar layers when **Automatic layering** is on. • **From leg face to bar edge**: Used when freezing a rebar set that uses the concrete cover value of the parent concrete part. The leg face is located on the face of the part. • **From leg face to bar center**: Used when freezing a rebar set that does not use the concrete cover value of the parent concrete part. The leg face is not on the face of the part, but is either entirely inside or outside the part.

See also

Modify a rebar set using leg faces and leg surfaces (page 549)

Rebar set properties (page 1068)

Leg surface properties (page 1076)

Leg surface properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set leg surfaces. The file name extension of the property file is `.rst_ls`.

Bar leg placement

Option	Description
Automatic layering	Select whether or not the rebar set bars at the leg surface are placed and layered automatically. The options are: • **On**: This is the default option. The concrete cover values are automatically inherited from the parent concrete part, and the bars are placed on different layers according to their creation order. With this option, the bars automatically react to changes in the surrounding objects. For example, a layer of bars may move when another layer of bars

Option	Description
	is created, moved, or deleted at the same concrete surface. • **Off**: This option is selected when the leg surface or rebar set is frozen (page 571). The bars do not automatically react to changes in the surrounding objects, but you can manually define the bar leg placement using **Offset**, for example.
Layer number	Available when **Automatic layering** is on. Defines the order of the bar layers. The smaller the layer number, the closer to the concrete surface the bar layer is. You can use both positive and negative values. By default, Tekla Structures arranges the bar layers according to their creation order. The bar layer that is created first is closest to the concrete surface. You can also adjust the layer order by using these options on the contextual toolbar:  Note that if you copy properties from one leg surface to another, the layer number is not copied.
Offset	The options are: • **From concrete cover**: Distance from the concrete cover to the bars when **Automatic layering** is on. A negative value moves the bars into the concrete cover area, or even outside the concrete. • **From previous layer**: Distance between two bar layers when **Automatic layering** is on. • **From leg face to bar edge**: Used when freezing a rebar set that uses the concrete cover value of the parent concrete part. The leg surface is located on the face of the part. • **From leg face to bar center**: Used when freezing a rebar set that does not use the concrete cover value of the parent concrete part.
Create holes	Select whether the bars at the leg surface are cut by the openings in the concrete.

See also

Modify a rebar set using leg faces and leg surfaces (page 549)

Rebar set properties (page 1068)

Leg face properties (page 1075)

Property modifier properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set property modifiers. The file name extension of the property file is `.rst_pm`.

General

Option	Description	
Name	User-definable name of the bars. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.	
Grade	Steel grade of the bars.	Grade-size-radius combinations are predefined in the rebar catalog. Click the **...** button in the property pane to open the **Select rebar** dialog. The dialog shows the available bar sizes for the chosen grade. You can also select whether the bars are main bars, or stirrups or ties.
Size	Diameter of the bars. Depending on the environment, the nominal diameter of the bars, or a mark that defines the diameter.	
Bending radius	Internal radius of the bends in the bars. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine. Automatic values are shown in square brackets, for example [120.00].	
Class	Used to group reinforcement. For example, you can display bars of different classes in different colors.	

Option	Description
Numbering	Numbering series of the bars.

Special

Option	Description
Follow edges	Select whether the property modifier attempts to follow the leg face edges that are located between the modifier end points.
Modify distribution	Select whether the property modifier affects the bar spacings. If you select **Yes**, the **Distribution** properties become available, and **Bars affected** and **First affected bar** are unavailable.
Bars affected	Select how many bars can be modified in the same location: • **1/1** = all bars are modified in the same cross section. • **1/2** = every second bar is modified in the same cross section. • **1/3** = every third bar is modified in the same cross section. • **1/4** = every fourth bar is modified in the same cross section.
First affected bar	Define which is the first bar to be modified, starting from the first end of the modifier. Enter a positive number, or use the arrow buttons to change the number.
Affect whole bar plane	Select whether the property modifier affects all bars of a rebar set that are in the same plane even though some of the bars are not touched by the property modifier (or its projection). For example, to modify bars on opposite sides of an opening using the same modifier, select **Yes**.
Grouping	Select whether and how the bars that are affected by the property modifier are grouped. The options are: • **Automatic**: Bars are grouped according to automatic rules.

Option	Description
	• **Manual**: Bars are grouped regardless of their geometry or arrangement. Note that this option does not group bars from different sides of splitters. • **No grouping**: Bars are not grouped, but they are individual bars. Use this option for overriding automatic and manual grouping.

Distribution

These properties are available when **Modify distribution** is set to **Yes**.

Option	Description
Start offset	The offsets at the start and end of the modifier. Automatic values are shown in square brackets, for example [32.00]. Note that the automatic offset values may change if the outermost rebar set bars are split by splitters and the split bars would end up in the concrete cover area.
End offset	
Creation method	How the bars are spaced. For more information, see Spacing options.
Number of reinforcing bars	
Target spacing value	
Exact spacing value	
Exact spacing values	
Exclude	Which bars are omitted from the rebar set. See also Examples.

Advanced: Rounding

Option	Description
Straight bars	Define whether the lengths of straight bars, first and last legs, and intermediate legs are rounded, and whether the bar lengths are rounded up, down, or to the nearest suitable number according to the rounding accuracy.
First and last legs	
Intermediate legs	
Rounding up at splitters	At splitter locations, define how much the bar lengths can be rounded up.

Advanced: Step tapering

Option	Description
Type	Define whether the bars are step tapered, and how the tapering steps are created. The options are **None**, **Distance**, and **Number of bars**.
	If you select the **Number of bars** option, enter the number of bars in one tapering step.
Straight bars	If you select the **Distance** option, enter the tapering step values for straight bars, first and last legs, and intermediate legs.
First and last legs	
Intermediate legs	

Advanced: Minimum lengths to be created

Option	Description
Minimum bar length	Use to prevent Tekla Structures from creating reinforcing bars that are too short. This setting is primarily for straight bars. Enter the minimum bar length as **Distance** or as **Coefficient of bar diameter**.
Minimum straight start/end leg length	Use for bent reinforcing bars. Enter the minimum leg length as **Distance** or as **Coefficient of bar diameter**.

More

Click the **User-defined attributes** button to open the user-defined attributes of the rebar set property modifiers. The file name extension of the user-defined attribute file is `.rst_pm.more`.

You can use user-defined attributes to define or override settings, such as bar layer prefix and number, or grouping settings.

See also

Modify a rebar set locally using modifiers (page 557)

Rebar set properties (page 1068)

End detail modifier properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set end detail modifiers. The file name extension of the property file is `.rst_edm`.

Special

Option	Description
Follow edges	Select whether the end detail modifier attempts to follow the leg face edges that are located between the modifier end points.
Bars affected	Select how many bars can be modified in the same location: • **1/1** = all bars are modified in the same cross section. • **1/2** = every second bar is modified in the same cross section. • **1/3** = every third bar is modified in the same cross section. • **1/4** = every fourth bar is modified in the same cross section.
First affected bar	Define which is the first bar to be modified, starting from the first end of the modifier. Enter a positive number, or use the arrow buttons to change the number.
End type	Select **Hook** or **Cranking**. If you select the empty option, no hooks or cranks are created, but you can define length adjustments, end preparations, and user-defined attributes.

Hook

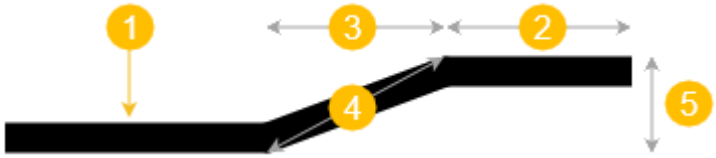
These properties are available when **End type** is **Hook**.

Option	Description	
Hook type	Shape of the hook.	The rebar catalog (`rebar_database.inp`) contains the predefined minimum bending radius and minimum hook length for all standard hooks. See Add hooks to reinforcing bars (page 585).
Angle	Angle of a custom hook.	
Radius	Internal bending radius of a standard hook or custom hook.	
Length	Length of the straight part of a standard or custom hook.	

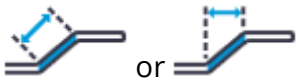
Option	Description	
Hook rotation	Rotation angle of a hook out of the bar plane. Use to create 3D bars.	For example: 

Cranking

These properties are available when **End type** is **Cranking**.



(1) = Location of the end detail modifier

Option	Description
Cranking type	Select **No cranking**, **Standard cranking**, or **Custom cranking**. Use the **No cranking** option to override other end detail modifiers that create cranks. With standard cranking, the crank dimensions are read from the rebar catalog (`rebar_database.inp`).
Crank straight length	With custom cranking, enter the length of the straight segment of the crank. This is **(2)** in the image above.
Cranked length	With custom cranking, select whether the length of the cranked segment is defined in the diagonal **(4)** or horizontal **(3)** direction:  or Then select and enter the needed distance or a multiplier of the bar diameter.
Cranked offset	With custom cranking, enter the offset distance of the straight segment of the crank. This is **(5)** in the image above. The default value is 2 * actual bar diameter.
Crank rotation	Defines to which angle the crank is rotated.

Length adjustment

Option	Description
Adjustment type	Select whether and how the bar length is adjusted (extended or shortened). • **No adjustment**: Bar length is not adjusted. • **End offset**: Bar length is adjusted according to the specified end offset. Use this option to keep the leg faces on the concrete faces and adaptive to the concrete faces, but still to extend or shorten the bar ends. • **Leg length**: Bar length is adjusted according to the specified leg length.
Length	Depending on the adjustment type, the length of the end offset or leg. With end offset, enter a positive value to extend the bars, or a negative value to shorten the bars. With leg length, enter a positive value to set the leg length.
Align bar ends	When the lengths of straight bars are rounded and/or step tapered, select whether the bar ends nearest to the end detail modifier are aligned or not. If you select **No**, the rounding and step tapering happens at the tapered edge of the rebar set, and if both edges are tapered, at the edge that has a larger angle.

End preparations

Option	Description
Method	Select the end method of the bars. The options are: • **Coupler** • **Coupler female** • **Coupler male** • **Threaded** • **Anchor**
Type	Select the end method type. The options are: • **Standard**

Option	Description
	• **Position** • **Bridging** • **Transition** • **Bolt** • **Weldable**
Product	Product name of the end detail. Can be shown in reports.
Code	Product code of the end detail. Can be shown in reports.
Thread type	Enter the type of the threading.
Threaded length	Length of the threading from the bar end.
Extra fabrication length	Additional length needed with some threading methods. Can be shown in reports, but does not affect the total length of the bar.

Use the advanced option `XS_REBAR_END_SPECIFIC_UDA_METHOD` to control whether the **End preparations** properties are created as start leg and end leg user-defined attributes with a `_START` or `_END` suffix, or as short leg and long leg user-defined attributes with an `_S` or `_L` suffix.

More

Click the **User-defined attributes** button to open the user-defined attributes of the rebar set end detail modifiers. The file name extension of the user-defined attribute file is `.rst_edm.more`.

See also

Modify a rebar set locally using modifiers (page 557)

Rebar set properties (page 1068)

Splitter properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set splitters. The file name extension of the property file is `.rst_sm`.

Some of the following settings depend on the splitter direction. An arrowhead symbol  close to the midpoint of each splitter indicates the direction and the left and right sides of the splitter. The arrow points from the start towards the end of the splitter.

Special

Option	Description
Follow edges	Select whether the splitter attempts to follow the leg face edges that are located between the splitter end points.
Bars affected	Select how many bars can be modified in the same location: • **1/1** = all bars are modified in the same cross section. • **1/2** = every second bar is modified in the same cross section. • **1/3** = every third bar is modified in the same cross section. • **1/4** = every fourth bar is modified in the same cross section.
First affected bar	Define which is the first bar to be modified, starting from the first end of the modifier. Enter a positive number, or use the arrow buttons to change the number.
Split type	Select **Lapping** or **Cranking**.
Split offset	Defines how far from the splitter the split is created. Positive values move the split to the right side and negative values to the left side of the splitter.

Lapping

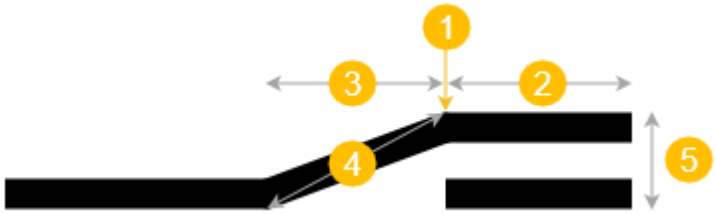
These properties are available when **Split type** is **Lapping**.

Option	Description
Lapping type	Select **Standard lapping** or **Custom lapping**.
Lap length	With custom lapping, enter the length of the lap splice. With standard lapping, the lap length is read from the rebar catalog (`rebar_database.inp`). **Note**: If the splitter is placed so that the lap length extends around a corner, Tekla Structures might remove part of the lap length if the last leg is shorter than the **Minimum straight start/end leg length** value of the rebar set or the model (defined in the **Options** dialog).

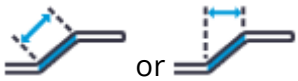
Option	Description
Lap side	Select the side of the lap splice from the splitter: • **Lap left** • **Lap right** • **Lap middle**
Lap placement	Select whether the lapping bars are parallel to each other, or on top of each other. **Note**: The **On top** option does not work with rebar sets that have leg surfaces.

Cranking

These properties are available when **Split type** is **Cranking**.



(1) = Location of the splitter

Option	Description
Cranking type	Select **Standard cranking** or **Custom cranking**. With standard cranking, the crank dimensions are read from the rebar catalog (`rebar_database.inp`).
Crank straight length	With custom cranking, enter the length of the straight segment of the crank. This is **(2)** in the image above.
Cranked length	With custom cranking, select whether the length of the cranked segment is defined in the diagonal **(4)** or horizontal **(3)** direction:  Then select and enter the needed distance or a multiplier of the bar diameter.
Cranked offset	With custom cranking, enter the offset distance of the straight segment of the crank. This is **(5)** in the image above. The default value is 2 * actual bar diameter.

Option	Description
Crank side	Select to which side of the splitter the crank is created, **Left** or **Right**.
Crank rotation	Defines to which angle the crank is rotated.

Staggering

Option	Description
Stagger type	Select whether and in which direction the splices are staggered. The options are: • **No stagger** • **Stagger left** • **Stagger right** • **Stagger middle**
Stagger offset	The offset of the adjacent bars if they are staggered.

See also

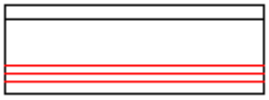
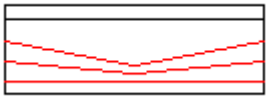
Modify a rebar set locally using modifiers (page 557)

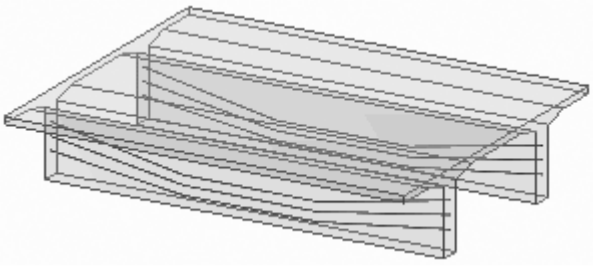
Rebar set properties (page 1068)

Reinforcement strand properties

Use the **Strand pattern** properties to view and modify the properties of strands. The file name extension of the property file is `.rbs`.

Option	Description
General	
Name	User-definable name of the strand. Tekla Structures uses strand names in reports and drawing lists, and to identify strands of the same type.
Grade	Steel grade of the strand.
Size	Diameter of the strand. Depending on the environment, the nominal diameter of the strand, or a mark that defines the diameter.

Option	Description
Bending radius	Internal radius of the bends in the strand. You can enter a separate value for each bend. Separate the values with spaces.
Class	Use to group reinforcement. For example, you can display strands of different classes in different colors.
Numbering	Mark series of the strand.
Special	
Pull per strand	Pre-stress load per strand (kN).
Number of cross sections	Number of cross sections of the strand pattern. For example: • Number of cross sections along strand profile = 1:  • Number of cross sections along strand profile = 2:  • Number of cross sections along strand profile = 3:  • Number of cross sections along strand profile = 4:  In this double-tee beam, the number of cross sections is 4:

Option	Description
	
Debonding	
Debonded strands	Enter the strand number. The strand number is the selection order number of the strand.
From start **Middle to start** **Middle to end** **From end**	Enter the length of the debonding. If you select the **Symmetry** checkbox, values from **From start** and **Middle to start** are copied to **From end** and **Middle to end**.
Symmetry	Define whether the end and start lengths are symmetrical.
IFC export	
IFC entity **Subtype (IFC4)** **User-defined type (IFC4)**	For IFC export, select the IFC entity type and subtype of the strands. The available subtypes depend on the selected IFC entity. You can select the IFC4 subtype among the predefined options, or you can select **USERDEFINED** and then enter any text in **User-defined type (IFC4)**.
More	
UDAs	You can create user-defined attributes to add information about reinforcement. Attributes can consist of numbers, text, or lists. You can use the values of user-defined attributes in reports and drawings. To set values for user-defined attributes, click the **User-defined attributes** button. You can also change the name of the fields, and add new ones, by editing the `objects.inp` file.

See also

Create a reinforcement strand pattern (page 541)

Debond reinforcement strands (page 542)

10 Disclaimer

© 2026 Trimble Inc. and affiliates. All rights reserved.

Use of the Software and of this Software Manual are governed by a License Agreement which determines whether you are an authorized user of the Software and the Software Manual. The warranties and disclaimers set forth in the License Agreement apply to the Software and the Software Manual. Neither the Trimble entity granting the license nor any of its affiliates assume responsibility that the text is free of technical inaccuracies or typographical errors. The right to make changes and additions to this manual is reserved.

Trimble and certain product names are registered trademarks of Trimble Inc. in the United States, the European Union and other countries and may have similar statutory protections. Trademarks of third parties are not mentioned in this Manual to suggest an affiliation with or endorsement by their owners.

Elements of the software described in this Manual may be the subject of pending patent applications in the European Union and/or other countries.

Portions of this software:

Portions of this software make use of Open CASCADE Technology software. Open Cascade Express Mesh Copyright © 2019 OPEN CASCADE S.A.S. All rights reserved.

FLY SDK - CAD SDK © 2012 VisualIntegrity™. All rights reserved.

This application incorporates Open Design Alliance software pursuant to a license agreement with Open Design Alliance. Open Design Alliance Copyright © 2002–2020 by Open Design Alliance. All rights reserved.

CADhatch.com © 2017. All rights reserved.

RapidXml C++ library © All rights reserved.

FlexNet Publisher © 2016 Flexera Software LLC. All rights reserved. This product contains proprietary and confidential technology, information and creative works owned by Flexera Software LLC and its licensors, if any. Any use, copying, publication, distribution, display, modification, or transmission of such technology in whole or in part in any form or by any means without the prior express written permission of Flexera Software LLC is strictly prohibited. Except where expressly provided by Flexera Software LLC

in writing, possession of this technology shall not be construed to confer any license or rights under any Flexera Software LLC intellectual property rights, whether by estoppel, implication, or otherwise.

To see the third party open source software licenses, go to Tekla Structures, click **File menu** --> **Help** --> **About Tekla Structures** --> **3rd party licenses** and then click the option.

Index

*

* character..... 202

.

.inp files
  in custom components..... 995,999

3

3D
  in snapping.....89
  model views..... 48

?

? character..... 202

A

adaptivity
  default settings.....438
  of individual model objects.....438
  of reinforcement..... 591
adding
  grid lines.....27
aligning
  objects..... 109
Analysis tab.....839
anchoring hooks..... 585
angles..... 728
Applications & components
  exporting custom components..... 987
  importing custom components.....987
applications in Applications and Components catalog..... 801
arcs
  measuring..... 728
arithmetic operators.....919
Array of objects (29).....154
ASCII file as a reference function..... 919
assemblies
  adding parts.....450
  assembly hierarchy.....440
  basic assemblies..... 440
  batch editing.....461
  changing the main assembly.....450
  changing the main part.....450
  comparing.....743
  creating..... 441,447
  examples.....441
  exploding.....453
  hierarchy..... 441
  higher-level assemblies.....440
  highlighting..... 450
  joining assemblies.....450
  nested assemblies..... 440,450
  nested assemblies..... 133
  numbering..... 755,769
  removing parts or sub-assemblies..... 453
  selecting..... 133
  showing invisible objects..... 682
  sub-assemblies.....447
  using bolts to create assemblies..441,447
  using welds to create assemblies 441,447
asterisk..... 202
attaching
  parts.....359
  rebar set bars to parts.....595
  reinforcement to part.....593
auto bolt
  creating bolts.....363
auto splitter..... 569
AutoConnection..... 820
  limitations..... 821
  restrictions..... 821
  rules.....821,831
  settings..... 821
  using..... 824

AutoDefaults..... 820,826
combining properties..... 834
connection check..... 836
iterating properties..... 834
rules..... 831
using..... 831
using reaction forces..... 838
using UDLs..... 838
auto
in snapping.....89
automatic splicing tool..... 582
AutomaticSplicingTool..... 582

B

background color
model views..... 73
base points
drawings..... 60
exporting..... 60
IFC export..... 60
reference models..... 60
basepoints..... 60
basic assemblies.....440,441
batch editing
assemblies..... 461
cast units..... 461
beams
cambering..... 353
concrete beams.....284
concrete polybeams..... 287
curved beams..... 229
orthogonal beams.....235
spiral beams..... 237,291
steel beams.....223
steel polybeams..... 226
warping.....354
bending..... 357
bending shapes
in rebar shape manager..... 608,616
of reinforcement.....603,604,605
rules..... 608
bent mesh.....536
bent plates
conical bent plates..... 245
conical bent plates..... 260
cylindrical bent plates..... 245,260
stand-alone bent plates..... 245
stand-alone bent plates..... 260
binding objects
to a plane..... 898
blind holes..... 376
bolt points.....660
bolted parts..... 363
bolts.....363
bolt group shape.....363
bolt spacing.....728
bolting sub-assemblies.....447
creating.....363
creating assemblies..... 447
holes..... 376
modifying..... 363
studs..... 375
box profiles.....1023
brackets.....202

C

c profiles.....1023
calculations.....917
cambering.....353
cambering parts.....353
cast units
adding objects..... 457
batch editing.....461
best practices.....457
cast unit type..... 454
casting direction.....458
creating.....454
exploding.....458
numbering..... 769
removing objects.....458
selecting..... 133
top-in-form face..... 458
cast-in-place cast unit.....454
cast-in-place
continuous concrete structures..... 471
pour breaks..... 482,484,485,487
pour objects.....475
pours..... 468,469,470,474,476
viewing parts.....471
viewing pour objects.....471
categories
in filtering..... 181
chamfering
corner chamfers.....420

corner chamfer dimensions..... 420
edge chamfers.....420
polybeams.....420
chamfers
in warping..... 354
changing
welds to polygon welds.....395
checking the model..... 697
circular hollow sections.....1023
circular reinforcing bar groups.....531
circular sections..... 1023
clash checking
clash symbols..... 732
clash types..... 732
clash types..... 732
modifying..... 739
printing.....739
symbols..... 732
viewing.....739
clash checking.....697,731
clip boxes..... 713
clip planes.....697,713
closing
custom components.....887
cold rolled profiles.....1023
colliding objects..... 731
colors
for pour objects.....476
object representation settings..... 686
columns
concrete columns.....281
positioning..... 349
steel columns.....220
combining
parts.....358
reinforcing bar groups.....581
comparing parts or assemblies..... 697,743
component catalog.....814
components in Applications & components
.....814
components
catalog..... 814
connections..... 801,803
detailing.....803
details.....801,803
exploding.....866
nested components.....866
properties.....803
selecting..... 133
showing invisible objects..... 682
sub-components..... 866
views..... 803
concrete parts
cast units.....454
concrete beam properties..... 284
concrete beams.....284
concrete column properties..... 281
concrete cover
of reinforcement..... 588
concrete item properties..... 320
concrete items.....320
concrete panel properties.....295
concrete panels.....295
concrete parts..... 217
beams.....284
casting direction.....458
columns.....281
items.....320
lofted slabs.....303
pad footings.....314
panels..... 295
polybeams.....287
showing as continuous.....471
slabs.....299
spiral beams..... 291
strip footings.....316
walls.....295
concrete polybeam properties.....287
concrete slab properties..... 299
concrete walls.....295
conical bent plates..... 245,260
connection design
Excel.....852
connection weld properties.....391
connections
custom connections.....866
custom connections.....863
construction objects
for lofted plates.....268
for lofted slabs.....303
construction arcs..... 650
construction circles.....650
construction lines..... 650,908
construction objects..... 650
construction polycurves.....655
construction arcs.....653

construction circles....652
construction lines....651
construction planes....651
copying with offset....656
modifying location....657
construction planes....650,908
construction polycurves....650
contextual toolbar
modifying part position....334
continuous concrete structures....471
contour plate properties....241
contour plates....241
control numbers....783
assigning to parts....784
directions....785
displaying in model....786
example....788
locking....788
order....785
removing....787
settings....1060
unlocking....788
control points....60
converting
welds to polygon welds....395
coordinate system....54
coordinates....23
copying
construction objects with offset....656
filters....210
object groups....692
objects....137,146,166
parts....217
pour breaks....487
property references....916
Radial array tool....152
with Array of objects (29) component 154
corner chamfers
properties....420
types and dimensions....420
creating bolts
auto bolt....363
creating horizontal parts....116
creating surface treatment
unpainted area....435
creating
assemblies....217,441
assembly hierarchy....441
basic assemblies....441
beams close to each other....223
bolts....363
cast units....454
cast units....217
clip planes....713
concrete beams....284
concrete columns....281
concrete items....320
concrete panels....295
concrete polybeams....287
concrete slabs....299
concrete walls....295
construction circles....217
construction lines....217
construction arcs....653
construction circles....652
construction lines....651
construction planes....217,651
construction polycurves....655
contour plates....241
curved beams....229
fittings....397
grids....23
higher-level assemblies....441
holes....376
lofted plates....268,303
model templates....211
model views....34
nested assemblies....441,450
object groups....692
orthogonal beams....235
pad footings....314
parts....217
points....217
pour breaks....485
pours....217
reinforcement....217
slabs
sloping....325
sloping slabs....325
spiral beams....237,291
standard-part models....799
steel beams....223
steel columns....220
steel items....320
steel polybeams....226
strip footings....316

studs..... 375
tile patterns.....431
twin profiles..... 232
view filters
selection filters..... 172
welds.....381
curved parts..... 229,357
curved reinforcing bar group..... 529
custom components
examples.....943
variables.....893
custom components
about custom components.....989
custom connection
adding to model.....879
custom connections..... 989
custom details..... 989
custom parts.....989
custom seams..... 989
custom component browser..... 887
custom component editor.....887
custom component browser..... 916
custom component dialog editor
changing the language..... 995
settings..... 995
custom components
.inp files..... 999
about custom components.....863
applications & components catalog... 863
binding objects.....898,908
component symbol.....863
construction planes and lines..... 908
copying property references.....916
default properties..... 1007
defining..... 866
dialog files..... 999
editing.....887
examples....
942,949,951,953,955,958,960,964,966,
980
exporting..... 987
formulas..... 919
importing.....987
locking..... 995
modifying the dialog box..... 995
nested components.....866
nested components.....883
preventing modifications..... 995
protecting with password..... 887
saving.....887
sub-components.....866,883
thumbnail images..... 866
types..... 863
wizard..... 866
custom hooks..... 585
custom inquiry
modifying default attributes.....724
custom parts
adding to model..... 885
customized mesh.....536
customizing
Custom Inquiry tool attributes..... 724
cuts
hints and tips.....412
in rebar sets..... 565
line cuts..... 412
part cuts..... 412
polygon cuts..... 412
cutting
beams.....400
columns.....400
cyclic dependencies 931
cylindrical bent plates..... 245,260

creating
timber objects.....331

D

data type conversion functions.....919
default adaptivity..... 438
default connection properties.....839
default connection settings..... 839
defining unpainted area
surface treatment..... 435
defining
custom components.....932
custom components.....866
weld cross sections..... 396
deleting
clip planes..... 713
dependencies
in variable formulas.....931
design group numbering..... 790
Design tab.....839

detaching
parts....359
detailing parts
detaching.... 359
exploding....359
details
custom details.... 866
custom details.... 863
diagnosing model....697,748
dialog editor
custom components....995
direct modification
modifying.... 119
direction
of rebar set modifiers....557
display settings....676
display detailing.... 439
display settings....671
displaying
control numbers....786
distance variables.... 893
distance variables.... 898
distance
reference distance.... 911
distances
measuring.... 728
drawing objects
mirroring.... 165
moving....155
rotating....161

E

edge chamfers
properties....420
editing
custom components....887
geometry....325
editors
custom component dialog editor....995
end detail modifiers.... 557
end offsets....348
exact lines.... 681
examples
adding option to create object in custom component....942
creating parametric variable....943
determining bolt group distance from beam flange....955
determining bolt size and bolt standard 953
determining number of bolt rows in custom component....958
drawing filters....202
modifying custom component dialog.... 1011,1012,1017,1019
modifying dialog of stiffener detail...1011
nested components....945
numbering....788,793,794,795,796
rebar set modifiers in custom components....981
replacing sub-components in custom component....949
snapping in model.... 103
sub-components.... 945
tile pattern definition....431
using construction planes in custom component....960
using Excel spreadsheets with custom components....980
using properties file in custom component....951
using template attribute in custom components....966
using user-defined attributes in custom components....964
view and selection filters....202
excel design.... 839
Excel
connection design....852
using with custom components....980
exploding
assemblies.... 453
cast units.... 458
components....866
parts....359
rebar assemblies....649
sub-assemblies in rebar assemblies...649
exporting
custom components....987
extensions in Applications and components catalog....801

F

family numbers....760,761
  example....794
filtering
  And/Or....198
  categories....181
  conditions....198
  copying filters....210
  object properties....181
  objects....168
  parentheses....198
  removing filters....210
  template attributes....198
  use for....168
  view filters
    selection filters....172
  wildcards....202
filters
  examples....202
finding distant objects....697,750
finding
  model objects....697
fit part end....397
fitting
  beams....400
  columns....400
  panels....403
  plates....403
  slabs....403
fittings....397
fixed profiles....332
flying through model....697,720
folded plates....1023
footings....314,316
formulas
  arithmetic operators....919
  ASCII file as a reference function....919
  data type conversion functions....919
  framing condition functions....919
  in custom components....917
  in rebar shape manager....615
  logical and comparison operators....919
  market size function....919
  mathematical functions....919
  reference functions....919
  statistical functions....919
  string operations....919
  trigonometric functions....919
framing condition functions....919
freezing
  rebar sets....571
functions....917

G

General tab....839
geometry
  editing....325
  modifying....325
  of reinforcement....594
global coordinate system....54
global point of origin....54
graphics card....76
grid lines
  adding....27
  modifying....29
  properties....29
grid views
  properties....51
grids
  coordinates....20,23
  creating....20,23
  deleting....20,23
  labels....20,23
  line extensions....20
  modifying....20,23
  origin....20
  properties....20,23
  radial....20,23
  rectangular....20,23
  work plane grid....56
grouping
  reinforcement....580

H

handles....116
  in pour breaks....487
  of reinforcement....584
hat profiles....1023
hatches
  in DX....671
hiding parts....671
hiding

cut lines..... 412
dimensions..... 125
part labels..... 744
rebar labels..... 744
rebar set guidelines..... 557
rebar set modifiers..... 557
reference lines..... 116
selected parts..... 682
top-in-form face..... 458
unselected parts..... 682
work area..... 52
high accuracy..... 681
higher-level assemblies..... 440,441
highlighting
assemblies..... 450
objects..... 135
holes..... 376
horizontal position..... 346

I

i beams (concrete)..... 1023
i beams (steel)..... 1023
i profiles..... 1023
identical
areas..... 215
parts..... 757
reinforcement..... 757
if you cannot select objects..... 135
import items..... 320
importing
custom components..... 987
points..... 660
inquiring..... 697
object properties..... 721
interrupting
object selection..... 135
irregular beams (concrete)..... 1023
items..... 217,320
changing the shape..... 320
limitations..... 320

J

joining assemblies..... 450
joints.def..... 839

L

l profiles..... 1023
labels
part labels..... 744
rebar labels..... 744
ledger beams (concrete)..... 1023
leg faces..... 549
leg surfaces..... 549
levels..... 349
line cuts..... 412
linear array tool..... 149
lines
exact..... 681
linking
parametric variables to object properties
..... 913
local coordinate system..... 54
lock x y z..... 92
lofted plates
properties..... 268
lofted slabs
properties..... 303
lofted walls
properties..... 303
log files
pour errors..... 746
solid errors..... 746
logical and comparison operators..... 919

M

macros
adding..... 811
editing..... 811
global..... 811
local..... 811
macros folder..... 801
recording..... 811
running..... 811
magnetic construction planes and lines.. 908
market size function..... 919
mathematical functions..... 919
measuring objects..... 697,728
angles..... 728
arcs..... 728
bolt spacing..... 728
distances..... 728

mesh
bent....536
customized....536
handles....584
modifying....572
polygonal....536
rectangular....536
ungrouping....579
mirroring
drawing objects....165
model objects....165
model templates
creating....211
modifying
options....211
model objects
create....114
delete....114
examples....114
hiding....682
showing only selected....682
model objects
mirroring....165
searching....697
model search toolbar....697
model views....31,34,44
background color....73
creating....34
general arrangement drawing....44
refreshing....49
rendering options....671
representation options....671
modeling direction....116
Modeling of floor bay (66)....354
modeling
identical areas....215
models
flying through model....720
moving....85
numbering....752
rotating....85
zooming....85
modifiers
changing the direction....557
in custom components....981
showing or hiding....557
visibility....557
modifying
concrete beams....284
concrete columns....281
concrete items....320
concrete panels....295
concrete polybeams....287
concrete slabs....299
concrete walls....295
construction objects....657
contour plates....241
curved beams....229
custom components....932
custom components....866
geometry....325
items....320,325
lofted plates....268,303
model templates....211
objects....119
orthogonal beams....235
pad footings....314
parts....217,332
pour breaks....487
pour object properties....477
rebar sets....545
reinforcement....545,572
steel beams....223
steel columns....220
steel polybeams....226
strip footings....316
twin profiles....232
moving
drawing objects....155
models in a view....85
objects....119,155,166
pour breaks....487

N

nested assemblies....440,441
nested assemblies....133,450
nested
components....866,883,945
no paint area....435
numbering....752,768
about numbering....752
assemblies....769
cast units....769
checking and repairing....781
clearing....782

control numbers....
783,784,785,786,787,788
examples....793,794,795,796
family numbers....760,761,794
history....780
identical parts....757
manually....781
modifying....781
numbering series....755
parts....767,783
preliminary numbers....771
preview....771
rebar assemblies....769
reinforcement....757,768
reinforcement sequence numbers....762
renumbering....783
series....753,754,756
settings....760,798,1057,1059,1060
standard-part model....799
user-defined attributes....759
welds....770
what affects....758
numbering
design group numbering....790
numbering settings....1057
numeric snapping....103

O

object groups
copying to another model....692
creating....692
deleting....692
filters....692
object representation settings....692
object groups....671
object properties
in filtering....181
object property report templates....723
object representation settings
colors....686
object groups....686
parts....686
transparency....686
object representations....671
objects
clash checking....731
copying....137,146,166
defining visibility....676
filtering....168
finding distant objects....750
inquiring properties....721
measuring....728
moving....155,166
numbering....767,768
placing objects in model....650
rotating....161
selecting....125
showing and hiding....695
offsets....348
operators....917
origin....54
ortho tool....99
orthogonal beam properties....235
orthogonal beams
positioning....349
orthogonal
snapping....99
others....1023
overlapping
numbering series....756
oversized holes....376

P

pad footing properties....314
pad footings....314
positioning....349
properties....314
panels....295,1023
panning....85
parallel snapping....92
parametric variables....893,943
parametric profiles....332
available in Tekla Structures....1023
predefined....1023
parametric variables....
942,945,949,951,953,955,958,960,964,966,
980
creating....913
linking....913
part add, see attaching parts....359
part cuts....412
part labels
showing and hiding....744
part position

on the work plane....340
part reference lines....116,334
partial-depth holes....376
parts
adding to assembly....450
assemblies....440,441
bending....357
cambering....353
changing the material....332
changing the profile....332
combining....358
comparing....743
concrete parts....217
coordinate systems....334
copying....217
creating....217
curved parts....357
custom parts....866
custom parts....863
cutting with another part....412
faces....334
handles....116,334
hiding....682
horizontal parts....116
identical parts....757
items....320
labels....744
location....334
modifying....217,332
modifying part shape....119
numbering....755,767,768,783
position....116,334
properties....217
reference lines....334
showing only selected parts....682
showing with exact lines....681
showing with high accuracy....681
splitting....361
steel parts....217
user-defined attributes....356
with exact lines....671
with high accuracy....671
performance
modeling tips....213
personal views....44
phases
in numbering....796
pour phase....474
plane of curvature....357
plane views....48
plane
in snapping....89
plates
bent plates....245,260
points....650,660
along arc using center and arc points 660
along extension of two points....660
at any position....660
at the intersection of a circle and a line....660
at the intersection of a part and a line....660
at the intersection of a plane and a line....660
at the intersection of two lines....660
at the intersection of two part axes....660
importing....660
on line....660
on plane....660
parallel to two points....660
projected points on line....660
properties....660
tangent to circle....660
polybeams....226,287
chamfers....420
polygon cuts....412
polygon mesh....536
polygon welds....381
converting....395
splitting double-sided to single-sided 395
position
depth....343
end offsets....348
horizontal....346
rotation....342
vertical....344
pour breaks
copying....487
handles....487
modifying....487
moving....487
removing....487
pour breaks....482
adaptivity....483
creating....485
visibility....484

pour objects....475
changing color and transparency....476
modifying properties....477
reinforcing....528
user-defined attributes....477
viewing....471
pour units....477
adding objects automatically....481
pours
changing color and transparency....476
errors....488,746
example workflow....491
introduction....468
pour breaks....482
pour objects....475
pour phase....474
pour representation....471
reinforcing....528
switching on/off....469,470
viewing....471
working with....491
precast cast unit....454
preliminary numbers....771
preparing parts for welding....392
prestressed strands
debonding....542
profiles
fixed....332
parametric....332,1023
predefined....1023
profile names....202
standardized values....332
twin profiles....232
properties
custom components....1007
grid lines....29
grids....23
twin profile....232
variable properties....936
property modifiers....557
property pane....114

Q

question mark....202

R

radial array tool....152
radial grid....23
radius....357
rebar assemblies....643
adding objects....646
changing the coordinate system....646
changing the main object....646
creating....644
exploding....649
modifying....646
removing objects....649
rebar assemblies
numbering....769
rebar cages....643,644
rebar classificator....595
rebar groups....520
circular....531
curved....529
spiral....533
tapered....533
rebar labels
showing and hiding....744
rebar sequence numbering....762
rebar set modifiers
direction....557
rebar sets....494
attaching to parts....595
cutting....565
freezing....571
in curved structures....513
leg faces....549
leg surfaces....549
modifiers....557
modifiers in custom components....981
modifying....545,557
rebar shape placing tool....507
showing or hiding modifiers....557
spacing....565
splicing....569
splitting....568,569
unfreezing....571
rebar shape manager....603
bending shapes....604,605,616
comparing bars to bending shapes....614
formulas....615
rules....608,616

rebar shape placing tool..... 507
RebarClassificator..... 595
rebars
adaptivity.....591
bar leg length.....602
bending types..... 617
combining..... 581
concrete cover..... 588
creating..... 494,520
geometry..... 594
grouping..... 580
handles..... 584
hooks..... 585
length..... 599
modifying..... 545,572
shape catalog..... 522,525
splitting..... 568,581
ungrouping..... 579
recording
macros..... 811
rectangular grid..... 23
rectangular hollow sections..... 1023
rectangular mesh..... 536
rectangular sections..... 1023
redrawing model views..... 49
reference distance variables..... 893
reference distance variables..... 911
reference functions..... 919
reference lines..... 116
reference models
clash checking..... 731
reference objects
rendering..... 671
representation..... 671
refreshing model views..... 49
reinforcement cages..... 643
reinforcement cages..... 644
reinforcement sequence numbers..... 762
reinforcement
adaptivity..... 591
attaching to part..... 593
bar groups..... 520
bar leg length..... 602
bar length..... 599
bending shapes..... 604,605
bending types..... 617
circular bar groups..... 531
classifying..... 595
combining..... 581
comparing shapes..... 614
concrete cover..... 588
curved bar groups..... 529
customized mesh..... 536
debonding strands..... 542
for pour objects..... 528
geometry..... 594
grouping..... 580
handles..... 584
hooks..... 585
identical..... 757
in templates..... 641
labels..... 744
layer information..... 595
mesh..... 536
modifying..... 544,545,572
numbering..... 757,768
omitting bars..... 578
prestressed strands..... 541
rebar sets..... 494
selecting from catalog..... 591
sequence numbers..... 762
shape catalog..... 522,525
shape codes..... 604,605
shape recognition..... 603
single bars..... 520
spiral bar group..... 533
splice..... 543
splicing..... 569,582
splitting..... 568,569,581
strands..... 541
tapered bar group..... 533
ungrouping..... 579
reinforcing bar groups..... 520
adaptivity..... 591
bar leg length..... 602
bar length..... 599
circular..... 531
combining..... 581
curved..... 529
excluding bars..... 578
geometry..... 594
grouping..... 580
handles..... 584
modifying..... 572
omitting bars..... 578
pours..... 528

shape catalog..... 522,525
spiral..... 533
splitting..... 581
tapered..... 533
ungrouping..... 579
reinforcing bars..... 520
bending types..... 617
combining..... 581
geometry..... 594
handles..... 584
hooks..... 585
leg length..... 602
length..... 599
modifying..... 572
pours..... 528
shape catalog..... 522,525
spacing..... 576
reinforcing pour objects..... 528
removing
filters..... 210
pour breaks..... 487
weld cross sections..... 396
rendering of parts and components..... 671
rendering
DirectX or OpenGL..... 76
of model views..... 76
of parts, components, reference objects, and building hierarchy..... 671
renumbering..... 783
repairing model..... 748
repairing
numbering errors..... 781
report templates
for inquiring object properties..... 723
reports
on reinforcement..... 615,641
representation
of cast-in-place concrete structures... 471
of parts, components, reference objects and building hierarchy..... 671
reshaping
objects..... 119
ribbons..... 114
right-click
selecting..... 135
rollover highlight..... 135
rotating
drawing objects..... 161
objects..... 161
parts..... 342
settings..... 161
rotation angle knob..... 334
rotation settings..... 161
round
holes..... 376
plates..... 241
slabs..... 299
rules
in bending shapes..... 608
in rebar shape manager..... 608,616
running
macros..... 811

S

saving
custom components..... 887
seams
custom seams..... 866
custom seams..... 863
search in model..... 697
search in model..... 697
search in selection..... 697
searching
model objects..... 697
secondary guidelines..... 557
select work plane..... 72
selecting
all objects..... 125
assemblies..... 133
by id..... 125
cast units..... 133
dates from model..... 201
handles..... 125
interrupting object selection..... 135
multiple objects..... 125
nested assemblies..... 133
objects..... 125,135
objects in components..... 133
on right-click..... 135
previous objects..... 125
unable to select objects..... 135
values from model..... 201
selection dial..... 334
selection filters..... 172
setting up

grid....19
views....19
work area....19
work plane....19
workspace....19
settings
adaptivity....438
bolt properties....363
concrete beam properties....284
concrete column properties....281
concrete item properties....320
concrete panel properties....295
concrete slab properties....299
contour plate properties....241
corner chamfer properties....420
custom component dialog editor....995
display settings....676
edge chamfer properties....420
grid line properties....29
grid properties....23
grid view properties....51
lofted plate properties....268,303
model templates....211
numbering....798
numbering settings....760,1057,1059,1060
object representation....686
orthogonal beam properties....235
pad footing properties....314
point properties....660
reinforcement properties....1061
rotation settings....161
snapping....113
steel beam properties....226
steel beam properties....223,229
steel column properties....220
steel item properties....320
strip footing properties....316
twin profile properties....232
view properties....50
weld properties....381
shape codes
of reinforcement....603,604,605,608
shape
modifying part shape....119
shapes
of items....320
shared views....44
shifting the work plane....56
showing and hiding
rebar set guidelines....557
rebar set modifiers....557
showing assemblies....671
showing component content....671
showing only selected parts....671
showing part reference lines....116
showing
cast-in-place concrete structures....471
cut lines....412
dimensions....125
invisible assembly objects....682
invisible component objects....682
part labels....744
parts with exact lines....681
parts with high accuracy....681
rebar labels....744
rebar set guidelines....557
rebar set modifiers....557
reference lines....116
top-in-form face....458
welds....393
work area....52
single bolts....363
single-part assemblies....440,441
single-part welds....381
slabs....299
sloping....325
warping....354
sloping slabs....325
slotted holes....376
snap symbols....89
snap grid....92,109
snap settings....113
snap switches....88,89
snapping....88,92,99
align objects....92
edge....92
example....103
extension line....92
line....92
lock coordinates....109
orthogonal directions....99
settings....113
snap priority....89
snap switches....89
snap symbols....89
snap zone....89

snap depth....89
temporary reference lines....109
to middle points....116
to numeric coordinates....103
using coordinates....103
snapping mode
absolute....103
global....103
relative....103
snapping toolbar....88
solid checker....746
solid errors....697,746
spacing in rebar sets....565
spacing reinforcing bars....576
special characters....202
spiral beams....237,291
splice....543
splicing
rebar set bars....569
reinforcement....569,582
splitters....557,569
splitting
parts....361
rebar set bars....569
rebar sets....568
reinforcement....569
reinforcing bar groups....581
stand-alone bent plates....245,260
standard-part models....799
standardized values for parametric profiles
....332
statistical functions....919
steel parts
assemblies....440,441
steel beam properties....223
steel column properties....220
steel connection properties....391
steel curved beam properties....229
steel item properties....320
steel items....320
steel parts....217
beams....223
bent plates....245,260
columns....220
contour plates....241
curved beams....229
items....320
lofted plates....268
orthogonal beams....235
polybeams....226
spiral beams....237
twin profiles....232
steel polybeam properties....226
strand pattern
of reinforcement....541
string operations....919
strip footing properties....316
strip footings....316
studs....375
sub-assemblies....440,441
sub-components....883
examples....866,945
surface treatment
add new subtype....430
adding....425
modifying....425
on all faces of part....425
on chamfered parts....425
on cut faces....425
on part face....425
on parts with openings and recesses....425
on selected areas....425
tiled surface treatment....425
surfaces....436
swap end handles....268
swap end handles....303
Swap Handles....116
switching between
model views....48
view angles....48
switching on/off
pour management....469,470
system components....803
system components....801

T

t profiles....1023
t profiles (concrete)....1023
tail information....381
tapped holes....376
templates
model templates....211
thumbnail images
of custom components....866
tiled surface treatment

creating new tile patterns..... 431
pattern definition example..... 431
pattern definitions..... 431
pattern elements.....431
timber objects..... 331
tips
creating beams close to each other....223
creating curved parts.....357
hiding cut lines..... 412
modeling identical areas..... 215
modeling large models.....213
numbering settings.....798
positioning columns, pad footings and orthogonal beams.....349
toggle coordinate lock..... 92
toolbars
model search..... 697
top-in-form face..... 458
tracking
along line.....103
transparency
object representation settings..... 686
of pour objects..... 476
trigonometric functions..... 919
Trimble Connect Visualizer..... 700
creating animations..... 701
managing materials..... 707
overriding materials.....707
scene settings..... 701
taking snapshots..... 701
vr mode..... 701
Trimble Connect Visualizer
user-defined materials..... 709
twin profiles..... 232

U

u profiles..... 1023
unfolding lofted plates..... 268
unfreezing
rebar sets..... 571
ungrouping
reinforcement.....579
updating model views..... 49
user-defined attributes
for parts.....356
in numbering..... 759
user-defined weld cross sections.....396

V

value types.....936
values
selecting from model.....201
variable formulas.....917
variable cross sections..... 1023
variables
creating dependencies..... 913
dependencies..... 931
distance variables..... 898
in custom components.....893
parametric variables..... 913
reference distance variables.....911
value types..... 936
variable properties.....936
vertical position.....344
view angles..... 48,697
assembly view angle..... 719
component view angle..... 719
connection view angle..... 719
part view angle..... 719
view depth..... 695
view depth clip planes.....713
view filters.....172
view planes..... 56
viewing
cast-in-place concrete parts.....471
models.....713
numbering history..... 780
pour objects.....471
views..... 31
properties.....50
switching between model views.....48
visibility of objects..... 676,695
visibility
of pour breaks..... 484
visualizing the model.....700
materials..... 707
visualizing the model.....697

W

walls.....295
warping..... 354
beams.....354
concrete slabs.....354
weld cross sections

defining..... 396
removing..... 396
weld preparation.....392
welded beam profiles.....1023
welded box profiles..... 1023
Welds in components.....839
welds..... 391
between parts.....381
creating.....381
creating assemblies..... 447
defining cross sections..... 396
numbering.....770,1059
polygon welds.....381
properties.....381
removing cross sections.....396
showing..... 393
single-part welds..... 381
user-defined cross sections..... 396
visibility in model..... 393
weld preparation.....392
weld types..... 381
welding sub-assemblies..... 447
wildcards..... 202
work area.....52,695
hiding..... 52
work plane handler toolbar
base points.....72
project base points..... 72
work plane..... 72
work plane
shifting.....56
show or hide..... 56
toolbar..... 72
wq profiles..... 1023

X

XML files
Trimble Connect Visualizer..... 707

Z

z profiles.....1023
zooming
in or out..... 85
to selected objects..... 85