TEKLA
A TRIMBLE COMPANY

Tekla Structures

Concrete Components Reference Guide

Product version 21.0
March 2015

©2015 Tekla Corporation

Contents

1 Concrete Detailing 9

1.1 Seating connections 9

Seating with dowel (75) 10
Defining bolt hole dimensions 11
Fitting the beam or column (75) 12
Defining grout (75, 76) 12
Chamfering corbels 13
Creating a recess (75, 76) 13
Defining recess dimensions 14
Bearing plate position (75, 76) 14
Two-sided seating with dowel (76) 14
Fitting beam or column (76) 16
Shaping beam ends (76, 78) 17
Seating with dowel to flange (77) 17
Creating a recess (77, 78) 19
Two-sided seating with dowel to flange (78) 19

1.2 Beam and column connections 21

Battering connection (13) 21
Corbel connection (14) 22
Corbel connection (14): Picture tab 24
Corbel connection (14): Parts tab 27
Corbel connection (14): Reinforcing bar tab 29
Concrete beam-beam (112) 32
Concrete beam-beam (112): Picture tab 34
Concrete beam-beam (112): Parts tab 44
Concrete beam-beam (112): Anchors tab 45
Concrete beam-beam (112): Parameters tab 47
Concrete beam-beam (112): Anchor rods tab 47
Concrete beam-beam (112): Socket tab 59

1.3 Panels and walls 60

Electric box in wall (84) 60
Electric box in wall (84): Picture tab 63
Electric box in wall (84): Parts tab 65
Electric box in wall (84): Electric box tab 67
Electric box in wall (84): Top conn / Bottom conn tab 70
Electric box in wall (84): UDA tab 72
Sandwich And Double Wall (1) 75
Sandwich And Double Wall (1): Parts tab 77
Sandwich And Double Wall (1): Vertical section tab 82
Sandwich And Double Wall (1): Horizontal section tab 86
Sandwich And Double Wall (1): Insulation tab 91
Sandwich And Double Wall (1): Outside Shell tab 93
Sandwich And Double Wall (1): UDA tab 96
Sandwich Wall Horizontal Seam (1) 97
Sandwich Wall Horizontal Seam (1): Picture tab 99

Sandwich Wall Horizontal Seam (1): Rabbets tab....102
Sandwich Wall Horizontal Seam (1): Extra foils tab....102
Sandwich Wall Vertical Seam (1)....104
Sandwich Wall Vertical Seam (1): Picture tab....105
Sandwich Wall Vertical Seam (1): Rabbets tab....107
Sandwich Wall Window (1)....109
Sandwich Wall Window (1): Picture tab....111
Sandwich Wall Window (1): Bottom detail tab / Left detail tab / Right detail tab....113
Sandwich Wall Window (1): Top detail tab....116
Sandwich Wall Window (1): Extra foils tab....119
Sandwich Wall Window (1): Additional parts tab....121
Sandwich Wall Window (1): Connections tab....124

1.4 Formwork components....124

Beam Formwork....125
Beam Formwork: Beam tab....126
Beam Formwork: Parameters tab....127
Column Formwork....129
Column Formwork: Column tab....130
Wall Formwork....132
Wall Formwork: Panel tab....133
Wall Formwork: Parameters tab....135
General Beam Form....136
Beam Form: Parameters tab....137
General Wall Form....138
Wall Form: Parameters tab....140
General Clamp....141
General Clamp: Parameters tab....141
General Clip....142
General Clip: Parameters tab....143
General Tie....144
General Tie: Parameters tab....145
General Tie and Clips....146
General Tie and Clips: Parameters tab....147

1.5 Openings....148

Hole Generation (32)....148
Hole Generation (32): Parameters tab....148
Polygon Hole Generation (33)....150
Polygon Hole Generation (33): Parameters tab....150

1.6 Flooring....151

Automatic seam recognition (30)....151
Seam Applicator....153
Seam Applicator: Parameters tab....154
Slab generation with polygon plate (61)....155
Slab generation with polygon plate (61) and Slab generation with points (62): Parameters tab 156
Slab generation with polygon plate (61) and Slab generation with points (62): Parts tab....156
Slab generation with polygon plate (61) and Slab generation with points (62): Analysis tab....156
Slab generation with points (62)....157
Slab generation with polygon plate (61) and Slab generation with points (62): Analysis tab....158
Modeling of floor bay (66)....159
Examples....160
Defining slab position....163
Defining projection type....164
Defining slab profile properties....165

Index list 166
Adjustment type 167
Profile/Seam width 167
Modeling of slab area (88) 167
Defining slab area properties 168
Sloping slab drainage 169
Sloping slab drainage: Parameters tab 170
Hollow Core Opening Tool 172
Hollow Core Opening Tool: Parameters tab 173
Hollow Core Opening Tool: Configuration file tab 175
Hollow Core Lifting Loops 176
Hollow Core Lifting Loops: Basic settings tab 177
Hollow Core Lifting Loops: Lifting device settings tab 178
Hollow Core Lifting Loops: Configuration file tab 178
Floor Tool 179
Floor Tool: Slab properties tab 180
Floor Tool: Insulation tab 181
Floor Tool: Bay contour tab 181
Floor Tool: Slab list tab 182
Floor Layout 183
Floor Layout: Layer tab 185
Floor Layout: General tab 186
Floor Layout: Advanced tab 188
Floor Layout: Detailing tab 189
Floor Layout: User defined tab 191
Floor Layout: Modifying a floor 192
Floor Layout CIP Filler 206

1.7 Concrete stairs 206

Concrete stairs (7) 207
Defining stair properties 208
Defining stair tolerances 208
Concrete stairs (65) 208
Concrete stairs (65): Stairs tab 211
Concrete stairs (65): Step shapes tab 214
Concrete stairs (65): Landings tab 216
Concrete stairs (65): Ridges tab 217
Concrete stairs (65): Stringers tab 220
Concrete stairs (65): Anti-skid/Kick plate tab 221
Concrete stairs (65): Attributes tab 223
Concrete stairs (65): UDA tab 223
Stairwells and elevator shafts (90) 223
Defining stairwell properties 225
Defining number of floors and height 226
Defining position 226
Defining length and width 226
Reinforced Concrete Stair (95) 227
Reinforced Concrete Stair (95): Parameters tab 230
Reinforced Concrete Stair (95): Parts tab 231
Reinforced Concrete Stair (95): Stairs and landings tab 232
Reinforced Concrete Stair (95): Ridges tab 233
Reinforced Concrete Stair (95): Stringers tab 235
Reinforced Concrete Stair (95): Anti-skid tab 236
Reinforced Concrete Stair (95): Bar A tab 237
Reinforced Concrete Stair (95): Bar B tab 237
Reinforced Concrete Stair (95): Bar C tab 239

Reinforced Concrete Stair (95): Bar D tab....240
Reinforced Concrete Stair (95): Bar E tab....242
Reinforced Concrete Stair (95): Bar F tab....243
Reinforced Concrete Stair (95): Bar G tab....244
Reinforced Concrete Stair (95): Bar H tab....245
Reinforced Concrete Stair (95): Bar I tab....246
Reinforced Concrete Stair (95): Bar J tab....247
Reinforced Concrete Stair (95): Bar K tab....248
Reinforced Concrete Stair (95): Bar L tab....249
Reinforced Concrete Stair (95): Mesh picture tab....250
Reinforced Concrete Stair (95): Mesh attributes tab....251
Reinforced Concrete Stair (95): Mesh bars tab....252
Reinforced Concrete Stair (95): Bottom anchor bars tab....252
Reinforced Concrete Stair (95): Top anchor bars tab....253
Reinforced Concrete Stair (95): Z anchor bars tab....253
Reinforced Concrete Stair (95): Landing end bars tab....254
Reinforced Concrete Stair (95): Configuration....255

1.8 Foundations....256

Concrete foundation (1030)....256
Concrete foundation (1030): Picture tab....258
Concrete foundation (1030): Parts tab....261
Concrete foundation (1030): Massive tab....263
Concrete foundation (1030): Piles tab....265
Concrete foundation (1030): Inj. tube and rebar tab....266
Concrete foundation (1030): Column tab....268
Concrete foundation (1030): Stirrups tab....269

2 Reinforcement....271

2.1 Reinforcement for foundations....271

Strip footing (75)....271
Strip footing reinforcement properties....272
Pile cap reinforcement (76)....273
Pad footing and pile cap shapes....275
Bar distribution....275
Lacer bars for pad footings and pile caps....276
Pad footing (77)....277
Starter bars....279
Starter bar stirrups....280
Starter bar location....281

2.2 Beam, column, and slab reinforcement....281

Mesh Bars....282
Mesh Bars: Picture tab....282
Mesh Bars: Detailing tab....284
Mesh Bars: Bar end conditions tab....286
Mesh Bars: Attributes tab....287
Mesh Bars by Area....287
Mesh Bars by Area: Picture tab....288
Mesh Bars by Area: Detailing tab....291
Mesh Bars by Area: Bar end conditions tab....293
Mesh Bars by Area: Attributes tab....294
Slab bars (18)....294
Slab reinforcement properties....295
Slab Reinforcement Tool....296
Slab Reinforcement Tool: Parameters tab....297

Slab Reinforcement Tool: Advanced tab....298
Beam reinforcement (63) 299
Beam reinforcement properties 301
Beam reinforcement (63): Stirrup spacing tab....302
Double tee reinforcement (64) 302
Double tee reinforcement properties 304
Beam end reinforcement (79) 305
Beam end reinforcing bars 307
Horizontal U bars 1 and 2 307
Vertical U bars 3A and 3B 307
Stirrups 5A and 5B 308
Corbel reinforcement (81)....308
Corbel reinforcement (81): Main bars tab....310
Corbel reinforcement (81): Stirrups/Transverse Stirrups/Diagonal Stirrups tabs....311
Corbel reinforcement (81): Additional bars tab....312
Round column reinforcement (82) 313
Column stirrup properties 314
Column top and bottom reinforcement 315
Rectangular column reinforcement (83)....316
Rectangular column reinforcement (83): Main bars tab....319
Rectangular column reinforcement (83): Bar ends tab....320
Rectangular column reinforcement (83): Side bars tab....321
Rectangular column reinforcement (83): Stirrups tab....322
Rectangular column reinforcement (83): Intermediate links tab....323
Rectangular column reinforcement (83): Top / Bottom tab....325
Hole reinforcement for slabs and walls (84) 326
Hole creation and reinforcement (85) 328
Reinforcement around holes 330
Stirrups at holes 331
Braced girder (88)....332
Braced girder (88): Picture tab....334
Braced girder (88): Parts tab....335
Braced girder (88): Geometry tab....345
Braced girder (88): Double wall tab....347
Braced girder (89) 349
Braced girder (89): Picture tab....351
Braced girder (89): Parts tab....353
Braced girder (89): Geometry tab....362
Braced girder (89): Double wall tab....364
Reinforcement mesh array in area (89) 366
Mesh generation and overlapping properties 367
Wall Panel Reinforcement....370
Wall Panel Reinforcement: Picture tab....371
Wall Panel Reinforcement: Reinforcement tab....372
Wall Panel Reinforcement: Opening tab....374
Wall Panel Reinforcement: Diagonals tab....374
Wall Panel Reinforcement: Column tab....375
Wall Panel Reinforcement: Beam tab....376
Wall Panel Reinforcement: Additional tab....376
Wall Panel Reinforcement: Attributes tab....377
Multiple Wire Size Mesh....377
Multiple Wire Size Mesh: Mesh Parameters tab....378
Multiple Wire Size Mesh: Longitudinal/Crossing Wires tab....378
Multiple Wire Size Mesh: Attributes tab....379
Multiple Wire Size Mesh: Configuration file for reinforcing bar patterns....380

Embedded anchors (8) 381
Embedded anchors (8): Input tab 385
Embedded anchors (8): Placement tab 389
Embedded anchors (8): Top placement tab 394
Embedded anchors (8): Thickening tab 396
Embedded anchors (8): Picture tab 397
Embedded anchors (8): Top part tab 400
Embedded anchors (8): Bottom part tab 400
Embedded anchors (8): Parts tab 401
Embedded anchors (8): Reinforcing bar tab 403
Embedded anchors (8): Hangup bars tab 404
Embedded anchors (8): Horizontal bar tab 406
Embedded anchors (8): Special bar tab 408
Embedded anchors (8): Advanced tab 410
Embed (1008) 411
Embed (1008): Picture tab 412
Embed (1008): Top part tab 414
Embed (1008): Bottom part tab 414
Embed (1008): Parts tab 415
Embed (1008): Placement tab 416
Embed (1008): Reinforcing bar tab 418
Embed (1008): Advanced tab 418
Embed (1008): Bolts 418
Continuous Beam Reinforcement 420
2.3 Lifting 421
Lifting anchor (80) 422
Lifting anchor properties 424
Custom components as lifting anchors 425
Creating recesses for lifting anchors 426
Anchor properties from file 426

3 Disclaimer 428

1 Concrete Detailing

This section introduces the concrete detailing tools available in Tekla Structures.

Click the links below to find out more:

- Seating connections on page 9
- Beam and column connections on page 20
- Panels and walls on page 60
- Formwork components on page 124
- Openings on page 147
- Flooring on page 151
- Concrete stairs on page 206
- Foundations on page 256

Precast components can only be applied to precast cast units. They cannot be applied to cast-in-place cast units.

1.1 Seating connections

Tekla Structures includes several seating connections that you can use to connect concrete columns and beams using anchor bolts. The seating connection tools are:

Component	Description
Seating with dowel (75) on page 10	Connects a column and beam using an anchor bolt.
Two-sided seating with dowel (76) on page 14	Connects a column and two beams using anchor bolts.
Seating with dowel to flange (77) on page 17	Connects the flanges of a beam to a column using anchor bolts.

Component	Description
Two-sided seating with dowel to flange (78) on page 19	Connects the flanges of two beams to a column using anchor bolts.

Seating with dowel (75)

Connects a column and a beam using an anchor bolt.

Parts created

- Anchor bolt
- Nut
- Washer plate
- Bearing plate
- Fittings for beam and column
- Hole for bolt
- Corbel (optional)
- Recess for nut and washer plate (optional)

Where to use

Situation	Description
	Connects a beam and a column using an anchor bolt. Washer plate and nut protrude from the beam.
	Connects a beam and a column using an anchor bolt and a beveled corbel. Washer plate and nut recessed into the beam.

Before you start Create the following parts:

- Concrete column (round or rectangular profile)
- Concrete beam (rectangular, HI, I, L, or inverted T profile)

Defining properties Use the tabs in the **Seating with dowel (75)** dialog box to define the following properties:

Tab	Contents	See also
Picture	Position of anchor bolt, bolt hole and bearing plate, bolt hole dimensions, option to fit beam or column, grout type, beam clearance	Defining bolt hole dimensions on page 11 Fitting the beam or column (75) on page 11 Defining grout (75, 76) on page 12
Parts	Bearing plate, grout, washer plate, nut, recess, and anchor bolt properties	Creating a recess (75, 76) on page 13
Corbel	Option to create corbel, corbel properties	Chamfering corbels on page 12
General		General tab
Analysis	Information used in structural analysis	Analysis tab

Picking order

1. Column
2. Beam

Defining bolt hole dimensions

Enter the following bolt hole dimensions:

Field	Description
1	Hole dimension in the direction of the beam
2	The distance from beam center line to hole center and bolt.
3	Hole dimension in the direction perpendicular to the beam.

Fitting the beam or column (75)

Select one of the following options to have Tekla Structures fit the column or the beam:

Option	Description
	Fits the column. (Default)
	Fits the beam. The beam must be sloped.

Defining grout (75, 76)

Select one of the following options to include and define grout:

Option	Description
	No grout. (Default)
	Bolt hole grouted. No nut or washer plate.
	Bolt hole grouted. Bolt, washer plate, and anchor bolt protruding.

Chamfering corbels

The options for chamfering corbels are:

Option	Description
	Beveled (Default)
	Straight
	Rounded

Creating a recess (75, 76)

Select one of the following options to define if the nut and washer plate are recessed into the beam:

Option	Description
	Nut and washer plate on the surface of the beam. (Default)
	Nut and bolt plate recessed into the beam.

Defining recess dimensions

If you choose to recess the nut and washer plate into the beam, enter the following dimensions to define the recess:

Bearing plate position (75, 76)

Select one of the following options to define the position of the bearing plate:

Option	Description
	Square with beam. (Default)
	Square with column.

Two-sided seating with dowel (76)

Connects a column and two beams using anchor bolts.

Parts created

- Anchor bolts (2)
- Nuts (2)
- Washer plates (2)
- Bearing plates (2)
- Fittings for beam and column (2)
- Holes for bolts (2)
- Corbels (optional) (2)
- Recesses for washer plate and nut (optional) (2)

Where to use

Option	Description
	Connects two beams to a column using anchor bolts and creates corbels. Nuts and washer plates on the surface of the beams.
	Connects two beams to a column using anchor bolts. Nuts and washer plates recessed into the beams. Second beam is sloped, and is cut to create a gap between the beam and the column.

Before you start

Create the following parts:

- Concrete column (round or rectangular profile)
- Two concrete beams (rectangular, HI, I, L, or inverted T profile)

Defining properties

Use the tabs in the **Two-sided seating with dowel (76)** dialog box to define the following properties:

Tab	Contents	More information
Picture	Dimensions defining the position of the anchor bolts relative to the beams and bearing plates. Options to cut and shape the ends of the beams and to fit column or beams.	To use the same dimensions for both beams, set **Use the same parameters for left and right beams** to **Yes** and enter the dimensions for the left beam only. Shaping beam ends (76, 78) on page 16 Fitting beam or column (76) on page 16
Parts	Bearing plate, bolt, nut and anchor bolt properties	

Tab	Contents	More information
Left beam **Right beam**	Left beam is the first beam picked and right is the second. Anchor bolt, bolt hole and recess properties. Clearance between beam and column if beam is sloped.	Defining bolt hole dimensions on page 11 Creating a recess (75, 76) on page 13 Defining recess dimensions on page 13
Corbel	Option to create corbel, corbel properties	Chamfering corbels on page 12
General		General tab
Analysis	Information needed for structural analysis	Analysis tab

Picking order

1. Column
2. Beam 1
3. Beam 2

Click the middle mouse button to finish picking and create the connection.

Fitting beam or column (76)

Select one of the following options to fit the beams or the column:

Option	Description
	Fits column. (Default)
	Fits beam.

Shaping beam ends (76, 78)

Select one of the following options to define how the end of the beams are cut and shaped:

Key	Options
1 = column 2 = first beam picked 3 = second beam picked	

Seating with dowel to flange (77)

Connects the flanges of a beam to a column using anchor bolts and an optional corbel.

Parts created

- Anchor bolts (2)
- Nuts (2)
- Washer plates (2)
- Bearing plate (1)
- Fittings for beam and column
- Holes for bolts (2)
- Corbel (optional)
- Recess for nut and washer plates

Where to use

Situation	More information
	Connects the flanges of a beam to a column using anchor bolts and a beveled corbel. Nuts and washer plates recessed.

Before you start

Create the following parts:

- Concrete column (Round or rectangular profile)
- Concrete beam with flange (rectangular, HI, I, L, or inverted T profile)

Defining properties

Use the tabs in the **Seating with dowel to flange (77)** dialog box to define the following properties:

Tab	Contents	More information
Picture	Dimensions defining the position of the anchor bolt relative to the beam and bearing plate, length of anchor bolt in beam and column, bolt hole dimensions and position	Defining bolt hole dimensions on page 11
Parts	Bearing plate, washer plate, nut and anchor bolt properties, option to create recess, recess properties	Creating a recess (77, 78) on page 18
Corbel	Options to create corbel, corbel properties	Chamfering corbels on page 12
General		General tab
Analysis	Information needed for structural analysis	Analysis tab

Creating a recess (77, 78)

To cut a recess in the beam web, select the **Yes** option in the **Create recess** dropdown list. Enter the following dimensions to define the recess:

Two-sided seating with dowel to flange (78)

Connects the flanges of two beams to a column using anchor bolts and optional corbels.

Parts created

- Anchor bolts (4)
- Nuts (4)
- Washer plates (4)
- Bearing plate (2)
- Fittings for beam and column
- Holes for bolts (4)
- Corbels (2) optional
- Recesses for nut and washer plate (4)

Where to use

Situation	Description
	Connects the flanges of two beams to a column using anchor bolts.

Situation	Description
	Connects the flanges of two beams to a column using anchor bolts and beveled corbels.

Before you start

Create the following parts:

- Concrete column (Round or rectangular profile)
- Concrete beam with flange (rectangular, HI, I, L, or inverted T profile)

Defining properties

Use the tabs in the **Two-sided seating with dowel (78)** dialog box to define the following properties:

Tab	Contents	More information
Picture	Dimensions defining the position of the anchor bolts relative to the beam and bearing plate, length of anchor bolts in beam and column, beam ends	Defining bolt hole dimensions on page 11 Shaping beam ends (76, 78) on page 16
Parts	Bearing plate, washer plate, nut and dowel properties	
Left beam **Right beam**	Left beam is the first beam picked and right is the second. Bolt hole, nut and washer plate properties. Option to create recess, recess properties.	Defining bolt hole dimensions on page 11 Creating a recess (77, 78) on page 18
Left corbel **Right corbel**	Option to create corbel, corbel properties	Chamfering corbels on page 12
General		General tab
Analysis	Information needed for structural analysis	Analysis tab

1.2 Beam and column connections

This section introduces components that can be used in concrete connections.

Click the links below to find out more:

- Battering connection (13) on page 21
- Corbel connection (14) on page 22
- Concrete beam-beam (112) on page 32

Battering connection (13)

Use to fit the secondary part, for example, in a custom component, either square to the main part, or at an angle.

Use for

Situation	Description
	Secondary is fit square to the main part.
	Secondary is fit to the main part at an angle.

Situation	Description

Before you start Create two parts.

Defining properties To enter the angle and the distance between the parts, go to the **Picture** tab.

If the part angle makes the gap larger than the value **A**, then the part is cut at the end.

Selection order

1. Select the main part
2. Select the secondary part.

Corbel connection (14)

Corbel connection (14) connects a beam to a column using a straight, beveled, or rounded corbel and reinforcing bars or fastening plates.

Objects created

- Corbel
- Bearing plate
- Drainage holes in bearing plate (1 or 2) (optional)
- Reinforcing bars (1 or 2) (optional)
  - Bolt plates for reinforcing bars
  - Nuts for reinforcing bars
  - Recesses for bolt plates and nuts
- Fastening plates (2) (optional)
  - Recesses for fastening plates

Use for

Situation	Description
	Connects a beam to a column using a reinforcing bar and a beveled corbel.
	Connects a beam to a column using fastening plates and a rounded corbel.

Selection order

1. Select the main part (column).
2. Select the secondary part (beam).

   The connection is created automatically when the secondary part is selected.

Part identification key

	Part
1	Bolt plate
2	Nut
3	Bearing plate
4	Corbel
5	Reinforcing bar

See also

Corbel connection (14): Picture tab on page 24

Corbel connection (14): Parts tab on page 27

Corbel connection (14): Reinforcing bar tab on page 29

Corbel connection (14): Picture tab

Use the **Picture** tab to control the corbel shape and dimensions, beam end shape and dimensions, and corbel side chamfers in the **Corbel connection (14)** connection.

When beam sloped, adjust

If the beam in the connection is sloped, define whether the beam or the corbel is cut.

When the beam end is cut, the corbel top stays horizontal. When the corbel is cut, the corbel top has the same slope as the beam.

Corbel dimensions and shape

	Description
①	Define the gap between the column and the beam.
②	Define the gap between the corbel and the beam.
③	Define the vertical corbel dimension.
④	Define the beveled corbel dimension.
⑤	Define the corbel width.

Option	Description
	Default Beveled corbel AutoDefaults can change this option.
	Beveled corbel
	Square corbel
	Rounded corbel

Corbel position

	Description
①	Define the horizontal corbel offset.
②	Define the height of the beam cut.
③	Define the gap between the corbel and extended beam.

Beam end shape You can fit the beam end to the column, or you can create a straight beam end.

Option	Description	Example
	Default Straight beam end AutoDefaults can change this option.	
	Straight beam end	
	Beam end is fit to the column based on the dimensions you define. Beam can extend symmetrically on both sides of the column, or with different dimensions.	

Option	Description	Example

Corbel thickness

Define the distances from column edges to set the corbel thickness.

Corbel side chamfers

Define whether the corbel sides are chamfered. By default, no chamfers are created.

Corbel connection (14): Parts tab

Use the **Parts** tab to control the corbel part properties and dimensions in the **Corbel connection (14)** connection.

Corbel parts

Part	Description
Bearing plate	Define the bearing plate thickness.
Cast unit	Define whether cast unit is formed.

Part	Description
Drainage hole	Define whether drainage holes for each reinforcing bar are created in the bearing plate.
Bolt plate	Define the bolt plate thickness.
Nut	Define the nut thickness.

Option	Description	Default
Pos_No	Define a prefix and a start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab.
Material	Define the material grade.	The default material is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab, in the **Part material** field.
Name	Define a name that is shown in drawings and reports.	

Bearing plate dimensions

	Description
①	Define the bearing plate distance from the column edge.
②	Define the bearing plate distance from the corbel edges.

Nut and bolt plate dimensions

	Description
①	Define the nut width.
②	Define the bolt plate width.

Corbel connection (14): Reinforcing bar tab

Use the **Reinforcing bar** tab to control the reinforcing bar and fastening plate properties, and the bolt plate and fastening plate recesses in the **Corbel connection (14)** connection.

Reinforcing bar properties

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Bending radius	Define the internal radius of the bends in the bar.
Name	Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcements. For example, you can display different reinforcement classes in different colors.
Number of bars	Select **1 Dowel** to create one reinforcing bar. Select **2 Dowels** to create two reinforcing bars. Then define the distance between the bars in the **Bar distance** field.

Reinforcing bar length

	Description
①	Define the distance between reinforcing bar center line and the column edge.
②	Define the vertical length of the reinforcing bar inside the corbel.
③	Define the length of the reinforcing bar inside the column.

Recess for bolt plate and nut

	Description
① ⑤	Define the size of the bolt plate recess in the x direction. Define the size of the bolt plate recess in the y direction.
② ④	Define the size of the nut recess in the y direction. Define the size of the nut recess in the x direction.
③	Define the reinforcing bar offset.

Reinforcing bar extra length

	Description
①	Define the extra length of the reinforcing bar.
②	Define the length of the reinforcing bar inside the recess.

Connecting devices

Define the connecting devices that connect the beam and the column.

Option	Description
	Default One or two reinforcing bars bent to the same angle as the corbel bevel Available for bevel corbels. AutoDefaults can change this option.
	One or two reinforcing bars bent to the same angle as the corbel bevel Available for bevel corbels.
	One or two reinforcing bars Default for straight and rounded corbels.
	Two fastening plates Use custom components as fastening plates.

Recess for fastening plate

	Description
①	Define the depth of column recess.
③	Define the depth of beam recess.
②	Define the offset of column recess.
④	Define the offset of beam recess.
Column recess **Beam recess**	Define the height and width of column and beam recess.

Using custom components as fastening plates

You can use custom components as fastening plates. Use the **Column component** and **Beam component** sections to define the fastening plates in the column and beam.

1. Select the following option in the**Connecting devices** list:

2. In the **Custom** list, select **Yes**.

3. Click the **...** button next to the **Component** field to open the **Select component** dialog box.
4. Browse for the custom component you want to use as fastening plate.

   The component you select must be a custom part and have two or more input points.
5. Select the component and click **OK**.
6. To use saved custom component properties, enter the name of the saved properties file in the **Custom settings** field.
7. If the direction or rotation of the fastening plate is not correct, select another option in the **Up direction** or **Rotation** list.

Concrete beam-beam (112)

Concrete beam - beam (112) creates a connection between a concrete beam and one or two secondary concrete beams.

Objects created

- Neoprene
- Steel plates
- Anchor rods
- Tubes
- Sockets

Use for

Situation	Description
	Connection between two concrete beams.
	Connection between three concrete beams.

Selection order

1. Select the main part (beam).
2. Select one or two secondary parts (beam).
3. Click the middle mouse button to create the connection.

Part identification key

	Part
1	Beam
2	Beam

See also

Concrete beam-beam (112): Picture tab on page 34

Concrete beam-beam (112): Parts tab on page 44

Concrete beam-beam (112): Anchors tab on page 45

Concrete beam-beam (112): Parameters tab on page 47

Concrete beam-beam (112): Anchor rods tab on page 47

Concrete beam-beam (112): Socket tab on page 59

Concrete beam-beam (112): Picture tab

Use the **Picture** tab to control the part dimensions and the shapes of the parts, and the recesses in the **Concrete beam-beam (112)** connection.

Dimensions

	Description	Example
1	Define the gap between the main part and the secondary part on the left and the right side.	
2	Define the vertical gap between the main part and the secondary part.	

	Description	Example
3	Define the length of the neoprene.	
4	Define the gap between the neoprene and the secondary part on the left and the right side.	
5	Define the length of the steel plate.	

	Description	Example
6	Define the gap between the steel plate and the secondary part on the left and the right side.	

	Description	Example
1	Define the distance from the centerline of the secondary parts to the outer contour of the main part on the left side.	
2	Define the distance from the centerline of the secondary parts to the outer contour of the main part on the right side.	

	Description	Example
3	Define the distance from the edge of the first secondary part to the centerline.	
4	Define the distance from the edge of the second secondary part to the centerline.	

	Description	Example
5	Define the depth of the recess in the main part.	
6	Define the depth of the recess in the secondary part.	

	Description	Example
7	Define the width of the recess in the first secondary part.	
8	Define the width of the recess in the second secondary part.	

	Description	Example
9	Define the width of the neoprene.	
10	Define the distance from the edge of the neoprene to the edge of the main part.	
11	Define the distance from the edge of the neoprene to the edge of the main part.	

	Description	Example
12	Define the width of the steel plate.	
13	Define the distance from the edge of the steel plate to the edge of the main part.	

	Description	Example
14	Define the distance from the edge of the neoprene to the edge of the main part.	
15	Define the distance from the edge of the recess in the main part to the edge of the secondary part.	

Concrete beam-beam (112): Parts tab

Use the **Parts** tab to control the neoprene and steel plate properties and how these parts are connected in the **Concrete beam-beam (112)** connection.

Neoprene and steel plate

Option	Description
t, b, h	Define the neoprene or the steel plate thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.

Option	Description
Name	Define a name for the neoprene or the steel plate.
Class	Define the part class number for the neoprene or the steel plate.
Comment	Add a comment for the neoprene or the steel plate.

Neoprene properties

A neoprene plate for absorbing shock and for damping sound can be used between the main and the secondary parts.

Option	Description
Neoprene to	Define to which part the neoprene is connected. The neoprene can also be a loose part. In the second list, define how the neoprene is connected to the part.
Holes in Neoprene	Define how the holes in the neoprene are created.
Diameter of Holes in Neoprene	Define the diameter of the holes in the neoprene.
	Define whether the neoprene is created as one single plate or as two plates and the edge distance from the neoprene to the secondary parts.

Steel plate properties

One or two additional steel plates can be used between the main and the secondary parts.

Option	Description
Add steel plate to the primary by	Define how the steel plate is connected to the main part.
Negative volume around steel plate	Define whether there is a negative volume around the steel plate.

Concrete beam-beam (112): Anchors tab

Use the **Anchors** tab to control the properties of anchor rods, nuts, washers and the top and bottom tubes in the **Concrete beam-beam (112)** connection.

Anchor rods, Nut, Washer,

Anchor rods can be used between the main and the secondary parts. Define the length and number of rods on the **Anchor rods** tab.

Plate, Tube top, Tube bottom

Nuts and washers can be created. Define the height of nuts and washers on the **Anchor rods** tab.

Tubes can be used between the main and the secondary parts. Define the height of the tubular profiles on the **Anchor rods** tab.

Option	Description
t, b, h	Define the part thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.
Comment	Add a comment for the part.

Anchor rods

Anchors to	Define to which part the anchor rods are connected. The anchor rods can also be loose parts. In the second list, define how the anchor rods are connected to the part.
Anchors all the same L	Define whether the anchor rods have the same length.
Type of reinforcing bars	Define the type of reinforcing bars.
Bottom section	Define the type of profile in the bottom section.

Washer

Define the position of the washer handles.

Washer plate

Define the position of the plate handles.

Define whether the washers and nuts are welded to the plates.

Tube top and tube bottom

You can add a tubular embed for creating a round hole for the anchors. The tube top starts at the bottom level of the nut.

Tubes around anchors to beam	Define how to connect the tubes to the beam.
Tubes all the same L	Define whether the tubes have the same length.

Concrete beam-beam (112): Parameters tab

Use the **Parameters** tab to control the connecting options in the **Concrete beam-beam (112)** connection.

Anchors only in secondary	Select whether anchor rods are included only in the secondary parts.
Fit secondaries perpendicular to	Select how to fit the secondary parts.

Concrete beam-beam (112): Anchor rods tab

Use the **Anchor rods** tab to control the number, dimensions and position of the anchor rods and the dimensions of the recesses in the main and secondary parts in the **Concrete beam-beam (112)** connection.

Dimensions

	Description	Example
1	Define the size of the holes in the beam.	

	Description	Example
2	Define the size of the recesses in the beam in both directions.	
3	Define the type of the holes in the beam.	

	Description	Example
4	Define the distance between the top of the anchor rod and the top of the beam.	
5	Define the distance between the bottom of the recess and the top of the anchor rod.	

	Description	Example
6	Define the distance between the bottom of the recess in the main part and the bottom of the recess in the secondary part.	
7	Define the distance from the centerline of the secondary parts to the outer contour of the main part.	

	Description	Example
8	Define the width of the anchor rod hook.	

	Description	Example
9	Define the height of the anchor.	
10	Define the angle of the anchor rod hook.	

	Description	Example
11	Define the radius of the anchor rod hook in the main part in the vertical direction.	

	Description	Example
12	Define the anchor rod egde distances from the main part.	
13	Define the number of anchor rods.	
14	Define the anchor rod spacing. Use a space to separate anchor rod spacing values. Enter a value for each space between anchor rods. For example, if there are 3 anchor rod rows, enter 2 values.	

	Description	Example
15	Define the anchor rod egde distances from the secondary parts.	
16	Define the number of anchors rods.	
17	Define the anchor rod spacing. Use a space to separate anchor rod spacing values. Enter a value for each space between anchor rods. For example, if there are 3 anchor rod rows, enter 2 values.	

	Description	Example
1	Define the distance between the top side of the top tube and the top of the part.	
2	Define the height of the nut.	

	Description	Example
3	Define the height of the washer.	
4	Define the distance between the bottom side of the top tube and the bottom side of the recess.	
5	Define the distance between the top side of the bottom tube and the bottom side of the recess.	

	Description	Example
6	Define the distance between the bottom side of the bottom tube and the bottom side of the part.	

Recesses Define whether anchors and recesses are created.

Option	Description
	Anchors and recesses
	Anchors
	Recesses

Concrete beam-beam (112): Socket tab

Use the **Socket** tab to control the socket properties and how the sockets are connected. If you create the sockets on the **Socket** tab, the anchors on the **Anchors rods** tab are automatically considered as sockets in the **Concrete beam-beam (112)** connection.

Sockets

Option	Description
Create socket	Select whether sockets are created and which parts are included.

Option	Description
Connect socket to prim by	Select how the sockets are connected to the main part.
Connection rod-connector	Select how to connect the rod connector.
Type of reinforcing bars	Select the rod type.
Leg rotation	Select the direction of the sockets. In the second list, define the angle of the sockets.

Connecting profile, Rod

Option	Description
t, b, h	Define the connecting profile and the rod thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade for the connecting profile and the rod.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number for the connecting profile and the rod.
Comment	Add a comment for the connecting profile and the rod.

Partname component

Option	Description
Partname component	If you use a custom component to create the socket, browse for the component and use the list of options to position the custom component.

1.3 Panels and walls

This section introduces components that can be used in concrete panels and walls.

Click the links below to find out more:

- Electric box in wall (84) on page 60
- Sandwich And Double Wall (1) on page 75
- Sandwich Wall Horizontal Seam (1) on page 97
- Sandwich Wall Vertical Seam (1) on page 104
- Sandwich Wall Window (1) on page 109

Electric box in wall (84)

Electric box in wall (84) creates electric boxes in walls.

Objects created

- Electric boxes
- Tubes

Use for

Situation	Description
	Electric box in a wall with top connection.
	Electric box in a wall with bottom connection.

Situation	Description
	Electric box in a wall with top and bottom connections.

Selection order

1. Select a concrete panel.
2. Pick a position for the electric box.
3. Pick a position for the top connection.
4. Pick a position for the bottom connection.

   The electric box and connections are created automatically.

Part identification key

	Part
1	Electric box
2	Panel

See also

Electric box in wall (84): Picture tab on page 63

Electric box in wall (84): Parts tab on page 65

Electric box in wall (84): Electric box tab on page 67

Electric box in wall (84): Top conn / Bottom conn tab on page 70

Electric box in wall (84): UDA tab on page 72

Electric box in wall (84): Picture tab

Use the **Picture** tab to control the shape, number and position of the tubes and the position of the electric boxes in the **Electric box in wall (84)** modeling tool.

Tube direction

Option	Description
	Select the tube shape in the up direction.
	Select the tube shape in the down direction.

Option	Description
	Select the tube direction in relation to the start point (yellow) and end point (magenta) of the wall.
	Select the number of tubes in the up direction.
	Select the number of tubes in the down direction.

Tubes to concrete / Electric parts to concrete

Select how the tubes and the electric parts are connected to the concrete part.

Option	Description
As subassembly	Add as a subassembly to the concrete part.
By cast unit	Add to the concrete part.
Welds	Weld to the concrete part.
Not	Do not connect to the concrete part.

Tube and electric box positions

Option	Description
	Select the position of the up connection in the concrete part.
	Select the position of the electric box in the concrete part.
	Select the position of the down connection in the concrete part.

Electric box in wall (84): Parts tab

Use the **Parts** tab to control the tube properties at the top and bottom of the concrete part and the position of the tube reference points in the **Electric box in wall (84)** modeling tool.

Tube top / Tube bottom

Option	Description	Default
t, b, h	Define the tube thickness, width and height.	`D19`
Pos_No	Define a prefix and a start number for the part position number.	`ET 1`
Material	Define the material grade.	`PVC`
Name	Define a name for the tube.	`E-tube`
Class	Define the part class number for the tube.	`2`
Comment	Add a comment for the tube.	

1st reference point Define a reference point for the tube. The reference point determines the position of the part mark in a drawing.

Option	Description	Example
At top	Start point is positioned at the top of the tube.	
At bottom	Start point is positioned at the bottom of the tube.	
At longest segment	Start point is positioned at the longest segment of the tube.	

Option	Description	Example
At smallest segment	Start point is positioned at the smallest segment of the tube.	

Electric box in wall (84): Electric box tab

Use the **Electric box** tab to control the shape, number and dimensions of the electric boxes and to define a possible connection piece (**Tube a**) in the **Electric box in wall (84)** modeling tool.

Electric box / Tube a

Option	Description	Default
t, b, h	Define the electric box and the connection piece (**Tube a**) thickness, width and height.	`D19`
Pos_No	Define a prefix and a start number for the part position number.	`ET 1`
Material	Define the material grade.	`PVC`
Name	Define a name for the electric box and the connection piece.	`E-tube`
Class	Define the part class number for the electric box and the connection piece.	`2`
Comment	Add a comment for the electric box and the connection piece.	

Rotation Control the rotation of the electric box and the connection piece.

Option	Example
Front	
Top	
Back	
Below	

Electric box shape

Option	Description
Profiles / Custom part Example custom part: 	You can use a profile from the profile catalog or select a custom component from the component catalog. If you use a custom component, browse for the component and use the list of options to position the custom component.

Option	Description
	Select the shape of the electric box and define the dimensions for the electric box.
	Select an extra shape for creating several different types of electric boxes. This shape is visible in drawings and it indicates the front and back side of the electric box.
Number =	Define the number of electric boxes in horizontal or vertical direction.
1	Select the top connection pieces for the electric box.
2	Select the left side connection pieces for the electric box.
3	Select the right side connection pieces for the electric box.
4	Select the bottom connection pieces for the electric box.
	Define the dimensions of the connection pieces.

Option	Description
	Define the positions of the connection pieces.
	Options to add the electric box to the part. The options are: • Create electric box • Create electric box + cut electric box • Create electric box + cut bounding box

Electric box in wall (84): Top conn / Bottom conn tab

Use the **Top conn** and **Bottom conn** tabs to control the shape and dimensions of the top and bottom connection boxes, and to define a possible connection piece (**Tube a**) in the **Electric box in wall (84)** modeling tool.

Top box / Bottom box / Tube a

Option	Description
t, b, h	Define the top and bottom connection box and the connection piece (**Tube a**) thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name for the connection box and the connection piece.
Class	Define the part class number for the connection box and the connection piece.
Comment	Add a comment for the connection box and the connection piece.

Rotation Control the rotation of the connection piece (**Tube a**).

Connection box shape

Top conn	Bottom conn	Description
Profiles / Custom part		You can use a profile from the profile catalog or select a custom component from the component catalog. If you use a custom component, browse for the component and use the list of options to position the custom component.
		Select the shape of the connection box and define the dimensions for the box.
		Define the dimensions of the connection box.
		Define the position of the connection piece.

Top conn	Bottom conn	Description
		Options to add the connection box to the part. The options are: • Create connection box • Create connection box + cut connection box • Create connection box + cut bounding box
		Select the connection pieces for the connection box.
		Select the number of connection boxes.

Connect Define how to the connection box is connected.

Electric box in wall (84): UDA tab

Use the **UDA** tab to add information in the parts' user-defined attributes (UDAs) in the **Electric box in wall (84)** modeling tool.

Option	Description		
Part	Select to which part the related information can be saved: **Nothing** **Tube top** **Tube bottom** **Tubes** **Electric box** **Top connection** **Bottom connection** **All**		
UDA name	Enter the name of the user-defined attribute. For example, to add a comment UDA, open the `objects.inp` file in a text editor and search for `comment`. The following attribute is shown: `attribute("comment", "j_comment", string, "%s", no, none, "0.0", "0.0")` The first text between the quotation marks is the UDA name, `comment`. The entered name is case sensitive.	**UDA name:** • comment • fabricator • art_number • type	**UDA:** • Comment • Fabricator name • Article number • Type
Type	Type of the user-defined attribute.	Use **String** for text, **Integer** for numbers, **Float** for numbers with decimals and **Option** for selecting an item in a list. You can find the UDA type in the `objects.inp` file	
Value	Enter the value that is saved to the user-defined attribute.		

Example

Part	UDA name	Type	Value
Top conn	art_number	String	EB_12345
Electric box		String	
Electric box		String	
Electric box		String	
Electric box		String	
Electric box		String	
Electric box		String	
Electric box		String	
Electric box		String	
Electric box		String	

Owner multi user | CC UDA | Calculation | Precast formslab | MAP coordinates | Unitechnik UDA

Parameters | Status | End Conditions | Analysis | Userfields | IFC export | Numbering steel

Comment	
Comment 2 (affects numbering)	
Shorten	
Camber	
Preliminary mark	
Preliminary assembly mark	
Model number	
Locked	
Hold	
User Phase (affects numbering)	
Numbering Order	
Control Number (Do not modify)	
Control Number Status (Do not modify)	
Fabricator name	
Type	
Nomination	
Article number	EB_12345
Fixed drawing main view	
Screw number	
Drawing no. architect	
Drawing no. engineer	
Product code	
Product description	
Product weight	
Product unit	
Initial GUID	

Sandwich And Double Wall (1)

Sandwich And Double Wall (1) creates a precast concrete wall. The wall consists of an inner shell, foil, insulation, and outer shell.

Objects created

- Inner shell
- Foil (optional)
- Insulation (optional)
- Outer shell (optional)

Use for

Situation	Description
	Sandwich wall, split shells.

Situation	Description
	Sandwich wall, brick wall on outer shell as surface treatment.

Selection order

1. Pick the first point.
2. Pick the second point.

   The wall is created automatically when the second point is picked.

Part identification key

	Part
1	Inner shell
2	Foil
3	Insulation
4	Outer shell

See also Sandwich And Double Wall (1): Parts tab on page 77

Sandwich And Double Wall (1): Vertical section tab on page 82

Sandwich And Double Wall (1): Horizontal section tab on page 85

Sandwich And Double Wall (1): Insulation tab on page 91

Sandwich And Double Wall (1): Outside Shell tab on page 93

Sandwich And Double Wall (1): UDA tab on page 96

Sandwich And Double Wall (1): Parts tab

Use the **Parts** tab to control the dimensions and positioning of the sandwich wall inner shell, foil, insulation, and outer shell in the **Sandwich And Double Wall (1)** modeling tool. The inner shell is always created.

Numbering settings

Set **Numbering settings** to **Enabled** to show **Prefix** and **Start number** for all parts.

Inner shell

Option	Description	Default
Thickness	Define the thickness of the inner shell.	150 mm

Foil

Option	Description	Default
Thickness	Define the thickness of the foil.	1 mm
Create foil	Define whether the foil is created.	Yes and subassembly

Insulation

Option	Description	Default
Thickness	Define the thickness of the insulation.	50 mm
Create insulation	Define whether the insulation is created.	Yes and subassembly

Outer shell

Option	Description	Default
Thickness	Define the thickness of the outer shell.	1 mm
Create outer shell	Define whether the outer shell is created:	Yes + cast unit

Option	Description	Default
Pos_No	Define a prefix and a start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	The default part start number is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab.
Material	Define the material grade.	The default material is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab, in the **Part material** field.
Name	Define a name that is shown in drawings and reports.	
Class	Define the part class number.	

Positioning

You can use **Standard** or **Advanced** positioning. With advanced positioning, there are more options for sandwich wall width and foil position.

Sandwich wall width

Option: Standard	Description
	No gaps between the parts. Sandwich wall thickness is the sum of all created parts.
	Define the total wall width. A gap is created between the outer shell and the insulation.

Option: Advanced	Description
	Define the total wall width and the gap between the inner shell and the insulation. Another gap is created between the outer shell and the insulation.

Option: Advanced	Description
	Define the total wall width and the gap between the outer shell and the insulation. Another gap is created between the inner shell and the insulation.
	Define the total wall width and the gap between the inner shell and the insulation. No gap is created between the outer shell and the insulation.

Foil position

Option	Description
	By default, the foil is placed on the outer side of the inner shell.

Option	Description
	Define another foil position. Set **Positioning** to **Advanced**.

Point offset

Option	Description
	Define the offset from the outer face of the inner shell.
	Define the offset from the inner face of the inner shell.
	Define the offset from the center of the sandwich wall.

Option	Description
	Define the offset from the inner face of outer shell.
	Define the offset from the outer face of outer shell.

Parametric profile

Option	Description
Parametric section	Set to **Enabled** to define the prefix for the parametric profiles of the sandwich parts.
Prefix for the parametric profile	Select the prefix for the parametric profile: • `Position h*b` and `Position b*h` to create plate profiles without a prefix. If **Parametric section** is set to **Disabled**, the parts have an `h*b` prefix, for example, `2000*100`. • `Position PLh*b` and `Position PLb*h` to create plate profiles whose prefix starts with `PL`. • `Par. section h*b` and `Par. section b*h` require you to enter a parametric plate profile prefix.
Par. section prefix	Define the prefix of the parametric profile.

Sandwich And Double Wall (1): Vertical section tab

Use the **Vertical section** tab to control the vertical level part properties in the **Sandwich And Double Wall (1)** modeling tool. You can define both the top and the bottom level. The levels can be set as absolute levels, or vertically relative (displaced) to the points.

Top level

Option	Description
n1, n2, n3, n4	Define the absolute top level height.
h1, h2, h3, h4	Define the part height from the bottom face.

Option	Description
d2, d3, d4	Define the vertical displacement from the adjacent part.

When you use relative levels, the levels of all other parts depend on the inner shell level.

Bottom level

Option	Description
m1, m2, m3, m4	Define the absolute bottom level height.
a1, a2, a3, a4	Define the bottom offset from the point.
b2, b3, b4	Define the vertical displacement from the adjacent part.

When you use relative levels, the levels of all other parts depend on the inner shell level.

Thicken innershell

You can define a thickening for both the top and the bottom side of the inner shell.

Option	Description	Example
	Inner shell is not thickened.	
	Inner shell is thickened towards the outer face of the insulation. Define the insulation height to prevent collisions.	
	Inner shell is thickened to the inner face of the outer shell.	

Option	Description	Example
	Inner shell is thickened to the outer face of the outer shell. Define the outer shell height to prevent collisions.	

Option	Description	Example
Thickness	Define the inner shell thickness. If you do not enter a value, the inner shell thickness defined on the **Parts** tab is used.	
Thickening parts	Select how the top and bottom thickening parts are added to the inner shell.	
Offset options	Select the offset options for the thickening parts. Offset can be defined for both wall ends. 	

Sandwich And Double Wall (1): Horizontal section tab

Use the **Horizontal section** tab to define the wall ends and to control how the wall parts are split in the horizontal direction in the **Sandwich And Double Wall (1)** modeling tool.

Wall end The wall end options are the same for both ends.

First wall end	Second wall end

Wall end offset

Option	Description	Example
	Inner shell end is fixed. Define the offset for the ends of the foil, insulation, and outer shell.	
	Insulation end is fixed. Define the offset for the ends of the inner shell, foil, and outer shell.	
	Outer shell end is fixed. Define the offset for the ends of the inner shell, foil, and insulation.	

Option	Description	Example
	Corner shape to the inside direction. For the outer shell angle, define the offset from the inner face of the inner shell. Insulation is along the inner shell.	
	Corner shape to the inside direction. Define the length of the outer shell angle.	
	Corner shape to the inside direction. For the outer shell angle, define the offset from the inner face of the inner shell. Insulation is along the outer shell.	
	Corner shape to the outside direction. Define the horizontal offset for the outer shell. There is no gap between the insulation and the inner shell angle.	
	Corner shape to the outside direction. Define the horizontal offset for the outer shell. For the inner shell angle, define the offset from the outer face of the inner shell.	

Option	Description	Example
	Corner shape to the outside direction. Define the horizontal offset for the outer shell. For the inner shell angle, define the length.	
	Angled shells are not created. Define horizontal offset for the inner shell, insulation, and outer shell.	

Corner shape

Option	Description
	Select the inner shell corner shape.
	Select the outer shell corner shape.

Option	Description
	Select the insulation corner shape.
Add corner to main shell	Add the corner to the shell. The options are: • `Add to 4 Cast Unit` (4 = outer shell) • `Add to 4 Sub assembly` • `Add to 1 Cast Unit` (1 = inner shell) • `Add to 1 Sub assembly`

Vertical chamfer

Option	Description
	Define whether a vertical chamfer is created. Define the horizontal and vertical chamfer dimensions, and the part prefix and the start number.

Horizontal offset

To use the options, set the **Split Front view** option to **Enabled**.

Option	Description
	Define the horizontal offset for the foil, insulation, and outer shell. The main gap between the inner shells is defined in the **Split Front view** section of the tab.

Option	Description
	Define the horizontal offset for the inner shell, foil, and outer shell. The main gap between the insulation parts is defined in the **Split Front view** section of the tab.
	Define the horizontal offset for the inner shell, foil, and insulation. The main gap between outer shells is defined in the **Split Front view** section of the tab.
	Define a 90-degree angle for the inner shell and the vertical offset towards the angled shell from the inner face of the outer shell. The main gap between the inner shells is defined in the **Split Front view** section of the tab. Additional options for the 90-degree angle: • Define the corner shapes for the inner shell. • Define how the corner is added to the inner shell. • Define the part prefix and the start number.

Mirroring

Option	Description
	Define whether the sandwich wall is mirrored.

Splitting parts To split wall parts, set the **Split Front view** option to **Enabled**.

Option	Description
	Parts are not split. Define the horizontal offset for both wall ends using the points as reference points.

Option	Description
	Parts are split. Define the section width and the number of sections. First end offset is the reference for the first section width at the first wall end. Second end offset is the reference for the last section width at the second wall end.
	Parts are split. Define the section width and the number of sections. First end offset is the reference for the first section width at the first wall end. Point (2) is the reference of the last section width at the second wall end.
	Parts are split. Define the section width and enter the number of sections. Point (1) is the reference of the start section at the first wall end. Second end offset is the reference for the last section width at the second wall end.
	Parts are split. Define the section width and enter the number of sections. Points (1, 2) are the references for the section width at both wall ends.

Option	Description
	Define the horizontal direction.

Sandwich And Double Wall (1): Insulation tab

Use the **Insulation** tab to control the vertical and/or horizontal splitting of the insulation in the **Sandwich And Double Wall (1)** modeling tool.

Dimensions

Option	Description
1	Define whether the insulation is split: • Insulation is not split. • Insulation is split in the horizontal direction. • Insulation is split in the vertical direction. • Insulation is split in the horizontal and vertical direction.

Option	Description
2	Define the number and width of the middle sections.
3	Define the gap between the sections.
4	Define the width of the first and last section in the horizontal direction.
5	Define the height of the top and bottom section in the vertical direction.
6	Define which wall end is used for distributing the insulation sections.
Point direction	Define the direction of the points. The options are horizontal or vertical. This option affects the representation of workshop drawings.

Sandwich And Double Wall (1): Outside Shell tab

Use the **Outside shell** tab to control the vertical and/or horizontal splitting of the outer shell in the **Sandwich And Double Wall (1)** modeling tool. You can also define chamfers and add a brick wall surface.

Dimensions

Option	Description
	Define the direction for the splitting.

	Description
1	Define whether the outer shell is split: • Outer shell is not split. • Outer shell is split in the horizontal direction. • Outer shell is split in the vertical direction. • Outer shell is split in the horizontal and vertical direction.
2	Define the number and width of the middle sections.
3	Define the gap between the sections.
4	Define the width of the first and last section in the horizontal direction.
5	Define the height of the top and bottom section in the vertical direction.

Chamfering

Option	Description
Vertical chamfers	Define the vertical chamfers. Select a chamfer shape from the list and define the chamfer dimensions.

Option	Description
Horizontal chamfers	Define the horizontal chamfers. Select a chamfer shape from the list and define the chamfer dimensions.
Chamfer side	Define the sides where the chamfers are created. For example:

Brick wall properties

Set the **Brick wall** option to **Enabled** to create a brick wall.

Option	Description	Example
 	Select the brick wall type for the outer shell and define the brick dimensions, mortar height and width.	

Option	Description	Example
Wall edges	Define whether the wall edges are created as loose parts or whether they are added into an assembly.	

Sandwich And Double Wall (1): UDA tab

Use the **UDA** tab to define user-defined attributes for the inner shell, foil, insulation, and outer shell in the **Sandwich And Double Wall (1)** modeling tool. You can define multiple UDAs for each part. UDAs can be used as filters, and they can be displayed in drawings and reports.

User-defined attributes

For example:

Sandwich Wall Horizontal Seam (1)

Sandwich Wall Horizontal Seam (1) creates a horizontal seam between two sandwich walls. Seam dimensions and rabbets can be defined for all layers: inner shell, foil, insulation, and outer shell. In addition, you can define an extra foil layer.

Objects created

- Seams

- Rabbets
- Foil
- Insulation
- Extra foil layer

Use for

Situation	Description
	Seams between sandwich wall shells.
	Seams with additional rabbets.

Before you start

To be able to select the needed parts, activate the **Select objects in components** switch.

Selection order

1. Select the inner shell of the first sandwich wall.
2. Select the inner shell of the second sandwich wall.

   The seam is created automatically when the secondary part is selected.

Part identification key

	Part
1	Concrete part (wall, column, beam, slab)
2	Point Multiple points can be picked.

See also

Sandwich Wall Horizontal Seam (1): Picture tab on page 99

Sandwich Wall Horizontal Seam (1): Rabbets tab on page 101

Sandwich Wall Horizontal Seam (1): Extra foils tab on page 102

Sandwich Wall Horizontal Seam (1): Picture tab

Use the **Picture** tab to control the horizontal seam properties in the **Sandwich Wall Horizontal Seam (1)** connection.

Seam position

Seams can be defined for inner shells, insulation and outer shells. Draft angles can be defined for the outer shells.

Define the seam position between the sandwich wall shells. The middle line between the walls is used as a reference when the seam position is calculated.

Example:

Shell classes

Define the part classes to be applied to the seams. By default, the **Sandwich Wall Horizontal Seam (1)** component creates inner shells with class 1, foil with class 2, insulation with class 3, and outer shells with class 4. Use these classes to apply seams.

To prevent the creation of seams between foils, insulation shells and outer shells, leave the **Insulation classes** box empty.

Rabbets in insulation

You can create rabbets in the insulation layer.

Option	Description
	No rabbet for the insulation layer.

Option	Description
	One side rabbet for the top and bottom of the insulation layer. Rabbet depth and width can be defined.
	Mirrored one side rabbet for the top and bottom of the insulation layer. Rabbet depth and width can be defined.
	Two sided rabbets for the top and bottom of the inlulation layer. Rabbet depth and width can be defined.

Offset You can define offset for the seam.

Sandwich Wall Horizontal Seam (1): Rabbets tab

Use the **Rabbets** tab to define rabbets in inner shells, insulation layers and outer shells in the **Sandwich Wall Horizontal Seam (1)** connection.

Rabbet side Select to which side of the sandwich wall the rabbets are applied. You can define the rabbets separately for both sides.

The options are:

- **Top side**
- **Bottom side**
- **Both sides**

Sandwich Wall Horizontal Seam (1): Extra foils tab

Use the **Extra foils** tab to define an extra foil layer in the seam in the **Sandwich Wall Horizontal Seam (1)** connection.

Foil

Option	Description	Default
t	Define the extra foil thickness.	`1 mm`
Pos_No	Define a prefix and a start number for the part position number.	The default part start number is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab.
Material	Define the material grade.	The default material is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab, in the **Part material** field.
Name	Define the name for the extra foil.	`FOIL`
Class	Define the part class number for the extra foil layer.	
Comment	Add a comment for the extra foil layer.	

Connection method

Option	Description
No connection	Select how the extra foil is attached to the sandwich wall. The options are: • **No connection** (foil is a loose part) • **Weld** • **Cast unit** • **As subassembly**

Option	Description
At Inside shell	Select the profile to which the extra foil is connected. The options are: • **At inside shell** • **At outside shell**
Position h*b	Select a prefix for the foil layer. • **Position h*b** and **Position b*h** create a profile without a prefix. • **Position PLh*b** and **Position PLb*H** create a profile which start with **PL** prefix. • **Par. section h*b** and **Par. section b*h**: enter a prefix in the box below.
Auto	Select the rotation of the foil layer. The options are: • **Auto** • **Below** • **Front** • **Top** • **Back**
Foil shape	Define shape of the extra foil.

Sandwich Wall Vertical Seam (1)

Sandwich Wall Vertical Seam (1) creates a vertical seam between two sandwich walls. Seam dimensions and rabbets can be defined for all layers: inner shell, foil, insulation, and outer shell.

Objects created

- Seam
- Rabbets
- Foil
- Insulation

Use for

Situation	Description
	Seams between sandwich wall shells.
	Seams with additional rabbets.

Limitations

The component works only if the sandwich walls are parallel.

Before you start

To be able to select the needed parts, activate the **Select objects in components** switch.

Selection order

1. Select the inner shell of the first sandwich wall.
2. Select the inner shell of the second sandwich wall.

   The seam is created automatically when the secondary part is selected.

Part identification key

	Part
1	Inner shell of the first sandwich wall
2	Inner shell of the second sandwich wall

See also Sandwich Wall Vertical Seam (1): Picture tab on page 105

Sandwich Wall Vertical Seam (1): Picture tab on page 105

Sandwich Wall Vertical Seam (1): Picture tab

Use the **Picture** tab to control the vertical seam properties in the **Sandwich Wall Vertical Seam (1)** connection.

Seam position

Seams can be defined for inner shells, insulation and outer shells. Draft angles can be defined for the outer shells.

Define the seam position between the sandwich wall shells. The middle line between the walls is used as a reference when the seam position is calculated.

Example:

Shell classes

Define the part classes to be applied to the seams. By default, the **Sandwich Wall Vertical Seam (1)** component creates inner shells with class 1, foil with class 2, insulation with class 3 and outer shells with class 4. Use these classes to apply seams.

For example, to prevent the creation of seams between foils, insulation shells and outer shells, leave the **Insulation classes** box empty.

Asymmetrical seams

You can create asymmetrical seams and/or rabbets in the insulation layer.

Option	Description
	Symmetrical seams. No rabbet for the insulation layer.

Option	Description
	Symmetrical seams. Rabbet depth can be defined for the insulation layer.
	Asymmetrical seams for the insulation layer and outer shell. No rabbet for the insulation layer.
	Asymmetrical seams for the insulation layer and outer shell. Rabbet depth can be defined for the insulation layer.

Offset / Mirror

You can define offset for the seam.

Sandwich Wall Vertical Seam (1): Rabbets tab

Use the **Rabbets** tab to define rabbets in inner shells, insulation layers and outer shells in the **Sandwich Wall Vertical Seam (1)** connection.

Rabbet side Select to which side of the sandwich wall the rabbets are applied.

- **Right side**

  Rabbets are equal on both sides, but the rabbet dimensions on the right side are applied to both sides.

- **Left side**

  Rabbets are equal on both sides, but the rabbet dimensions on the left side are applied to both sides.

- **Both sides**

  Rabbet dimensions can be defined separately for both sides.

- **Both sides mirrored**

  Rabbet dimensions can be defined separately for both sides, but sides are mirrored.

Rabbet position Typically the vertical seams go from the top to the bottom of the sandwich wall. Define the seam offsets for the top and bottom sides. Offsets are applied only to the inner and outer shells.

Sandwich Wall Window (1)

Sandwich Wall Window (1) creates a rectangular opening in a sandwich wall. The opening is created through up to four parts (inner shell, foil, insulation, and outer shell). You can select whether the foil is created. A frame can be created, as well as extra foils and additional parts. The frame can be a wooden frame or a concrete border, added to the inner shell.

Objects created

- Rectangular opening
- Wooden frame or concrete border
- Extra foils
- Additional parts

Use for

Situation	Description
	Opening in a sandwich wall, with a wooden frame and foil layers.
	Opening in a sandwich wall, with a concrete border in the inner shell.

Limitations

You cannot use the Tekla Structures version 20.0 **Sandwich Wall Window (1)** modeling tool to modify sandwich wall windows that have been created with earlier Tekla Structures versions. If you need to modify such a window, you need to recreate it.

Selection order

1. Pick the first position.
2. Pick the second position.

   The opening is created automatically when the second position is picked.

Part identification key

	Part
1	Sandwich wall element
2	First picked position
3	Optional: second picked position

See also

Sandwich Wall Window (1): Picture tab on page 111

Sandwich Wall Window (1): Bottom detail tab / Left detail tab / Right detail tab on page 113

Sandwich Wall Window (1): Top detail tab on page 116

Sandwich Wall Window (1): Extra foils tab on page 119

Sandwich Wall Window (1): Additional parts tab on page 121

Sandwich Wall Window (1): Connections tab on page 124

Sandwich Wall Window (1): Picture tab

Use the **Picture** tab to control the dimensions and parts of the opening in the **Sandwich Wall Window (1)** modeling tool.

Dimensions

Option	Description
1	Set the points for the opening: • **2 points** Pick two points to create the opening (points 2 and 3 in the image). • **Point 1 B H** Pick one point to create the opening (point 2 in the image). This point is the opening reference point. Define the width and height of the opening. • **Point 2 B H** Pick one point to create the opening (point 3 in the image). This point is the opening reference point. Define the width and height of the opening. • **2 Points H** Pick two points to create the opening (points 2 and 3 in the image). Define the height of the opening.
2	Define the vertical offset from the insertion point.
3	Define the horizontal offset from the insertion point.
4	Define the width of the opening.
5	Define the height of the opening.

Select other layers

Option	Description
Insulation, **Wall**	Define the parts where the opening is created. If you do not enter any numbers, the opening is only created to the part you selected when applying the component.
Foil	To create the foil, select **Yes** in the **Foil** option. By default, the foil is not created. The frame shape options shown on the **Bottom detail**, **Left detail**, **Right detail**, and **Top detail** tabs depend on whether the foil is created not.

Sandwich Wall Window (1): Bottom detail tab / Left detail tab / Right detail tab

Use the **Bottom detail** tab, **Left detail** tab, and **Right detail** tab to control the size, position, number, and shape of the opening bottom, left side, and right side in the **Sandwich Wall Window (1)** modeling tool.

Frame

Part	Description	Default
Frame	Define the frame profile by selecting the profile from the profile catalog.	`50*50`

Option	Description	Default
Pos_No	Define a prefix and a start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	Prefix:`empty`, Start number: `1` The default part start number is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab.
Material	Define the material grade.	The default material is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab, in the **Part material** field.
Name	Define a name that is shown in drawings and reports.	
Class	Define the part class number.	

Frame shape

The options for inner and outer frame are the same regardless of whether the foil is created or not.

Inner	Outer	Description
		Default Cut or extension is not created. AutoDefaults can change this option. Any of the three options is used depending on the frame size.
		Cut or extension is not created.
		Extension is created. If the frame is narrower than the insulation, the concrete part is extended to fill the gap.
		Cut is created. If the frame is wider than the insulation, the concrete part is cut so that the frame fits.

Middle	
	Foil is not created. Cut is not created.
	Foil is not created. Frame is not created.
	Foil is created. Cut is not created.
	Foil is created. Cut is created.
	Foil is created. Frame is not created.

Additional recess Define the vertical and horizontal dimensions for the additional recess. By default, the recess is not created.

Frame corner cuts

Option	Description
	Frame corners are not cut.
	Define the horizontal and vertical dimensions of the frame corner cuts.

Sandwich Wall Window (1): Top detail tab

Use the **Top detail** tab to control the size, position, number, and shape of the opening top side in the **Sandwich Wall Window (1)** modeling tool.

Part

Part	Description	Default
Frame	Define the frame profile by selecting the profile from the profile catalog.	`50*50`
Foil2	Define the thickness of the foil.	

Option	Description	Default
Pos_No	Define a prefix and a start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	Prefix: `empty`, Start number: `1` The default part start number is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab.
Material	Define the material grade.	The default material is defined on the **Tools** --> **Options** -->

Option	Description	Default
		Options... --> **Components** tab, in the **Part material** field.
Name	Define a name that is shown in drawings and reports.	
Class	Define the part class number.	

Top frame shape The options for inner and outer frame are the same regardless of whether the foil is created or not.

Inner	Outer	Description
		Default Cut or extension is not created. AutoDefaults can change this option. Any of the three options is used depending on the frame size.
		Cut or extension is not created.
		Extension is created. If the frame is narrower than the insulation, the concrete part is extended to fill the gap.
		Cut is created. If the frame is wider than the insulation, the concrete part is cut so that the frame fits.

Middle	Description
	Foil is not created. Cut is not created.
	Foil is not created. Frame is not created.
	Foil is created. Cut is created.
	Foil is created. Frame is not created.

Additional recess Define the vertical and horizontal dimensions for the additional recess. By default, the recess is not created.

Frame corner cuts

Option	Description
	Frame corners are not cut.
	Define the horizontal and vertical dimensions of the frame corner cuts.

Drip mold

Option	Description
	Drip mold is not created.
	Drip mold is created. Define the mold dimensions.
	Drip mold is created. Define the mold dimensions. The drip mold can be created to any of the shape options. For example, if the shape has a bevel, the drip mold is created at the same angle as the bevel.

Sandwich Wall Window (1): Extra foils tab

Use the **Extra foils** tab to control the creation of extra foils on the top, bottom, right side, and left side of the opening in the **Sandwich Wall Window (1)** modeling tool.

Extra foil

Part	Description	Default
Extra foil	Define the thickness of the foil.	50*50

Option	Description	Default
Pos_No	Define a prefix and a start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	Prefix: `empty`, Start number: `1` The default part start number is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab.
Material	Define the material grade.	The default material is defined on the **Tools** --> **Options** --> **Options...** --> **Components** tab, in the **Part material** field.
Name	Define a name that is shown in drawings and reports.	
Class	Define the part class number.	

Extra foil dimensions

	Description	Default
1	Define whether extra foils are created.	
2	Define the inside offset.	`2* screw diameter`
3	Define the foil width.	`2* screw diameter`

Shells to cut

Option	Description
	All shells are cut.
	The inner shell is not cut.

Option	Description
	The inner shell and the foil are not cut.
	The inner shell and the foil are not cut. An insulation extension is created. Define the width and the material grade of the insulation extension.
	The outer shell is cut.

Sandwich Wall Window (1): Additional parts tab

Use the **Additional parts** tab to control the size, position, number, and alignment of additional parts in the **Sandwich Wall Window (1)** modeling tool.

Parts Select whether additional parts, custom component parts, or no additional parts are created.

Part	Description	Default
Parts	Define the profile of the additional parts by selecting the profile from the profile catalog.	`50*50`
Custom component parts	Define the custom component parts by selecting the component from the component catalog. You can also use a configuration file to define the properties.	

Option	Description	Default
Pos_No	Define a prefix and a start number for the part position number. Some components have a second row of fields where you can enter the assembly position number.	Prefix: `empty`, Start number: `1`
Material	Define the material grade.	
Name	Define a name that is shown in drawings and reports.	
Class	Define the part class number.	

Define distances Select whether to define the distances between the parts by entering the distance dimensions on this tab or by using an external text file. You can use an external file to automate the creation of additional parts when window sizes differ.

The name of the external file is `SandwichPanelWindowAdditionalParts.dat`. The file is located in the `..\Environments\Common\system folder`.

The image below shows an example of a local standard for creating additional parts. M is a module that is 100 mm.

The `SandwichPanelWindowAdditionalParts.dat` configuration file that contains the needed specifications for the example above is as follows:

```
BOTTOM;
0;599;1;1
600;1200;1;2;145
1201;1800;1;3;145
TOP;
1201;1800;1;1
LEFT;
0;599;1;1
600;1200;1;2;145
1201;2100;1;3;145
RIGHT;
0;599;1;1
600;1200;1;2;145
1201;2100;1;3;145
```

The keywords `BOTTOM`, `TOP`, `LEFT` and `RIGHT` define the side of the window the settings are applied to.

The syntax in the file is as follows: `bmin;bmax;cm;nd;dmax;d1;d2;d3`.

`bmin`	Smallest window dimension to apply the setting.
`bmax`	Biggest window dimension to apply the setting.
`cm`	Creation method:

	• 1 means use the number of parts for nd. • 2 means use maximum spacing for nd.
nd	Depending on the creation method (cm): • Number of parts, when cm is 1. • Maximum allowed distance between parts, when cm is 2.
d1	Distance from the start point to the first additional part (optional).
d2	Distance from the first additional part to the second (optional).
d3	Distance from the second additional part to the third (optional).

The additional parts are placed symmetrically, so distances d1, d2 and d3 are also used from the end point. If the optional distances d1, d2 and d3 are not defined, the parts are placed at equal distances.

In the image above 0;599;1;1 means that when the window width is equal or bigger than 0, but equal or less than 599, the creation method is by number of parts and 1 part should be created.

600;1200;2;300;145 would mean that when the window width is equal or bigger than 600, but equal or less than 1200, the creation method is use maximum spacing. The maximum allowed spacing between additional parts is 300 mm. The first part (and last part) distance is 145 mm. If the window width is 1200, three more parts are needed (1200 - 2x145)/300=3.03. The spacing between the additional parts will then be (1200 - 2x145)/4 = 227.5 mm.

Distance dimensions

	Description
1	Define the horizontal distance from the first additional part to the foil edge.
2	Define the vertical distance from the first additional part to the foil edge.
3	Select the additional part distribution.
4	Define the maximum distance between the additional parts, or the number of additional parts.

Placement Select whether additional part placement is the same or different on each side.

Option	Description
	Define the placement dimensions for the bottom, left side, and right side. If additional part placement is the same on each side, define the bottom dimensions only.
	Define the placement dimensions for the top side.

Sandwich Wall Window (1): Connections tab

Use the **Connections** tab to control how the wooden frame, additional parts, extra foil, and extensions are connected to the shells in the **Sandwich Wall Window (1)** modeling tool.

Add

Option	Description
Wooden frame to, **Additional parts**, **Extra foil to**	Select to which part the wooden frame, additional parts, and extra foil are connected. The **Extra foil to** option is shown if you have selected to create the foil on the **Picture** tab.

Connect extensions to their shells

Option	Description
Inside shell extension, **Outside shell extension**, **Insulation shell extension**, **Foil extension**	Select how the extension parts are connected to their main parts. The **Foil extension** option is shown if you have selected to create the foil on the **Picture** tab.

1.4 Formwork components

This section introduces formwork components.

Click the links below to find out more:

- Beam Formwork on page 125
- Column Formwork on page 128
- Wall Formwork on page 132
- General Beam Form on page 136
- General Wall Form on page 138
- General Clamp on page 141
- General Clip on page 142
- General Tie on page 144
- General Tie and Clips on page 146

Beam Formwork

Beam Formwork creates a timber formwork around a beam.

Availability

To use the component, go to Tekla Warehouse, locate the **Formwork components** and click **Insert into model**.

The components are then available in the **Component Catalog**.

Objects created

- Timber formwork (bottom and side plywood panels, supports, walers)
- Ties
- Clips

Use for

Situation	Description
	Beam formwork

Limitations

Beam Formwork works only with rectangular beams.

Before you start

Create a concrete beam.

Selection order

1. Select the beam.

The beam formwork is created automatically.

Part identification key

	Part
1	Bottom and side plywood panels
2	Bottom and side supports
3	Waler
4	Tie and clip

See also Beam Formwork: Beam tab on page 126

Beam Formwork: Parameters tab on page 127

Beam Formwork: Beam tab

Use the **Beam** tab to control the dimensions, spacings and part properties of the plywood panels and supports in the **Beam Formwork** detail.

Dimensions

	Description
①	Define the spacing between side supports.
②	Define the beam form length.
③	Define the distance to first bottom support from the form edge.
④	Define the spacing between bottom supports.
⑤	Define the bottom support length.

Plywood panel and support properties

Option	Description
Bottom Profile	Define the thickness, width and height for the bottom plywood panel.
Side Profile	Define the thickness, width and height for the side plywood panel.
Bottom Support	Define the thickness, width and height for the bottom support.
Side Support	Define the thickness, width and height for the side support.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.

Beam formwork properties

Option	Description
Beam	Define a name, prefix and start number for the beam formwork cast unit.

Beam Formwork: Parameters tab

Use the **Parameters** tab to control the waler properties and the dimensions and spacings of the clips and walers in the **Beam Formwork** detail.

Dimensions

	Description
1	Define the waler extension length.
2	Define the number of walers and clips in vertical direction.
3	Define the vertical spacing between clips.
4	Define the vertical distance to first clip from the form edge.
5	Define the horizontal distance to first clip from the form edge.
6	Define the horizontal spacing between clips.

Waler properties

Option	Description
Waler Profile	Define the thickness, width and height for the walers.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.

Tie and clip properties

Option	Description
Tie Component Name	Define the component that you want to use as tie by selecting it from the component catalog.
Tie Attribute File	Select an attribute file for the component.
Clip Component Name	Define the component that you want to use as clip by selecting it from the component catalog.
Clip Attribute File	Select an attribute file for the component.

Column Formwork

Column Formwork creates a timber formwork around a column.

Availability

To use the component, go to Tekla Warehouse, locate the **Formwork components** and click **Insert into model**.

The components are then available in the **Component Catalog**.

Objects created

- Timber formwork (plywood panels and supports on sides)
- Clamps

Use for

Situation	Description
	Column formwork

Limitations

Column Formwork works only with rectangular columns.

Before you start

Create a concrete column.

Selection order

1. Select the column.

The column formwork is created automatically.

Part identification key

	Part
1	Plywood panel
2	Support
3	Clamp

See also Column Formwork: Column tab on page 130

Column Formwork: Column tab

Use the **Column** tab to control the properties of clamps, plywood panels and supports in the **Column Formwork** detail.

Dimensions

	Description
1	Define the number of clamps.
2	Define the spacing between clamps.
3	Define the distance to first clamp from the bottom.

Clamp properties

Option	Description
Clamp Name	Define the component that you want to use as clamp by selecting it from the component catalog.
Attribute File	Select an attribute file for the component.

Plywood panel and support properties

Option	Description
Plywood Profile	Define the thickness, width and height for the plywood panels.
Side Profile	Define the thickness, width and height for the side supports.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.

Column formwork properties

Option	Description
Panel	Define a name, prefix and start number for the column formwork cast unit.

Wall Formwork

Wall Formwork creates a timber formwork around a panel.

Availability

To use the component, go to Tekla Warehouse, locate the **Formwork components** and click **Insert into model**.

The components are then available in the **Component Catalog**.

Objects created

- Timber formwork (plywood panels, studs, frame and walers)
- Ties
- Clips

Use for

Situation	Description
	Wall formwork

Limitations

Wall Formwork works only with single panels.

Before you start

Create a concrete panel.

Selection order

1. Select the panel.

The wall formwork is created automatically.

Part identification key

	Part
1	Stud
2	Walers
3	Plywood panel
4	Tie and clip
5	Frame

See also Wall Formwork: Panel tab on page 133

Wall Formwork: Parameters tab on page 135

Wall Formwork: Panel tab

Use the **Panel** tab to control the properties of studs, frame and plywood panels in the **Wall Formwork** detail. Define the dimensions of wall form, studs and frame.

Dimensions

	Description
1	Define the distance to first stud from the form edge.
2	Define the spacing between studs.
3	Define the width of edge lap.
4	Define the height of wall form.
5	Define the length of wall form.

Frame, stud and plywood panel properties

Option	Description
Edge Profile	Define the thickness, width and height for the frames.
Stud Profile	Define the thickness, width and height for the studs.
Plywood Profile	Define the thickness, width and height for the plywood panels.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.

Wall formwork properties

Option	Description
Panel	Define a name, prefix and start number for the wall formwork cast unit.

Wall Formwork: Parameters tab

Use the **Parameters** tab to control the waler profile and the dimensions and spacings of the clips and walers in the **Wall Formwork** detail.

Dimensions

	Description
1	Define the horizontal distance to first clip from the form edge.
2	Define the horizontal spacing between clips.
3	Define the length of waler extension.
4	Define the number of walers and clips in vertical direction.
5	Define the vertical spacing between clips.
6	Define the vertical distance to first clip from the form edge.

Waler properties

Option	Description
Waler Profile	Define the thickness, width and height for the walers.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.

Option	Description
Class	Define the part class number.

Tie and clip properties

Option	Description
Clip Component Name	Define the component that you want to use as clip by selecting it from the component catalog.
Clip Component File	Select an attribute file for the component.
Tie Component Name	Define the component that you want to use as tie by selecting it from the component catalog.
Tie Component File	Select an attribute file for the component.

General Beam Form

General Beam Form creates plywood panels and supports. You can use **General Beam Form** as a stand-alone component or as a subcomponent for the **Beam Formwork** detail.

Availability

To use the component, go to Tekla Warehouse, locate the **Formwork components** and click **Insert into model**.

The components are then available in the **Component Catalog**.

Objects created

- Plywood panels on sides and bottom
- Supports on sides and bottom

Use for

Situation	Description
	Beam form in a beam formwork.

Limitations

General Beam Form works only with rectangular beams.

Selection order

1. Pick the start point.
2. Pick the end point.

The beam form is created automatically.

Part identification key

	Part
1	Side support
2	Side plywood panel
3	Bottom plywood panel
4	Bottom support

See also Beam Form: Parameters tab on page 137

Beam Form: Parameters tab

Use the **Parameters** tab to control the dimensions and properties of beam form in the **Beam Form** modeling tool.

Dimensions

	Description
①	Define the spacing between side supports.

	Description
②	Define the spacing between side plywood panels.
③	Define the height of side plywood panels.
④	Define the distance to first bottom support from the form edge.
⑤	Define the spacing between bottom supports.
⑥	Define the length of bottom support.

Plywood panel and support properties

Option	Description
Bottom Profile	Define the thickness, width and height for the bottom plywood panel.
Side Profile	Define the thickness, width and height for the side plywood panel.
Bottom Support	Define the thickness, width and height for the bottom support.
Side Support	Define the thickness, width and height for the side support.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.

Beam form properties

Option	Description
Beam	Define a name, prefix and start number for the beam form cast unit.

General Wall Form

General Wall Form creates a timber form containing a plywood panel, a frame and studs. You can use **General Wall Form** as a stand-alone component or as a subcomponent for the **Wall Formwork** detail.

Availability

To use the component, go to Tekla Warehouse, locate the **Formwork components** and click **Insert into model**.

The components are then available in the **Component Catalog**.

Objects created

- Plywood panel
- Studs
- Frame

Use for

Situation	Description
	Wall form in a wall formwork.

Selection order

1. Select the start point.
2. Select the end point.

The wall form is created automatically.

Part identification key

	Part
1	Stud
2	Frame

	Part
3	Plywood panel

See also Wall Form: Parameters tab on page 140

Wall Form: Parameters tab

Use the **Parameters** tab to control the properties of the wall form in the **Wall Form** modeling tool.

Dimensions

	Description
1	Define the distance to first stud from the form edge.
2	Define the spacing between studs.
3	Define the width of edge lap.
4	Define the height of wall form.
5	Define whether the wall form is created on the left or right side.

Stud, frame and plywood panel properties

Option	Description
Edge Profile	Define the thickness, width and height for the frame.
Stud Profile	Define the thickness, width and height for the studs.
Plywood Profile	Define the thickness, width and height for the plywood panel.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.

Wall form properties

Option	Description
Panel	Define a name, prefix and start number for the wall form cast unit.

General Clamp

General Clamp creates a clamp. You can use **General Clamp** as a stand-alone component or as a subcomponent for the **Column Formwork** detail.

Availability

To use the component, go to Tekla Warehouse, locate the **Formwork components** and click **Insert into model**.

The components are then available in the **Component Catalog**.

Objects created

Clamps

Use for

Situation	Description
	Clamps in a column formwork.

Selection order

1. Select the start point.
2. Select the end point.

The clamps are created automatically.

See also

General Clamp: Parameters tab on page 141

General Clamp: Parameters tab

Use the **Parameters** tab to control the properties and dimensions of clamps in the **General Clamp** modeling tool.

Dimensions

	Description
1	Define the length of the clamp.
2	Define the distance to first slot.
3	Define the number of slots.
4	Define the spacing between slots.
5	Define the profile of slots.

Clamp properties

Option	Description
Clamp Profile	Define the thickness, width and height for the clamps.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.

General Clip

General Clip creates a clip. You can use **General Clip** as a stand-alone component or as a subcomponent for the **Beam Formwork** and **Wall Formwork** details.

Availability To use the component, go to Tekla Warehouse, locate the **Formwork components** and click **Insert into model**.

The components are then available in the **Component Catalog**.

Objects created Clip

Use for

Situation	Description
	Clip and tie in a beam formwork.

Selection order

1. Select the start point.
2. Select the end point.

The clip is created automatically.

See also General Clip: Parameters tab on page 143

General Clip: Parameters tab

Use the **Parameters** tab to control the dimensions and properties of clips in the **General Clip** modeling tool.

Dimensions

	Description
1	Define the length of the clip.

	Description
②	Define the width of the slot.
③	Define the length of the slot.

Clip properties

Option	Description
Clip Profile	Define the thickness, width and height for the clip.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.

General Tie

General Tie creates a tie. You can use **General Tie** as a stand-alone component or as a sub-component for the **Beam Formwork** and **Wall Formwork** details.

Availability

To use the component, go to Tekla Warehouse, locate the **Formwork components** and click **Insert into model**.

The components are then available in the **Component Catalog**.

Objects created

- Tie
- Spacers

Use for

Situation	Description
	Tie and clip in a wall formwork.

Selection order

1. Pick the start point.

2. Pick the end point.

The tie is created automatically.

Part identification key

	Part
1	Spacer
2	Tie

See also General Tie: Parameters tab on page 145

General Tie: Parameters tab

Use the **Parameters** tab to control tie dimensions and properties in the **General Tie** modeling tool.

Dimensions

	Description
1	Define the length of the tie end extension.
2	Define the length of the spacer.
3	Define whether or not to create spacers.

Tie and spacer properties

Option	Description
Tie Profile	Define the thickness, width and height for the tie.
Spacer Profile	Define the thickness, width and height for the spacers.

Option	Description
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.

Option	Description
Class	Define the part class number.

General Tie and Clips

General Tie and Clips creates a tie and clips. You can use **General Tie and Clips** as a stand-alone component or as a subcomponent for the **Beam Formwork** and **Wall Formwork** details.

Availability

To use the component, go to Tekla Warehouse, locate the **Formwork components** and click **Insert into model**.

The components are then available in the **Component Catalog**.

Objects created

- Tie
- Clips
- Spacers

Use for

Situation	Description
	Tie and clip in a wall formwork.

Selection order

1. Select the start point.
2. Select the end point.
3. The tie and clips are created automatically.

Part identification key

	Part
1	Clip
2	Spacer
3	Tie

See also General Tie and Clips: Parameters tab on page 147

General Tie and Clips: Parameters tab

Use the **Parameters** tab to control the properties of the **General Tie and Clips** modeling tool.

Dimensions

	Description
1	Define the length of the tie end extension.

Tien and clip properties

Option	Description
Tie Component Name	Define the component that you want to use as tie by selecting it from the component catalog.
Tie Attribute File	Select an attribute file for the component.
Clip Component Name	Define the component that you want to use as clip by selecting it from the component catalog.
Clip Attribute File	Select an attribute file for the component.

1.5 Openings

This section introduces components that can be used in concrete openings.

Click the links below to find out more:

- Hole Generation (32) on page 148
- Polygon Hole Generation (33) on page 150

Hole Generation (32)

Hole Generation (32) creates a hole in the object, or splits the object in two if the hole cuts the whole object. Use this component, for example, in welded profiles, slabs, or panels generated with a component.

Objects created

- Cuts

Use for

Situation	Description
	A hole is created when a point is picked.

Selection order

1. Select the part or object created by a component to be cut.
2. Click the middle mouse button.
3. Pick the position relative to the hole being generated.

See also Hole Generation (32): Parameters tab on page 148

Hole Generation (32): Parameters tab

Use the **Parameters** tab to control the hole properties in the **Hole Generation (32)** modeling tool.

Option	Description
	Define the hole dimensions and the location of the picked point. The selected **Partcut** option affects which of these dimensions are available.
	To rotate the hole, define the rotation angle.
	Define the recess depth.
Partcut	Select the hole type. The options are: • **Rectangular** creates a rectangular hole using width and height. • **Circular** creates a round hole using height as a diameter. • **Profile** creates a cut using the selected profile.
Profile	If you selected **Profile** as the hole type, select a suitable profile from the **Profile Catalog**.
Coordination system	Select the coordination system. The options are: • **Use global xy-plane** creates the hole according to the global coordinate system. • **Use local** creates the hole according to the work plane. If the part is sloped or skewed, the **Use local** option is used automatically.
Cutpart name	Define a name for the cut part.

Option	Description
Rotated	Select whether the cut is rotated 90 degrees in the XY plane.

Polygon Hole Generation (33)

Polygon Hole Generation (33) creates a polygon-shaped hole in the object. Use this component, for example, in welded profiles, slabs, or panels generated with a component.

Objects created

- Cut

Use for

Situation	Description
	A polygon-shaped hole in concrete slabs, created by picking five points.

Selection order

To create a polygon-shaped hole:

1. Select the part or object (component) to be cut.

   You can cut other components, such as **Modeling of floor bay (66)** with this component.

2. Click the middle mouse button to end selecting.
3. Pick the corner points of the polygon-shaped hole.
4. Click the middle mouse button to end picking.

Polygon Hole Generation (33): Parameters tab

Use the **Parameters** tab to control the cuts created by **Polygon Hole Generation (33)**.

Field	Description
	Cutting depth

1.6 Flooring

This section introduces components that can be used in concrete floors.

Click the links below to find out more:

- Automatic seam recognition (30) on page 151
- Seam Applicator on page 153
- Slab generation with polygon plate (61) on page 154
- Slab generation with points (62) on page 157
- Modeling of floor bay (66) on page 159
- Modeling of slab area (88) on page 167
- Sloping slab drainage on page 169
- Hollow Core Opening Tool on page 172
- Hollow Core Lifting Loops on page 176
- Floor Tool on page 179
- Floor Layout on page 182
- Floor Layout CIP Filler on page 205

Automatic seam recognition (30)

Automatic seam recognition (30) is used to add predefined custom seams between parallel parts, such as slabs or wall elements. Especially useful this tool is when adding seams for warped floors.

Use for

Situation	More information
	Use to connect parts with predefined custom seams.

Before you start Create parallel parts, for example, concrete slabs or walls.

Create custom seam.

Defining properties Use the tabs in the **Automatic seam recognition (30)** dialog box to define the following properties:

Property	Description
Seam name	Enter the seam name, or use the browse button (...) to locate the seam in the **Select component** dialog box.
Seam property file	Enter the name, or use the browse button (...) to locate the name of the seam attribute file (optional).
Seam direction	Select to change the direction.
Seam input part order	Select to reverse the main and secondary part. You may need to change the **Seam up direction** too.
Seam up direction	Define the direction of the seam.
Re-calculate seam points for	Use to re-calculate the seam points for all parts or only for warped parts. • **All parts** use for fully or partially warped decks or floors. This option creates all seam points in the same way, no matter, if the slab is warped or not. • **Warped parts** use for wall panels and other situations, where warping is not used, or the **All parts** option does not give desired result.
Position of re-calculated points	Control the location of the re-calculated seam points. The options in the list box are same as the circles in the following image.

Picking order

1. Pick the main part.
2. Pick secondary parts. Click the middle mouse button to finish picking and create the seams between the parts.

See also For more information on custom components, see .

Seam Applicator

Seam Applicator creates a seam between two double tee slabs, or between a double tee slab and a wall or a spandrel. The double tees can also be warped and/or cambered. To be able to use **Seam Applicator** you need to have a custom seam. The custom seam can be created in the model, or it can be imported to the model.

Objects created

- Seams

Use for

Situation	Description
	Two double tee slabs with seams
	Double tee slab and wall with seams

Before you start Create a custom seam.

When you define the custom seam in **Custom Component Wizard**, make sure that you select the **Allow multiple instances of connection between same parts** checkbox on the **Advanced** tab.

Selection order

1. Select the main part.

2. Select the secondary part.

   The seam is created automatically when the secondary part is selected.

   One of the selected parts must be a double tee slab and the other part can be a double tee slab, a wall, or a spandrel.

If the seam cannot be created between the selected parts for some reason, a dummy beam is created instead. The beam indicates that the seam creation did not succeed.

See also Seam Applicator: Parameters tab on page 154

Seam Applicator: Parameters tab

Use the **Parameters** tab to control the properties of the seam in the **Seam Applicator** modeling tool.

Option	Description
Seam name	Define the custom seam that connects the parts by selecting it from the component catalog.
Seam property file	Select the attribute file for the custom seam.
Seam direction	Define the seam direction in relation to the main and the secondary part.
Seam up direction	Define the seam rotation.
Offset start point	Define the seam start point offset from the edge of the double tee slab start point.
Offset end point	Define the seam end point offset from the edge of the double tee slab end point.
Number of copies	Define the number of seams.
Distance to first seam	Define the distance between the double tee slab edge and the first seam start point.
Spacing values	Define the space between the seams.
Copy at equal distances (Ignore spacing values)	Define whether seams are created at equal distances. If you select **Yes**, the values in **Spacing values** field are ignored.

Slab generation with polygon plate (61)

Creates slabs based on the shape of an auxiliary plate.

Parts created Concrete slabs.

Where to use

Situation	More information
	Hollow core slab created using an auxiliary plate. To change the shape or size of the plate, use the **Polygon shape** command. See .

Before you start Create a contour plate or a concrete slab in the shape of the slabs you want to create. This is the auxiliary plate.

Defining properties Use the tabs in the **Slab generation with polygon plate (61)** dialog box to define the following properties:

Tab	Contents	More information
Parameters	Slab dimensions, gaps between slabs, slab direction and distribution	Slab generation with polygon plate (61) and Slab generation with points (62): Parameters tab on page 155
Parts	Slab properties	Slab generation with polygon plate (61) and Slab generation with points (62): Parts tab on page 156
Analysis	Analysis properties of slab.	Slab generation with polygon plate (61) and Slab generation with points (62): Analysis tab on page 156
Loading	Loads properties used for analysis.	

Selection order Auxiliary plate.

Slab generation with polygon plate (61) and Slab generation with points (62): Parameters tab

Use these options to define slab properties:

Property	Description
	Gap between slabs. To create overlapping slabs, enter a negative value. Use this to allow for joint reinforcement, grouting, etc.
	Distance from the outer edges of the slabs and the corner of the auxiliary plate. To create a slab larger than the auxiliary plate, enter a negative value. Use this option to allow space for other parts, such as bearing beams.
Generation direction	Defines the direction in which Tekla Structures creates the slabs.
Adjustment	Defines which slab(s) to truncate or delete for an even slab distribution. The options are **first**, **last**, and **first and last**.

Slab generation with polygon plate (61) and Slab generation with points (62): Parts tab

Define a profile by selecting it from the Profile Catalog. To select a slab type from the Profile Catalog, click the button against the **Slab** field on the **Parts** tab.

Slab generation with polygon plate (61) and Slab generation with points (62): Analysis tab

Use the **Analysis** tab to define the analysis properties of the parts created using the **Slab generation with polygon plate (61)** and **Slab generation with points (62)** modeling tools.

Option	Description	Only use for
Analysis type	The option you select in the **Analysis type** list limits the other properties you can define (see the **Only use for** column) Select how Tekla Structures analyzes the slabs. • **Ignore**: Slabs are not analyzed. • **Beam**: Analyze each slab as a beam. • **Plate**: Analyze each slab as a plate.	

Option	Description	Only use for
	• **Rigid diaphragm**: Analyze slabs as a rigid diaphragm.	
Beam axis	Select the location of the beam axis.	Beam
Restraints	Select the support conditions of the beam ends. The options are **Pinned** and **Fixed**.	Beam
Plate plane	Select the plane on which to create the elements and nodes. If you select **Top plane**, Tekla Structures creates the elements on the top surface of the slab.	Plate
Element type	Select the shape of the elements.	Plate
Element size	**x** and **y**: Define the approximate dimensions of the elements in the local x and y direction of the slab. For triangular elements, define the approximate dimensions of the bounding box around each element. **Holes**: Define the approximate size of the elements around openings.	Plate
Filter	Nodes that belong to a part matching the filter will be connected to the rigid diaphragm. For example, you can use **column_filter** to connect only column nodes to rigid diaphragms.	Rigid diaphragm

See also

Slab generation with points (62)

Creates slabs based on the corner points you pick.

Parts created Concrete slabs.

Where to use

Situation	More information
	Double tee slabs created by picking eight points. You cannot change the slab area once the slabs are created.

Defining properties Use the tabs in the **Slab generation with points (62)** dialog box to define the following properties:

Tab	Contents	More information
Parameters	Slab dimensions, gaps between slabs, slab direction and distribution	Slab generation with polygon plate (61) and Slab generation with points (62): Parameters tab on page 155
Parts	Slab properties	Slab generation with polygon plate (61) and Slab generation with points (62): Parts tab on page 156
Analysis	Analysis properties of slab.	Slab generation with polygon plate (61) and Slab generation with points (62): Analysis tab on page 156
Loading	Loads properties used for analysis.	

Selection order Pick any number of points to indicate corners of the slab. Close the shape by picking the first point you picked.

Slab generation with polygon plate (61) and Slab generation with points (62): Analysis tab

Use the **Analysis** tab to define the analysis properties of the parts created using the **Slab generation with polygon plate (61)** and **Slab generation with points (62)** modeling tools.

Option	Description	Only use for
Analysis type	The option you select in the **Analysis type** list limits the other properties you can define (see the **Only use for** column) Select how Tekla Structures analyzes the slabs. • **Ignore**: Slabs are not analyzed. • **Beam**: Analyze each slab as a beam. • **Plate**: Analyze each slab as a plate. • **Rigid diaphragm**: Analyze slabs as a rigid diaphragm.	
Beam axis	Select the location of the beam axis.	Beam
Restraints	Select the support conditions of the beam ends. The options are **Pinned** and **Fixed**.	Beam
Plate plane	Select the plane on which to create the elements and nodes. If you select **Top plane**, Tekla Structures creates the elements on the top surface of the slab.	Plate
Element type	Select the shape of the elements.	Plate

Option	Description	Only use for
Element size	**x** and **y**: Define the approximate dimensions of the elements in the local x and y direction of the slab. For triangular elements, define the approximate dimensions of the bounding box around each element. **Holes**: Define the approximate size of the elements around openings.	Plate
Filter	Nodes that belong to a part matching the filter will be connected to the rigid diaphragm. For example, you can use **column_filter** to connect only column nodes to rigid diaphragms.	Rigid diaphragm

See also

Modeling of floor bay (66)

Creates an area of concrete slabs between two support lines that you define, for example, between two main frames. Support lines can be polylines or sloped.

Parts created Concrete slabs.

Where to use

Situation	More information
	Slab area created between two support lines.

Before you start Create two support lines with two or more points.

Defining properties Use the tabs in the **Modeling of floor bay (66)** dialog box to define the following properties:

Tab	Contents	More information
Parameters	The position of the slab.	Defining slab position on page 163
Profiles	The properties of slab profiles.	Defining slab profile properties on page 165

Picking order

1. Pick points to define the first support line. To finish, middle-click away from the last point picked.
2. Pick points to define the second support line. To finish, middle-click away from the last point picked.
3. Pick a point to indicate the origin of the slabs.
4. Pick a point to indicate the direction of the slabs.

See also Examples on page 160

Examples

These examples show how to use the **Modeling of floor bay (66)** component.

Example 1:

1. Pick 2 points (1, 2) to define support line 1.
2. To finish, middle-click.

3. Pick 2 points to define support line 2.

4. To finish, middle-click.

5. Pick a point to indicate the origin of the slab.
6. Pick a point to indicate the direction of the slab.

Tekla Structures creates the floor bay.

Example 2:

1. Pick 2 points to define support line 1.

2. To finish, middle-click.

3. Pick 4 points to define support line 2.

4. To finish, middle-click.
5. Pick a point to indicate the origin of the slab.
6. Pick a point to indicate the direction of the slab.

Tekla Structures creates the floor bay.

Defining slab position

Use these options to define the position of the slab:

Property	Description
	The position of the slab relative to the origin line picked when creating the slab. The options are **center of profile** (default) or **center of seam**.
 	Select the slab spacing and enter the appropriate dimension. The options are **seam** (default) or **center to center**. If you select the seam option, the bottom dimension is enabled. If you select the center-to-center otion, the upper dimension is enabled.
	Slab offsets from support lines 1 and 2.

Property	Description
	Vertical position of the slab. The options are **Top** (default), **Center** or **Bottom**. Enter an optional offset value.
Projection type	Defines how the slabs are spaced on a sloping floor bay. See Defining projection type on page 164.

Defining projection type

Use the **Projection type** field to define how to handle slab spacing or seams on a sloping floor bay. The options are **Global** (default), **Support line 1**, **Support line 2**, or **Both support lines**.

- **Global:** the centers of the slabs remain constant, seam spacing adjusts.

- **Support line 1:** Seam spacing on support line 1 does not adjust (effect exaggerated here):

- **Support line 2:** Seam spacing on support line 2 does not adjust (effect exaggerated here):

- **Both support lines:** Seam spacing is calculated at the average plane position between both support lines

1 Along this line, center to center and seam width are exactly as entered in the dialog box.

2 Support line 2

3 Plane between support lines 1 and 2

4 Support line 1

Defining slab profile properties

Use the options on the **Profiles** tab to define a default profile for all the slabs in the floor.

Index list

Use the **Index list** to specify different profiles, seam width, and various cutting options for specific slabs or seams. See also Adjustment type on page 167.

Input the values individually, or in a list:

Examples for slabs

If the origin point is the center of slab:

If the origin point is the center of seam:

Examples for seams

If the origin point is the center of slab:

If the origin point is the center of seam:

Adjustment type

Use to define the type of adjustment for the selected slab. The options are:

- **Default** – Use the default slab profile.
- **Profile** – Use the specific slab profile.
- **Cut left side** – Cut the left side of the profile. If no profile is defined, uses the default slab profile.
- **Cut right side** – Cut the right side of the profile. If no profile is defined, uses the default slab profile.
- **Cut both sides** – Cut both sides of the profile. If no profile is defined, uses the default slab profile.
- **Seam width** – Adjust the seam width.

Profile/Seam width

Depending on which adjustment type option you choose, defines:

- The width of the default slab profile.
- The width of the specified profile.
- Seam width.

Modeling of slab area (88)

Use to create various types of concrete slabs, including fanned layouts. This component creates slabs between two support lines, which can be polylines or sloped. To create a fanned layout, use support lines that are curved or of different lengths.

Parts created Concrete slabs.

Where to use

Situation	More information
	Slab area created between two support lines.

Situation	More information
	Fanned layouts of slab area

Before you start Create the two support lines with two or more points.

Defining properties Use the tabs in the **Modeling of slab area (88)** dialog box to define the following properties:

Tab	Contents	More information
Parameters	Distance from the outer edges of the slabs to the support lines, slab direction and distribution, fan angle.	Defining slab area properties on page 168
Parts	Slab properties.	
Advanced	Element and seam index properties.	

Picking order

1. Pick points to define the first support line. Click the middle mouse button outside the point you just picked.
2. Pick points to define the second support line. Click the middle mouse button outside the point you just picked.
3. Pick two points to indicate the direction of the slabs. The first point is the origin for the slabs.

Defining slab area properties

Use these options to define slab properties:

Property	Description
	Distances from the slab to the support lines.
Origin point	Origin point of the slab. Origin point is the point you pick after picking the support line points.
Support line plane	The options are **Middle**, **Bottom**, and **Top**.

Property	Description
Adjustment	Defines which slab(s) to truncate or delete, or how the slab area is generated. The slab closest to the first support line's origin is the first.
	The fan angle.
Slab	Slab generation with polygon plate (61) and Slab generation with points (62): Parts tab on page 156
Standard spacing	Slab spacing.
Advanced tab	Use the **Advanced** tab to modify individual slabs and seams. You can index them and then define width or angle for the indexed objects.
Index	The number of the slab, seam, or fanned seam that is different from the others.
Width	The width of the indexed slab or seam.
Angle	The fan angle of the indexed seam.

Sloping slab drainage

Sloping slab drainage creates a concrete slab topping and a drainage hole.

Objects created

- Drainage hole

Use for

Situation	Description
	Drainage holes with valley (1) and hip (2) created.

Limitations Slab topping does not adapt to slab chamfers. You need to define the slab chamfers and topping chamfers separately.

Before you start Create or split the slabs so that there is one drainage point in each slab.

Selection order

1. Select the main part (slab).
2. Pick a point to create the drainage hole.

   The drainage hole is created automatically when the point is picked.

Part identification key

	Part
1	Drainage hole

See also Sloping slab drainage: Parameters tab on page 170

Sloping slab drainage: Parameters tab

Use the **Parameters** tab to control the type, dimensions and properties of the drainage hole in the **Sloping slab drainage** modeling tool.

Drainage hole type

Option	Description
+	A drainage hole is not created.

Option	Description
	A rectangular drainage hole is created.
	A round drainage hole is created.

Drainage hole dimensions

Option	Description
	Define the side dimensions of the rectangular drainage hole.
	Define the diameter of the round drainage hole.

Drainage hole properties

Option	Description
Define slab inclination by	Select how the slab is inclined, by a combination of slope (S) or slope percentage (S%), and topping thicknesses (d1, d2). The **Slope**, **d1**, and **d2** options are available accordingly.
Slope	Define the slope of the topping, for example, 0.01 (S) or 2 (S%).
d1	Define the thickness of the topping at the drainage point or at the hole.
d2	Define the thickness of the topping at the edge of the slab or at the hip.
Create cast unit	Select how the cast unit is created: • **Yes** Topping is part of the slab cast unit. • **No** Each part of the topping forms a cast unit.

Option	Description
	• **Topping** Topping is a separate cast unit.
Material	Define the material of the topping by selecting it from the material catalog.
Type	Select whether a valley or a hip is created. If you select **Hip**, you can only pick slab corners.

Hollow Core Opening Tool

Hollow Core Opening Tool creates holes and recesses to hollow core slabs. **Hollow Core Opening Tool** works with single hollow core slabs, or with floors that are created with the **Modeling of floor bay (66)** modeling tool.

Objects created

- Holes and recesses

Use for

Situation	Description
	Hollow core slab with small and large openings

Limitations

Hollow Core Opening Tool calculates the location for the openings based on the given values and the slab profile data. If needed, you can use an external configuration file in XML format to provide different rules for the openings and the slab profile data.

You can define small and large openings. Small openings are placed in the middle of the hollow cores so that no webs are broken. Large openings break webs.

Before you start

Create a hollow core slab.

Selection order

1. Select the main part (slab).
2. Pick a position for the opening.

See also

Hollow Core Opening Tool: Parameters tab on page 172

Hollow Core Opening Tool: Configuration file tab on page 175

Hollow Core Opening Tool: Parameters tab

Use the **Parameters** tab to control the properties of the openings and recesses in the **Hollow Core Opening Tool** modeling tool.

Option	Description
	Select the shape of the openings and recesses. • Square - in the middle of the slab • Round - in the middle of the slab • At one end of the slab • At both ends of the slab
Hollow positions	If you select to add openings or recesses at one end or at both ends of a slab, enter the positions of the hollows that are cut. The positions are numbered from left to right at the start of the part. The part is viewed from the top in the start-to-end direction. The openings and recesses are created in the same voids at the end of the part.

Option	Description
Size	Define the size of the opening in longitudinal and cross direction, or the diameter for a round opening. Opening width defines whether the opening is small or large.
Opening Type	Select the type of the opening.
Place exactly by point	Select whether the opening is created exactly to the picked position.
Centralize in zone	Select how the center of an opening is centered • **By hollow**: The opening is centered to the hollow core. • **By web**: The opening is centered to the web.
Cut full width of hollow	Select whether the opening is cut based on the minimum width of hollows or the number of hollows. To use this option, center the opening **By hollow** in the **Centralize in zone** option.
Rules file (XML)	Select the external XML file where you have defined the opening and slab profiles.
Cutpart name	Define a name for the cut part.
	Define the recess depth.

Option	Description
	Define the alignment for the opening.
	Select whether the thick side part of the opening is removed.

Hollow Core Opening Tool: Configuration file tab

Use an external `.xml` file to define the openings and the slab profiles in the **Hollow Core Opening Tool** modeling tool.

Example An example `Zones.xml` file is located in `..\ProgramData\Tekla Structures\<version>\environments\common\system`.

Zones

```
<ZoneRules>
        <!--Profile element has name and NumberOfSmall attributes which define number of small
zones in width direction. Profile name corresponds to TS profile name prefix that is P27, P32
etc. despite actual profile can be "P27(265x1200)" for example-->
        <Profile Name="P27" NumberOfSmall="5">
                <!--SmallZone element describes its parameters and must be the only in each
                profile. -->
                <SmallZone Start="77" Width="150" Interval="73" />
                <!--LargeZone element describes large zone (which breaks webs). There must be as
                many LargeZone elements as large zones supposed to be defined for a given
                profile. -->
                <LargeZone Start="325" Width="550" />
                <LargeZone Start="775" Width="425" />
                <LargeZone Start="0" Width="425" />
        </Profile>
        <Profile Name="P32" NumberOfSmall="4">
                <SmallZone Start="90" Width="180" Interval="95" />
                <LargeZone Start="375" Width="450" />
                <LargeZone Start="675" Width="525" />
                <LargeZone Start="0" Width="525" />
        </Profile>
</ZoneRules>
```

Hollow Core Lifting Loops

Hollow Core Lifting Loops creates lifters for hollow core slabs. **Hollow Core Lifting Loops** works with single hollow core slabs, or with floors that are created with **Modeling of floor bay (66)** modeling tool.

Objects created

- Lifters

Use for

Situation	Description
	Hollow core slab with recesses, hollow core fillings and lifting devices.

Limitations You also need a lifting device. The lifting device can be created as a custom part, or it can be imported to the model. Alternatively, you can use an external configuration file in XML format to define the properties for the recess, the lifting device and the slab.

Before you start Create a hollow core slab and a lifting device.

Selection order

1. Select the main part (slab).

   The recess and hollow core fillings are created.

See also Hollow Core Lifting Loops: Basic settings tab on page 177

Hollow Core Lifting Loops: Lifting device settings tab on page 178

Hollow Core Lifting Loops: Configuration file tab on page 178

Hollow Core Lifting Loops: Basic settings tab

Use the **Basic settings** tab to control the properties of the lifters and hollow core fillings in the **Hollow Core Lifting Loops** modeling tool.

Configuration file

Option	Description
Configuration file	Select the external XML file where you have defined the recess, the lifting device and the slab profile properties.
Override	Define whether the values on the **Basic settings** and **Lifting device settings** tabs override the properties defined in the external XML file.

Dimensions

Option	Description
1	Define the minimum distance from the lifter insertion point to the end of the slab.
2	Define the maximum distance from the lifter insertion point to the end of the slab.
3	Define the minimum distance from the lifter insertion point to the edge of an opening.
4	Define the minimum distance between two lifter insertion points.
5	Define the length of the hollow core filling.
6	Define the profile of the recess.
7	Define the depth of the recess.

Hollow core filling

Option	Description
Use automatic fill profile	Define whether the automatic hollow core filling profile is used.
Profile	Define the hollow core filling profile.
Prefix	Define a prefix for the part position number.
Start Number	Define a start number for the part position number.
Name	Define a name for the lifter. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group the lifters. For example, you can display lifters of different classes in different colors.

See also Hollow Core Lifting Loops: Lifting device settings tab on page 178

Hollow Core Lifting Loops: Configuration file tab on page 178

Hollow Core Lifting Loops: Lifting device settings tab

Use the **Lifting device settings** tab to control the properties of the lifting device in the **Hollow Core Lifting Loops** modeling tool.

Option	Description
Lifter component	Define the custom part that represents the actual lifting device.
Saved properties to use	Select which saved properties to use for the lifting device custom part.
Lifter max load	Define the maximum load for the lifting device.
Rotation	Select the rotation of the lifting device custom part around its axis.
Angle	Define the rotation angle of the lifting device custom part.
Up Direction	Select the lifting device up direction in relation to the local coordinate system.
Offset	Define the offset of the lifting device in relation to the local coordinate system in the corresponding directions.

See also Hollow Core Lifting Loops: Basic settings tab on page 177

Hollow Core Lifting Loops: Configuration file tab on page 178

Hollow Core Lifting Loops: Configuration file tab

Use an external XML file to define the lifting device, the lifter, and the slab properties in the **Hollow Core Lifting Loops** modeling tool.

Example

Configuration file

```
<LiftingsConfiguration>
        </LiftingDevices>
                <Lifter Name="Lifter2" AttrFile="standard" MaxLoad="3000"
                RecessProfile="PRMD300*200-200*100"
                RecessDepth="60" UpDirection="+Z"
                Rotation="0" OffsetX="0" OffsetY="0" OffsetZ="0" />
                <Lifter Name="Lifter3" AttrFile="standard" MaxLoad="3000"
                RecessProfile="PRMD250*200-150*100" RecessDepth="50"
                UpDirection="+Z" Rotation="0" OffsetX="0" OffsetY="0" OffsetZ="0" />
        </LiftingDevices>
        <Profiles>
                <Profile Name="P27" MinEdgeDistance="200" MaxEdgeDistance="300"
                MinOpeningDistance="200" MinLiftersDistance="400"
                SideCoreDistance="0" CoreFillingLength="500"/>
                <Profile Name="P32" MinEdgeDistance="200" MaxEdgeDistance="300"
                MinOpeningDistance="200" MinLiftersDistance="400"
                SideCoreDistance="0" CoreFillingLength="500"/>
        </Profiles>
</LiftingsConfiguration>
```

Floor Tool

Floor Tool creates a floor bay with optional insulation.

Objects created

- Hollow core slabs

Use for

Situation	Description
	Floor bay with insulation

Selection order

Make sure that you have defined the slab properties.

1. Pick the contour points of the floor bay.

   The floor bay is automatically created when you close the polygon.

The list of picked contour points on the **Bay contour** tab and the list of slabs on the **Slab list** tab are created when the slabs are created.

See also

Floor Tool: Slab properties tab on page 179

Floor Tool: Insulation tab on page 181

Floor Tool: Bay contour tab on page 181

Floor Tool: Slab list tab on page 182

Floor Tool: Slab properties tab

Use the **Slab properties** tab to control the profile, material and location of the hollow core slabs in the **Floor Tool** modeling tool.

Slab attributes

Option	Description
Profile	Define the profile by selecting it from the profile catalog.
Name	Define a name for the slab.
Material	Define the material grade by selecting it from the material catalog.
Class	Define the part class number.
Prefix	Define the part prefix.
Start number	Define the start number for the cast unit.
Cut slab ends	Define whether the slab ends are cut or not. If you select **Yes**, define the minimum end angle for the slabs.
Rotation	Select an option to rotate the slab.
Vertical location	Define the offset location in the vertical direction.
Offset	Define the offset in the vertical direction.
Minimum end angle	Define the minimum end angle for the slabs. If the line to which the slab ends has a bigger angle than the defined minimum end angle, the part will be aligned to the line.

Additional attributes

Option	Description
Exposure Class	Define the exposure class for the hollow core slab. The exposure class will be saved as a user-defined attribute.
Relative strength	Define the relative strength of the hollow core slab. The relative strength will be saved as a user-defined attribute.
Fire rating	Define the fire rating for the hollow core slab. The fire rating will be saved as a user-defined attribute.
User-defined attributes	User-defined attributes are located in the `FloorTool.ini` file.

Slab direction

Option	Description
Slab direction	Define the direction of the slab. If you select **Angle**, you need to pick points in the model to define the new direction. The angle you have picked is

Option	Description
	shown in the **Angle** field. You can also define the angle directly in the field to rotate the slabs. Note that picking two points does not change the local +X direction.

Floor Tool: Insulation tab

Use the **Insulation** tab to control the insulation properties of the hollow core slabs in the **Floor Tool** modeling tool.

First define whether to create insulation or not. Insulation is created separately for each slab.

Insulation

Option	Description
Name	Define a name for the insulation.
Material	Define a material grade by selecting it from the material catalog.
Thickness	Define the thickness of the insulation.
Class	Define the part class number.
Offset at slab start/end	Define the offset between the slab start/end positions and the corresponding positions in the insulation parts.
Left/Right offset	Define the offset between the slab and the insulation for the rightmost and leftmost slab.

Floor Tool: Bay contour tab

Use the **Bay contour** tab to control the coordinates of the picked contour points and the horizontal offset of the lines connecting the contour points in the **Floor Tool** modeling tool. If the coordinates are not visible on the tab, click the **Get** button.

You need to create the hollow core slabs before using the **Bay countour** tab.

Bay contour properties

Option	Description
XY-constant Z	Define the Z coordinate for all contour points at the same specific height.
XYZ (sloping floor)	Define the Z constant for each contour point separately. The floor may be sloped.
Offset method	Define the offset method.
Default end offset	Define the default end offset.
Default side offset	Define the default side offset.

Bay contour table

The table lists the contour points showing their X, Y, and Z coordinates. You can modify the Z coordinate in the table only if you have selected **XYZ (sloping floor)**. You can modify the **Horizontal offset** on all rows of the table.

The table also shows the lines connecting the points. You can only modify the **Horizontal offset**. Entering a positive value enlarges the slab area and entering a negative value shrinks the slab area.

Floor Tool: Slab list tab

Use the **Slab list** tab to list all the hollow core slabs created in the **Floor Tool** modeling tool. If the slab list is not visible on the tab, click the **Get** button.

Slab properties

Option	Description
Slab number	Shows the order of the hollow core slabs starting from the first picked point.
Width	Shows the width of a narrowed slab.
Narrowed from	Define whether the slab is narrowed. The options are: • not narrowed (default) • **Right side** • **Left side**
Type	Define the slab type. • Use **Slab** for normal hollow core slabs. • Use **PIP** for a precast infill piece. • Use **GAP** for an empty space between the slabs. You do not need to define a profile when you select this option. • Use **CIP** to create a concrete part using the width defined in the **Width** column. The width range that you can use is defined in the `FloorTool.ini` file. The slabs listed in the `FloorTool.ini` file can only have a width that is inside the defined range. The slabs that are not listed can have any width. The value you give is rounded down to the nearest allowed value.
Profile	Define the profile by selecting it from the profile catalog.

You can **Add** and **Delete** slabs, and move them **Up** and **Down** in the list by using the buttons on the left. To restore the default values, click **Restore defaults**.

Floor Layout

Floor Layout creates floors built of precast filigree and hollow core slabs. The floors may contain several layers, such as precast slabs, insulation and a cast-in-place topping. **Floor Layout** is most useful for non-rectangular floors that contain one or more openings that split slabs into many pieces, and when slab widths are not constant on the whole floor area. You can modify the floors using direct modification.

Objects created

- Slabs
- Insulation
- Cast-in-place topping

Use for

Situation	Description
	Create floor structures that have layers, for example, hollow core slabs at the bottom, insulation as the middle layer, and cast-in-place topping.

Selection order

1. Pick the corner points of the floor.

   You can pick the points at grid lines or the reference points of supporting parts. You can change the offsets later.

   The default slab direction is from the first (P1) point to the second (P2) point. You can change this later if needed.

2. Click the middle mouse button to create the floor.

Floor Layout: Layer tab

Use the **Layer** tab to control the layer type, profile, and how the layer is created in the **Floor Layout** modeling tool.

Layer list

Layers	Description
	The layer list shows the current layers of the floor. The first layer in the list is the top layer and the last layer is the bottom layer. In the example image on the left, the `FILIGREE` layer is the top layer and the `CIP` layer is the bottom layer. Use the buttons to do the following: • Click to add a new layer. • Click to remove the selected layer. • Click to change the order of the layers by moving the selected layer up or down in the list.

Layer properties

Select a layer in the layer list to define the layer properties.

Option	Description
Layer name	Enter a name for the layer. This is the layer name shown in the layer list. The name is also shown in the direct modification toolbar when you are modifying offsets . This name is not visible in the model so you can enter any suitable name.
Layer type	Select the layer type. The **Precast** layer is the primary layer and any insulation parts are added to precast units. The parts created for a **Cast in place** layer are not added to any other cast unit.
Create layer as	Select how the layer parts are created: as beam parts, custom parts, or a single part.
Layer component **Component attributes**	When the layer parts are created as custom parts, you can select a component and the component attributes.
Layer thickness or profile	When the layer parts are created as beam parts or as a single part, you can select the profile or thickness of the layer parts. With parametric profiles, you can replace the width of the profile with the text `[W]`. When you do this, the beam is never cut but the

Option	Description
	desired width is achieved by setting the profile name based on the slab width. For example, the profile could be `BL80*[W]`.
Rotation	Select the rotation of the layer parts.
Part name **Class** **Material** **Part prefix**, **Start no** **Cast unit prefix**, **Start no**	Define the name, class, material, part prefix and start number of the layer parts.

Floor Layout: General tab

Use the **General** tab to control the default part and gap width, depth position of the layers, and the direction for lining up the slabs in the **Floor Layout** modeling tool.

Option	Description
Default part width	Enter the default width for the slabs in the primary layer. If you define the width of an individual slab in the slab part properties, the **Default part width** value is not used.
Default gap width	Enter the default seam gap width in the primary layer. If you define the width of an individual gap, the **Default gap width** value is not used.
Depth position	Select how the layers are positioned in relation to the plane of input points.

Option	Description
Line up direction	Select the direction for the slabs. By default, the width of the last slab may differ from the width of the other slabs.

Customizing allowed slab width

You can customize the allowed slab widths in the `FloorLayout.ini` file. This file can be located under the model folder or in any of the folders defined in the `XS_FIRM`, `XS_PROJECT` or `XS_SYSTEM` advanced options.

The image below shows an example of the `FloorLayout.ini` file:

```
//
// Slab not listed below can have any width. Slab listed below can
// only have width inside the given range.
// User input values will be rounded down to the nearest allowed value.
//
//                                Zone 1       Zone 2       Zone 3       Zone 4
// Slab widths     Profile          min. max.    min. max.    min. max.    min.
max.
SLAB_WIDTH         P15(150X1200)    500-650      700-800      850-950
1000-1100
SLAB_WIDTH         P18(175X1200)    500-650      700-800      850-950
1000-1100
SLAB_WIDTH         P20(200X1200)    500-650      700-800      850-950
1000-1100
```

Floor Layout: Advanced tab

Use the **Advanced** tab to control the minimum angles, and the width and length of the slabs in the **Floor Layout** modeling tool.

Option	Description
Minimum end angle **Minimum start angle**	Controls how the slab ends are modeled. When the angle of the floor edge is smaller than the minimum angle, you can select a suitable option from the list to make the slab end straight.
	Minimum slab width. Slabs narrower than the minimum width are not created.

Option	Description
	Minimum slab length. Slabs shorter than the minimum length are not created.
	Minimum neck width of the slabs. If the width of the neck is less than the minimum neck width defined, the slab is split to two slabs and the neck is left out. Otherwise, the slab is created as one slab.

Floor Layout: Detailing tab

Use the **Detailing** tab to select the components that are applied to all slabs in the floor.

Option	Description
Component name	Select a component that is applied to the slabs in the floor.
Component attr file name	Select the settings that are used with the component you have selected. The list contains all the files that are available for the selected component.

Adding a component to the component name list

The component name list is empty by default. You can add components to the list:

1. Select **Add new** from the list.

   The **Detailing component data** dialog box is displayed.
2. Enter a suitable name in the **Name in dropdown** box.
3. Click ... to open the **Component Catalog**.
4. Select a component and click **OK**.

   The component name or number is added to the **Component name** box.
5. Click **OK**.

When you have added the needed components, the names of the components are shown in the component name list as shown in the example image below. Select a component from the list:

The list is stored in the `\attributes` folder under the model folder.

If you want to predefine the list to contain your favorite components in any new models, you can copy the `LayoutConfiguration.datx` file from the `\attributes` folder under the model folder to any of the folders defined in the `XS_FIRM`, `XS_PROJECT` or `XS_SYSTEM`.

Floor Layout: User defined tab

Use the **User defined** tab to set the user-defined attribute values for the slabs in the **Floor Layout** modeling tool.

If you have defined the user-defined attributes in the part properties dialog box and want to use those values, leave the **User defined** tab empty.

The content of the tab may vary depending on your environment, see below how to customize the content.

Customizing the User defined tab

You can customize the content of the **User defined** tab using the `floorlayout.objects.inp` file. Note that you cannot create new user-defined attributes with the file. The user-defined attributes on the tab are existing user-defined attributes that have been defined in the `objects.inp` file. The `floorlayout.objects.inp` file has the same format as the `objects.inp` file.

The `floorlayout.objects.inp` file can be located in any of the folders set in the `XS_FIRM`, `XS_PROJECT` or `XS_SYSTEM` advanced options. The file is never read from the model folder.

Below is an example of the `floorlayout.objects.inp` file:

```
part(0,"Part")
{
     tab_page("Parameters")
     {
          attribute("comment", "j_comment", string, "%s", no, none, "0.0",
"0.0")
          {
                value("", 0)
          }
          attribute("xs_shorten", "j_xs_shorten", distance, "%d", yes,
none, "0.0", "0.0")
          {
                value("0.0", 0)
          }
          attribute("USER_FIELD_1", "j_user_field_1", string, "%s", no,
none, "0.0", "0.0")
          {
                value("", 0)
          }
          attribute("USER_FIELD_2", "j_user_field_2", string, "%s", no,
none, "0.0", "0.0")
          {
                value("", 0)
          }
          attribute("USER_FIELD_3", "j_user_field_3", string, "%s", no,
none, "0.0", "0.0")
          {
                value("", 0)
          }
          attribute("USER_FIELD_4", "j_user_field_4", string, "%s", no,
none, "0.0", "0.0")
          {
                value("", 0)
          }
     }
}
```

Floor Layout: Modifying a floor

You can use direct modification to modify the floor. Before you start, ensure that the **Direct modification** switch is active. Select the floor to display the direct modification toolbar.

To modify the floor, select the appropriate command from the toolbar.

Button	Description
	Add a polygonal opening.
	Add a rectangular opening.
	Add a break line to split the slab into two slabs.
	Modify the openings or exterior boundary of the floor. This command is active by default. Make the floor warped by offsetting one or more vertices of the exterior boundary polygon from the plane.
	Modify the offsets at floor boundaries
	Modify the layout of the floor: • Modify the width or profile of any of the slabs in the floor. • Modify the side of the cut when the slab is cut to a smaller width. • Modify the gaps between the slabs. • Modify the direction of the slabs. • Modify the order of slabs when the slabs do not the have the same width or profiles.
	Add or modify the detailing components that are applied only to certain slabs in the floor layout. The detailing components are listed on the **Detailing** tab. You can apply the same detailing component settings to all slabs in the floor layout using the floor layout attributes.

Adding a polygonal opening

1. Click .
2. Pick the corner points of the opening.

You can pick the points at grid lines or you can pick the reference points of supporting parts. You can change the offsets later.

Adding a rectangular opening

1. Click .
2. Pick the corner points of the opening.

You can pick the points at grid lines or you can pick the reference points of supporting parts. You can change the offsets later.

Adding a break line to split the slab into two slabs

1. Click .
2. Pick the end points of the break line.

Modifying the openings and exterior boundary of the floor

1. The button is active by default.

   The handles are shown for the exterior boundary and for all openings in the floor.

2. Do any of the following:

   - To move an edge, drag any of the edge lines. Hold down the left mouse button when dragging.

- To move a corner point, drag the handle in the corner.

- To add new points to the edge, drag the handle in the middle of the edge.

- To delete any of the points, select the handle in the corner and press the **Delete** key.
- To make the floor warped, offset the vertices of the exterior boundary so that all vertices are not in the same plane.

  Drag a handle normally to move the vertices in the original floor plane.

Hold down the **Alt** key while dragging to move the vertex perpendicular to the floor plane.

Hold down the **Alt** and **Shift** keys to move the vertex both in the plane and off from the plane by snapping to any point in the model.

Modifying the offset at floor boundaries

1. Click .

   The handles for modifying the offset are shown. If there are offsets, the offset dimensions are also shown.

2. To adjust the offset, do one of the following:

   - Drag the edge offset lines.

- Select one or more offset lines by holding down the **Shift** key.

  When you have the offset lines selected, right-click on any of the selected lines.

  Enter the offset value to the toolbar that is displayed and click .

Note that when you use the offset command, the direct modification toolbar shows the floor layout layers. Each layer may have different offsets. You can select which layers to modify. The simplest way is to select all layers and have the offsets equal in each layer.

FILIGREE
CIP
140.00
140.00
Floor Layout
Save
Load
standard
FILIGREE
CIP

140.00
140.00
140.00
140.00
140.00
FILIGREE
CIP
50.00
50.00
50.00
50.00
50.00
200.00
200.00
140.00
140.00
200.00
200.00
140.00

Modifying the layout of the floor

1. Click .

   The following direct modification elements are shown:

2. To modify the layout of the floor, do any of the following:
   - To modify slab width, do one of the following:
     - To move all seams, select one seam and start dragging.
     - To add a new seam, select a seam, hold down the **Ctrl** key and start dragging. When you stop dragging and release the mouse key, a new seam is added to that location.
     - To make one slab smaller, select a seam, hold down the **Alt** key and start dragging. The slab on the left or right side is made smaller and all slabs on the other side will move with the seam.
   - To modify the profile, width, and cut side of a slab, right-click the slab handle.

When the profile of the slab is fixed (not parametric), the slab is cut on either side when the width is smaller than the original width. The **Default** cut side is leading

edge for the first slab and trailing edge for all others in relation to the layout direction.

You can modify the properties of multiple slabs at the same time by holding down the **Shift** key when selecting slabs.

- To modify the gap width between the slabs, do one of the following:
  - Drag the end of the gap dimension.
  - Right-click a seam line handle, enter the width and click .

You can modify multiple gaps at same time by holding down the **Shift** key when selecting seam line handles.

- To move a slab or change the order of slabs, drag the slab handles.

- To change the direction of slabs, drag the slab direction handle.

- To remove a slab from the layout, select the slab handles and press the **Delete** key.

Adding and modifying detailing components

1. Click to open the detailing toolbar.

2. Do any of the following to add, remove or modify the detailing strips that control how and to which slabs the detailing components are added in the floor:

Button/ Option	Description
+	Add a new detailing strip. Pick the start and end points for the strip. In practice, the selected detailing component will be added into the slabs that the strip touches, see the example image below. You can also add new detailing strips by dragging an existing strip and holding down the **Ctrl** key.
✕	Delete a detailing strip. Select the strip that you want to delete. You can select multiple strips by holding down the **Shift** key.

Button/ Option	Description
Component name 	Select the actual detailing component from the **Component name** list. The content of the list may vary depending on your environment. You can also add new components to the list by selecting the **Add new** option. Note that you can only add detailing components that use one input part.
Component attribute file name 	Select the detailing component attributes that are used for the slab. Note that the content of the list varies depending on the component that you have selected.
	Modify the detailing component and/or attribute file of the detailing strip. Select the detailing strip and select the component name and/or attribute file from the list. Click the button to modify the selected detailing strips. You can select multiple strips by holding down the **Shift** key.
	Close the detailing toolbar to continue the other modifications of the floor layout. Note that the detailing toolbar is also closed when you select any other modification tool in the **Floor Layout** direct modification toolbar.

The image below shows an example of a floor layout containing three detailing strips.

Floor Layout CIP Filler

Floor Layout CIP Filler creates CIP filler parts into the empty areas in floors created with the **Floor Layout** modeling tool. The empty areas are spaces created by offsets around the exterior and interior boundaries. There may also be empty areas when the slab is too narrow to be created with the **Floor Layout** modeling tool.

Floor Layout CIP filler properties

Option	Description
Minimum width	Minimum width of the part. If the empty space is smaller than this width, the part is not created in this location.
Cast unit prefix, Start no	Cast unit prefix of the CIP filler part Cast unit start number of the CIP filler part
Name, Class, Material, Finish	Define the name, class, material and finish of the CIP filler parts.

1.7 Concrete stairs

Tekla Structures includes the following tools you can use to create concrete stairs and stairwells:

- Concrete stairs (7) on page 207

- Concrete stairs (65) on page 208
- Stairwells and elevator shafts (90) on page 223
- Reinforced Concrete Stair (95) on page 227

Concrete stairs (7)

Creates concrete stairs.

Where to use

Situation	More information
	A concrete staircase connected to two concrete slabs.

Before you start

Create two concrete slabs to connect the stairs to.

Defining properties

Use the tabs in the **Concrete stairs (7)** dialog box to define the following properties:

Tab	Contents	More information
Parameters	Run and rise dimensions, stair slab thickness	Defining stair properties on page 207
Part	Stairs properties	
Notch	Tolerance between the stairs and adjacent parts	Defining stair properties on page 207

Picking order

1. Pick a point to indicate the level of the lowest step.
2. Pick a point to indicate the level of the highest step.
3. Click two parts to which to connect the stairs. Tekla Structures fits the stairs to these parts using the values in the **Tolerance** fields on the **Notch** tab.

Click the middle mouse button to finish picking and create the stairs.

Defining stair properties

The illustration below shows the stairs properties that you can define.

1. Length
2. Length at top
3. Tolerances
4. Length at bottom
5. Slab thickness
6. Width
7. Height

In the **Approximate dimension** list box, select whether you want to specify the rise (**Height**) or the run (**Length**).

Defining stair tolerances

Tolerances are the gaps that Tekla Structures leaves between the ends of the stairs and adjacent structures. Use the **Notch** tab to define length, height, and width tolerances.

Concrete stairs (65)

Concrete stairs (65) creates precast stairs with optional landings, ridges, stringers, and anti-skid and kick plate profiles. It contains five different step shapes and options to create round or chamfered step edges.

Objects created

- Stairs
- Landings
- Ridges
- Stringers
- Anti-skid
- Kick plate

Use for

Situation	Description
	Staircase with • square step shape • stringers on both sides • top and bottom landings
	Staircase with • chamfered step corners • ridges on both sides
	Staircase with • rounded step corners • anti-skid profile • bottom landing

Situation	Description
	Staircase with • square step shape • rounded step corners • ridges on both sides • anti-skid and kick plate profiles

Selection order

Selection order depends on the selected **Creation method** option on the **Stairs** tab.

When **Creation method** is set to **Default**:

1. Pick the first point to indicate the start point of stairs.
2. Pick the second point to indicate the direction of stairs.

The stairs are created automatically.

Part identification key

	Part
1	Landings
2	Stringers
3	Ridges
4	Kick plates
5	Steps
6	Anti-skids

See also

Concrete stairs (65): Stairs tab on page 211

Concrete stairs (65): Step shapes tab on page 214

Concrete stairs (65): Landings tab on page 216

Concrete stairs (65): Ridges tab on page 217

Concrete stairs (65): Stringers tab on page 220

Concrete stairs (65): Anti-skid/Kick plate tab on page 221

Concrete stairs (65): Attributes tab on page 223

Concrete stairs (65): UDA tab on page 223

Concrete stairs (65): Stairs tab

Use the **Stairs** tab to define how the stairs are created and what are the dimensions of the stairs in the **Concrete stairs (65)** modeling tool.

Creation method

Option	Description
Creation method	Select the creation method of the stairs. **Creation method** defines how many and which points you need to pick when you create the stairs. • **Default** Number of steps is calculated automatically using the vertical distance between the two input points. • **One point – N & L & H**: define N, L, and H (number, length, and height of the steps). • **Two points – N & L**: define N and L. Height of the steps is calculated automatically using the vertical distance between the two input points. • **Two points – N & H**: define N and H. Length of the steps is calculated automatically using the horizontal distance between the two input points. • **Two points – L & H**: define L and H. Number of steps is calculated automatically using the vertical distance between the two input points. • **Two points – N**: define N. Height and length of steps is calculated automatically using the horizontal and vertical distance between the two input points. You can define the length of the top step.
No of steps, N	Define the number of steps, if the **Two points – L & H** option is not selected.
Position	Define the horizontal position of the staircase, relative to the line joining the picked points.
Distance	Enter a distance to move the stairs in the horizontal direction from the line joining the picked points. The selected **Position** option has an effect on this dimension.

Option	Description
Rotation	Define the angle between the line joining the steps and the line joining the selected points.
Cast unit type	Select the cast unit type: • **Precast** • **Cast in place**

Stair dimensions

	Description
1	Define height of the step.
2	Define length of the step.
3	Define length of the top step.
4	Define height of the top step.

	Description
5	Define width of the top corbel.
6	Define slope from the corbel to the underside of the stairs.
7	Define thickness of the slab.
8	Define chamfering for the underside of stairs.
9 10	Define chamfer or radius values.
11	Define height of the bottom step.
12	Define height of the stair foot.
13	Define chamfering for the bottom of stairs.
14	Define slope from the foot nose to the foot base.
15	Define length of the bottom cut.
16	Define length of the stair foot.
17	Define width of the staircase, including stringers.

Create foot Define whether a foot is created at the bottom of the stairs.

Concrete stairs (65): Step shapes tab

Use the **Step shapes** tab to define the shape of steps in the **Concrete stairs (65)** modeling tool.

All other steps/ Last step at foot Select the step shape. You can define the shape of the last step at the bottom if you set **Last step same as other step** to **No**.

Option	Description
	Default Beveled From the list below the step shape options, select whether you want to enter a dimension or an angle. AutoDefaults can change this option.

Option	Description
	Straight step
	Top straight, bottom beveled Enter two dimensions.
	Top beveled, bottom straight Enter the vertical dimension and then select whether you want to enter an angle or a horizontal dimension.
	Top and bottom staight Enter vertical dimensions and then select whether you want to enter an angle or a horizontal dimension.

out Select the shape of the step front edge and enter the required dimensions.

- **Default**

- **Radius**: Creates rounded edge.

- **Chamfer**: Creates chamfered edge.
- **No**: Creates square edge.

in Select the shape of the step inner corner and enter the required dimensions.

- **Default**
- **Radius**: Creates rounded inner corner.

- **Chamfer**: Creates chamfered inner corner.
- **No**: Creates square inner corner.

Concrete stairs (65): Landings tab

Use the **Landings** tab to create top and/or bottom landings in the **Concrete stairs (65)** modeling tool.

Create landings Define whether one or two landings are created, or whether there are no landings.

Landing dimensions and extensions

	Description
1	Define thickness of the bottom landing.
2	Define length of the bottom landing.
3	Define length of the top landing.
4	Define thickness of the top landing.
5 7	Define left and right extensions of the bottom landing.
6 8	Define left and right extensions of the top landing.

Concrete stairs (65): Ridges tab

Use the **Ridges** tab to create horizontal and/or vertical ridges on both sides or only on the other side of the stairs in the **Concrete stairs (65)** modeling tool.

Dimensions

	Description
1	Define thickness of the vertical ridge.
2	Define width of the vertical ridge.
3	Define width of the horizontal ridge.
4	Define thickness/height of the horizontal ridge.

Horizontal ridges

Part	Description
Create	Define whether horizontal ridges are created.
Create at foot	Define whether horizontal ridges are created at the foot of the stairs. This option works in the same way as the **Create** option.

Part	Description
Inside chamfer ①	Select the type of the inside chamfer and enter the required dimensions.
Corner chamfer ②	Select the type of the corner chamfer and enter the required dimensions.
Slope ③	Set the slope as an angle or a dimension. The slope makes the ridge inclined.
Foot corner chamfer	Define whether corner chamfers are created at the stair foot.

Top and foot ridges

Define the horizontal ridge length at the stair top step and at the stair foot. When no foot is created, vertical ridges continue to the bottom level of the stairs.

Vertical ridges

Part	Description
Create	Define whether vertical ridges are created.
Slope	Set the slope as an angle or a dimension. The slope makes the ridge inclined.

Concrete stairs (65): Stringers tab

Use the **Stringers** tab to create stringers and ridges on the left, right, or both sides of the stairs in the **Concrete stairs (65)** modeling tool.

Parts

Part	Description
Create stringers	Define whether stringers are created.
Create top ridge	Select an option to create top ridges.
Create bottom ridge	Select an option to create bottom ridges.
Slope	Use to incline the inner surface of the stringers. You can define the slope using an angle or a dimension.
Inside chamfer	Define whether the inside edge of stringer is chamfered or not.
Outside chamfer	Define whether the outside edge of stringer is chamfered or not.
Stringer height	Define how the stringer height is determined.

Dimensions

	Description
1	Define height of the stringer bottom ridge.
2	Define horizontal offset of the bottom ridge from the bottom step.
3	Define height of the stringer top ridge.
4	Define width of the stringer.
5	Define the difference in height that is the perpendicular distance between the step edge and stringer edge. You can enter this dimension if **Stringer height** is **Diff in height**.
6	Define total height of the stringer. You can enter this dimension if **Stringer height** is **Default** or **Total height**.

Concrete stairs (65): Anti-skid/Kick plate tab

Use the **Anti-skid** and **Kick plate** tabs to create slip resistant surfaces in the **Concrete stairs (65)** modeling tool.

Parts

Part	Description
Creation anti-skid **Creation kick plate**	Define whether anti-skids or kick plates are created.

Part	Description
Create anti-skid on foot	Define whether anti-skids are created at the stair foot.
Create cutout	Define whether you want to create cutouts with the anti-skid profile. By default, the cutouts are not created.
Include in cast unit	Define whether anti-skids or kick plates are included in cast unit.
Profile	Define the anti-skid or kick plate profile by selecting it from the profile catalog.
Rotation	Select an option to rotate the anti-skid or kick plate profile.

Anti-skid/kick plate position

	Description
1	Define anti-skid or kick plate distance from the left edge of the step.
2	Define anti-skid or kick plate distance from the right edge of the step.

	Description
1	Define distance from the step front edge.
2	Define depth for the anti-skid or kick plate.

Concrete stairs (65): Attributes tab

Use the **Attributes** tab to control the properties of different stair parts in the **Concrete stairs (65)** modeling tool.

Part attributes

Option	Description
Material	Define the material grade for the part.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.
Name	Define a name for the part. Tekla Structures uses the name in drawings and reports.
Class	Define the part class number.

Concrete stairs (65): UDA tab

Use the **UDA** tab to provide additional information about the parts created in the **Concrete stairs (65)** modeling tool.

The user-defined attributes are case sensitive. Check the correct spelling of an attribute from the `objects.inp` file.

Stairwells and elevator shafts (90)

Creates a rectangular stairwell or elevator shaft to represent the structure in the design stage. When you detail the structure, you can apply seams, connections, and reinforcement to complete the details of a stairwell or elevator shaft. This component uses the Concrete stairs (65) on page 208 component for creating stairs.

Parts created

- Wall panels
- Landings (optional)
- Flights of stairs (optional)
- Roof slab (optional)

- Openings (optional)

Where to use

Situation	More information
	A stairwell over four floors created using the default options. As shown, the points picked and the dimensions on the **Stairs and landings** tab determine the length of the stairwell. Walls consist of one or more precast concrete panels. 1 First point picked 2 Second point picked
	Stairwells always contain a first floor and a top floor. Indicate the number of additional floors on the **Plan view** tab, **Levels** field. This stairwell has 2 additional floors, for a total of 4 floors. 1 Top floor 2 Additional floor 3 Additional floor 4 1st or ground floor
	Elevator shaft created by selecting the **Elevator shaft** option on the **Stairs and landings** tab, **Type** field, for all floors.

Limitations The minimum floor height between floors is 200 mm.

Defining properties

Use the tabs in the **Stairwells and elevator shafts (90)** dialog box to define the following properties:

Tab	Contents	More information
Plan view	Number of floors or levels. Position of picked points. Length and width of shaft. Wall panel dimensions.	Defining stairwell properties on page 225
Openings	Opening dimensions for each wall panel and for the first, additional, and top floors.	Each wall may have a single opening, or no opening.
Stairs and landings	Stair and landing options and dimensions. Option to create elevator shaft (in the **Type** list boxes).	
Attributes	Basic part properties for walls, stairs, and landings. Option to use the saved properties from the Concrete stairs (65) on page 208 component for the first, additional, and top stairs.	

Picking order

1. On the **Plan view** tab, check the **Position** option. The default is in the middle of walls 3 and 4.

2. Pick a point to indicate the position of wall 3.
3. Pick a point to indicate the position of wall 4.

Defining stairwell properties

Use the **Plan view** tab in the **Stairwells and elevator shafts (90)** dialog box to define the properties of the stairwell or elevator shaft.

Defining number of floors and height

To define the number of levels and floor heights, enter a height for each floor level in the **Levels** field, for example:

You can also define negative floor levels, for example:

You can also use multiplier to define many floors, as in the following US imperial example:

Tekla Structures also uses these values to calculate the total height of the stairwell.

Defining position

To create the stairwell or elevator shaft, you pick two points in the model. Use the **Position** field to define the location of the stairwell or elevator shaft relative to these points. The options are:

- Middle (default)
- Left
- Right

Enter the offset distance from the selected position. Here, the offset is from the middle of wall.

Defining length and width

To manually define the total length and total width of the stairwell or elevator shaft, enter the following values:

L - total length

W - total width

Leave both fields blank to have Tekla Structures automatically calculate these dimensions from the the points picked:

1 First point picked.

2 Second point picked.

3 Total width, W

4 Total length, L

The dimensions defined on the **Stairs and landings** tab may override the total length or width that is automatically calculated or that you manually enter.

Reinforced Concrete Stair (95)

Reinforced Concrete Stair (95) creates reinforced concrete stairs.

Objects created

- Stairs
- Landings (optional)
- Ridges (optional)
- Stringers (optional)
- Anti-skid (optional)
- Main bars and stirrups of stair reinforcement (**Bar A** - **Bar L** tabs) (optional)
- Meshes in stairs and landings (optional)
- Mesh bars (optional)
- Anchor bars (optional)
- Landing end bars (optional)

Use for

Situation	Description
	Reinforced concrete stairs.
	Reinforced concrete stairs with chamfered steps, ridges and stringers.
	Reinforced concrete stairs with reinforced landings.

Selection order

1. Pick the first point to indicate the start point of the stairs.
2. Pick the second point to indicate the direction of the stairs.
3. Select any number of parts to be cut by the stairs (optional).
4. Click the middle mouse button to create the stairs.

See also Reinforced Concrete Stair (95): Parameters tab on page 230
Reinforced Concrete Stair (95): Parts tab on page 231
Reinforced Concrete Stair (95): Stairs and landings tab on page 232
Reinforced Concrete Stair (95): Ridges tab on page 233
Reinforced Concrete Stair (95): Stringers tab on page 235
Reinforced Concrete Stair (95): Anti-skid tab on page 236
Reinforced Concrete Stair (95): Bar A tab on page 237
Reinforced Concrete Stair (95): Bar B tab on page 237
Reinforced Concrete Stair (95): Bar C tab on page 239
Reinforced Concrete Stair (95): Bar D tab on page 240
Reinforced Concrete Stair (95): Bar F tab on page 243
Reinforced Concrete Stair (95): Bar E tab on page 241
Reinforced Concrete Stair (95): Bar G tab on page 244
Reinforced Concrete Stair (95): Bar H tab on page 245
Reinforced Concrete Stair (95): Bar I tab on page 246
Reinforced Concrete Stair (95): Bar J tab on page 247

Reinforced Concrete Stair (95): Bar K tab on page 248

Reinforced Concrete Stair (95): Bar L tab on page 249

Reinforced Concrete Stair (95): Mesh picture tab on page 250

Reinforced Concrete Stair (95): Mesh attributes tab on page 251

Reinforced Concrete Stair (95): Mesh bars tab on page 252

Reinforced Concrete Stair (95): Bottom anchor bars tab on page 252

Reinforced Concrete Stair (95): Top anchor bars tab on page 253

Reinforced Concrete Stair (95): Z anchor bars tab on page 253

Reinforced Concrete Stair (95): Landing end bars tab on page 254

Reinforced Concrete Stair (95): Configuration on page 255

Reinforced Concrete Stair (95): Parameters tab

Use the **Parameters** tab to control the shape of the stairs, number of steps, stair creation method and the stairs width in the **Reinforced Concrete Stair (95)** modeling tool.

Stair dimensions

	Description
1	Horizontal dimension of the steps area.
2	Vertical dimension of the steps area.
3	Step length.
4	Step height.
5	Shape options for stair bottom.
6	Shape options for stair top.
7	Stair width.

Stair creation method

Option	Description
P1 P2 step length	Create stairs between points P1 and P2. Stair dimension are defined by the distance between P1 and P2 and step length.
P1 P2 step height	Create stairs between points P1 and P2. Stair dimension are defined by the distance between P1 and P2 and step height.
P1 step length step height N steps	Create stairs from point P1 to point P2. Stair dimensions are defined by P1 and step height, length and number of steps.
P2 step length step height N steps	Create stairs from point P1 to point P2. Stair dimensions are defined by P2 and step height, length and number of steps.

Option	Description
P1 horizontal and vertical step distance	Create stairs from point P1 to point P2. Stair dimensions are defined by P1 and horizontal and vertical step distance.
P2 horizontal and vertical step distance	Create stairs from point P1 to point P2. Stair dimensions are defined by P2 and horizontal and vertical step distance.

Reinforced Concrete Stair (95): Parts tab

Use the **Parts** tab to control the material, name, class, positioning, cast unit type and step chamfering in the **Reinforced Concrete Stair (95)** modeling tool.

Step chamfer settings

Option	Description
Middle for all	All steps are chamfered according to the **Middle steps** settings.
Bottom different	Bottom step is chamfered according to the **1st step at bottom** settings. All other steps are chamfered according to the **Middle steps** settings.
Top different	Top step is chamfered according to the **Last top step** settings. All other steps are chamfered according to the **Middle steps** settings.
Bottom and top different	Bottom step is chamfered according to the **1st step at bottom** settings. Top step chamfered are according to the **Last top step** settings. All other steps are chamfered according to the **Middle steps** settings.

Step chamfer settings are set in the same way for **1st step at bottom**, **Middle steps** and **Last top step**.

Option	Description
	Size of step cut is defined by the distance.

Option	Description
	Size of step cut is defined by the angle.

Corner chamfers types

Corner chamfers **1.** and **2.** can be defined by the **Radius x** or by the sides of the **Chamfer x, y**.

Reinforced Concrete Stair (95): Stairs and landings tab

Use the **Stairs and landings** tab to control the size and type of the bottom landing and the top landing in the **Reinforced Concrete Stair (95)** modeling tool.

Bottom landing cut option

Option	Description
	Bottom landing with no cut.
	Bottom landing with cut defined by its length and distance from the top side of landing.
	Bottom landing with cut defined by its length and distance from the bottom side of landing.
	Bottom landing with L shape cut defined by its three dimensions and distance from the top side of landing.

Option	Description
	Bottom landing cut defined by a profile and its position in landing.

Top landing cut option

Option	Description
	Top landing with no cut.
	Top landing with cut defined by its length and distance from the top side of landing.
	Top landing with cut defined by its length and distance from the bottom side of landing.
	Top landing with L shape cut defined by its three dimensions and distance from the top side of landing.
	Top landing cut defined by a profile and its position in landing.

Landing dimensioning

	Description
	Use the top view setting to define the dimensions of the bottom landing and the top landing.

Reinforced Concrete Stair (95): Ridges tab

Use the **Ridges** tab to create horizontal and/or vertical ridges on both sides or only on one side of the stairs in the **Reinforced Concrete Stair (95)** modeling tool.

Dimensions

		Description
	1	Thickness of the vertical ridge.
	2	Width of the vertical ridge.
	3	Width of the horizontal ridge.
	4	Thickness/height of the horizontal ridge.

Horizontal ridges

Option	Description
Create	Define whether horizontal ridges are created.
Create at foot	Define whether horizontal ridges are created at the foot of the stairs. This option works in the same way as the **Create** option.
Chamfering 	**1 Inside chamfer**: Select the type of the inside chamfer and enter the required dimensions. **2 Corner chamfer**: Select the type of the corner chamfer and enter the required dimensions. **3 Slope**: Set the slope as an angle or a dimension. The slope makes the ridge inclined.
Foot corner chamfer	Define whether corner chamfers are created at the stair foot.

Vertical ridges

Option	Description
Create	Define whether vertical ridges are created.
Slope	Set the slope as an angle or a dimension. The slope makes the ridge inclined.

Reinforced Concrete Stair (95): Stringers tab

Use the **Stringers** tab to create the stringers and ridges on the left, right, or both sides of the stairs in the **Reinforced Concrete Stair (95)** modeling tool.

		Description
	1	Height of the stringer bottom ridge.
	2	Horizontal offset of the bottom ridge from the bottom step.
	3	Height of the stringer top ridge.
	4	Width of the stringer.
Slope		Use to incline the inner surface of the stringers. You can define the slope using an angle or a dimension.
Inside chamfer		Define whether the inside edge of stringer is chamfered or not.
Outside chamfer		Define whether the outside edge of stringer is chamfered or not.

Stringer height

Stringer height based on the total height or distance from inner/outer step corner.

Reinforced Concrete Stair (95): Anti-skid tab

Use the **Anti-skid** tab to create slip resistant surfaces in the **Reinforced Concrete Stair (95)** modeling tool.

Option	Description
Creation anti-skid	Define whether anti-skids are created.

Option	Description
Create anti-skid on foot	Define whether anti-skids are created at the stair foot.
Create cutout	Define whether you want to create cutouts with the anti-skid profile. By default, the cutouts are not created.
Include in cast unit	Define whether anti-skids are included in the cast unit.
Profile	Define the anti-skid profile by selecting it from the profile catalog.
Rotation	Select an option to rotate the anti-skid profile.

Reinforced Concrete Stair (95): Bar A tab

Use the **Bar A** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Reinforced Concrete Stair (95): Bar B tab

Use the **Bar B** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Bar B bottom ending options

Define geometry and concrete cover thickness of the bar B bottom ending.

Option	Description
	Bar B simple bottom ending.
	Bar B bottom ending bent once to fit the bottom footing of the stairs.
	Bar B bottom ending bent twice to fit the bottom footing of the stairs.
	Bar B bottom ending bent up to three times to fit the bottom footing of the stairs.

Bar B top ending options

Define geometry and concrete cover thickness of the bar B top ending.

Option	Description
	Bar B simple top ending.
	Bar B top ending bent once to fit the top geometry of the stairs.

Option	Description
	Bar B top ending bent twice to fit the top geometry of the stairs. Last leg length is defined by the cover thickness.
	Bar B top ending bent twice to fit the top geometry of the stairs.

Reinforcing bar spacing

Option	Description
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Reinforced Concrete Stair (95): Bar C tab

Use the **Bar C** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Option	Description
	Geometry and concrete cover thickness.

Option	Description
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Extra I bars

Option	Description
	The purple color in the dialog box represents the extra I bars.
	Define whether to create the extra I bars. Select the check boxes next to the purple points.
	Cover thickness of the reinforcing bars.

Reinforced Concrete Stair (95): Bar D tab

Use the **Bar D** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Extra I bars Extra I bars are straight bars crossing bar D. You can create up to four extra I bars.

Option	Description
	The purple color in the dialog box represents the extra I bars.
	Define whether to create the extra I bars. Select the check boxes next to the purple points.
	Cover thickness of the reinforcing bars.

Reinforced Concrete Stair (95): Bar E tab

Use the **Bar E** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Create options

Option	Description
No	Bar is not created.
Stirrup	Bar is created as stirrup.
Pin	Bar is created as pin.

Option	Description

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Reinforced Concrete Stair (95): Bar F tab

Use the **Bar F** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Reinforced Concrete Stair (95): Bar G tab

Use the **Bar G** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Extra I bars Extra I bars are straight bars crossing bar G. You can create up to four extra I bars.

Option	Description
	The purple color in the dialog box represents the extra I bars.
	Define whether to create extra I bars. Select the check boxes next to the purple points.
	Cover thickness of the reinforcing bars.

Reinforced Concrete Stair (95): Bar H tab

Use the **Bar H** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Create option

Option	Description
	• Do not create bar H • Create bar H • Create only bottom bar H reinforcing bar group

Bar H geometry option

Option	Description
	Two bent reinforcing bar groups. Define geometry and concrete cover thickness.
	Top reinforcing bar group is straight, bottom reinforcing bar group is bent on both sides. Define geometry and concrete cover thickness.
	Two bent reinforcing bar groups. Define geometry and concrete cover thickness.

Option	Description
	Spacing, number of reinforcing bars and concrete cover thickness of the reinforcing bar group.

Reinforced Concrete Stair (95): Bar I tab

Use the **Bar I** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Option	Description
	Positioning, spacing and concrete cover thickness.
	Concrete cover thickness of the reinforcing bar groups.

Reinforced Concrete Stair (95): Bar J tab

Use the **Bar J** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool. You can create several bar J reinforcing bar groups. Each group has its own color representation on the **Bar J** tab.

Color group options

Option	Description
	Use the color group option to define which combination of bar J reinforcing bar groups (blue, green, brown) is created.

Option	Description
	Positioning and concrete cover thickness for reinforcing bar groups based on the color representation.
	Concrete cover thickness for all reinforcing bar groups.

Reinforced Concrete Stair (95): Bar K tab

Use the **Bar K** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Option	Description
	Geometry and concrete cover thickness.
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Extra I bars Extra I bars are straight bars crossing bar K. You can create up to three extra I bars.

Option	Description
	The purple color in the dialog box represents the extra I bars.
	Define whether to create extra I bars by selecting the check boxes next to the purple points.
	Cover thickness of the reinforcing bars.

Reinforced Concrete Stair (95): Bar L tab

Use the **Bar L** tab to define the geometry, concrete cover thickness, reinforcing bar spacing and reinforcing bar properties of a stair reinforcing bar group in the **Reinforced Concrete Stair (95)** modeling tool.

Option	Description
	Define whether an L shaped reinforcing bar group is created at the stair footing.
	Define offsets for the reinforcing bar groups in the two top steps.
	Define geometry for reinforcing bar groups in all steps, except for the two top steps.

Option	Description
	Spacing, number of bars and concrete cover thickness of the reinforcing bar group.

Bar L geometry options

Option	Description
	L shape reinforcing bar groups.
	L shape reinforcing bar groups with hooks.

Reinforced Concrete Stair (95): Mesh picture tab

Use the **Mesh picture** tab with the **Mesh attributes** tab to define bottom landing meshes, stair meshes and top landing meshes in the **Reinforced Concrete Stair (95)** modeling tool.

Side view

		Description
	1	Bottom landing mesh (top). Define offsets and cover thickness of the mesh.
	2	Bottom landing mesh (bottom). Define offsets and cover thickness of the mesh.
	3	Stair mesh (top). Define offsets and cover thickness of the mesh.
	4	Stair mesh (bottom). Define offsets and cover thickness of the mesh.
	5	Top landing mesh (top). Define offsets and cover thickness of the mesh.
	6	Top landing mesh (bottom). Define offsets and cover thickness of the mesh.

Top view

		Description
	1	Cover thickness of bottom landing mesh (top).
	2	Cover thickness of bottom landing mesh (bottom).
	3	Cover thickness of stair mesh (top).
	4	Cover thickness of stair mesh (bottom).
	5	Cover thickness of top landing mesh (top).
	6	Cover thickness of top landing mesh (bottom).

Reinforced Concrete Stair (95): Mesh attributes tab

Use the **Mesh attributes** tab with the **Mesh picture** tab to define the properties, positioning and creation type of bottom landing meshes, stair meshes and top landing meshes in the **Reinforced Concrete Stair (95)** modeling tool.

Cross bar location

	Description
	Cross bar above.
	Cross bar below.

Creation type

	Description
	Create mesh as reinforcement mesh.
	Create mesh as two independent reinforcing groups.

Reinforced Concrete Stair (95): Mesh bars tab

Use the **Mesh bars** tab to define the geometry, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties of the mesh bars in the top and bottom landing in the **Reinforced Concrete Stair (95)** modeling tool. You can create up to three reinforcing bar groups.

		Description
	1	Cover thicknesses of all mesh bar groups in the stair bottom landing. Use the color representation in the dialog box.
	2	Vertical cover thicknesses of all mesh bar groups in the stair bottom landing. Use the color representation in the dialog box.
	3	Cover thicknesses of all mesh bar groups in the stair top landing. Use the color representation in the dialog box.
	4	Vertical cover thicknesses of all mesh bar groups in the stair top landing. Use the color representation in the dialog box.

Reinforced Concrete Stair (95): Bottom anchor bars tab

Use the **Bottom anchor bars** tab to define the geometry, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties of the bottom anchor bars in the **Reinforced Concrete Stair (95)** modeling tool. You can create up to three reinforcing bar groups.

Creation type

Option	Description
No	No reinforcing bar group created.
Type1	L shape anchor reinforcing bar group.
Type2	Simple straight anchor reinforcing bar group.

Option	Description

Reinforced Concrete Stair (95): Top anchor bars tab

Use the **Top anchor bars** tab to define the geometry, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties of the top anchor bars in the **Reinforced Concrete Stair (95)** modeling tool. You can create up to three reinforcing bar groups.

Creation type

Option	Description
No	No reinforcing bar group created.
Type1	L shape anchor reinforcing bar group.
Type2	Simple straight anchor reinforcing bar group.

Reinforced Concrete Stair (95): Z anchor bars tab

Use the **Z anchor bars** tab to define the geometry, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties of the Z anchor bars in the **Reinforced Concrete Stair (95)** modeling tool. You can create up to three reinforcing bar groups.

		Description
	1	Cover thicknesses of all Z anchor bar groups in the stair bottom landing. Use the color representation in the dialog box.
	2	Vertical cover thicknesses of all Z anchor bar groups in the stair main part. Use the color representation in the dialog box.
	3	Cover thicknesses of all Z anchor bar groups in the stair top landing. Use the color representation in the dialog box.
	Bot L	Length of the Z anchor bars that extend from the stair bottom landing.
	Top L	Length of the Z anchor bars that extend from the stair top landing.

Reinforced Concrete Stair (95): Landing end bars tab

Use the **Landing end bars** tab to reinforce the stair landings and to define the geometry, concrete cover thickness, reinforcing bar spacing, and bar properties of the landing end bars in the **Reinforced Concrete Stair (95)** modeling tool.

Bottom end Reinforcement of bottom landing.

		Description
	1	Landing end bars. U-shape geometry follows the shape of the landing based on the defined cover thicknesses. Color representation in the picture: blue, green.
	2	Define whether to create extra cross bars. Select the check boxes next to the purple points.
	3	Define whether to create extra cross bars. Select the check boxes next to the red points.

Top end Reinforcement of top landing.

		Description
	1	Landing end bars. U-shape geometry follows the shape of the landing based on defined cover thicknesses. Color representation in the picture: blue, green.
	2	Define whether to create extra cross bars. Select the check boxes next to purple points.
	3	Define whether to create extra cross bars. Select the check boxes next to red points.

Side cover thickness

Option	Description
	Side cover thickness of all reinforcing bar groups defined on the **Landing end bars** tab.

Reinforced Concrete Stair (95): Configuration

Use the **Configuration** tab to define the default cover thicknesses and bending radiuses of all reinforcing bars created by the **Reinforced Concrete Stair (95)** modeling tool.

Cover thickness for reinforcing bar groups A - C

Define the default cover thickness for reinforcing bar groups A - C.

Option	Description
Value	Default cover thickness defined by a value.
Rebar diameter	Default cover thickness multiplied by reinforcing bar diameter.

General cover thickness bars D - L

Define the default cover thickness for reinforcing bar groups D - L and all reinforcing bars except the reinforcing bar groups A - C and meshes.

Option	Description
Value	Default cover thickness defined by a value.
Rebar diameter	Default cover thickness multiplied by reinforcing bar diameter.

Bending radius

Define the default bending radius of all reinforcing bars.

Option	Description
Rebar_database.inp	Default bending radius defined by `rebar_database.inp` file.
Relative to diameter	Default bending radius according to the reinforcing bar diameter.
Bending radius	Default bending radius defined by value.

Mesh cover thickness Define the default cover thickness for the meshes created by the **Reinforced Concrete Stair (95)** modeling tool.

1.8 Foundations

This section introduces components that can be used in concrete foundations.

Click the links below to find out more:

- Concrete foundation (1030) on page 256

Concrete foundation (1030)

Concrete foundation (1030) creates a concrete foundation plate at the bottom of a selected concrete column.

Objects created

- Concrete foundation plate
- Injection tubes and injection hoses in concrete column
- Up to 4 concrete piles under the foundation plate (optional)
- Stirrups for the reinforcing bars

Use for

Situation	Description
	Concrete foundation plate with chamfers, piles, injection tubes with curved injection hoses, reinforcing bars and stirrups.

Situation	Description
	Concrete foundation plate with more than 4 edges, recesses in the foundation plate, and multiple reinforcing bars.

Selection order

1. Select a concrete column.
2. Pick a point.

   The concrete foundation plate is created automatically when the point is picked.

Part identification key

	Part
1	Concrete column
2	Point The point defines the new bottom level of the column.

See also

Concrete foundation (1030): Picture tab on page 258

Concrete foundation (1030): Parts tab on page 261

Concrete foundation (1030): Massive tab on page 263

Concrete foundation (1030): Piles tab on page 265

Concrete foundation (1030): Inj. tube and rebar tab on page 266

Concrete foundation (1030): Column tab on page 268

Concrete foundation (1030): Stirrups tab on page 269

Concrete foundation (1030): Picture tab

Use the **Picture** tab to control the dimensions of the reinforcing bars and injection tubes in the **Concrete foundation (1030)** detail.

Dimensions

	Description	Default
1	Define the length of the injection tube extension from the top of reinforcing bar.	180 mm
2	Define the reinforcing bar height in the column.	

	Description	Default
3	Define the reinforcing bar hook length.	`10 * reinforcing bar diameter`
4 5	Define the reinforcing bar top and bottom cover thickness. The bottom cover thickness is used only if you leave the box for the top cover thickness empty.	`0,5 * foundation plate thickness`
6	Define the pile thickness. The pile thickness value on the **Parts** tab overrides this value.	`300 mm`
7	Define the distance from the bottom of the foundation plate to the bottom of the pile.	`5000 mm`
8	Define the pile depth in the foundation plate.	`50 mm`
9	Define the foundation plate thickness.	`800 mm`
10	Define the distance between the column and the foundation plate.	`0`
11	Define the vertical offset from the picked point.	`0`
12	Define the reinforcing bar height.	`800 mm`
13	Define the injection tube height.	`580 mm`
14	Define the vertical offset for the rear injection tubes if the tubes are pointing to the same direction.	

Injection tube

Option	Description
	Injection tube in one piece, 90 degree angle.

Option	Description
	Injection tube in one piece. Use value **a** to define the slope of the tube.
	Injection tube and a 90 degree injection hose.
	Injection tube and a sloped injection hose. Use value **a** to defne the slope of the hose.
	Straight injection tube. No injection tube opening is created.

	Description	Default
1	Define the height of a sloped injection tube or hose angle.	30 mm
2	Define the height of a curved injection hose.	0 mm

Concrete foundation (1030): Parts tab

Use the **Parts** tab to control the profile properties in the **Concrete foundation (1030)** detail.

Concrete massive, Piles

Define the profile properties of the concrete foundation plate and the concrete piles.

If you leave the pile thickness value empty on the **Parts** tab, the pile thickness value defined on the **Picture** tab is used instead together with the **Pile profile type prefix (e.g. D)**.

Option	Description
t, b, h	Define the part thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number.
Comment	Add a comment for the part.

Reinforcing bars

Option	Description
Type of reinforcing bars	Select the profile type for the reinforcing bars. • **Default**: the same as **Reinforcing bar** • **Poly-profile**: profile catalog • **Reinforcing bar**: reinforcing bar catalog
Reinforcement bars	Define the size of the reinforcing bars.

Option	Description
Radius	Define the radius of the reinforcing bar hook.
Reinf. bars belong to	Define to which part the reinforcing bars belong. • **Default**: the same as **Columns** • **Concrete massive**: reinforcing bars belong to the concrete foundation plate. • **Column**: reinforcing bars belong to the column.

Injection tubes and curve

Option	Description
Injection tubes **Curve**	Define the size of the injection tubes. Define the size of the injection hoses.
Article number	Define an article number for the injection hose and the injection tube. The entered values are saved to the user-defined attribute of the part.
	Define how the injection hoses are connected to the injection tubes. • **Default**: injection hoses are loose parts. • **Part-add gain**: injection hoses are added to the injection tubes. • **Weld gain**: injection hoses are welded to the injection tubes. • **Cast unit column**: injection hoses are added to the column. • **Weld to column**: injection hoses are welded to the column.

Option	Description
Pile profile type prefix (e.g. D)	Define the default parametric profile prefix for the piles. This value works only if you have set the pile thickness on the **Picture** tab. You can override this value by setting the pile thickness on the **Parts** tab.
Reinf. bar profile type prefix (e.g.D)	Define the default parametric profile prefix for the reinforcing bar profiles.

Concrete foundation (1030): Massive tab

Use the **Massive** tab to control the shape and the dimensions of the concrete foundation plate and the recesses in the **Concrete foundation (1030)** detail.

Concrete foundation plate type

Option	Description
Plate type for rectangular massive	Select the plate type for the rectangular foundation plate.
Massive direction	Select the direction of the foundation plate.

Concrete foundation plate dimensions

	Description
1	Define the offset from the concrete column in the x direction.
2	Define the offset from the concrete column in the y direction.
3	Select the shape of the foundation plate.
4	Define the width of the foundation plate.

	Description
5 6	Define the length of the straight part of the foundation plate.
7	Define the offset from the foundation plate center line.
8	Define the width of the straight part of the foundation plate.
9	Define the length of the foundation plate.

Recess dimensions

	Description
1	Select the type of the recess.
2	Define the width of the recesses.
3	Define the depth of the recesses.
4	Define the distance between the bottom of the reinforcing bar and the bottom of the recess.
5	Define the distance between the center line of the reinforcing bar and the left side of the recess.
6	Define the distance between the center line of the reinforcing bar and the right side of the recess.

Option	Description
Chamfer type for rectangular massive	Define the chamfer type for the rectangular foundation plate.

Option	Description
Chamfer X	Define the chamfer size in the x direction.
Chamfer Y	Define the chamfer size in the y direction.

Concrete foundation (1030): Piles tab

Use the **Piles** tab to control the position of the piles in the **Concrete foundation (1030)** detail.

Pile positions

	Description
1	Select the position and the offset of the piles.
2	Define the distance between the first pile and the column in the x direction.
4	Define the distance between the second pile and the column in the x direction.
6	Define the distance between the third pile and the column in the x direction.
8	Define the distance between the fourth pile and the column in the x direction.
3	Define the distance between the first pile and the column in y direction.
5	Define the distance between the second pile and the column in y direction.
7	Define the distance between the third pile and the column in y direction.
9	Define the distance between the fourth pile and the column in y direction.
10	Define the pile offset from the concrete column in the x direction.
	Define the pile offset from the concrete column in the y direction.

	Description
11	

Option	Description
Piles direction	Define the direction of the piles.
Join piles to massive	Define how the piles are connected to the foundation plate.

Concrete foundation (1030): Inj. tube and rebar tab

Use the **Inj. tube and rebar** tab to control the creation, number and position of the reinforcing bars and injection tubes in the **Concrete foundation (1030)** detail.

Option	Description
Create	Define which parts are created.
Injection tubes bend radius	Define the radius of the injection tubes.

Injection tube properties

	Description
1	Select the direction of the bent bars.
2	Select the direction of the injection hoses.
3	Define the number of injection tubes.
4	Define the center-to-center distance between the injection tubes.
5	Define the level of the curved injection parts. This is useful if the curved parts are pointing to the same direction. You can define the levels on the **Picture** tab. Example:

	Description
6	Define the distance from the center of the injection tube to the outer contour of the concrete foundation.

Concrete foundation (1030): Column tab

Use the **Column** tab to control the dimensions of the column cut-out in the **Concrete foundation (1030)** detail.

Column dimensions

	Description
1	Define the size of the column cut-out.
2	Define the height of the column cut-out.

Concrete foundation (1030): Stirrups tab

Use the **Stirrups** tab to control the stirrup properties in the **Concrete foundation (1030)** detail.

Stirrup properties

Option	Description
	Define whether the stirrups are created. Define the length of the laps.
Grade	Define the grade of the stirrups.
Size	Define the size of the stirrups.
End conditions left	Define the stirrup shape on the left.
End conditions right	Define the stirrup shape on the right.
Bend lengths left	Define the stirrup length on the left.
Bend lengths right	Define the stirrup length on the right.
Creation method	Define the creation method of the stirrups.

Option	Description
	Define a comment, name, class, serie and start number for the stirrups.

	Description
1	Select the start point of stirrups, either from the bottom of the column or the top of the reinforcing bars.
2	Define the end of the stirrups from the bottom of the reinforcing bars.

2 Reinforcement

This section introduces components that can be used in reinforcement.

2.1 Reinforcement for foundations

Tekla Structures includes the following components that you can use to automatically create reinforcement for foundations:

Component	Description
Strip footing (75) on page 271	Creates reinforcement for a concrete strip footing.
Pile cap reinforcement (76) on page 273	Creates reinforcement for a concrete pile cap.
Pad footing (77) on page 277	Creates reinforcement for a concrete pad footing.
Starter bars on page 279	Creates starter bars in a footing and a pedestal for a column.
	Creates starter bars in a footing for a column.

Strip footing (75)

Creates reinforcement for a concrete strip footing.

Bars created

- Longitudinal bars for the top and bottom surfaces and sides of the footing
- Stirrups

Use for

Situation	More information
Straight strip footings that have rectangular cross sections	

Situation	More information
	Main bars entirely inside the footing, no side bars, stirrup laps at stirrup corners.
	Main bars protruding from the footing, two bars on both sides, stirrup laps in the middle of the top surface.

Do not use for

Footings that have:

- Irregular cross sections
- Skew or cut corners

Before you start

- Create the concrete strip footing.
- Calculate the required area of reinforcement.

Defining properties

Use the following tabs in the **Strip footing (75)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Picture	Concrete cover thickness, location of side bars and first stirrups	
Main bars	Grade, size, number, spacing, and bond lengths of the top, bottom, left, and right bars	Strip footing reinforcement properties on page 272
Stirrups	Grade, size, spacing, and bend type of stirrups	
Attributes	Numbering properties, name, and class of the top, bottom, left, and right bars, and stirrups	

Picking order

1. Concrete strip footing.

Strip footing reinforcement properties

Bond length of main bars

Bond lengths define how far main bars extend into adjacent structures at the ends of strip footings. Use the **Bond length 1** fields on the **Main bars** tab for the first end of the footing (with the yellow handle), and the **Bond length 2** fields for the second end of the footing (with the magenta handle).

You can define bond lengths separately for:

- Top bars
- Bottom bars
- Bars on the left side of the footing
- Bars on the right side of the footing

Bend type for stirrups

To define the location of the stirrup laps in the strip footing, select an option from the **Bend type** list box on the **Stirrups** tab. The options are:

Option	Examples
At mid	
At corner	

Pile cap reinforcement (76)

Creates reinforcement for a concrete pile cap.

Bars created

- Bars in two directions for the top and bottom surfaces of the pile cap
- Lacer bars

Use for

Situation	More information
	Bottom bars concentrated over the piles, top bars under the column. Two lacer bars.

Situation	More information
	Bars evenly distributed on the bottom and top surfaces. No lacer bars.
Rectangular footings with or without cut corners, footings that are skewed on one or both sides	Pad footing and pile cap shapes on page 275
Bars on the top, bottom, or both surfaces of the footing	
Straight or bent bar ends	

Before you start

- Create the concrete pile cap.
- Calculate the required area of reinforcement.

Defining properties

Use the following tabs in the **Pilecap reinforcement (76)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Picture	Concrete cover thickness, primary bar direction	
Primary top bars	Grade, size, hooks, bend lengths, distribution (by number or spacing), and arrangement/location of the top and bottom bars in two directions	Bar distribution on page 275
Secondary top bars		
Primary bottom bars		
Secondary bottom bars		
Lacer bars	Option to create lacer bars, grade, size, number, spacing, location, type, orientation, and laps of lacer bars	Lacer bars for pad footings and pile caps on page 276
Attributes	Numbering properties, name, and class of the top, bottom, and lacer bars	

Picking order

1. Concrete pile cap
2. Piles and/or columns

3. Click the middle mouse button to finish.

Pad footing and pile cap shapes

Use the **Pile cap reinforcement (76)** and **Pad footing (77)** tools to create reinforcement for the following shapes of foundations:

Shape	Description
	Rectangular
	Skewed on two sides
	Skewed on one side
	Rectangular with cut corners

Bar distribution

Pad footings In pad footings, you can arrange the main reinforcing bars into:

- One zone of bars that have the same bar properties
- Three zones of bars that have different bar properties

You set the options separately for primary and secondary bars. Select an option from the **Arrangement** list box on the relevant tab in the **Pad footing (77)** dialog box.

Pile caps Define the pile cap reinforcement using the following properties in the **Pile cap reinforcement (76)** dialog box:

Field	Description
To suit pile/column	Select **Yes** to concentrate main bars over piles and under columns.

Field	Description
% of pile/column width	The area where the bars are concentrated, as a percentage of the width of the pile or column. For example, if the pile diameter or width is 500 mm, enter 120 in the **% of pile width** field to concentrate bars in a 600 mm-wide area over the pile.
Bar portioning (%)	The proportion of bars concentrated over a pile or under a column.
Bar portioning (number of bars)	The number of bars concentrated over a pile or under a column.

You can define these properties separately for:

- Primary top bars
- Secondary top bars
- Primary bottom bars
- Secondary bottom bars

Lacer bars for pad footings and pile caps

Lacer bars are reinforcing bars that loop around the sides of a concrete footing.

You can create up to six different groups of lacer bars in a footing. Each group can contain different values for:

- Grade
- Bar size
- Number of bars
- Spacing
- Shape
- Dimensions

To create lacer bars for a footing:

1. Open the footing reinforcement properties dialog box and click the **Lacer bars** tab.
2. In the **Lacer bar option** list box, select **Yes** to create lacer bars.
3. Enter properties for each lacer bar group:

Property	Description
	Quantity, spacing, and location of lacer bar groups. Tekla Structures only uses information from some of the fields, in this order of priority: 1. **Number of bars** and **Spacing** 2. **Number of bars**, **Start**, and **End** 3. **Spacing**, **Start**, and **End**
Type	The number and location of laps. The options are:
Orientation	The options are **Default**, **Front**, and **Back**.
p1...p4 **L1...L4**	The exact locations and lengths of lacer bar laps. Locations are measured from the corner of the bar to the midpoints of laps.
	The location of the topmost lacer bar, measured from the end of main bars. Entering a value here overrides the location defined in the **End** field.

Pad footing (77)

Creates reinforcement for a concrete pad footing.

Bars created

- Bars in two directions for the bottom surface of the pad footing

- Lacer bars

Use for

Situation	More information
	Rectangular footing, 90-degree hooks at the primary bar ends, 180-degree hooks at the secondary bar ends, no lacer bars.
	Rectangular footing, three zones of primary bars with different spacing, straight bar ends, three lacer bars.
	Footing skewed on two sides, two groups of lacer bars with different spacing.
Rectangular footings with or without cut corners, footings that are skewed on one or both sides	Pad footing and pile cap shapes on page 275

Before you start

- Create the concrete pad footing.
- Calculate the required area of reinforcement.

Defining properties

Use the following tabs in the **Pad footing (77)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Picture	Concrete cover thickness	
Primary bar	Arrangement, grade, size, hooks, bend lengths, and distribution (by number or spacing) of the primary and	Bar distribution on page 275
Secondary bar		

Tab	Contents	See also
	secondary bars, primary bar direction	
Lacer bar	Grade, size, number, spacing, location, type, orientation, and laps of lacer bars	Lacer bars for pad footings and pile caps on page 276
Attributes	Numbering properties, name, and class of the primary, secondary, and lacer bars	

Picking order

1. Concrete pad footing

Starter bars

Starter bars for pillar (86) and **Starter bars for footing (87)** create starter bars in a footing for a column. Starter bars may go through a pedestal or straight to the column. Starter bars can be in a rectangular or circular form.

Bars created

- Starter bars (straight or L-shaped)
- Stirrups (optional)

Use for

Situation	More information
	Starter bars for footing (87) places starter bars in a footing in rectangular or circular form. Starter bars can be straight or L-shaped, and can have stirrups.
	Starter bars for pillar (86) creates straight or L-shaped starter bars that go through a rectangular or circular pedestal. Starter bars can have stirrups.

Before you start

- Create the footing.
- Calculate the required area of reinforcement.

Defining properties

Use the following tabs in the **Starter bars for pillar (86)** and **Starter bars for footing (87)** dialog boxes to define the component properties:

Tab	Contents	See also
Picture	**86**: Bar dimensions and locations, number and spacing of bars, stirrup type, concrete cover thickness	Starter bar stirrups on page 280
	87: Bar dimensions and locations, number and spacing of bars, concrete cover thickness	
Location	**87** only: Location of bars, stirrup type	Starter bar location on page 281 Starter bar stirrups on page 280
Parameters	Grade, size, name, class, and numbering properties of bars	

Picking order

1. Footing
2. Pedestal (only for **Starter bars for pillar (86)**)

Starter bar stirrups

Use these options to define stirrup laps in footings:

Option	Description
	Laps on the side of the stirrups 45-degree hooks at bar ends
	Laps at stirrup corners 135-degree hooks at bar ends
	Laps at stirrup corners 90-degree hooks at bar ends
	If the starter bars are in a circular form, you must define the angle of stirrup overlap.

Starter bar location

When you use **Starter bars for footing (87)**, you must define the location of starter bars. Enter the distance from the center of the bar group to the center of the footing, in two directions:

2.2 Beam, column, and slab reinforcement

Tekla Structures includes the following components that you can use to automatically create reinforcement for beams, columns, and slabs:

- Mesh Bars on page 282
- Mesh Bars by Area on page 287
- Slab bars (18) on page 294
- Slab Reinforcement Tool on page 296
- Beam reinforcement (63) on page 299
- Double tee reinforcement (64) on page 302
- Beam end reinforcement (79) on page 305
- Corbel reinforcement (81) on page 308
- Round column reinforcement (82) on page 313
- Rectangular column reinforcement (83) on page 316
- Hole reinforcement for slabs and walls (84) on page 326
- Hole creation and reinforcement (85) on page 328
- Braced girder (88) on page 331
- Braced girder (89) on page 349
- Reinforcement mesh array in area (89) on page 366
- Wall Panel Reinforcement on page 370
- Multiple Wire Size Mesh on page 377
- Embedded anchors (8) on page 380
- Embed (1008) on page 411
- Continuous Beam Reinforcement on page 420

Mesh Bars

Mesh Bars creates reinforcement for concrete slabs or walls.

Objects created

- Primary bars
- Crossing bars

Use for

Situation	Description
	Reinforcement for the bottom or the top surface of the concrete element, or for both.

Selection order

1. Select a concrete slab.

   The reinforcement is created automatically.

See also

Mesh Bars: Picture tab on page 282

Mesh Bars: Detailing tab on page 284

Mesh Bars: Bar end conditions tab on page 286

Mesh Bars: Attributes tab on page 287

Mesh Bars: Picture tab

Use the **Picture** tab to define how the top and bottom bars are created in the **Mesh Bars** modeling tool.

Option	Description
Create bars	• Top bars with primary bars above secondary bars • Top bars with secondary bars above primary bars • No bars The same options are available for the bottom bars as well. The value defined in the box is the cover thickness for the top/bottom layer.
Same as bottom	Select whether the top bars are created using the same properties as the bottom bars. If you select **No**, enter the top bar properties.
Bar type	Select whether the bars are created as a bar group or as a mesh.
Size	Select the diameter of a bar. You can set the size separately for the top primary and secondary bars, and for the bottom primary and secondary bars.
Grade	Define the strength of the steel used in the bars. You can set the grade separately for the top primary and secondary bars, and for the bottom primary and secondary bars.
Spacing type	Select whether the spacing is based on the number of bars or the spacing value. • **By spacings**: enter the distance between the bars in the **Spacing** box. • **Number of bars**: enter the number of bars in the **Number of bars** box.
Spacing	Distance between the bars. Tekla Structures calculates the number of bars. You can set the spacing separately for the top primary and secondary bars, and for the bottom primary and secondary bars.

Option	Description
Number of bars	Number of bars. Tekla Structures calculates the distance between bars. You can set the number of bars separately for the top primary and secondary bars, and for the bottom primary and secondary bars.
Primary bars direction	The direction of the primary bars. Use to change bar direction. The options are: • **Auto** • **Use slab x direction** • **Use slab y direction** • **Use global x direction** • **Use global y direction** If the **Auto** option is selected, the primary bars are created along the two longest parallel sides of a slab. If there are no parallel sides, then the bars are created along the **slab x direction** instead.
Up direction	Select which sides are regarded as the top and bottom sides of the slab. If the **Auto** option is selected, then the bars are created on the side with the greatest area.

Mesh Bars: Detailing tab

Use the **Detailing** tab to control how the bars are distributed in the **Mesh Bars** modeling tool.

Option	Description
Bar edge distances	Define the edge distance of the bars.
Adjustment	Select how the bars are distributed. The same options are available for both the primary and the secondary bars.
	 Equal distribution by target spacing value.

Option	Description
	 By exact spacing value with flexible first space without first bar.
	 By exact spacing value with flexible last space without last bar.
	 By exact spacing value with flexible first and last space without first and last bars.
	 By exact spacing value with flexible first space.
	 By exact spacing value with flexible last space.
	 By exact spacing value with flexible first and last space.
Bar grouping	Select whether tapered bars are grouped or not. If you group the tapered bars, select how the bars along tapered the edge are handled.

Option	Description
	• Tapered bars are handled normally. • Creates as many bars as possible with same length in the tapered edge.
Holes	Select how bars next to openings are handled. • **Cut bars in holes**: If you select **Yes**, the bars are cut in openings but otherwise they are not cut. • **Minimum hole size**: The minimum value by which the bar is shortened. Lower values are ignored. • **Cover thickness in holes**: Distance between the bar end/ start point to the edge of the opening.
Minimum length of a bar to be created	Bars that are shorter than this value are not created.

Mesh Bars: Bar end conditions tab

Use the **Bar end conditions** tab to control the cover thickness and bar hooks in the **Mesh Bars** modeling tool.

Option	Description
Cover thickness on sides	• **Different each side**: Each end of a bar group can have different distances from part edges. • **Same all sides**: All ends of the bar groups have the same distance from part edge.
End conditions	• Creates a 90° hook at a given end of bar • Creates a 135° hook at a given end of bar.

Option	Description
	• Creates a 180° hook at a given end of bar. • Creates a straight bar without a hook at the given end of a bar.
Bending radius	Bending radius at bar ends
Bending length	Bending length at bar ends

Mesh Bars: Attributes tab

Use the **Attributes** tab to control the bar group or mesh properties in the **Mesh Bars** modeling tool.

Option	Description
Name	Name of the bar group or mesh.
Prefix	Prefix for the part position number of the bar group or mesh.
Start number	Start number for the part position number of the bar group or mesh.
Class	Class of the bar group or mesh. If you create a mesh, only the class of primary bars is taken into account.

Mesh Bars by Area

Mesh Bars by Area creates reinforcement for concrete slabs or walls.

Objects created

- Primary bars
- Crossing bars

Use for

Situation	Description
	Reinforcement for the bottom or top surface of the concrete element, or for both.

Selection order

1. Select a concrete slab.
2. Select a group of points to define a working polygon.

   The reinforcement is created automatically.

See also

Mesh Bars by Area: Picture tab on page 288

Mesh Bars by Area: Detailing tab on page 291

Mesh Bars by Area: Bar end conditions tab on page 293

Mesh Bars by Area: Attributes tab on page 294

Mesh Bars by Area: Picture tab

Use the **Picture** tab to define how the top and bottom bars are created in the **Mesh Bars by Area** modeling tool.

Option	Description
Create bars	• Top bars with primary bars above secondary bars

Option	Description	
	• Top bars with secondary bars above primary bars • No bars The same options are available for the bottom bars as well. The value defined in the box is the cover thickness for the top/bottom layer.	
Mesh area perimeter	Select the outline that the mesh follows. The meshes in the examples are created by picking the same points but with different mesh area perimeters. • Part • Polygon • Polygon + part	• Part, for example: • Polygon, for example:

Option	Description
	• Part + polygon, for example:
Same as bottom	Select whether the top bars are created using the same properties as the bottom bars. If you select **No**, enter the top bar properties.
Bar type	Select whether the bars are created as a bar group or as a mesh.
Size	Select the diameter of a bar. You can set the size separately for the top primary and secondary bars, and for the bottom primary and secondary bars.
Grade	Define the strength of the steel used in the bars. You can set the grade separately for the top primary and secondary bars, and for the bottom primary and secondary bars.
Spacing type	Select whether the spacing is based on the number of bars or the spacing value. • **By spacings**: Enter the distance between the bars in the **Spacing** box. • **Number of bars**: Enter the number of bars in the **Number of bars** box.
Spacing	Distance between the bars. Tekla Structures calculates the number of bars. You can set the spacing separately for the top primary and secondary bars, and for the bottom primary and secondary bars.
Number of bars	Number of bars. Tekla Structures calculates the distance between bars. You can set the number of bars separately for the top primary and secondary bars, and for the bottom primary and secondary bars.
Primary bars direction	The direction of the primary bars. Use to change bar direction. The options are: • **Auto** • **Use slab x direction**

Option	Description
	• **Use slab y direction** • **Use global x direction** • **Use global y direction** If the **Auto** option is selected, the primary bars are created along the two longest parallel sides of a slab. If there are no parallel sides, then the bars are created along the **slab x direction** instead.
Up direction	Select which sides are regarded as the top and bottom sides of the slab. If the **Auto** option is selected, then the bars are created on the side with the greatest area.

Mesh Bars by Area: Detailing tab

Use the **Detailing** tab to control how the bars are distributed in the **Mesh Bars by Area** modeling tool.

Option	Description
Bar edge distances	Define the edge distance of the bars.
Adjustment	Select how the bars are distributed. The same options are available for both the primary and the secondary bars.
	 Equal distribution by target spacing value.
	 By exact spacing value with flexible first space without first bar.
	 By exact spacing value with flexible last space without last bar.

Option	Description
	 By exact spacing value with flexible first and last space without first and last bars.
	 By exact spacing value with flexible first space.
	 By exact spacing value with flexible last space.
	 By exact spacing value with flexible first and last space.
Bar grouping	Select whether tapered bars are grouped or not. If you group the tapered bars, select how the bars along tapered edge are handled. • Tapered bars are handled normally. • Creates as many bars as possible with same length in the tapered edge.

Option	Description
Holes	Select how bars next to openings are handled. • **Cut bars in holes**: If you select **Yes**, the bars are cut in openings but otherwise they are not cut. • **Minimum hole size**: The minimum value by which the bar is shortened. Lower values are ignored. • **Cover thickness in holes**: Distance between the bar end/ start point and the edge of the opening.
Minimum length of bar to be created	Bars that are shorter than this value are not created.

Mesh Bars by Area: Bar end conditions tab

Use the **Bar end conditions** tab to control the cover thickness and bar hooks in the **Mesh Bars by Area** modeling tool.

Option	Description
Cover thickness on sides	• **Different each side**: Each end of a bar group can have different distances from part edges. • **Same all sides**: All ends of the bar groups have the same distance from part edge.
End conditions	• Creates a 90° hook at a given end of bar • Creates a 135° hook at a given end of bar. • Creates a 180° hook at a given end of bar. • Creates a straight bar without a hook at the given end of a bar.
Bending radius	Bending radius at bar ends
Bending length	Bending length at bar ends

Mesh Bars by Area: Attributes tab

Use the **Attributes** tab to control the bar group or mesh properties in the **Mesh Bars by Area** modeling tool.

Option	Description
Name	Name of the bar group or mesh.
Prefix	Prefix for the part position number of the bar group or mesh.
Start number	Start number for the part position number of the bar group or mesh.
Class	Class of the bar group or mesh. If you create a mesh, only the class of primary bars is taken into account.

Slab bars (18)

Creates reinforcement for a concrete slab.

Bars created

- Primary slab bars
- Crossing slab bars

Use for

Situation	More information
	Slab bars (18) creates reinforcement for the bottom or top surface of the slab, or for both. See Slab reinforcement properties on page 295.

Before you start

- Create the concrete slab.
- Calculate the required area of reinforcement.

Defining properties Use the following tabs in the **Slab bars (18)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Picture	Concrete cover thickness, which bars to create (bottom/top/both), primary bar direction, which sides are regarded as top and bottom sides	Slab reinforcement properties on page 295
Bottom bars	Grade, size, spacing, and generation type of bottom bars, concrete cover thickness on slab sides, crossing bar location in relation to primary bars, end hooks and bend lengths	Slab reinforcement properties on page 295
Top bars	Grade, size, spacing, and generation type of top bars, concrete cover thickness on slab sides, crossing bar location in relation to primary bars, end hooks and bend lengths	
Attributes	Numbering properties, name, and class of bars	

Picking order

1. Concrete slab

Slab reinforcement properties

Tekla Structures creates slab reinforcement using the properties in the **Slab bars (18)** dialog box:

Field	Description
Create bars	Defines on which surface(s) to create bars. The options are: • **Both sides** • **Bottom side** • **Top side**
Primary bar direction	The direction of primary bars. Use to change bar direction. The options are: • **Use slab x direction** • **Use slab y direction** • **Use global x direction**

Field	Description
	• **Use global y direction**
Up direction	Defines which sides are regarded as the top and bottom sides of the slab.
Bar generation type	Defines whether Tekla Structures treats the bars as a group or a mesh.
Cross bar location	Defines whether the crossing bars are located above or below the primary bars.
End conditions	The hook angles at bar ends. The options are **Straight**, 90, 135, and 180 degrees.
Bend lengths	The bend lengths at bar ends.

Slab Reinforcement Tool

Slab Reinforcement Tool creates reinforcement to irregularly shaped concrete slabs. The reinforcement is created as reinforcing bar groups. It is also possible to splice the reinforcing bars.

Objects created

- Reinforcement

Use for

Situation	Description
	Slab with reinforcement

Before you start Create a slab.

Selection order

1. Select the main part (slab).
2. Pick points to define the direction of the reinforcing bars.
3. If the area to be reinforced is set to **By polygon**, pick the polygon points to define the reinforced area.

See also

Slab Reinforcement Tool: Parameters tab on page 296

Slab Reinforcement Tool: Advanced tab on page 298

Slab Reinforcement Tool: Parameters tab

Use the **Parameters** tab to control the reinforcement properties, cover thickness, and reinforcing bar spacing and area in the **Slab Reinforcement Tool** modeling tool.

Reinforcing bars

Option	Description
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.
Name	Define the name for the reinforcing bars. Tekla Structures uses the name in drawing and reports.
Size	Define the diameter of the reinforcing bar.
Grade	Define the strength of the steel used in the reinforcing bars.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.

Cover thickness

Option	Description
Cover thickness	Define the concrete cover thickness on the different sides of the slab.
Depth	Define the concrete cover, either from the top or from the bottom of the slab.
Depth value	Define the depth value for concrete cover thickness.

Spacing and area

Option	Description
Spacing Method	Define the reinforcing bar spacing method.
Number of Bars	Define the number of reinforcing bars in a reinforcement.
Spacing	Define the space between the reinforcing bars.
Adjustment	Define how the reinforcing bars are placed in the slab. The placing is based on the slab width and the cover thickness at sides. • **Target spacing**: The reinforcing bars are placed at even spacing. • **Exact spacing cover at left/right side**: The reinforcing bars are placed starting from the left/right side. • **Exact spacing centered both covers**: The reinforcing bars are centered and at equal distance from the sides.
Area to be Reinforced	Define the area to be reinforced in the slab. • **By part perimeter**: The reinforcing bars are created in the whole area of the slab. • **By polygon**: The reinforcing bars are created to a picked polygon shaped area in the slab.

Slab Reinforcement Tool: Advanced tab

Use the **Advanced** tab to control the reinforcing bar end hooks, splicing, and opening properties in **Slab Reinforcement Tool** modeling tool.

Bar Ends

Option	Description
Start/End	Define the type of the hooks to be used at the reinforcing bar ends. If you select **Custom Hook**, you can define a custom hook's **Angle**, **Radius** and **Length**.
Angle	Define the hook angle by entering a value between `-180` and `+180` degrees.
Radius	Define the internal bending radius of the hook.
Length	Define the length of the straight part of the hook.

Splicing

Option	Description
Splice Bars	Define whether the reinforcing bars are spliced or not.
Maximum Length of Bar	Define the maximum reinforcing bar length after which the bars are spliced.
Splicing in Same Cross Section	Define how many reinforcing bars can be spliced in the same location. • **1/1** = all reinforcing bars are spliced in the same cross section. • **1/2** = every second reinforcing bar is spliced in the same cross section. • **1/3** = every third reinforcing bar is spliced in the same cross section. • **1/4** = every fourth reinforcing bar is spliced in the same cross section.
Splicing Symmetry	Define the symmetry that is applied when the reinforcing bars are spliced. • **Not Symmetrical**: The splice pattern of the reinforcing bar is not symmetrical and the uneven length is only at one side. • **Symmetrical with different lengths at sides**: The splice pattern of the reinforcing bar is symmetrical with uneven lengths at the sides. • **Symmetrical with different length at center**: The splice pattern of the reinforcing bar is symmetrical with uneven length in the center.
Splicing Offset	Define the offset of the splice center point from the point where the reinforcing bars originally met.

Option	Description
Minimum splitting distance	Define the minimum longitudinal distance between two splices in consecutive bars.
Splicing Type	Define the type of the splice.
Lapping Length	Define the length of the lapping connection.

Openings

Option	Description
Cut Bars at Opening	Define whether reinforcing bars are cut in slab openings.
Cover thickness for openings	Define the concrete cover thickness on the sides of the openings.
Minimum Size of Large Opening	Define the minimum size of an opening that is considered to be large. This field is available if you set **Cut Bars at Opening** to **At Large Opening Only**.

Beam reinforcement (63)

Creates reinforcement for a concrete beam.

Bars created

- Longitudinal main bars
- Corner and side bars
- Main stirrups
- Stirrups for one or two ledges

Use for

Situation	More information
	Rectangular beams

Situation	More information
	L-shaped beams (RCL profiles)
	RCX profiles
	Inverted T-beams (RCDL profiles)

Before you start

- Create the concrete beam.
- Calculate the required area of reinforcement.

Defining properties

Use the following tabs in the **Beam reinforcement (63)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Parameters	Concrete cover thickness, number, size, and location of bars, stirrup size, option to place main bars only inside the main stirrups or distributed to the ledges	Beam reinforcement properties on page 301

Tab	Contents	See also
Stirrup spacing	Distances from the part ends to the first stirrups, number of spaces and spacing of main and ledge stirrups in five zones along the length of the part, option to create different stirrups in two ledges, ledge stirrup spacing method	Beam reinforcement (63): Stirrup spacing tab on page 302
Advanced	Grade, name, class, and numbering properties of the bottom, top, and side bars, and main and ledge stirrups	

Picking order 1. Concrete beam

Beam reinforcement properties

Use the following fields on the **Parameters** tab in the **Beam reinforcement (63)** dialog box to define the beam reinforcement properties:

1 Main stirrup size

2 Size of corner bars

3 Ledge stirrup size

4 Number of bars

5 Size of bars

Beam reinforcement (63): Stirrup spacing tab

Use the **Stirrup spacing** tab in the **Beam reinforcement (63)** dialog box to define the spacing of the beam stirrups.

Additional stirrup spacing type

Defines how the ledge stirrups are distributed along the length of the beam. The options are:

Option	Description
Target	Tekla Structures creates equal spaces between the bars, aiming the spacing value as closely as possible to the value you specify. This is the default option.
Exact	The first and last space of a zone adjust themselves to even out bar distribution. The spaces in the middle of each zone are exactly the size you specify.

Double tee reinforcement (64)

Creates prestressed strands and other reinforcement for a concrete double tee.

Bars created

- Prestressed strands in double tee ribs
- Stirrups in double tee ribs
- Reinforcement mesh on the top surface of the double tee

Use for

Situation
Straight strands
Different strand pattern and number of strands
When you use different stirrup shapes and spacings

Before you start

- Create the concrete double tee.
- Calculate the required area of strands and other reinforcement.

Defining properties

Use the following tabs in the **Double tee reinforcement (64)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Parameters	Concrete cover thickness, size, number, and pattern of strands, stirrup size and type, size and spacing of longitudinal and crossing bars in the mesh	Double tee reinforcement properties on page 303
Stirrup spacing	Distances from the part ends to the first stirrups, number of spaces and spacing of stirrups in five zones along the length of the part	
Advanced	Grade, name, class, and numbering properties of the strands, stirrups, and mesh bars	

Picking order

1. Concrete double tee

Double tee reinforcement properties

Strand pattern Use the following options to define the strand pattern in double tee ribs:

Grid	Staggered 1st	Staggered 2nd

Stirrups Use the following options to define the stirrups and stirrup hooks in double tees:

Option	Example
Closed	
Open	
Open with 90 degree hooks	
Open with 135 degree hooks	

Option	Example
Open with 180 degree hooks	

Beam end reinforcement (79)

Creates reinforcement for the end of a concrete beam or strip footing.

Bars created

- Horizontal U-shaped bars (types 1 and 2)
- Vertical U-shaped bars (types 3A and 3B)
- Oblique bar (type 4)
- Stirrups (types 5A and 5B)

1 Horizontal U bars (type **2**)

2 Horizontal U bars (type **1**)

3 Oblique bar (type **4**)

4 Vertical U bars (type **3A**)

5 Stirrups (type **5A**)

6 Vertical U bars (type **3B**)

7 Stirrups (type **5B**)

Use for

Situation	More information
Standard beams	Use bars 3A and 5A for the beam end.
Dapped beams	
Beams with anchor bolt hole in the notched area	
Beams and strip footings that have a rectangular cross section	

Do not use for Parts that have irregular cross sections.

Before you start

- Create the concrete beam or strip footing.
- Calculate the required area of reinforcement.

Defining properties Use the following tabs in the **Beam end reinforcement (79)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Picture	Concrete cover thickness, distances from the concrete surface to the bars, angle of bar 4	
Bars	Bar dimensions of each bar type, which bars to create, option to splice bars 3A and 3B	Beam end reinforcing bars on page 306
Groups	Number and spacing of bars in each group of bar types. If the spacing varies, enter each value individually. In the **Bar 4** list box, select **Yes** to create the bar.	
Attributes	Grade, size, class, and numbering properties of the bars	

Picking order

1. Select concrete beam or strip footing.
2. Pick position.

Beam end reinforcing bars

Horizontal U bars 1 and 2 on page 307

Vertical U bars 3A and 3B on page 307

Stirrups 5A and 5B on page 308

Horizontal U bars 1 and 2

Use the following options to create bars in the lower area of the beam end, in the horizontal planes (bar type 1):

Option	Description
	Two bars on each plane. One in the middle of the beam end, the other extending to the sides of the beam.
	One bar on each plane, extending to the sides of the beam.
	One bar on each plane, in the middle of the beam end.
	Two identical, overlapping bars on each plane.

To create bars in the upper area of the beam end, around a hole, enter dimensions for bar 2.

Vertical U bars 3A and 3B

To create vertical U-shaped bars, enter dimensions for:

- Bar 3A: for the notched area of the beam.

- Bar 3B: for the higher part of the beam.

Splicing bars 3 You can create vertical U bars (bar type 3) of two bars joined with a splice. To do this, use the following list box on the **Bars** tab:

If you choose to splice bars, you can select the splice type:

Option	Description
Lap up	Creates a lap splice above the horizontal center line of the beam end.
Lap down	Creates a lap splice below the horizontal center line of the beam end.
Lap both	Creates a lap splice centered to the horizontal center line of the beam end.
Coupler	Creates a coupler.
Weld joint	Creates a welded joint.

For lap splices, you can define the lap length **L** and whether the bars are on top of each other or parallel to each other.

For all splice types, you can define the offset of the splice center point from the horizontal center line of the beam end.

Stirrups 5A and 5B

To create stirrups for beam ends, enter dimensions for:

- Bar 5A: for the notched area of the beam.
- Bar 5B: for the higher part of the beam.

Corbel reinforcement (81)

Corbel reinforcement (81) creates reinforcement for one or two corbels in a concrete column. The two corbels must have the same top level, thickness, and horizontal location.

Objects created
- Main bars

- Stirrups
- Additional bars

Use for

Situation	Description
	Two corbels, beveled and rounded, with the same top level, thickness, and horizontal location. Two additional bars crossing each other.
	One straight corbel. One additional bar.

Limitations Do not use **Corbel reinforcement (81)** for two very different corbels.

Before you start

- Create the concrete column and beam(s).
- Create the corbel(s). Use **Corbel connection (14)** or a **Seating connection** to create the corbel(s).

Selection order

1. Select the main part (concrete column).
2. Select the secondary part(s) (one or more corbels).
3. Click the middle mouse button to create the reinforcement.

See also Corbel reinforcement (81): Main bars tab on page 310

Corbel reinforcement (81): Stirrups/Transverse Stirrups/Diagonal Stirrups tabs on page 311
Corbel reinforcement (81): Additional bars tab on page 312

Corbel reinforcement (81): Main bars tab

Use the **Main bars** tab to control the main bar length, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties in **Corbel reinforcement (81)** modeling tool.

Option	Description
①	Define the main bar length.
②	Define the distance from the concrete surface to the main bar.

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Name	Define a name for the main bars. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Option	Description
Concrete cover	Define the concrete cover thickness.
Number of bars	Define the number of bars in the reinforcement.

Option	Description
Bar spacing	Define the space between the reinforcing bars.

Corbel reinforcement (81): Stirrups/Transverse Stirrups/Diagonal Stirrups tabs

Use the **Stirrups**, **Transverse Stirrups** and **Diagonal Stirrups** tabs to control the stirrup creation, stirrup type, concrete cover thickness, reinforcing bar spacing, and reinforcing bar properties in the **Corbel reinforcement (81)** modeling tool.

Stirrups

Option	Description
Stirrup location	Define the distance from the concrete surface to the stirrups.
Number of bars	Define the number of bars in the reinforcement.
Bar spacing	Define the space between the reinforcing bars.
Name	Define a name for the stirrups. Tekla Structures uses the name in drawings and reports.
Size	Define the diameter of the reinforcing bar.

Stirrup type

Option	Description
	The stirrup consists of a bent single reinforcing bar. Define the hook length. The default values (for the standard 90 degree stirrup hook) are read from the `rebar_database.inp` file. Select a suitable overlap position for this stirrup type.
	The stirrup consist of two overlapping U bars that face each other. Define minimum and maximum overlapping length.

Concrete cover

Option	Description
Thickness	Define the concrete cover thickness.
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Name	Define a name for the stirrups. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Option	Description
Concrete cover	Define whether the concrete cover thickness is same on all sides.

Corbel reinforcement (81): Additional bars tab

Use the **Additional bars** tab to control the creation of additional bars in corbels, concrete cover thickness, and additional bar properties in the **Corbel reinforcement (81)** modeling tool.

Additional bars

Option	Description
	No additional bars.
	One additional bar.
	Two additional bars parallel to each other.
	Two additional bars crossing each other.
	Two pairs of additional bars crossing each other.

Option	Description
Concrete cover	Define the concrete cover thickness.
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Name	Define a name for the additional bars. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Define a prefix for the part position number.

Option	Description
Start number	Define a start number for the part position number.

Round column reinforcement (82)

Creates reinforcement for a concrete column that has a round cross section.

Bars created

- Longitudinal main bars
- Stirrups
- Column end reinforcement

Use for

Situation	More information
	Main bars bent outside the column. Continuous spiral stirrup.
	Straight main bars protruding from the column, or entirely inside the column. Separate stirrups. End of column reinforced.
Ends of column cut or fitted	
Exact or variable stirrup spacing	

Do not use for Rectangular columns.

Before you start

- Create the concrete column.
- Calculate the required area of reinforcement.

Defining properties Use the following tabs in the **Round column reinforcement (82)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Main bars	Horizontal and vertical bond lengths at the top and bottom	

Tab	Contents	See also
	of the column, thickness of concrete cover over stirrups Number, rotation, grade, size, class, name, and numbering properties of the bars	
Stirrups	Thickness of concrete cover over stirrups at the top and bottom of the column, spacing and number of stirrups or turns in each stirrup group, stirrup types, overlap angle	Column stirrup properties on page 314 Group 1 is the top stirrup group, 5 is the bottom one. Group 3 is always created.
Stirrup attributes	Grade, size, name, class, and numbering properties of each stirrup group	
Top	Concrete cover thickness, dimensions, rotation, number, grade, size, class, name, and numbering properties of the top reinforcement	Column top and bottom reinforcement on page 315
Bottom	Concrete cover thickness, dimensions, rotation, number, grade, size, class, name, and numbering properties of the bottom reinforcement	

Picking order 1. Concrete column

Column stirrup properties

Use these options to define the stirrups in round and rectangular columns:

Option	Description	Applies to
	Separate stirrups	Round columns
	Single, continuous spiral stirrup	
	Separate spiral stirrups	

Option	Description	Applies to
	Angle of stirrup overlap	
	Laps at stirrup corners 135-degree hooks at bar ends	Rectangular columns
	Laps at stirrup corners 90-degree hooks at bar ends	

Use these options in the **Bars/laps** list box to define the spacing of stirrups:

Option	Description
Exact space, flexible at ends	Tekla Structures uses exactly the spacing value you specify, and evens out the stirrup distribution at the column ends.
Target space	Tekla Structures creates the stirrups at even spacings and tries to use the spacing value you specify.
Distance list	Tekla Structures creates the stirrups in the middle stirrup group using the defined spacing distances. Tekla Structures uses as many distances from the distance list as possible. The distances in the list may not all fit within the middle stirrup group area so that for example the last spacing distance may be shorter than the distance value in the list. The **Distance list** option is available for rectangular columns.

Column top and bottom reinforcement

Use these options to define the rotation of the reinforcement at the top or bottom of columns:

Option	Description	Applies to
	No rotation. Transverse bars are perpendicular to the longer side of the column.	Rectangular columns

Option	Description	Applies to
	Rotation angle is 90 degrees. Transverse bars are parallel to the longer side of the column.	
	Rotation angle of the reinforcement at the ends of round columns.	Round columns

Rectangular column reinforcement (83)

Rectangular column reinforcement (83) modeling tool creates reinforcement for a concrete column that has a rectangular cross section.

Objects created

- Longitudinal main bars: corner bars (4), side bars
- Stirrups
- Intermediate links
- Column end reinforcement

Use for

Situation	Description
	Rectangular concrete column with corner bars and side bars bent outside the column. Side bars on long sides. Intermediate links tie side bars at every second stirrup.

Situation	Description
	Rectangular concrete column with straight corner and side bars entirely inside the column. End of column reinforced.

Limitations Do not use for round columns.

Selection order

1. Select the main part (column).

   The component is created automatically when the part is selected.

Part identification key

	Part
1	Corner bars
2	Side bars
3	Stirrups
4	Intermediate links

See also

Rectangular column reinforcement (83): Main bars tab on page 319

Rectangular column reinforcement (83): Bar ends tab on page 320

Rectangular column reinforcement (83): Side bars tab on page 321

Rectangular column reinforcement (83): Stirrups tab on page 322

Rectangular column reinforcement (83): Intermediate links tab on page 323

Rectangular column reinforcement (83): Top / Bottom tab on page 325

Rectangular column reinforcement (83): Main bars tab

Use the **Main bars** tab to control the corner bar properties, the symmetry options, rotation, and concrete cover thickness in the **Rectangular column reinforcement (83)** modeling tool.

Basic corner bar properties

Define the grade, size and bending radius of the corner bars. The active fields depend on the selected symmetry option.

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Bending radius	Define the bending radius of the corner bars.

Define the concrete cover thickness.

Symmetry options

Select the symmetry option. Using the symmetrical conditions you can define which of the corner bars have the same grade, size and bending radius properties. The corners bars that have same properties are symmetrical.

Rotation

In square columns, you can select the perpendicular sides of a column if the sides require different reinforcement. You can rotate all reinforcement in a square column by 90 degrees.

Additional corner bar properties

Option	Description
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Define a name for the main bars. Tekla Structures uses the name in drawing and reports.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Rectangular column reinforcement (83): Bar ends tab

Use the **Bar ends** tab to control the length of vertical and horizontal extensions for the corner and side bars at the top and bottom of the column, and the cranking in the **Rectangular column reinforcement (83)** modeling tool.

Bar creation

Select whether the extensions of corner bars and side bars are created symmetrically on both sides of the column. If you select **Not symmetrical**, you can enter separate extension values for the opposite sides of the column.

Vertical extension

Define the length of the vertical extension outside the column for corner bars and side bars.

The active fields depend on the **Bar creation** options: **Symmetrical** or **Not symmetrical**.

Horizontal extension

Define the length of the horizontal extension for corner bars and side bars.

The active fields depend on the **Bar creation** options: **Symmetrical** or **Not symmetrical**.

Corner bar direction

Select how to rotate the bent corner bars.

Cranking

You can create cranked reinforcing bars at the top of the column.

In the **Corner bar direction** option list, select to activate the cranking options. When you select this option, you can enter the horizontal distance dimensions for the cranked bars.

To successfully create the cranked reinforcing bars, ensure that the bending radius is not too large.

Define the distance from the column corner to the point where the cranking starts

.

Rectangular column reinforcement (83): Side bars tab

Use the **Side bars** tab to control the number of side bars, their spacing and placing, symmetry options, and side bar properties in the **Rectangular column reinforcement (83)** modeling tool.

Number of side bars

Define the number and spacing of side bars. You can define two sets of side bars on each side of the column.

You can define the side bars separately for each side of the column.

The active fields depend on the selected symmetry option.

Placing of side bars

Select the horizontal and vertical placing for the side bars. Select whether the bars are placed starting from the corner or at equal spaces.

	Distance between corner bars and side bars.
	Distance between side bars.

Symmetry options

Select the symmetry option. Using the symmetrical conditions you can define which side bars are symmetrical, and which side bars use the same properties.

Side bar properties

Define the side bar properties for side bars 1 and side bars 2.

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Bending radius	Define the bending radius of the side bars.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Define a name for the side bars. Tekla Structures uses the name in drawing and reports.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Rectangular column reinforcement (83): Stirrups tab

Use the **Stirrups** tab to control the stirrup properties in the **Rectangular column reinforcement (83)** modeling tool.

Stirrup dimensions

1. Define the thickness of the concrete cover over the stirrups at the top and the bottom of the column. The default cover thickness is 50 mm.

   Group **1** is the top stirrup group, **5** is the bottom stirrup group. Group **3** is always created.
2. Define the spacing of stirrups in each stirrup group.
3. Define the number of stirrups in each stirrup group.
4. Select how the stirrups are distributed.
5. If you select the **Distance list** option, enter different spacing values for the groups.
6. Define the cover thickness for each stirrup group.

Bar lapping at stirrup corners

Select the how the bars lap at the stirrup corners. The options are 135-degree hooks at the bar end or 90-degree hooks at the bar end.

Create as one group

Select whether all the stirrups are created as a single group. If you select **Yes**, all stirrup groups are created and they have the same properties as **Group 3**.

Ignore cuts If you have a recess or a hole in the column, you can select to ignore the cuts at the top and bottom of the column when stirrups are created.

Stirrup properties

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Name	Define a name for the stirrups. Tekla Structures uses the name in drawing and reports.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Rectangular column reinforcement (83): Intermediate links tab

Use the **Intermediate links** tab to control the intermediate links in order to tie all side bars in the **Rectangular column reinforcement (83)** modeling tool.

If you have rectangular columns that have very large cross sections, the side bars may be a far from the corners of the stirrups. You will need to create intermediate links to tie all side bars, and to prevent them from buckling when they are in compression.

Intermediate links are created for each stirrup group.

Intermediate links are created between **Side bars 1** or **Side bars 2** that are symmetrical.

For **Side bars 2** intermediate links are created only if no **Side bars 1** are created.

Properties of intermediate links

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Name	Define a name for the stirrups. Tekla Structures uses the name in drawing and reports.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Spacing of intermediate links

	Spacing is the same as stirrup spacing.
	Spacing is double the stirrup spacing (intermediate links at every second stirrup).
	Same as above, but for alternate stirrups.
	No intermediate links are created.

Intermediate link type

Select the type of the link.

Intermediate link pattern

Select whether intermediate links go in one direction or in a crossing pattern.

Hook orientation

Select the hook orientation for both sides.

Closed stirrups as intermediate links

Select whether to use closed stirrups as intermediate links.

If you select **Yes**, select the type of the bar lapping at the stirrup corners.

Zone without intermediate links

Define the zone length where intermediate links are not created. In this zone the stirrups tie the side bars. The distance is measured from the stirrup corner.

If you have selected the **Yes** option in **Use closed stirrups always when possible**, you can define the distance from the corner of the closed stirrup.

Rectangular column reinforcement (83): Top / Bottom tab

Use the **Top** and **Bottom** tabs to control the top and bottom reinforcement thickness of concrete cover, the number of bars and their spacing and rotation in the **Rectangular column reinforcement (83)** modeling tool.

Number and spacing of top or bottom bars

Define the number and spacing of the top or bottom bars.

You can define the number of transverse bars in both cross directions.

Column top / bottom reinforcement

1. Define the concrete cover thickness.
2. Define the length of the ultimate leg.

Placing of top or bottom bars

Select the horizontal and vertical placing for the top or bottom bars. Select whether the bars are placed starting from the corner or at equal spaces.

	Distance between corner bars and side bars.
	Distance between side bars.

Rotation

Select how to rotate the reinforcement at the top or bottom of columns.

	No rotation. Transverse bars are perpendicular to the longer side of the column.

	Rotation angle is 90 degrees. Transverse bars are parallel to the longer side of the column.

Top and bottom reinforcement properties

Option	Description
Grade	Define the strength of the steel used in the reinforcing bars.
Size	Define the diameter of the reinforcing bar.
Bending radius	Define the bending radius of the top or bottom bars.
Class	Use the **Class** to group reinforcement. For example, you can display reinforcement of different classes in different colors.
Name	Define a name for the top or bottom bars. Tekla Structures uses the name in drawing and reports.
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Hole reinforcement for slabs and walls (84)

Creates a hole in a concrete slab or wall and reinforcement around the hole.

Bars created

- Straight bars along hole edges
- Diagonal bars close to hole corners
- U-shaped edge bars

Use for

Situation	More information
Rectangular or round holes in concrete slabs and walls	

Situation	More information
	Straight and edge bars only, no diagonal bars.
	Diagonal and edge bars only, no straight bars.
	Hole rotated from the direction of the slab. One diagonal bar at each corner.
	Different number of bars on each side of the hole in the wall. No diagonal bars.

Before you start

- Create the concrete slab or wall.
- Calculate the required area of reinforcement.

Defining properties

Use the following tabs in the **Hole reinforcement for slabs and walls (84)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Picture	Shape of the hole (rectangular or round): Hole dimensions, bar dimensions and location, concrete cover thickness Rotation angle of the hole: 	
Horizontal and vertical bars	Which bars are closest to the surface of the concrete (vertical or horizontal), number, grade, size, numbering properties, name, and class of each bar group along the sides of the hole	The slab or wall direction defines the direction of the hole and which bars are to the left and right of, and above and below the hole.
Edge bars	Spacing, grade, size, numbering properties, name, and class of bar group on each side	
Diagonal bars	Number, grade, size, numbering properties, name, and class of bar group at each corner of the hole	

Picking order

1. Center of the hole
2. Concrete slab or wall

Hole creation and reinforcement (85)

Creates a hole in a concrete part and reinforcement around the hole.

Bars created

- Straight bars at hole edges
- Stirrups
- Z-shaped bars

Use for

Situation	More information
	Rectangular hole, stirrups on each side of the hole, no Z-shaped bars.
	Round hole, Z-shaped bars, no stirrups around the hole.
Concrete beams or columns	
Round or rectangular holes	Hole only. No additional reinforcement around it.
With or without Z-shaped bars or stirrups	

Do not use for Parts that have round or irregular cross sections.

Before you start

- Create the concrete part.
- Calculate the required area of reinforcement.

Defining properties Use the following tabs in the **Hole creation and reinforcement (85)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Picture	Hole shape, direction, and dimensions, bar dimensions and location, concrete cover thickness, angle of Z-shaped bars	Reinforcement around holes on page 330

Tab	Contents	See also
Bars	The number, grade, size, name, class, and numbering properties of: • Straight bars above and below the hole • Z-shaped bars on the left and right side of the hole	
Stirrups	Number, spacing, grade, size, name, class, and numbering properties of stirrups	
	Additional stirrup settings	Stirrups at holes on page 331

Picking order

1. Center of the hole
2. Concrete part

Reinforcement around holes

Use these options to create holes and additional reinforcement:

Option	Description
	Creates a hole and additional reinforcement around it.
	Only creates a hole, no reinforcement.

Use these options to define the direction of holes:

Option	Description
	Hole along the local y direction of the part.
	Hole along the local z direction of the part.

Use these options to define Z-shaped bars around round holes:

Option	Description
	Concrete cover measured in the direction of the radius of the hole.
	Concrete cover measured from the corner of the bounding box around the hole.

Use these options to define the direction of Z-shaped bars:

Option	Description

Stirrups at holes

Use these options to define the stirrups around holes:

Option	Description
	Single stirrups
	Bundled stirrups

Hooks

Option	Description
	135-degree hooks
	90-degree hooks

Braced girder (88)

Braced girder (88) creates braced girders between selected points to the selected concrete part. The girders are cast into the concrete part and they also work as a part of the reinforcement and as a connector between the precast and cast-in-place concrete.

Objects created

The component creates girders that consist of the following parts:

- Bottom reinforcing bars (2)
- Top reinforcing bar
- Connecting reinforcing bars (2)

Instead of using reinforcing bars, you can use profiles from the **Profile Catalog** to create the braced girders.

Use for

Situation	Description
	Braced girders are created in the precast concrete slab.
	Braced girders are created in the precast concrete sandwich panel.
	Braced girders are created in a wide plate floor with an opening.

Situation	Description
	Braced girders are created in a hollow wall.
	Profile as a girder.

Before you start Create a concrete part.

Selection order

1. Select the concrete part.

Part identification key

	Part
1	Top reinforcing bar
2	Connecting reinforcing bar
3	Bottom reinforcing bar

See also

Braced girder (88): Picture tab on page 333

Braced girder (88): Parts tab on page 335

Braced girder (88): Geometry tab on page 345

Braced girder (88): Double wall tab on page 347

Braced girder (88): Picture tab

Use the **Picture** tab to control the shape and dimensions of reinforcing bars in the **Braced girder (88)** modeling tool.

Reinforcing bar dimensions

	Description
①	Define the distance between the end of the top reinforcing bar and the part end.
②	Define the distance between the end of the connecting reinforcing bar and the part end.
③	Define the distance between the end of the bottom reinforcing bar and the part end.
④	Define the radius of the connecting reinforcing bar.
⑤	Define the distance between bendings.

Select the location of the top and bottom reinforcing bars. Define the distance between the bottom reinforcing bars, and the distance between the top and bottom reinforcing bars.

Reinforcing bar properties

Define the reinforcing bar properties for the top, bottom and the connecting reinforcing bars.

Option	Description
Size	Define the size of the reinforcing bar.
Grade	Define the grade of the reinforcing bar.
Comment, **Name**, **Class**, **Serie**, **Start number**	Define a comment, name, and class for the reinforcing bar, and a prefix and a start number for the part position number.
Create	Select whether the reinforcing bar in question is created or not.

Braced girder (88): Parts tab

Use the **Parts** tab to control the girder creation, positioning and number of girders in the **Braced girder (88)** modeling tool.

Girder creation

Option	Description
Creation method	Select the type of the girders.
Profile	If you use a profile as a girder, select the profile from the **Profile Catalog**. In addition, define a prefix and a start number for the part position number, a material and a name.

Workplane orientation

Option	Description
Workplane position	Select the work plane position.
	Use the **Position in plane**, **Rotation** and **Position in depth** options to define the orientation of the girder on the work plane.

Girder position in z direction

Select how the girders are positioned in the z direction.

Examples	

Geometry

Option	Description
Geometry	Select the geometry for the connecting reinforcing bars. Examples:

Option	Description

Girder position in y direction

Select how the girders are positioned in the y direction.

Examples

Examples

Extra girders

Select whether additional girder groups are created.

Examples

For example, if you have selected this option:

Examples
Extra girder groups is set to **None**: No additional girder groups are created between 2 existing braced girders.

Examples
Extra girder groups is set to **By number**: Additional girder groups are created based on the entered number. Distances between the groups are equally divided.

Examples
Extra girder groups is set to **By distance**: The number of additional girder groups is based on the defined distance. The distances between the girder groups are equally divided.

Girder group

Number is the number of girders in the girder group.

Distance list is the distance between the girders in the girder group.

e = is the position of the girders from the reference line.

Option	Description
   	• Girder group is positioned middle of the reference line. • Girder group is positioned on the right side of the reference line. • Girder group is positioned on the left side of the reference line.

Examples	
Existing girders:	

Examples	

Braced girder (88): Geometry tab

Use the **Geometry** tab to control how openings and part length affect the girder creation in the **Braced girder (88)** modeling tool.

Option	Description
Create always girders	Select whether girders are always created. If you select **Yes**, girders are created even when the girder is placed fully outside the concrete part.
Openings	Select whether girders are created in openings.
Length	Select how girders adapt to the part shape.

Option	Description
Minimum length	Define the minimum length of the girder.

Examples of minimum length
No minimum length defined:

Examples of minimum length
Defined minimum length (no girder created):

Braced girder (88): Double wall tab

Use the **Double wall** tab to select how a second concrete element affects the girders in the **Braced girder (88)** modeling tool.

Look up sec concrete element: Select whether a second concrete element affects the creation of the girders.

The example below shows a hollow wall where inside and outside shells have different geometry.

Examples
A hollow core wall where inside and outside shells have different geometry. **Look up second concrete element** = **No**

Examples
A hollow core wall where inside and outside shells have different geometry. **Look up second concrete element** = **Yes**

Examples

Braced girder (89)

Braced girder (89) creates braced girders between selected points to the selected concrete part.

Objects created The component creates girders that consist of the following parts:

- Bottom reinforcing bars (2)
- Top reinforcing bar
- Connecting reinforcing bars (2)

Instead of using reinforcing bars, you can use profiles from the **Profile Catalog** to create the braced girders.

Use for

Situation	Description
	Braced girders are created in the precast concrete slab.
	Braced girders are created in the precast concrete sandwich panel.
	Braced girders are created in a wide plate floor with an opening.
	Braced girders are created in a hollow wall.

Situation	Description
	Profile as a girder.

Before you start Create a concrete part.

Selection order

1. Select the concrete part.
2. Pick the starting point of a girder.
3. Pick the end point of a girder.

Part identification key

	Part
1	Top reinforcing bar
2	Connecting reinforcing bar
3	Bottom reinforcing bar

See also

Braced girder (89): Picture tab on page 351

Braced girder (89): Parts tab on page 353

Braced girder (89): Geometry tab on page 362

Braced girder (89): Double wall tab on page 364

Braced girder (89): Picture tab

Use the **Picture** tab to control the shape and dimensions of reinforcement bars in the **Braced girder (89)** modeling tool.

Reinforcing bar dimensions

	Description
1	Define the distance between the end of the top reinforcing bar and the picked point.
2	Define the distance between the end of the connecting reinforcing bar and the picked point.
3	Define the distance between the end of the bottom reinforcing bar and the picked point.
4	Define the radius of the connecting reinforcing bar.
5	Define the distance between bendings.

Select the location of the top and bottom reinforcing bars. Define the distance between the bottom reinforcing bars, and the distance between the top and bottom reinforcing bars.

Reinforcing bar properties

Define the reinforcing bar properties for the top, bottom and the connecting reinforcing bars.

Option	Description
Size	Define the size of the reinforcing bar.
Grade	Define the grade of the reinforcing bar.

Option	Description
Comment, **Name**, **Class**, **Serie**, **Start number**	Define a comment, name, and class for the reinforcing bar, and a prefix and a start number for the part position number.
Create	Select whether the reinforcing bar in question is created or not.

Braced girder (89): Parts tab

Use the **Parts** tab to control the girder creation, positioning and number of girders in the **Braced girder (89)** modeling tool.

Girder creation

Option	Description
Creation method	Select the type of the girders.
Profile	If you use a profile as a girder, select the profile from the **Profile Catalog**. In addition, define a prefix and a start number for the part position number, a material and a name.

Workplane orientation

Option	Description
Workplane position	Select the work plane position.
	Use the **Position in plane**, **Rotation** and **Position in depth** options to define the orientation of the girder on the work plane.

Girder position in z direction

Select how the girders are positioned in the z direction.

Example
Braced girders are positioned to picked points. The braced girders are related to the top side of the part.

Geometry

Option	Description
Geometry	Select the geometry for the connecting reinforcing bars. Examples:

Option	Description

Girder position in y direction

Select how the girders are positioned in the y direction.

Examples

Examples

Extra girders

Select whether additional girder groups are created.

Examples
For example, if you have selected this option:

Examples
Extra girder groups is set to **None**: No additional girder groups are created between 2 existing braced girders.

Examples
Extra girder groups is set to **By number**: Additional girder groups are created based on the entered number. Distances between the groups are equally divided.

Examples
Extra girder groups is set to **By distance**: The number of additional girder groups is based on the defined distance. The distances between the girder groups are equally divided.

Girder group

Number is the number of girders in the girder group.

Distance list is the distance between the girders in the girder group.

e = is the position of the girders from the reference line.

Option	Description
   	• Girder group is positioned middle of the reference line. • Girder group is positioned on the right side of the reference line. • Girder group is positioned on the left side of the reference line.

Examples	
Existing girders:	

Examples	

Braced girder (89): Geometry tab

Use the **Geometry** tab to control how openings and part length affect the girder creation in the **Braced girder (88)** modeling tool.

Option	Description
Create always girders	Select whether girders are always created. If you select **Yes**, girders are created even when the girder is placed fully outside the concrete part.
Openings	Select whether girders are created in openings.
Length	Select how girders adapt to the part shape.

Option	Description
Minimum length	Define the minimum length of the girder.

Examples of minimum length
No minimum length defined:

Examples of minimum length
Defined minimum length (no girder created):

Braced girder (89): Double wall tab

Use the **Double wall** tab to select how a second concrete part affects the girders in the **Braced girder (89)** modeling tool.

Look up sec concrete element: Select whether a second concrete element affects the creation of the girders.

The example below shows a hollow wall where inside and outside shells have different geometry.

Examples
A hollow core wall where inside and outside shells have different geometry. **Look up second concrete element** = **No**

Examples
A hollow core wall where inside and outside shells have different geometry. **Look up second concrete element** = **Yes**

Examples

Reinforcement mesh array in area (89)

Creates reinforcement meshes for an entire concrete part, or for a defined polygonal area.

Parts created Reinforcement meshes.

Use for

Situation	Description
	Array of overlapping rectangular meshes.

Situation	Description
	Array of meshes in a polygonal area that you define. Meshes clipped to fit the defined area.

Before you start

- Create the concrete part.
- Set the work plane parallel to the plane where you want to create the mesh array.

Defining properties

Use the following tabs in the **Reinforcement mesh array in area (89)** dialog box to define the properties of the objects that the component creates.

Tab	Contents	See also
Picture	Mesh catalog type, cover thickness and location, mesh generation, cuts, mesh area perimeter.	Mesh generation and overlapping properties on page 367
Overlapping	Overlapping values, mesh offset values.	Mesh generation and overlapping properties on page 367
Attributes	Numbering properties, name, and class of reinforcement meshes.	

Selection order

1. To attach the reinforcement mesh array:
   - to the entire part, pick the starting point of the mesh
   - to a selected area, pick points to define the polygonal shape of the mesh
2. Click the middle mouse button to create the reinforcement mesh array.

Mesh generation and overlapping properties

Use the following properties to define how Tekla Structures creates the mesh array.

Picture tab

Property	Options	Description
Cover thickness		Define the mesh location and give values for **Cover thickness on plane** and **Cover thickness from plane**. The default value for **Cover thickness on plane** is given.

Property	Options	Description
Generation direction	**X** **Y** rotation angle	Generation direction. To rotate the mesh generation direction around the alignment point, enter an angle.
Direction of bottom layer	**Same as top layer** **Opposite of top layer**	Select **Opposite of top layer** to mirror the bottom layer.
Cut by father part cuts	**Yes** **No**	Select **Yes** to cut the mesh with the cuts made in the father part.
Clipping outside	**Yes** **No**	Select **Yes** to clip the meshes to fit the part or the selected area.
Mesh area perimeter	**Polygon** **Part** **Polygon + Part**	Select the outline which the mesh follows. The meshes in the examples are created by picking the same points but with different mesh area perimeters.
	example **Polygon**	

Property	Options	Description
	example **Part**	
	example **Polygon + Part**	

Overlapping tab

Property	Options	Description
Overlapping	**Long side** • **Min. overlapping** • **Max. overlapping** **Short side** • **Min. overlapping** • **Max. overlapping**	Define the minimum and maximum overlap along the longitudinal (**Long Side**) and crossing direction (**Short Side**) of the mesh. The default values are given.

Property	Options	Description
Mesh offset		Define how the meshes are placed.
Mesh start offset	**1/4** **1/3** **1/2** **2/3** **3/4** **Specified** **Symmetric**	Define mesh start offset for the first mesh. The length of the first mesh is divided by the selected value. For example, if you select the **Mesh start offset** to be **1/4**, the length of the first mesh is divided by **1/4**. If you select **Specified**, the **Start length offset** field becomes visible and you can give the offset for the first mesh in the field.

See also Reinforcement mesh array in area (89) on page 366

Wall Panel Reinforcement

Wall Panel Reinforcement is used for detailing precast wall panels. It includes the reinforcement for the panel edges, around the openings, a mesh for the whole panel and additional bars, for example, with a wall shoe connection.

Objects created

- Reinforcement
- Stirrups
- Mesh

Use for

Situation	Description
	Wall panel with openings and reinforcement in the panel edges and around openings.

Selection order

1. Select the concrete part to reinforce.

   The reinforcement is created automatically.

See also

Wall Panel Reinforcement: Picture tab on page 371

Wall Panel Reinforcement: Reinforcement tab on page 372

Wall Panel Reinforcement: Opening tab on page 374

Wall Panel Reinforcement: Diagonals tab on page 374

Wall Panel Reinforcement: Column tab on page 375

Wall Panel Reinforcement: Beam tab on page 376

Wall Panel Reinforcement: Additional tab on page 376

Wall Panel Reinforcement: Attributes tab on page 376

Wall Panel Reinforcement: Picture tab

Use the **Picture** tab to select how the corners of a wall panel are reinforced in the **Wall Panel Reinforcement** modeling tool.

Option	Description	
Corner reinforcement	Select how the corners are reinforced and the bar is offset on both sides of the corner.	

Option	Description	
Minimum breaking distance	Cuts smaller than this value are ignored and the reinforcing bars are not cut.	
Rebar count		Creates a single layer of reinforcement.
		Creates two layers of reinforcement.
Cover thickness	Distance from the edge of the wall panel to the side of the outermost bar.	
Door minimal height	Cuts greater than this value are considered as door openings and the additional reinforcing bars that are needed, for example, in shoe connections are not created.	
Create stirrups priority	Select whether horizontal or vertical reinforcing bars are on top.	
Minimum edge length	The edge stirrups are not created if the dimension is less than the defined minimum value. 	

Wall Panel Reinforcement: Reinforcement tab

Use the **Reinforcement** tab to create the reinforcing bars around the wall panel edges in the **Wall Panel Reinforcement** modeling tool.

Option	Description	
Wall panel edge reinforcement type		Creates normal reinforcement.
		Creates U reinforcement.
Mesh	Select whether a mesh is created for the whole panel. If a mesh is created, you can define the mesh properties, or use custom settings. You can use the **Mesh Bars** component to create the mesh.	
Horizontal reinforcing bars	Define the horizontal reinforcing bar properties for the wall panel edges. Note that horizontal bars have three sets of properties, depending on the length of the bar. For example, if the length of the main bar is less than 3000, it uses the first field on left, if in between 3000 and 6000, the middle field and if greater than 6000, the right field. 	
Vertical reinforcing bars	Define the vertical reinforcing bar properties for the wall panel edges. Note that vertical bars have two sets of properties, depending on the length of the bar.	

Option	Description
U Reinforcement	Define the U reinforcing bar properties. Each side of the wall panel has a separate set of U reinforcing bar properties.

Wall Panel Reinforcement: Opening tab

Use the **Opening** tab to create the reinforcing bars around the wall panel opening in the **Wall Panel Reinforcement** modeling tool.

Option	Description	
Opening reinforcement type	Can be set for all four opening edges.	
		Creates normal reinforcement.
		Creates U reinforcement.
Horizontal reinforcing bars	Define the horizontal reinforcing bar properties. Note that horizontal bars have separate sets of properties for the top and the bottom bars.	
Vertical reinforcing bars	Define the vertical reinforcing bar properties.	
U Reinforcement	Define the U reinforcing bar properties. Each side of the opening has a separate set of U reinforcing bar properties.	

Wall Panel Reinforcement: Diagonals tab

Use the **Diagonals** tab to create diagonal reinforcing bars at the corners of the openings in the **Wall Panel Reinforcement** modeling tool.

Option	Description
Create diagonals	Select whether diagonal reinforcing bars are created or not.
Diagonals count	Number of diagonal reinforcing bars.
Size	Select the diameter of a bar.

Option	Description
Grade	Define the strength of the steel used in the bars.
Bending radius	Define the bar bending radius.
L1 and L2	L1 + L2: Length of the diagonal reinforcing bars.
C	Distance between the diagonal reinforcing bar and the corner of the opening.

Wall Panel Reinforcement: Column tab

Use the **Column** tab to a create column reinforcement in the **Wall Panel Reinforcement** modeling tool.

Option	Description	
Column reinforcement		Column reinforcement is as high as the part.
		Column reinforcement does not reach higher than the top edge of opening.
		Column reinforcement is approximately the **L** value higher than opening edge.
Column reinforcement properties	Select whether column reinforcing bars are created or not. • **Minimum column height H**: cuts and openings smaller than the minimum value are not taken into account in the column creation. • **Maximum column width B**: if the width between two openings or cuts or edges is greater than the maximum value, then the column reinforcement is not created. • **Column height offset L1**: distance between the column reinforcement end and the opening edge. • **Column height offset L2**: same as L1 but for the bottom part of column reinforcement.	
Main reinforcing bars	Define the main reinforcing bar properties. Note that main bars have three sets of properties, depending on the width of the column.	

Option	Description
Stirrup type	Select the stirrup type.
Reinforcing stirrups	Define the stirrup properties. Note that stirrups have three sets of properties, depending on the width of the column.

Wall Panel Reinforcement: Beam tab

Use the **Beam** tab to create a beam reinforcement on top of the opening in the **Wall Panel Reinforcement** modeling tool.

Option	Description
Beam reinforcement properties	Select whether beam reinforcing bars are created or not. • **Maximum beam height Hmax**: distance from the top edge of the wall to the top edge of the opening. If the height is greater than the maximum, then the beam reinforcement is not created. • **Minimum beam length Lmin**: distance between the sides of the opening. If the width of the opening is less than the minimum, then the beam reinforcement is not created.
Top	Define the beam top reinforcement properties. Note that the reinforcing bars have two sets of properties, depending on the length of the beam.
Bottom	Define the beam bottom reinforcement properties. Note that the reinforcing bars have two sets of properties, depending on the length of the beam.
Stirrup type	Select the stirrup type.
Reinforcing stirrups	Define the stirrup properties. Note that stirrups have two sets of properties, depending on the length of the beam.

Wall Panel Reinforcement: Additional tab

Use the **Additional** tab to create additional horizontal and vertical reinforcing bars in the **Wall Panel Reinforcement** modeling tool.

Option	Description
Additional reinforcing bars	Select whether additional reinforcing bars are created or not. Define the additional reinforcing bar properties.

Wall Panel Reinforcement: Attributes tab

Use the **Attributes** tab to control the reinforcing bar properties of the created parts in the **Wall Panel Reinforcement** modeling tool.

Option	Description
Prefix	Define a prefix for the part position number.
Start number	Define a start number for the part position number.
Name	Define a name for the part. Tekla Structures uses the name in drawings and reports.
Class	Define the part class number.

Multiple Wire Size Mesh

Multiple Wire Size Mesh creates reinforcement meshes with multiple reinforcing bar sizes. The created reinforcement mesh is optimized for a mesh welding machine. **Multiple Wire Size Mesh** can be used for slabs and wall panels. The slabs and wall panels can be of any polygonal shape with any number of openings. The reinforcing bars in the mesh can be of different size, and the spacings between the bars can vary.

Objects created

- Reinforcement meshes

Use for

Situation	Description
	Three zones of longitudinal reinforcing bars

Before you start

Create a polygon slab or a wall panel with any number of openings. Enter zone values to define the width of the area to be reinforced in both the longitudinal and crossing directions. Each direction can be divided up to five zones.

Selection order

1. Select the main part (slab or wall panel).

   The reinforcement mesh is created.

See also

Multiple Wire Size Mesh: Mesh Parameters tab on page 378

Multiple Wire Size Mesh: Longitudinal/Crossing Wires tab on page 378

Multiple Wire Size Mesh: Attributes tab on page 379

Multiple Wire Size Mesh: Configuration file for reinforcing bar patterns on page 380

Multiple Wire Size Mesh: Mesh Parameters tab

Use the **Mesh Parameters** tab to control the mesh creation and cover thickness in the **Multiple Wire Size Mesh** modeling tool.

Option	Description
	Define the concrete cover thickness.
Create Rebars	Define to which side of the slab or the wall panel the reinforcing bars are created.
Longitudinal Direction	Define whether the x or the y direction is used as the longitudinal direction when the reinforcing bars are created. By default, the x direction is the longitudinal direction.

Multiple Wire Size Mesh: Longitudinal/Crossing Wires tab

Use the **Longitudinal Wires** or the **Crossing Wires** tab to control the mesh creation in the longitudinal or the crossing direction, and the reinforcement zones in the **Multiple Wire Size Mesh** modeling tool.

Option	Description
Wire sizes	Define the sizes of the reinforcing bars used in the pattern. Separate the sizes with a space.
Min overhang	Define the minimum length of the reinforcing bar extension.
Max overhang	Define the maximum length of the reinforcing bar extension.
Min spacing	Define the minimum space between the reinforcing bars. The value must be divisible by the grid size. For example, if the **Min spacing** is `150`, **Max spacing** is `300` and the **Grid size** is `50`, the spacings are `150`, `200`, `250` and `300`.

Option	Description
Max spacing	Define the maximum space between the reinforcing bars. The value must be divisible by the grid size. For example, if the **Min spacing** is `150`, **Max spacing** is `300` and the **Grid size** is `50`, the spacings are `150`, `200`, `250` and `300`
Grid size	Define the mesh grid size. For example, if the **Min spacing** is `150`, **Max spacing** is `300` and the **Grid size** is `50`, the spacings are `150`, `200`, `250` and `300`. This value depends on the welding machine.
Min wire length	Define the minimum reinforcing bar length.
Min number of welding points	Define the minimum number of crossing reinforcing bar intersections.
Pattern width to fulfill	Define the width of the pattern that is filled with the mesh. This value is used when the zone width is greater than the given value.
Zone width value type	Define the width of the area to be reinforced as a percentage of the whole slab or wall panel width, or as the actual length.
Width, **Real width**, **Mesh area per length**	Define the mesh width, real width and area per length in each zone.
Load pattern from file	Select the external text file where you have defined the patterns for longitudinal and crossing reinforcing bars. If you use the external patterns file, **Multiple Wire Size Mesh** tries to determine a pattern with a suitable area per length value for every zone and apply these values to the zones.

Multiple Wire Size Mesh: Attributes tab

Use the **Attributes** tab to control the reinforcement properties in the longitudinal and in the crossing directions in the **Multiple Wire Size Mesh** modeling tool.

Option	Description
Prefix	Define a prefix for the part position number.
Start Number	Define a starting number for the part position number.
Grade	Define the strength of the steel used in the reinforcing bars.

Option	Description
Name	Define a name for the longitudinal and the crossing reinforcing bars. Tekla Structures uses the name in drawings and reports.
Class	Use **Class** to group the longitudinal and the crossing reinforcing bars. For example, you can display the longitudinal and the crossing reinforcing bars of different classes in different colors.

Multiple Wire Size Mesh: Configuration file for reinforcing bar patterns

Use an external text file to define the patterns for longitudinal and crossing reinforcing bars in the **Multiple Wire Size Mesh** modeling tool.

Every pattern starts from the **Pattern** line, followed by, for example, the pattern name. Each **Bar Size** and **Spacing** pair is on a separate row. The **Bar Size** and **Spacing** values are separated by a space.

The file has the following format:

```
< Pattern > < Name >
<Bar Size> <Spacing>
<Bar Size> <Spacing>
.................................
<Bar Size> <Spacing>
```

Examples

Pattern 1

```
10 200
10 200
8 100
8 300
10 400
```

Pattern 2

```
10 200
10 200
8 100
8 100
```

Embedded anchors (8)

Embedded anchors (8) creates one or more embeds in a concrete part.

Objects created

- Embeds
- Thickening parts
- Reinforcing bars
- Hangup bars
- Crane cables

Use for

Situation	Description
	Two custom anchors.

Situation	Description
	Bolt anchors that are custom parts, hangup bars and crane cables.
	Cast-in lifting anchors that are custom parts.

Situation	Description
	Cast-in wire rope lifting hoops that are custom parts. Adjusting anchors are in the front side. The component is applied two times.

Selection order

1. Select a concrete part.
2. Pick at least one point on the face where you want to insert the embed.

   You can also select multiple points, depending on how the embeds are divided.
3. Click the middle mouse button to create the embeds.

Part identification key

	Part
1	Concrete part (wall, column, beam, slab)
2	Point You can select multiple points.

See also

Embedded anchors (8): Input tab on page 385

Embedded anchors (8): Placement tab on page 389

Embedded anchors (8): Top placement tab on page 394

Embedded anchors (8): Thickening tab on page 396

Embedded anchors (8): Picture tab on page 397

Embedded anchors (8): Top part tab on page 400

Embedded anchors (8): Bottom part tab on page 400

Embedded anchors (8): Parts tab on page 401

Embedded anchors (8): Reinforcing bar tab on page 402

Embedded anchors (8): Hangup bars tab on page 404

Embedded anchors (8): Horizontal bar tab on page 406

Embedded anchors (8): Special bar tab on page 408

Embedded anchors (8): Advanced tab on page 410

Embedded anchors (8): Input tab

Use the **Input** tab to control the custom parts used for the embeds, the input points for placing the embeds and the component direction properties in the **Embedded anchors (8)** modeling tool.

Input points sequence

Select the distribution method for the embeds. You can also adjust the placement of embeds on the **Placement** tab.

Option	Description
	Select a concrete part and pick one position point for the anchor.
	Select a concrete part and pick multiple position points for the anchors.
	Select a concrete part and pick three points to define the plane. Pick a point to place the embed.
	Select a concrete part and pick three points to define the plane. Pick multiple points to place the embeds.

Up direction

Define how the embeds are rotated.

This option is not available if the **three-point-plane** option is selected from the **Input points sequence** list.

You can use the local directions +X,-X,+Y,-Y,+Z,-Z to define a specific face of a typical cube-shaped part.

- X local:

+X global:

Positioning examples

Embed handling

Select whether the embed is handled as a modeling tool or as a detail.

In general, a modeling tool responds to the selected points and a detail scans the main part's contour.

In certain situations, a modeling tool responds better to changes in the main part contour.

Embed offset

Define embed offsets in X- and/or Y-direction, from the selected position points.

Custom part

- Define the name of the custom component to be used for the embeds. If the embeds are defined on the **Picture** tab, leave this box empty.
- Define the name of the configuration file for the custom component.
- Select whether the custom parts must be used for the embeds.

  If set to **Not**, define the embeds' shape on the **Picture** tab.

  If set to **Yes**, define the connecting method.
- Select how the embeds are rotated.

- Select how the embeds are rotated around a position point.

Embedded anchors (8): Placement tab

Use the **Placement** tab to control the placing of the embeds, crane cable angles and safety factor properties and to select the lifting anchors in the **Embedded anchors (8)** modeling tool.

You can define the input point sequence, lift system and embed dimensions both in reference line and cross section direction.

Input points sequence

Select the distribution method of the embeds.

Use the **a** and **b** boxes on the right to enter the values.

Option	Description
	Embed is placed on a selected position point.
	Embed is placed on a center of gravity (COG) point in the part length direction.

Option	Description
	Multiple embeds. Define dimensions **a** and **b**. Reference = COG
	Define dimensions **a** and **b**. Reference = COG If embeds are likely to be placed outside the part, then value **d** is used.
	Define dimensions **a** and **b** as percentages of the part length. Reference = COG
	Define dimension **a** as a percentage of the part length. Reference = COG
	Define dimension **a**. The right embed is calculated from COG.
	Define dimension **a** as a percentage of the total part length. Reference = COG
	Embed is placed in the middle of the part.
	Multiple embeds. Define dimensions **a** and **b**. Reference = middle of the part
	Define dimensions **a** and **b** as percentages of the part length. Reference = middle of the part

Option	Description
	Define dimensions **a** and **b**. The distances are from the embeds to the part ends.
	Define dimensions **a** and **b** as percentages of the part length. The distances are from the embeds to the part ends.

Lift system

Select the crane cable lift system.

Dimensions

Boxes for defining embed dimensions **a** and **b**, crane cable angle **alpha** and crane cable height **h**.

Extra embeds

Define extra embeds. Use the boxes on the left and right to define distances. You can enter multiple distances. Use a space to separate the values.

Use the middle box to define the number of additional embeds.

Mirroring

- Define the gravity point (POG).
- Select whether embeds are mirrored.

  Mirroring is useful for asymmetric custom embeds.

- Select whether both embeds, or only the left or the right embed is displayed.

Lift.dat configuration file

You can use a configuration file to calculate the lifting capacities of lifting anchors.

Set **Browse configuration file** to **Yes**, and define the full path to the configuration file. The default file is `lift.dat`, located in the `profil` folder.

The `lift.dat` configuration file contains a list of all custom lifting anchors belonging to the default Tekla Structures installation. The anchors are sorted based on fabricator and type, component name and lifting capacity. You can add more custom components to the list in the configuration file. The file can be opened in any text editor, for example Notepad.

lift.dat - Kladblok

File Edit Format View Help

```
//Type  Type-client       Name CuCoPa      Lift-force ^(kg)
//-------------------------------------------------------

// PFEIFER-Plaatstaalanker
1    PFEIFER-Plaatanker   pfeifer_HP-Rd12_0-5T        500
1    PFEIFER-Plaatanker   pfeifer_HP-Rd16_1-2T       1200
1    PFEIFER-Plaatanker   pfeifer_HP-Rd20_2-0T       2000
1    PFEIFER-Plaatanker   pfeifer_HP-Rd24_2-5T       2500
1    PFEIFER-Plaatanker   pfeifer_HP-Rd30_4-0T       4000
1    PFEIFER-Plaatanker   pfeifer_HP-Rd36_6-3T       6300

// PFEIFER-Golfanker
2    PFEIFER-Golfanker    pfeifer_HA-Rd12_0-5T        500
2    PFEIFER-Golfanker    pfeifer_HA-Rd16_1-2T       1200
2    PFEIFER-Golfanker    pfeifer_HA-Rd20_2-0T       2000
2    PFEIFER-Golfanker    pfeifer_HA-Rd24_2-5T       2500
2    PFEIFER-Golfanker    pfeifer_HA-Rd30_4-0T       4000
2    PFEIFER-Golfanker    pfeifer_HA-Rd36_6-3T       6300
2    PFEIFER-Golfanker    pfeifer_HA-Rd42_8-0T       8000
2    PFEIFER-Golfanker    pfeifer_HA-Rd52_12-5T     12500
2    PFEIFER-Golfanker    pfeifer_HA-Rd56_15-0T     15000
2    PFEIFER-Golfanker    pfeifer_HA-Rd60_20-0T     12500

// PFEIFER - Hijslussen
3    PFEIFER-Hijslus      pfeifer_BS_0-8T         800
3    PFEIFER-Hijslus      pfeifer_BS_1-2T         1200
3    PFEIFER-Hijslus      pfeifer_BS_1-6T         1600
3    PFEIFER-Hijslus      pfeifer_BS_2-0T         2000
3    PFEIFER-Hijslus      pfeifer_BS_2-5T         2500
3    PFEIFER-Hijslus      pfeifer_BS_4-0T         4000
3    PFEIFER-Hijslus      pfeifer_BS_5-2T         5200
3    PFEIFER-Hijslus      pfeifer_BS_6-3T         6300
3    PFEIFER-Hijslus      pfeifer_BS_8-0T         8000
```

- **Type**

  Groups where lifting anchors can be selected.

- **Type client**

  Description of types.

- **Name CuCoPa**

  Name of custom component part (CuCoPa) as listed in the component database.

- **Lift force (kg)**

  Lifting capacity in kilograms.

- **Dir**

  Direction. Component direction during creation of CuCoPa.

- **Attribute file**

  Link to configuration file.

The anchor type can be selected from the list. The first list option corresponds to the first type (`1`) in the `lift.dat` configuration file.

The last list option is **All types and use preference list**. This option searches from all groups. A preference can be defined in the **Preference type list**. Start with the highest preference group and separate the groups with a space.

Several fabricators have lifting anchors with the same lifting capacity, but with different anchor lengths. Use the order in the configuration file to define which anchor is selected. First found anchor with correct capacity = 1, second found anchor = 2, and so on.

If you have selected to use a custom part, ensure that the corresponding custom parts are available in the model. Names of the custom parts should not contain spaces, because they are not read from the `lift.dat` configuration file .

Safety factor Define the safety factor. The weight to be lifted up is multiplied with this factor in the anchor calculation.

Embedded anchors (8): Top placement tab

Use the **Top placement** tab to define the embed offsets and how the information message box and crane cables are shown in the **Embedded anchors (8)** modeling tool.

Offset Define the embed offsets at both ends of the concrete part. An embed which is positioned exactly in the middle of the concrete part stays in that position and does not get an offset.

Show information message boxes Define whether a message box containing the embed calculation results is shown. The message box shows the weight to be lifted up, the real lift-up weight including a safety factor, and the selected anchor.

The message box appears only if you have set **Browse Configuration File** to **Yes** on the **Placement** tab.

Show crane cables Define whether crane cables are shown. If set to **Yes**, crane cables are shown as construction lines or profiles with the **alpha** angle, defined on the **Placement** tab.

Embedded anchors (8): Thickening tab

Use the **Thickening** tab to define the thickening parts to the concrete part in the **Embedded anchors (8)** modeling tool.

Thickenings are useful especially with sandwich walls.

Thickening properties

Option	Description
	Select the side where thickening is applied. You can also define the thickening thickness.
	Select the thickening part reference, and define the thickening width and thickening height.

Option	Description
	Select how to connect the thickening parts to the main concrete part.
	Select whether an extra insulation layer is added in the thickening location.

Example

Embedded anchors (8): Picture tab

Use the **Picture** tab to control the embed dimensions, connection method and rotation in the **Embedded anchors (8)** modeling tool.

Embed dimensions

Enter values to dimension the negative volume and screw depth for a second, bottom, embed part.

Select the shape of the cutout.

Option	Example
Circle	
Square	

Option	Example
Half moon x	
Half moon y	

As negative volume Select whether cutout profiles are handled as negative volume or as positive volume.

Positioning Select how to position the top and the bottom parts of the embed.

Option	Example

Option	Example

How to connect top part to concrete element

Select whether the top part of embed is created, and if set to **Yes**, select how the part is connected to concrete part.

How to connect bottom part

Select whether the bottom part of embed is created, and if set to **Yes**, select how the part is connected to the concrete part.

Embedded anchors (8): Top part tab

Use the **Top part** tab to define the top part of the embed in the **Embedded anchors (8)** modeling tool.

Top part dimensions

Use the diameter and height boxes to define the shape of the top part of the embed. The main diameters of parts **1**, **2**, **3**, **4** and **5** can also be defined on the **Parts** tab.

Embedded anchors (8): Bottom part tab

Use the **Bottom part** tab to define the bottom part of the embed in the **Embedded anchors (8)** modeling tool.

Bottom part dimensions

Use the diameter and height boxes to define the shape of the bottom part of the embed. The main diameters of parts **1**, **2**, **3**, **4** and **5** can also be defined on the **Parts** tab.

Embedded anchors (8): Parts tab

Use the **Parts** tab to define the embed top and bottom part properties in the **Embedded anchors (8)** modeling tool.

Part properties Define the part properties for both the top and the bottom part. If the profile properties are left empty, the lengths and diameters defined on the **Top Part** and **Bottom Part** tabs are used.

Option	Description
t, b, h	Define the part thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name for the part.
Class	Define the part class number.
Comment	Add a comment for the part.

You can define UDAs for the top and bottom parts.

Embedded anchors (8): Reinforcing bar tab

Use the **Reinforcing bar** tab to define extra reinforcing bars for the embeds in the **Embedded anchors (8)** modeling tool.

You can define the reinforcing bar shape and hook properties, and the reinforcing bar profile properties in two directions.

Grade, Size

The **Grade** and **Size** options work together. Open the reinforcing bar catalog to select a grade and a size for **bar A** and **bar B**.

End conditions left / right

Select the shape of the reinforcing bar.

Bend lengths left / right, Bend radius

Define the bend length for the hooks and the bend radius.

Rotation

Select how to rotate the reinforcing bars and define the rotation angle.

Dimensions Define the reinforcing bar dimensions and the number of bars.

Embedded anchors (8): Hangup bars tab

Use the **Hangup bars** tab to define extra reinforcing bars for the embeds in the **Embedded anchors (8)** modeling tool.

You can define the reinforcing bar shape and hook properties, and the reinforcing bar profile properties.

Grade, Size The **Grade** and **Size** options work together. Open the reinforcing bar catalog to select a grade and a size.

End conditions left / right

Select the shape of the reinforcing bar.

Bend lengths left / right, Bend radius

Define the bend length for the hooks and the bend radius.

Rotation

Select how to rotate the hangup reinforcing bars and define the rotation angle.

Dimensions

Option	Description	Default
1	Select how to place the hangup reinforcing bars. Use the values **a**, **b**, **c**, **d**, **e**, **f** and the angles **alpha** and **beta** to define the dimensions of the hangup reinforcing bars.	
2	Define the number of hangup reinforcing bars.	0
3	Define the cover thicknesss.	
4	Define the distance between hangup reinforcing bars.	100 mm

Embedded anchors (8): Horizontal bar tab

Use the **Horizontal bar** tab to define extra horizontal reinforcing bars for the embeds in the **Embedded anchors (8)** modeling tool.

You can define the reinforcing bar shape and hook properties, and the reinforcing bar profile properties.

Grade, Size The **Grade** and **Size** options work together. Open the reinforcing bar catalog to select a grade and a size.

End conditions left / right Select the shape of the reinforcing bar.

Bend lengths left / right, Bend radius Define the bend length for the hooks and the bend radius.

Rotation Select how to rotate the horizontal reinforcing bars and define the rotation angle.

Side view You can create sloped horizontal reinforcing bars. Define the number of bars and the distance between them. Use value **d** to define cover thickness for the top side and value **f** to define the reinforcing bar length.

Top view Select a reinforcing bar shape.

Use values **a**, **b**, **c**, **d**, **e**, **f** and the angles **alpha** and **beta** to define the dimensions of the horizontal reinforcing bars.

Embedded anchors (8): Special bar tab

Use the **Special bar** tab to define extra U-shaped reinforcing bars for the embeds in the **Embedded anchors (8)** modeling tool.

You can define the reinforcing bar shape and hook properties, and the reinforcing bar profile properties in two directions.

Grade, Size The **Grade** and **Size** options work together. Open the reinforcing bar catalog to select a grade and a size for **bar A** and **bar B**.

End conditions left / right Select the shape of the reinforcing bar.

Bend lengths left / right, Bend radius Define the bend length for the hooks and the bend radius.

Rotation Select how to rotate the U-shaped reinforcing bars and define the rotation angle.

Shape Select a reinforcing bar shape.

Use the values **a**, **b**, **c**, **d**, **e**, **f** and the angles **alpha** and **beta** to define the dimensions of the U-shaped reinforcing bars.

Embedded anchors (8): Advanced tab

Use the **Advanced** tab to define UDAs and reinforcing bar properties for the reinforcing bars, hangup bars, special bars and horizontal bars in the **Embedded anchors (8)** modeling tool.

Reinforcing bar properties

Option	Description
Comment	Add a comment for the reinforcing bars.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number for the reinforcing bars.
Serie	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Embed (1008)

Embed (1008) creates one or more embeds in concrete parts. You can create multiple embeds to be used as lifting anchors with one insertion point.

Objects created

- Embeds
- Reinforcing bars

Use for

Situation	Description
	Embeds

Selection order

1. Select a concrete part.
2. Pick one point on the part face where you want to insert the embed.

   The detail is created automatically when you pick the point.

See also

Embed (1008): Picture tab on page 412

Embed (1008): Top part tab on page 414

Embed (1008): Bottom part tab on page 414

Embed (1008): Parts tab on page 415

Embed (1008): Placement tab on page 416

Embed (1008): Reinforcing bar tab on page 418

Embed (1008): Advanced tab on page 418

Embed (1008): Bolts on page 418

Embed (1008): Picture tab

Use the **Picture** tab to control the embed dimensions, connection method and rotation in the **Embed (1008)** detail.

Embed dimensions

Define if the embed needs to be recessed. You can define the dimensions of the recess, distance from the recess plane to the top of the embed, select the shape of the cutout and whether cutouts are handled as negative volume or as positive volume.

Option	Example
Circle	

Option	Example
Half moon X	
Square	
Half moon Y	

Top part/Bottom part Set the part class and rotation. Each option rotates the embed 90 degrees counterclockwise. You can define also a fixed rotation angle.

How to connect top part to concrete element

Select whether the top part of the embed is created, and if set to **Yes**, select how the part is connected to the concrete part.

How to connect bottom part

Select whether the bottom part of the embed is created, and if set to **Yes**, select how the part is connected to the concrete part.

Select an embed from the Component Catalog

Select a custom part from the Component Catalog. Use the **Yes and subassembly** option to add the embed as subassembly to the main part. The default direction is **2 point +x**.

Embed (1008): Top part tab

Use the **Top part** tab to define the top part of the embed in the **Embed (1008)** detail.

Top part dimensions

Use the diameter and height boxes to define the shape of the top part of the embed. You can define the main diameters of parts **1**, **2**, **3**, **4** and **5** also on the **Parts** tab.

Examples

Embed top part defined on the **Top part** tab, embed bottom part defined on the **Bottom part** tab.

Embed (1008): Bottom part tab

Use the **Bottom part** tab to define the bottom part of the embed in the **Embed (1008)** detail.

Bottom part dimensions

Use the diameter and height boxes to define the shape of the bottom part of the embed. You can define the main diameters of parts **1**, **2**, **3**, **4** and **5** also on the **Parts** tab.

For examples, see Embed (1008): Top part tab on page 414.

Embed (1008): Parts tab

Use the **Parts** tab to define the embed top and bottom part profiles in the **Embed (1008)** detail.

Part properties

Both the top and the bottom part are build from multiple profiles. You can define profiles for each section.

Define the part properties for the top, bottom and formwork part. If the profile properties are left empty, the lengths and diameters defined on the **Top Part** and **Bottom Part** tabs are used.

Option	Description
t, **b**, **h**	Define the part thickness, width and height.
Pos_No	Define a prefix and a start number for the part position number.
Material	Define the material grade.
Name	Define a name for the part.
Comment	Add a comment for the part.

You can define UDAs for the top and bottom parts.

Embed (1008): Placement tab

Use the **Placement** tab to define the embed placing, positioning and embed distribution in the **Embed (1008)** detail.

Positioning Select how to position the top and the bottom part of the embed.

Placing Define the placing type of the embed. Depending on the component insertion points, multiple embeds can be created.

Option	Description
	Embed is placed on a selected position point.
	Embed is placed on a center of gravity (COG) point in the part length direction.

Option	Description
	Multiple embeds. Define dimensions **a** and **b**. Reference = COG
	Define dimensions **a** and **b** as percentages of the part length. Reference = COG
	Embed is placed in the middle of the part, along the length of the concrete part.
	Multiple embeds. Define dimension **a** as a percentage of the total part length. Reference = COG
	Embed is placed in the middle of the part.
	Define dimensions **a** and **b**. Reference = middle of the part
	Define dimensions **a** and **b** as percentages of the part length. Reference = middle of the part
	Define dimensions **a** and **b**. The distances are from the embeds to the part ends.
	Define dimensions **a** and **b** as percentages of the part length. The distances are from the embeds to the part ends.

Dimensions Define embed dimensions **a** and **b**.

Center of gravity Define concrete part COG (center of gravity) for the embeds.

Number of extra anchors The spacing distance between multiple embeds.

Embed (1008): Reinforcing bar tab

Use the **Reinforcing bar** tab to define extra reinforcing bars for the embeds in the **Embed (1008)** detail.

You can define the reinforcing bar shape properties, and the reinforcing bar profile properties in two directions.

You can define the number, shape, dimension and covering thickness of the reinforcing bars.

Embed (1008): Advanced tab

Use the **Advanced** tab to define bar properties for reinforcing bars A and B in the **Embed (1008)** detail.

Reinforcing bar properties

Option	Description
Comment	Add a comment for the reinforcing bars.
Name	Define a name that is shown in drawings and reports.
Class	Define the part class number for the reinforcing bars.
Serie	Define a prefix for the part position number.
Start number	Define a start number for the part position number.

Embed (1008): Bolts

Click the **Bolts** button to open the **Bolts** dialog box where you can define extra embeds and embed offsets in the **Embed (1008)** detail.

Embed (1008) uses only the **Horizontal Dist.** and **Vertical Dist.** options, the other options are not taken into account.

Example

In the example below, the bolt placement uses fixed dimensions from part edges, defined on the **Placement** tab. Extra embeds are defined in the **Bolts** dialog box.

Optionally, you can also use the **Sec/first** and **Positioning** options to define a distance from the edge of the beam.

Continuous Beam Reinforcement

You can reinforce a continuous beam using a macro called **ContinuousBeamReinforcement**. The macro creates main top and bottom bars, stirrups, fittings, and additional top and bottom bars using system components. **Longitudinal reinforcement (70)** creates the main and additional top and bottom bars, **Stirrup reinforcement (67)** creates stirrups, and **Battering connection (13)** creates fittings.

To reinforce a continuous beam:

1. Go to **Tools** --> **Macros...** , and run **ContinuousBeamReinforcement**.
2. Select the beams and click **Next**.
3. Select the columns and click **Next**.
4. If needed, modify the reinforcement properties, and click **Finish**.

   The reinforcements are created.

Limitations The beams must be aligned for the reinforcing to be successful.

Example

2.3 Lifting

This section introduces components that can be used for lifting.

Click the links below to find out more:

- Lifting anchor (80) on page 421

Lifting anchor (80)

Creates two lifting anchors (or anchor groups) for a concrete part and places them symmetrically on either side of the part's center of gravity.

1 30% of part length (default)

2 Center of gravity

Objects created

- Lifting anchors (2 or more)
- Recesses for anchors (optional)

Use for

Situation		More information
		Straight anchor with straight legs (Type A in the anchor properties file, see Anchor properties from file on page 426)
		Straight anchor with L-shaped legs (Type D)
		Angle anchor with straight legs (Type B)

Situation	More information
	Angle anchor with L-shaped legs (Type C)
	Custom components as anchors
	Anchors recessed into the part.
	Skew and/or rotated anchors

Before you start Create the concrete part.

Defining properties Use the following tabs in the **Lifting anchor (80)** dialog box to define the properties of the objects that this component creates:

Tab	Contents	See also
Picture	Anchor dimensions and locations, number and spacing of anchors, option to use custom components as anchors, option to create recesses, recess dimensions, skew angle of anchors, option to rotate anchors, basic anchor properties	Lifting anchor properties on page 424 Custom components as lifting anchors on page 425 Creating recesses for lifting anchors on page 426

Picking order 1. Concrete part

Lifting anchor properties

The properties of lifting anchors are:

Field	Description
Anchor dimensions	The options are: • **Use dialog values** • **Select from file**, see Anchor properties from file on page 426. You need to enter different dimensions for different anchor types.
	The distance between an anchor (or anchor group) and the center of gravity of the part. Default is 30% of the part length.
	The distance between an anchor (or group) and the center line of the part.
Number of bars/ spacings	The number and spacing of anchors in a group. If the spacing varies, enter each value individually.
Custom Component	See Custom components as lifting anchors on page 425.
Custom settings	
Up direction	
	The option to define if the anchors are recessed into the part. See Creating recesses for lifting anchors on page 426.

Field	Description
	The skew angle of anchors. Anchor heads are skewed towards the part's center of gravity.
Rotate anchor	The option to rotate anchors. The options are: • **No**: Anchors parallel to the part. • **Yes**: Anchors perpendicular to the part.
Side	The side of the part where Tekla Structures creates the anchors. The options are **Front**, **Top**, **Back**, **Below**, **Start end**, and **Finish end**.
Grade, Size, Name, Prefix, Start number, Class	

Custom components as lifting anchors

To use custom components as lifting anchors:

1. In the **Custom** list box, select **Yes**.
2. Click the **...** button next to the **Component** field to open the **Select component** dialog box.
3. Browse for the custom component you want to use as lifting anchor.

   The component you select must be a custom part and have two or three input points.

   The component should be created so that the first and the second input points are on the concrete surface and along the longitudinal axis of the concrete part.

   For more information on how to create custom parts that can be used as lifting anchors, log in to the Tekla Extranet and go to **Self learning** --> **Manuals & instructions** , and open the **Standard embed creation instruction** document.

4. Select the component and click **OK**.
5. To use saved custom component properties, enter the name of the saved properties file in the **Custom settings** field.
6. If the anchor position is not correct, select another option in the **Up direction** list box.

See also For more information on custom components, see .

Creating recesses for lifting anchors

Select one of the following options to define if the lifting anchors are recessed into the concrete part:

Option	Description
	Anchors on the surface of the part. (Default)
	Anchors recessed into the part.

If you choose to recess the anchors into the part, use the following fields to define the shape and dimensions of the recesses:

Anchor properties from file

You can define lifting anchor properties by entering values in the dialog box (see Lifting anchor properties on page 424), or you can create a file containing the anchor properties you want to use. Use any standard text editor to create the file and save it as `LiftingAnchors.dat` in the model folder.

To use the anchor properties you define in a file:

1. In the **Lifting anchor (80)** dialog box, select **Select from file** from the **Anchor dimensions** list box.
2. Click the **...** button to browse for the file.

Example Create a row in the file for each lifting anchor. Enter the following properties, separated by spaces:

- Anchor capacity [kN], including safety and material factors

- Type [A, B, C, or D], see the **Use for** table in Lifting anchor (80) on page 421
- Grade [characters]
- Size [characters]
- Anchoring length 1 [mm]
- Anchoring length 2 [mm]
- Protruding dimension of the anchor outside the part [mm]
- Penetrating dimension of the anchor inside the part [mm]
- Bending radius [mm]
- Hook length [mm] (0 = no hook)
- Angle of leg 1 [degrees]
- Angle of leg 2 [degrees]

Here is an example of a lifting anchor file (Tekla Structures ignores the comments enclosed in /* */):

LiftingAnchors.dat

```
10 B A500HW 10 400 400 100   0 30  75 30 30  /*      /\       */
20 B A500HW 12 600 600 150   0 36 100 30 30  /*     /  \      */
30 B A500HW 16 900 900 200   0 80 150 30 30  /*   \/    \/    */
10 B A500HW 10 400 400 100   0 30   0 30 30  /*      /\       */
20 B A500HW 12 600 600 150   0 36   0 30 30  /*     /  \      */
30 B A500HW 16 900 900 200   0 80   0 30 30  /*    /    \     */
10 C A500HW 10 400 400 100 200 30  75 30 30  /*      /\       */
20 C A500HW 12 600 600 150 300 36 100 30 30  /*  _  /  \  _   */
30 C A500HW 16 900 900 200 500 80 150 30 30  /* |__/    \__|  */
10 C A500HW 10 400 400 100 200 30   0 30 30  /*      /\       */
20 C A500HW 12 600 600 150 300 36   0 30 30  /*     /  \      */
30 C A500HW 16 900 900 200 500 80   0 30 30  /* ___/    \___  */
```

Tekla Structures will use the first anchor in the file that:

1. Has the shape and type you define using the graphic list boxes in the **Lifting anchor (80)** dialog box.
2. Can carry the part's weight with other identical anchors. The total number of anchors is two times the number you specify in the **Number of bars** field.

3 Disclaimer

© 2015 Tekla Corporation and its licensors. All rights reserved.

This Software Manual has been developed for use with the referenced Software. Use of the Software, and use of this Software Manual are governed by a License Agreement. Among other provisions, the License Agreement sets certain warranties for the Software and this Manual, disclaims other warranties, limits recoverable damages, defines permitted uses of the Software, and determines whether you are an authorized user of the Software. All information set forth in this manual is provided with the warranty set forth in the License Agreement. Please refer to the License Agreement for important obligations and applicable limitations and restrictions on your rights. Tekla does not guarantee that the text is free of technical inaccuracies or typographical errors. Tekla reserves the right to make changes and additions to this manual due to changes in the software or otherwise.

In addition, this Software Manual is protected by copyright law and by international treaties. Unauthorized reproduction, display, modification, or distribution of this Manual, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the full extent permitted by law.

Tekla, Tekla Structures, Tekla BIMsight, BIMsight, Tedds, Solve, Fastrak and Orion are either registered trademarks or trademarks of Tekla Corporation in the European Union, the United States, and/or other countries. More about Tekla trademarks: http://www.tekla.com/tekla-trademarks. Trimble is a registered trademark or trademark of Trimble Navigation Limited in the European Union, in the United States and/or other countries. More about Trimble trademarks: http://www.trimble.com/trademarks.aspx. Other product and company names mentioned in this Manual are or may be trademarks of their respective owners. By referring to a third-party product or brand, Tekla does not intend to suggest an affiliation with or endorsement by such third party and disclaims any such affiliation or endorsement, except where otherwise expressly stated.

Portions of this software:

D-Cubed 2D DCM © 2010 Siemens Industry Software Limited. All rights reserved.

EPM toolkit © 1995-2004 EPM Technology a.s., Oslo, Norway. All rights reserved.

Open CASCADE Technology © 2001-2014 Open CASCADE SA. All rights reserved.

FLY SDK - CAD SDK © 2012 VisualIntegrity™. All rights reserved.

Teigha © 2003-2014 Open Design Alliance. All rights reserved.

PolyBoolean C++ Library © 2001-2012 Complex A5 Co. Ltd. All rights reserved.

FlexNet Copyright © 2014 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 licenses, go to Tekla Structures, click **Help** --> **About** and click the **Third party licenses** button.

The elements of the software described in this Manual are protected by several patents and possibly pending patent applications in the European Union and/or other countries. For more information go to page http://www.tekla.com/tekla-patents.

Index

A

Automatic seam recognition (30)..... 151

B

battering connection (13).....21
beam end reinforcement (79)..... 305
beam ends
    concrete components 76, 78.....16
Beam Formwork..... 125
beam reinforcement (63).....299
bearing plate
    concrete components 75, 76.....14
bolt hole dimensions in concrete components.....11
Braced girder (88).....331
Braced girder (89).....349

C

cast-in-place.....420
cast-in-situ.....420
CIP.....420
Column Formwork..... 128
column reinforcement
    round columns (82)..... 313
    stirrups..... 314
    top and bottom.....315
Concrete beam-beam (112).....32
concrete components.....9
    stairs (7).....207
    Stairwells and elevator shafts (90)..... 223
concrete detailing.....9
Concrete foundation (1030).....256
concrete slab
    generation with points (62)..... 157
    generation with polygon plate (61)..... 154
Concrete stairs (65).....208
concrete stairs
    about.....206
    properties..... 207,225
connections
    seating (concrete).....9
continuous beam
    reinforcing.....420
Corbel connection (14).....22
Corbel reinforcement (81).....308
corbels
    chamfering..... 12
custom components
    as lifting anchors..... 425

D

double tee reinforcement (64).....302

E

Electric box in wall (84)..... 60
elevator shaft.....223
Embed (1008).....411
Embedded anchors (8).....380

F

fitting options
    concrete component 75.....11
    concrete component 76.....16
Floor Layout.....182
Floor Layout CIP Filler..... 205
Floor Tool.....179
flooring decks..... 154

G

General Beam Form.....136
General Clamp..... 141
General Clip.....142
General Tie.....144

General Tie and Clips..... 146
General Wall Form.....138
grout
concrete components 75, 76.....12

H

Hole Generation (32)..... 148
hole reinforcement
creation and reinforcement (85)..... 328
for slabs and walls (84).....326
Hollow Core Lifting Loops..... 176
Hollow Core Opening Tool..... 172

L

lacer bars in footings.....276
lifting anchor (80)..... 421

M

mesh bars.....282
mesh bars by area.....287
Modeling of floor bay (66)..... 159
Modeling of slab area (88).....167
Multiple Wire Size Mesh 377

P

pad footing reinforcement (77).....277
panels..... 154
pile cap reinforcement (76)..... 273
Polygon Hole Generation (33).....150
Polygon Hole Generation (33) parameters..... 150

R

recess nut and washer plate
concrete components 75, 76.....13
concrete components 77, 78.....18
Reinforced concrete stair (95)..... 227
Reinforcement mesh array in area (89).....366
reinforcement
for foundations..... 271
reinforcing
continuous beam.....420
round column reinforcement (82)..... 313

S

Sandwich And Double Wall (1)..... 75
Sandwich Wall Horizontal Seam (1).....97
Sandwich Wall Vertical Seam..... 104
Sandwich Wall Window (1)..... 109
Seam Applicator.....153
seating connections.....9
seating with dowel (75)..... 10
seating with dowel to flange (77).....17
slab bars (18)..... 294
slab generation with points (62).....157
slab generation with polygon plate (61).....154
Slab Reinforcement Tool.....296
slab
properties..... 155
type..... 156
slabs.....154
Sloping slab drainage..... 169
stairs, concrete
properties..... 207,225
stairs
concrete.....207,223
stairwell.....223
starter bars (86, 87)..... 279
strip footing reinforcement (75)..... 271

T

tolerances
in concrete stairs.....208
two-sided seating with dowel (76).....14
two-sided seating with dowel to flange (78)..... 19

W

Wall Formwork..... 132