Tekla Structures Basic Training

Tekla Structures 10.0

April 26, 2004

Contents

8 Concrete Detailing

In this lesson

This lesson explains how to modify and finalize concrete members and build cast units, in Tekla Structures. You will also learn how to put in reinforcements both interactively and with the help of macros.

Introduction

Concrete detailing can be done manually or by using system connections, which help you to automate the detailing. See more about concrete detailing in Tekla Structures **Help: Detailing > Reinforcement > Concrete detailing** and about connections in **Help: Detailing > Getting Started > Basics > Connection concepts.**

For fully automatic connection creation, you can use AutoConnection functionality to select and apply connections with predefined properties to selected parts. Use AutoConnection to have Tekla Structures automatically create similar connections for similar framing conditions. See more about AutoConnections in Tekla Structures **Help: Detailing > Getting Started > AutoConnection > Using AutoConnection**.

You can create reinforcement interactively, i.e. manually bar by bar or in rebar groups. Rebar group consists of several identical reinforcing bars. You can also use reinforcing macros for automatic and parametric reinforcement creation. Reinforcement created in the latter way is updated automatically if the dimensions of the reinforced concrete part change. See more about reinforcements in Tekla Structures **Help: Detailing > Getting started with reinforcement**.

8.1 Connecting Concrete Members Using Connections and Macros

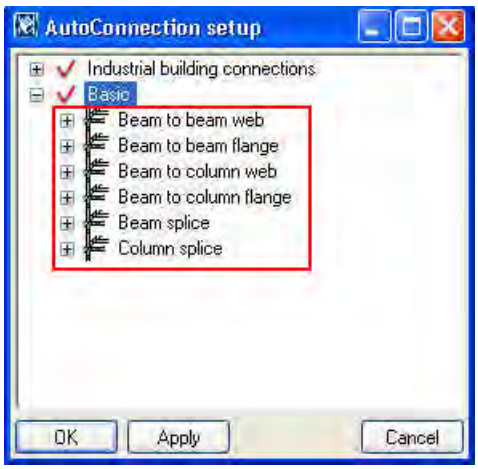
In this exercise we will use system connections to connect the concrete beams and columns and to fit the hollow core slabs to steel beams. You can create connection macros either by using AutoConnections or manually. See more about connections in Tekla Structures **Help: Detailing > Getting Started > Basics > Connection consepts** and about AutoConnections in **Help: Detailing > Getting Started > AutoConnection > Using AutoConnection**.

AutoConnections is the recommended way to create connections. Use AutoConnections to automatically select and apply connections with predefined properties to selected parts.

Create Corbels Using AutoConnections and AutoDefaults

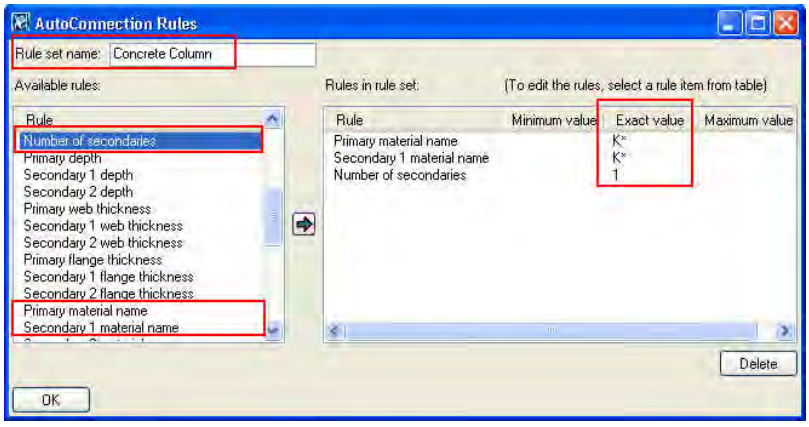
Define conditions for AutoConnections

There are six basic cases for AutoConnections:

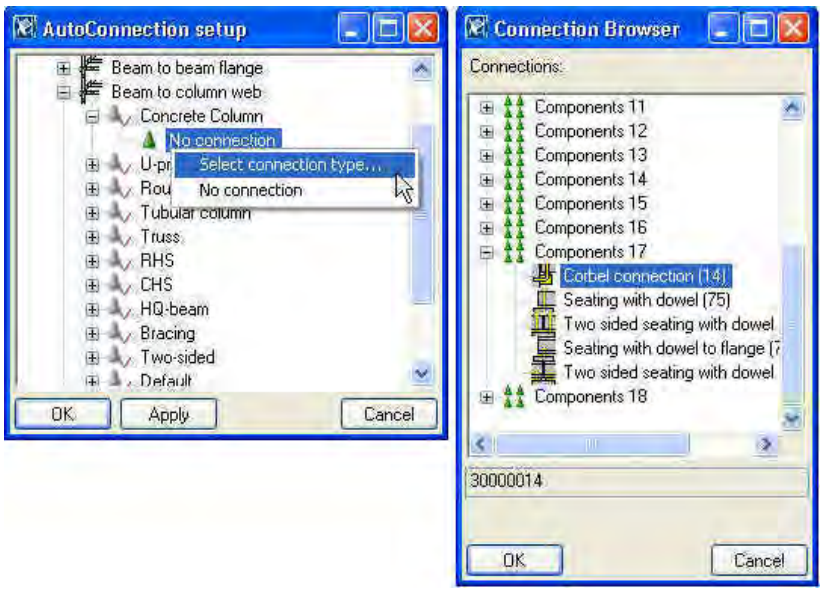


For each of these it's possible to define several framing conditions. We will now define conditions to use the **Corbel connection (14)**, which will create corbels and connect beams to columns if only one beam is to be connected. For two beams we will use **Two sided seating with dowel (76)** connection.

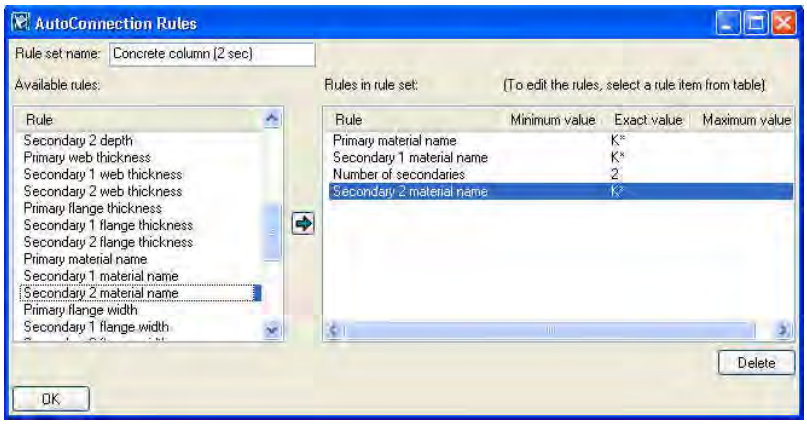
1. Open the **AutoConnection setup** dialog by selecting **AutoConnection…** from **Setup** pull down menu.
2. Right click on **Beam to column web** rule group and select **Create additional rule sets**. Tekla Structures creates a **New** rule set under **Beam to column web** rule group.
3. Open the **Beam to column web** rule group and right click on **New** rule set and select **Edit rule set…** from popup list.
4. Change the **Rule set name** to describe the use (Concrete Column).
5. Select **Primary material name** and **Secondary 1 material name** from **Available rules** list and move them to the right by clicking the arrow in the middle of the dialog.
6. Write **K*** to **Exact value** field for both rules.
7. Add a new rule **Number of secondaries** by selecting it from the **Available rules** list and clicking the arrow.
8. Set the **Exact value** to 1.



9. Press **OK** to close the dialog and save the rule settings.
10. Open the **Concrete Column** rule set and right click on the **No connection** icon to select the **Corbel connection (14)**. Now pick **Select connection type…**, which opens **Connection Browser**.

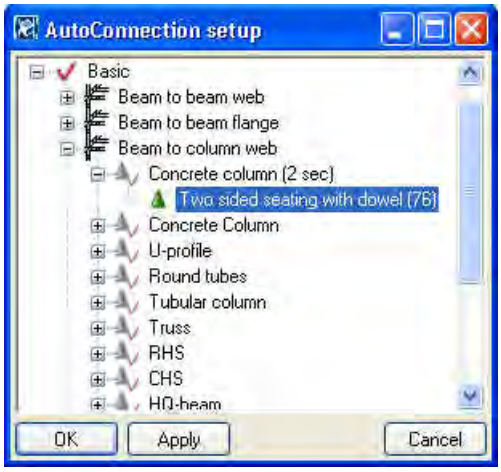


11. In **Connection Browser** select **Corbel connection (14)**. Close the browser by clicking **OK**.
12. You have now completed one rule definition. Next copy the rule set to apply also in **Beam to column flange** condition by right clicking on **Concrete Column** rule set and selecting **Copy rule set** from popup menu.
13. Open the **Beam to column flange** rule group and right click on any rule set and select **Paste rule set** from popup menu. You have now defined beam to column connection for concrete members to be always **Corbel connection (14)** if number of secondaries is 1.
14. Next create another rule set for **Two sided seating with dowel (76)** connection in case there are two beams to be connected to column. Repeat steps 2 – 8. **Rule name** is: Concrete column (2 sec)
15. Set the **Exact value** to 2.
16. Add a new rule **Secondary 2 material name** by selecting it from the **Available rules** list and clicking the arrow.
17. Write **K*** to **Exact value** field.



18. Press **OK** to close the dialog and save the rule settings.
19. Move the rule set above the Concrete column rule set using right button command **Move up**.
20. Open the **Concrete column (2 sec)** rule set and right click on the **No connection** icon to select the **Two sided seating with dowel (76)**. Now pick **Select connection type…**, which opens **Connection Browser**.

21. In **Connection Browser** select **Two sided seating with dowel (76)**. Close the browser by clicking **OK**.



22. Next copy the rule set to apply also in B**eam to column flange** condition by right clicking on **Concrete column (2 sec)** rule set and selecting **Copy rule set** from popup menu.

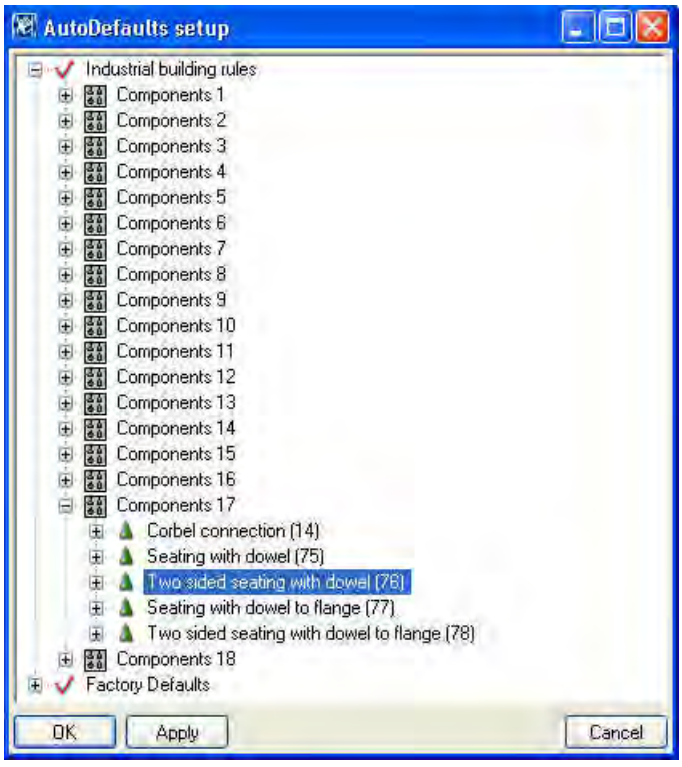
23. Open the **Beam to column flange** rule group and right click on any rule set and select **Paste rule set** from popup menu.

Define AutoDefaults

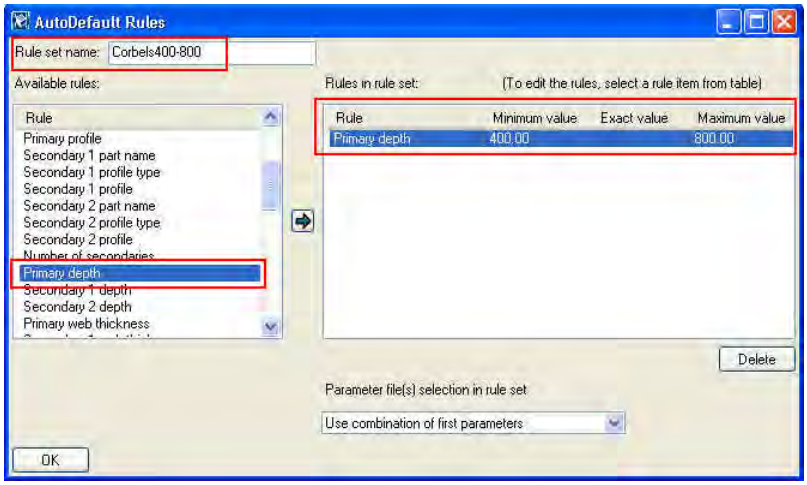
With **AutoDefaults** you can create rules defining when to use different pre-defined connection properties. When the connections need to be modified (for example changing the beam size), Tekla Structures automatically redefines the connection properties using AutoDefaults rules defined by you. See more in **Help: System > AutoConnection > AutoDefaults setup > AutoDefaults setup**. We will now add an AutoDefault rule for **Two sided seating with dowel (76)** and use it in Industrial building rule group, which we created in Lesson 2.

1. Open **AutoDefaults setup** dialog from **Setup > AutoDefaults…** pulldown menu.

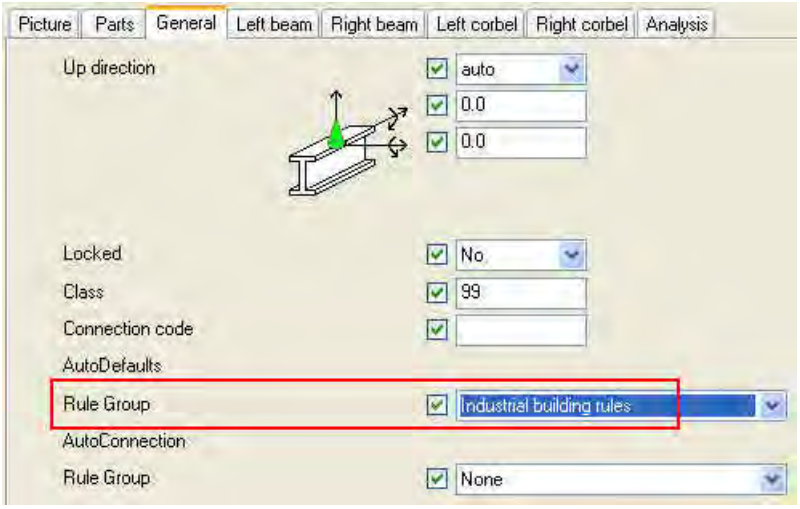
2. Browse to **Two sided seating with dowel (76)** under **Industrial building rules** group.



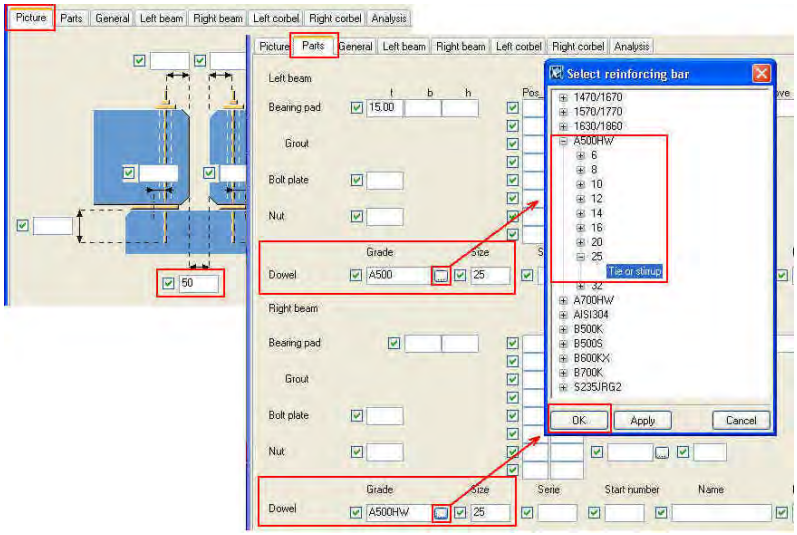
3. Right click to create an additional rule set.
4. Select the **New** rule set under **Two sided seating with dowel (76)** and select **Edit rule set…** from right click popup menu.
5. Write new name for the rule set: Corbels400-800.
6. Select **Primary depth** from **Available rules** list and move it to the right using arrow button in the middle of the dialog.
7. Set **Minimum value** to 400 and **Maximum value** to 800.

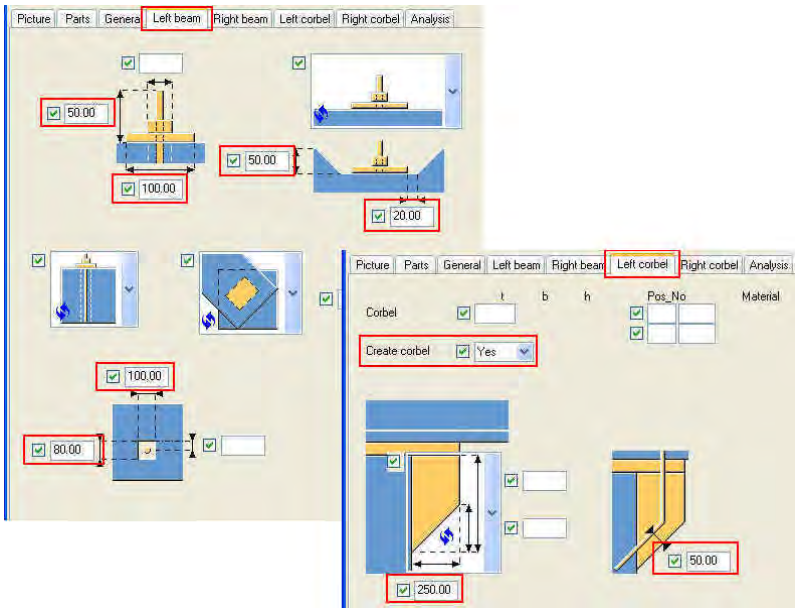


8. Press **OK** to save the settings and close the dialog.
9. Select connection parameters Standard.j30000076 under the Corbels400-800 rule set and **right click > Additional connection parameters...**
10. Select the new connection parameters and **right click > Edit connection parameters**. The connection properties dialog opens.
11. On the **General** tab, select the **Industrial building rules** group; it can also be **None**.



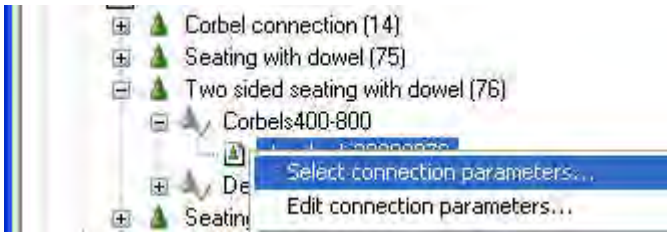
12. On all the tabs, set the fields that you want AutoDefaults to override to Default by selecting the options marked with the arrow symbol.
13. Set all needed options, values and dimensions on all tab pages as below.





14. Save the settings using **Save as** field, put name Corbels400-800.

15. Close the dialog with **OK**.
16. Right click on the connection parameters and pick **Select connection parameters…**



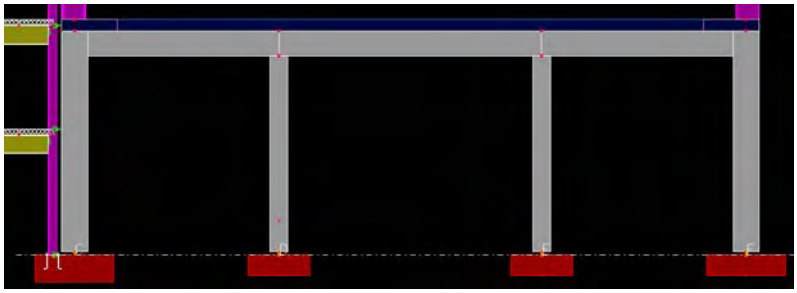
17. Select Corbels400-800.j30000076 from **Attribute File List**.

18. Close the **Attribute File List** with **OK**.
19. Close the **AutoDefault setup** dialog with **OK**.

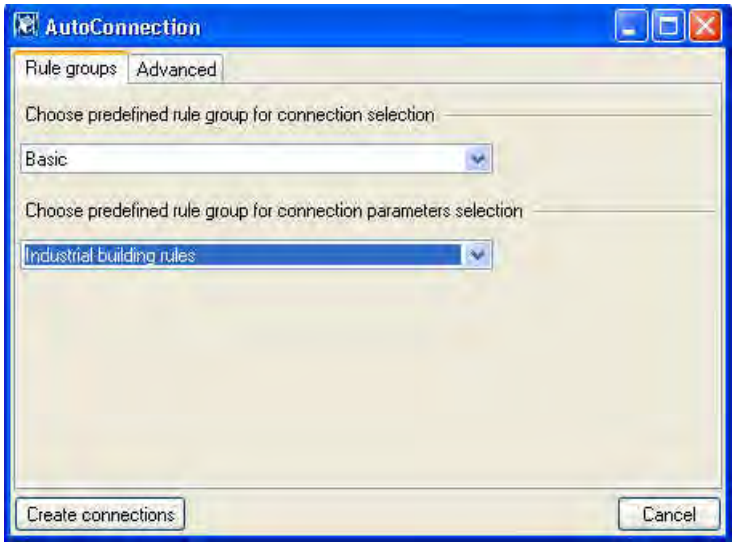
Create connections

We will now use the AutoConnections and apply the AutoDefault rules, which we just defined.

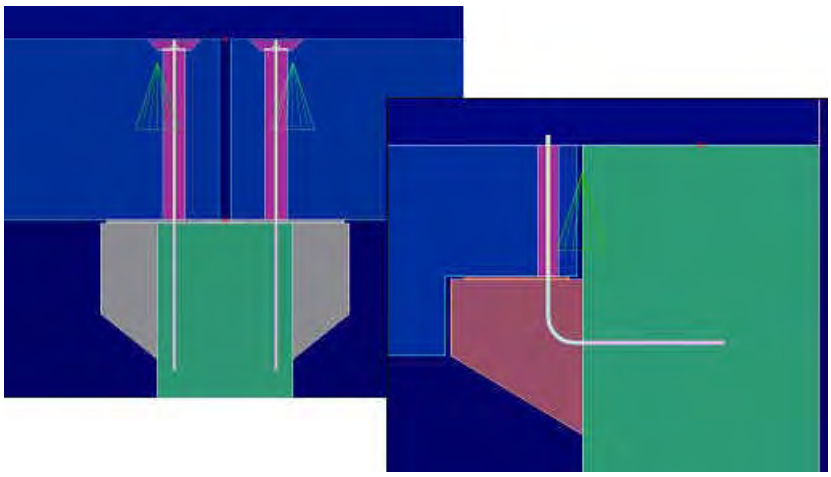
1. Select all concrete beams and columns between grid lines **C** and **F**.



2. From **Detailing** pull down menu select **AutoConnection…**
3. Select Industrial building rules group for connection parameters.



4. Press **Create connections** button. TeklaStructures creates connectons between concrete beams and columns using **Corbel connection (14)**, which will create corbels and connect beams to columns if only one beam is to be connected. For two beams **Two sided seating with dowel (76)** connection is used.

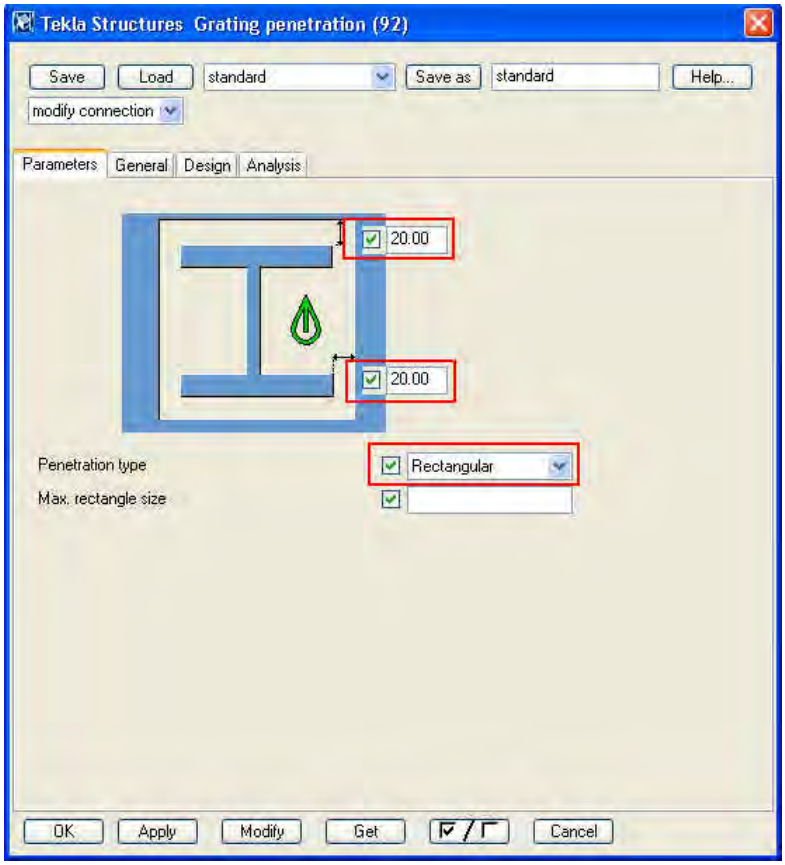


Create Cuts to Hollowcore Slab Corners

Define cut size

We will next fit the hollowcore slab corners to the steel columns. For this we'll use connection **Grating penetration (92)**.

1. Double click the **Grating penetration (92)** icon to open the properties dialog.
2. Set **Penetration type** to Rectangular and clearance around I-profile to 20 mms.



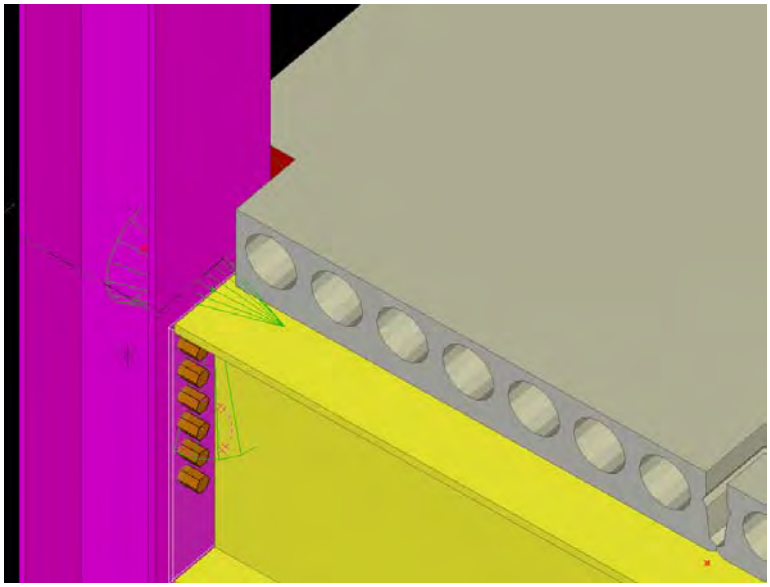
3. Close the dialog with **OK**.

Create the cuts

Next zoom in to any hollowcore slab corner, which collides to I-profile column and create the cutting.

1. Pick the **Grating penetration (92)** icon.

2. Pick the hollowcore slab.
3. Pick the I-profile column.



Tekla Structures creates a cut to hollowcore slab at 20 mm distance around the I-profile column.

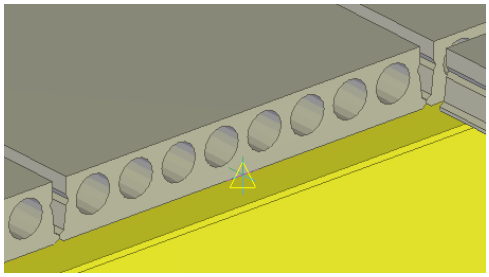
4. Repeat steps 2 and 3 for all other hollowcore slab corners, which collide to I-profile columns.

Create In-situ Slabs

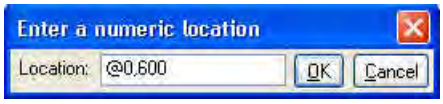
Before we add reinforcements to concrete members we need to create additional 600 mms wide in-situ slabs beside the hollowcore slab area in first and second floors. For this, zoom in to first floor in grid lines **A** and **5 – 6**.

1. Double click the **Create concrete slab** icon.
2. Set the profile height to 175 mm.

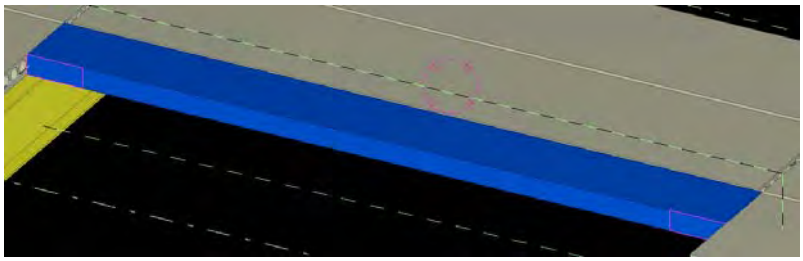
3. Press **OK** to close the dialog.
4. Start picking the polygon shape at the middle of the hollowcore slab in grid line 5.



5. Write: **R** then add **0, 600**, to the **Enter a numeric location** dialog and press **OK**.



6. Write: **R** then add **6050**, to the **Enter a numeric location** dialog and press **OK**.
7. Write: **R** then add **0, -600**, to the **Enter a numeric location** dialog and press **OK**.
8. Close the polygon with middle button.



Now select the slab and copy it **3500.00** mms in z-direction using **Copy > Translate...** right button command.

8.2 Add Reinforcement to Concrete Members

Create Rebars Using System Rebar Macros

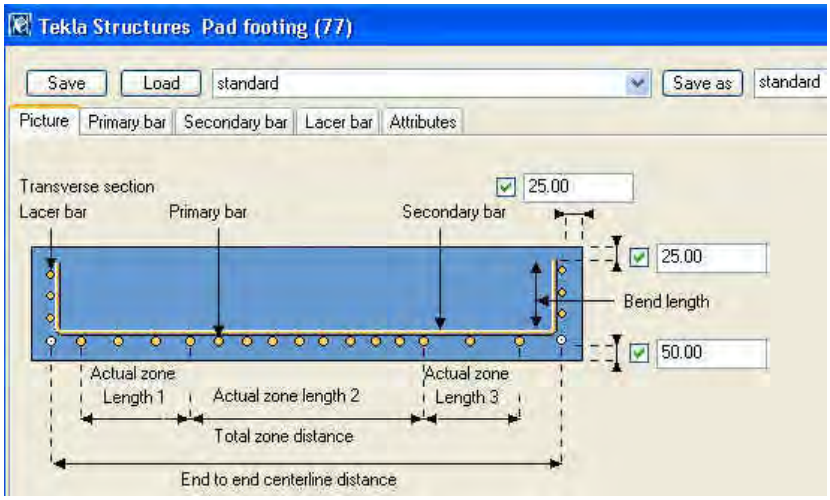
Once you have created and detailed a model of concrete parts, you may need to reinforce the parts. Reinforcing macros create the entire reinforcement of a part in one go. The reinforcement is created according to the macro attributes. Reinforcement created using macro is updated automatically if the dimensions of the reinforced concrete part change.

See more about reinforcements in Tekla Structures **Help: Detailing > Getting started with reinforcement** and in **Help: Detailing > Basic reinforcement properties**.

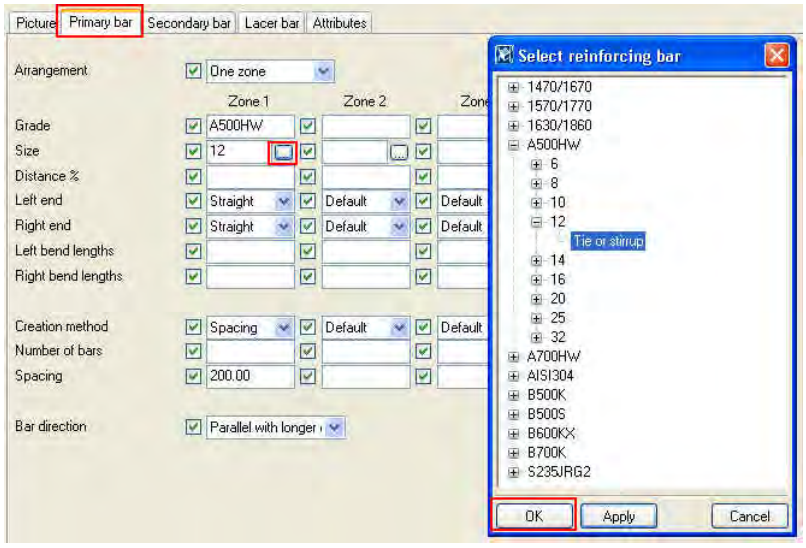
Pad Footings

We will first put reinforcements into 1800*1800 pad footings, which are 650 mms height. For this we use the **Pad footing (77)** reinforcing macro. See more in Tekla Structures **Help: Detailing > Reinforcement > Reinforcement reference > Pad footing (77)**

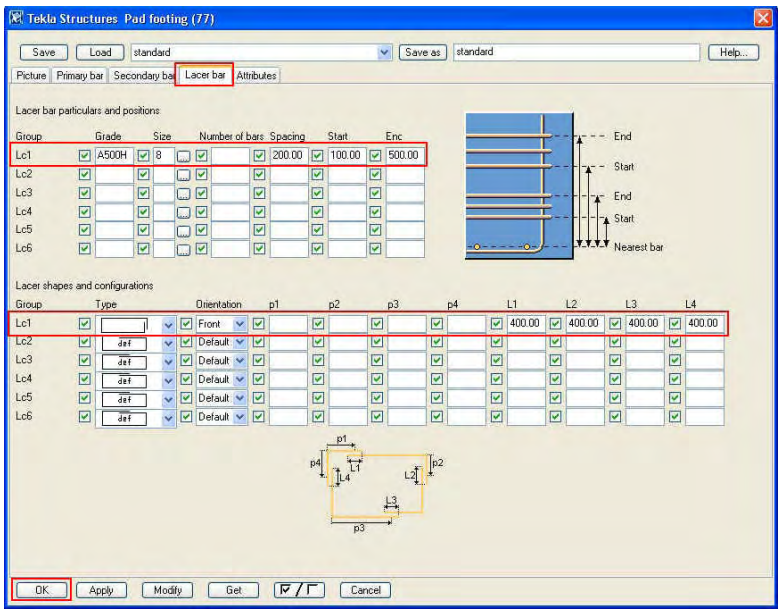
1. Double click the **Pad footing (77)** reinforcing macro icon to define the properties to be used.
2. Set cover thicknesses on **Picture** tab page.



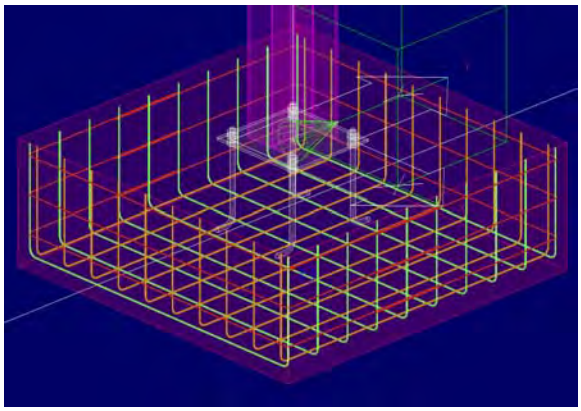
3. Next define primary bars as described below. Select the grade and size from **Select reinforcing bar** dialog, which opens when you push button in size field. Exit the dialog with **OK**.



4. Use the same settings for secondary bars and put 12 mm reinforcement in 200 mm intervals.
5. Then you need to set the lacer bars as described below.



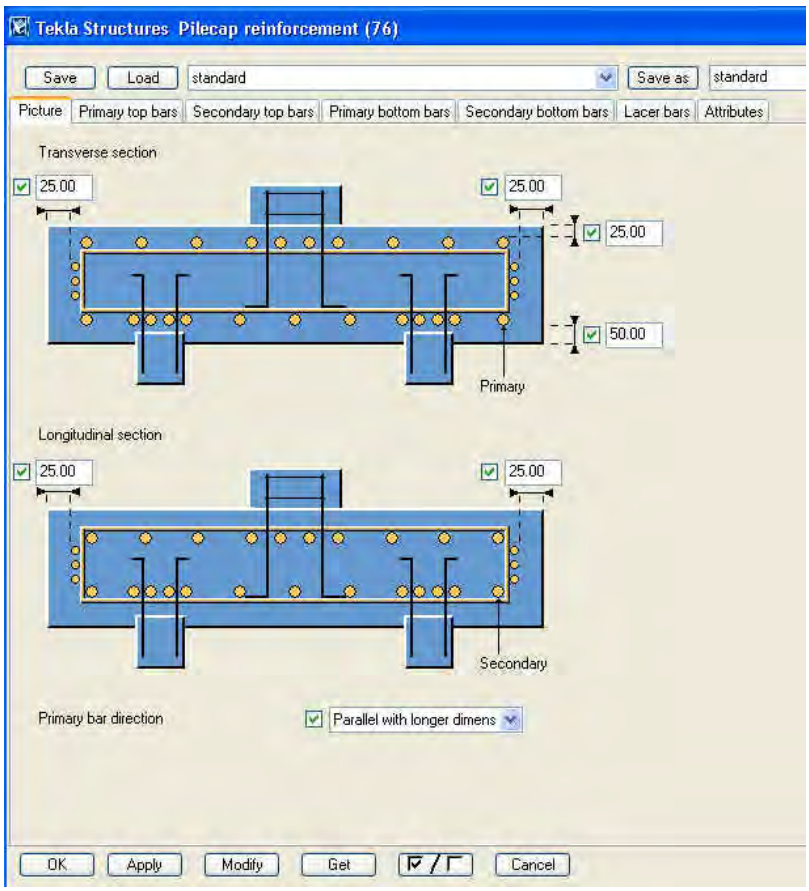
6. Save the values in **Save as** field with name 1800*1800-650.
7. Close the dialog with **OK**.
8. Now pick the pad footings sized 1800*1800*650. The macro generates the reinforcement inside the pad footings.



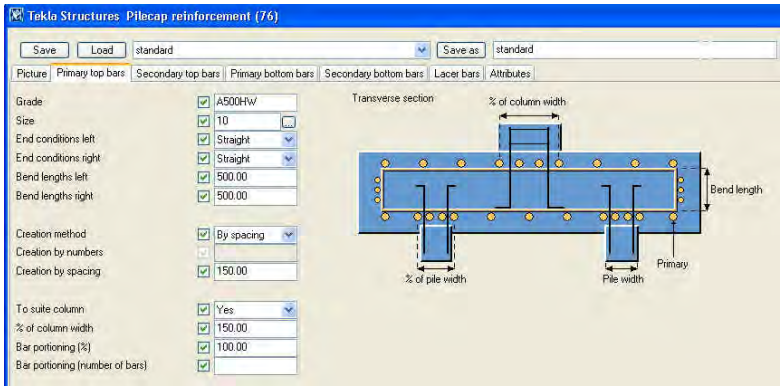
9. Next we will put reinforcement to **2100*2100*650** pad footings, which are located in grid lines **D** and **E**. Repeat steps 1 to 8, now using 12 mm bars in 150 mm intervals both for primary and secondary bars. Save the connection settings with name 2100*2100-650.
10. Still using the macro **Pad footing (77)**, put reinforcements to pad footings sized **2700*2700*650** on grid line **F**. This time use 16 mm bars in 150 mm intervals for primary and secondary bars. Save the connection settings with name 2700*2700-650.

For pad footings on grid line **C** we will use a different macro, **Pilecap reinforcement (76)**, which creates reinforcement also to top of the footing. See more in Tekla Structures **Help: Detailing > Reinforcement > Reinforcement reference > Pilecap reinforcement (76)**

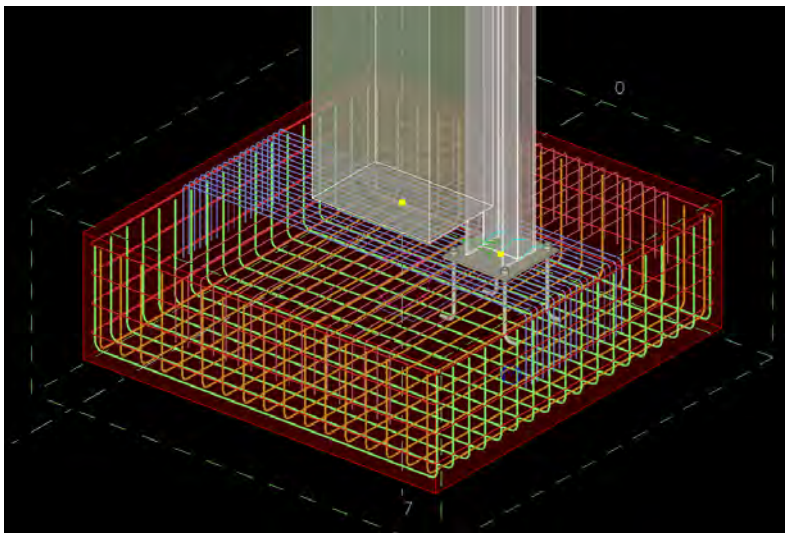
1. Double click the **Pilecap reinforcement (76)**, reinforcing macro icon to define the properties to be used.
2. Set cover thicknesses on **Picture** tab page.



3. Next define **top primary and secondary bars** as described below. Select the grade and size from **Select reinforcing bar** dialog, which opens when you push button in size field. Exit the dialog with **OK**. With these settings top bars will be created only under columns 150% of column size in both directions.



4. Now define **bottom primary and secondary bars** using 16 mm bars in 150 mms intervals, bond length in both sides is 700 mms. Bottom bars don't need to suite column, so you can use default in that field and leave % fields empty.
5. Then set 10 mm diameter lacer bars at 200 mm intervals starting at 100 and ending to 700 mms.
6. Save the values in **Save as** field with name 2700*2700-850.
7. Close the dialog with **OK**.
8. Now pick first the pad footing sized 2700*2700*850, then the columns on top of the footing. End with middle button. The macro generates the reinforcement inside the pad footings putting top bars under the columns.

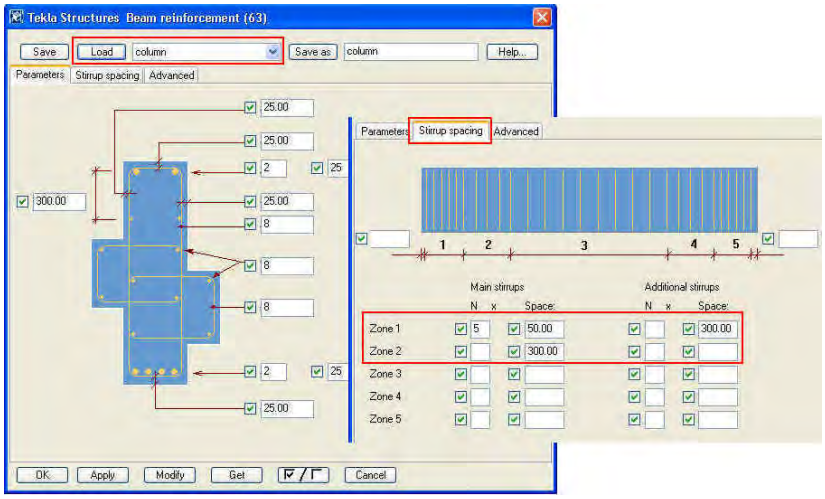


9. Repeat step 8 for all pad footings on grid line **C**.

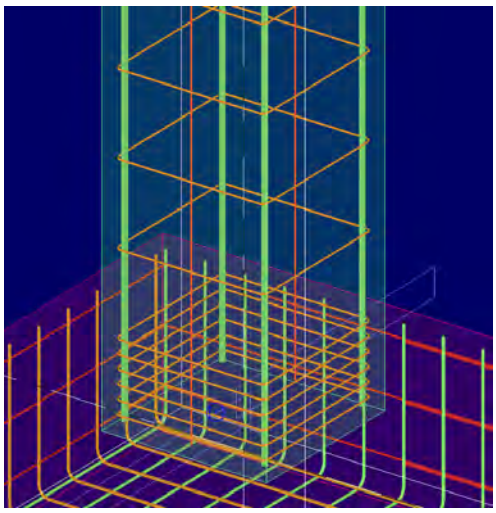
Columns

We will next put reinforcements into columns using **Beam reinforcement (63)** reinforcing macro. See more in Tekla Structures **Help: Detailing > Reinforcement > Reinforcement reference > Beam reinforcement (63)**

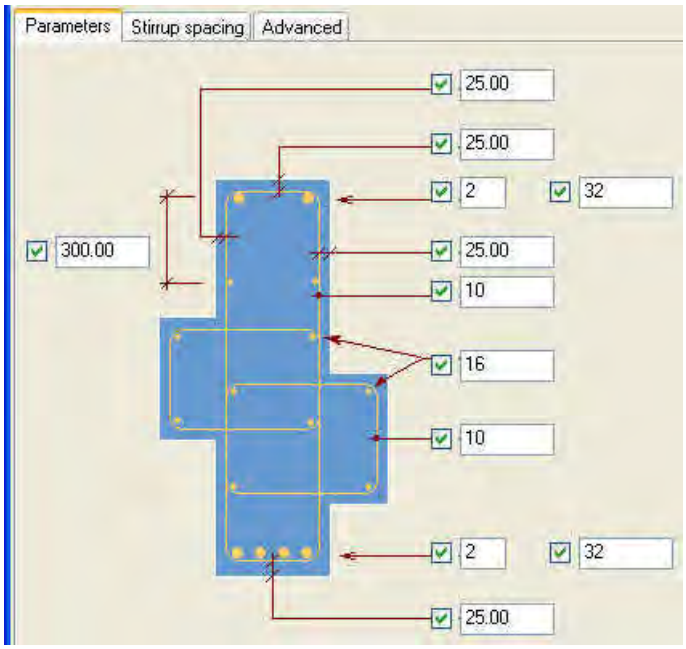
1. Double click the **Beam reinforcement (63)** reinforcing macro icon to define the properties to be used.
2. Load predefined column settings by selecting **column** option from pulldown list and pressing **Load** button in front of the list.
3. Open **Stirrup spacing** tab page and empty the zone 3 field settings.



4. Save the values in **Save as** field with name column600*600-corbel.
5. Press **OK** to close the dialog.
6. Pick one by one all the columns in grid lines **D** and **E**. The macro generates reinforcements inside the columns.



7. Next we will put reinforcement to 900*600 columns, which are located in grid lines **C** and **F**. Reopen the **Beam reinforcement (63)** reinforcing macro dialog and use the settings described below.

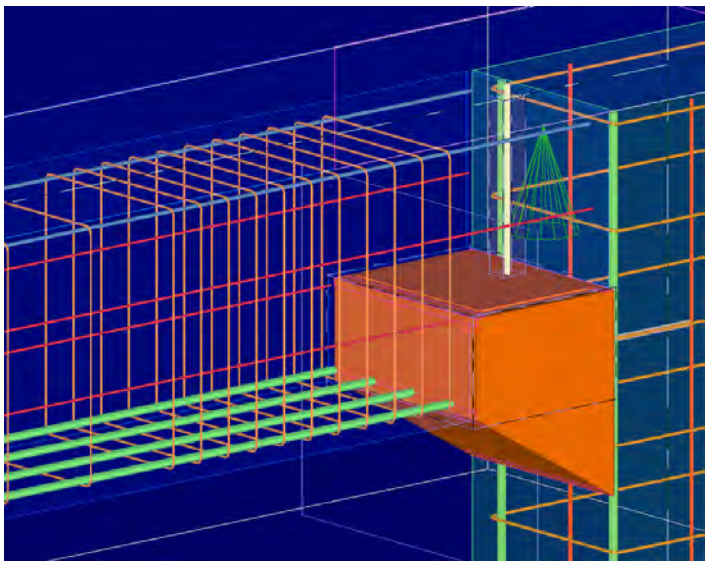


8. Save the values in **Save as** field with name column900*600-corbel.
9. Press **OK** to close the dialog.
10. Pick one by one all the columns in grid lines **C** and **F**. The macro generates reinforcements inside the columns.

Beams

We will next put reinforcements into beams using the same **Beam reinforcement (63)** reinforcing macro, which we used for columns. See more in Tekla Structures **Help: Detailing > Reinforcement > Reinforcement reference > Beam reinforcement (63)**

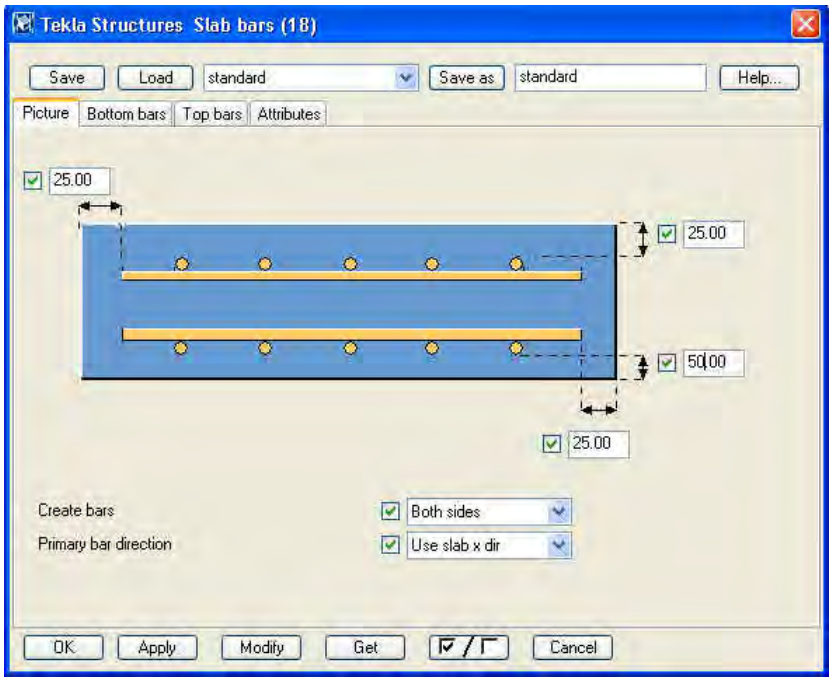
1. Double click the **Beam reinforcement (63)** reinforcing macro icon to define the properties to be used.
2. Load predefined settings by selecting **standard** option from pulldown list and pressing **Load** button in front of the list.
3. Press **OK** to close the dialog.
4. Pick one by one all the concrete beams between grid lines **C** and **F**. The macro generates reinforcements inside the beams cutting the bottom bars at the corbels.



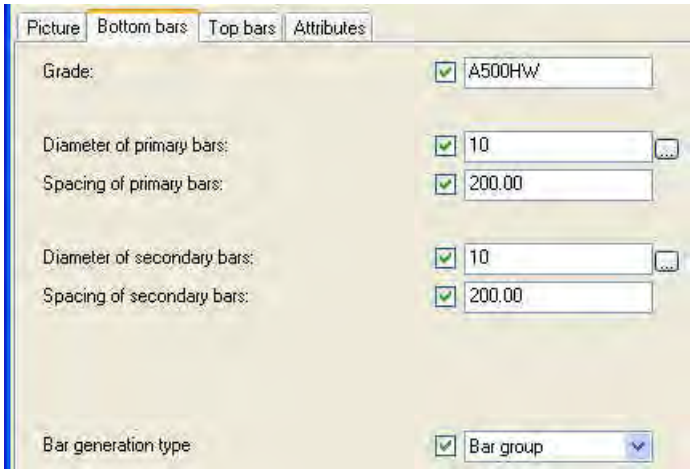
Slabs

We will next put reinforcements into cast in-situ slabs using **Slab bars (18)** reinforcing macro. See more in Tekla Structures **Help: Detailing > Reinforcement > Reinforcement reference > Slab reinforcement (18)**

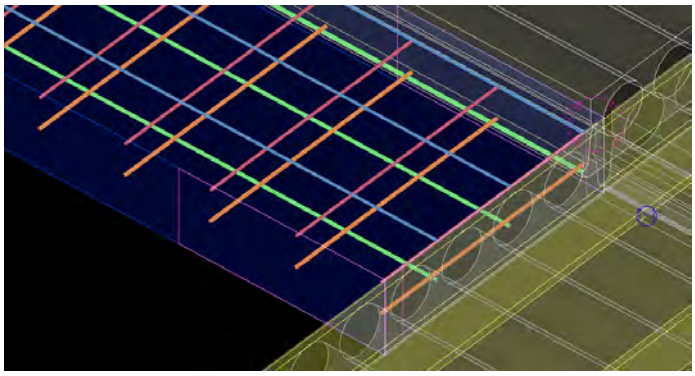
1. Double click the **Slab bars (18)** reinforcing macro icon to define the properties to be used.
2. Define cover thickness to be used. And set the bars to be created to both sides using slab x direction.



3. Open **Bottom bars** tab page and set the bar diameter to 10 mm, spacing in both directions to 200 mms.



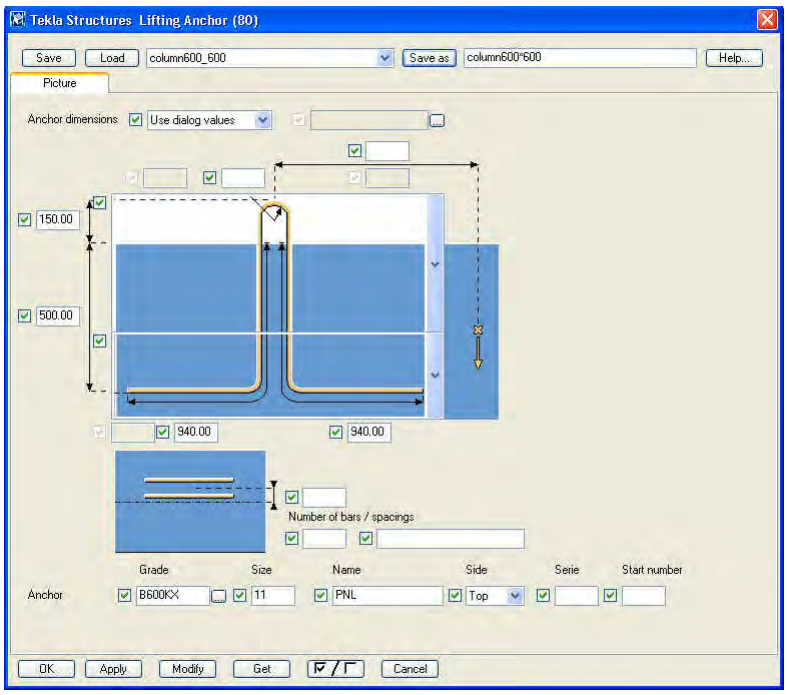
4. Use the same settings for top bars.
5. Save the values in **Save as** field with name Slab175.
6. Close the dialog with **OK**.
7. Pick the cast in-situ slabs one by one. The macro generates reinforcements inside the slabs.



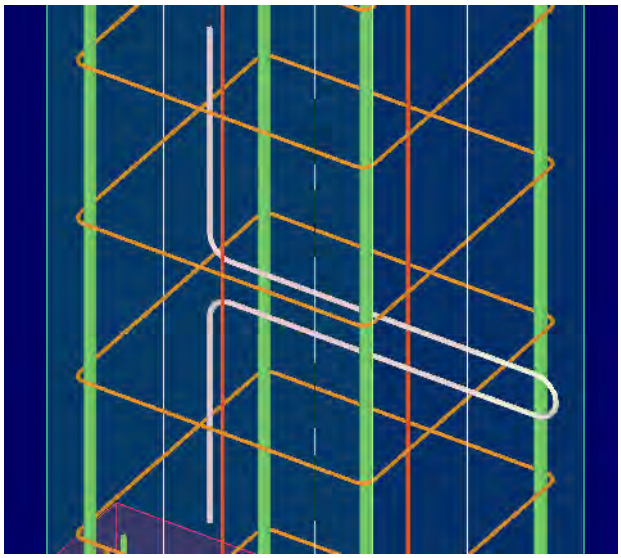
Lifting hooks

We will next put lifting anchors into beams and columns using **Lifting anchor (80)** macro. The macro generates lifting anchors according to center of gravity. See more in Tekla Structures **Help: Detailing > Reinforcement > Reinforcement reference > Lifting anchor (80)**.

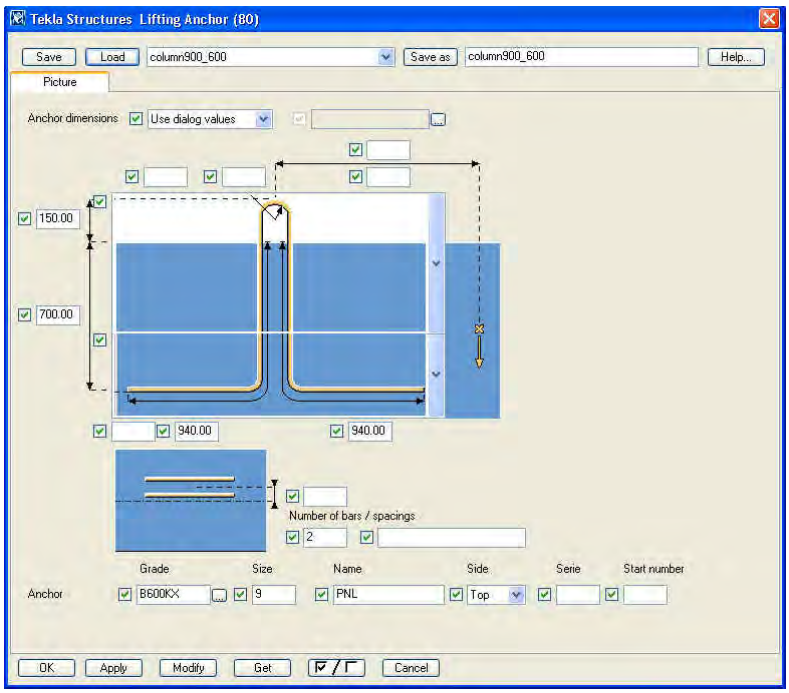
1. Double click the **Lifting anchor (80)** macro icon to define the properties to be used.
2. Select anchor shape to be used.
3. Define anchor dimensions.
4. Select anchor size and grade.
5. Save the values in **Save as** field with name column600*600.
6. Press **OK** to close the dialog.



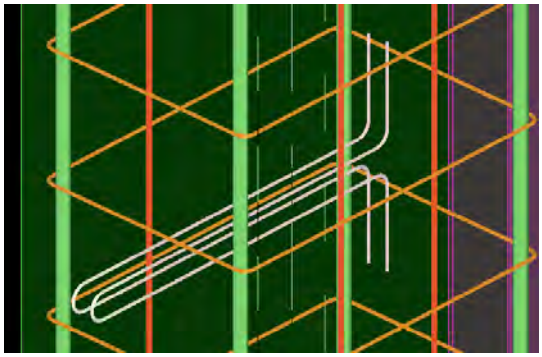
7. Pick one by one all the columns in grid lines **D** and **E**. The macro generates lifting anchors inside the columns.



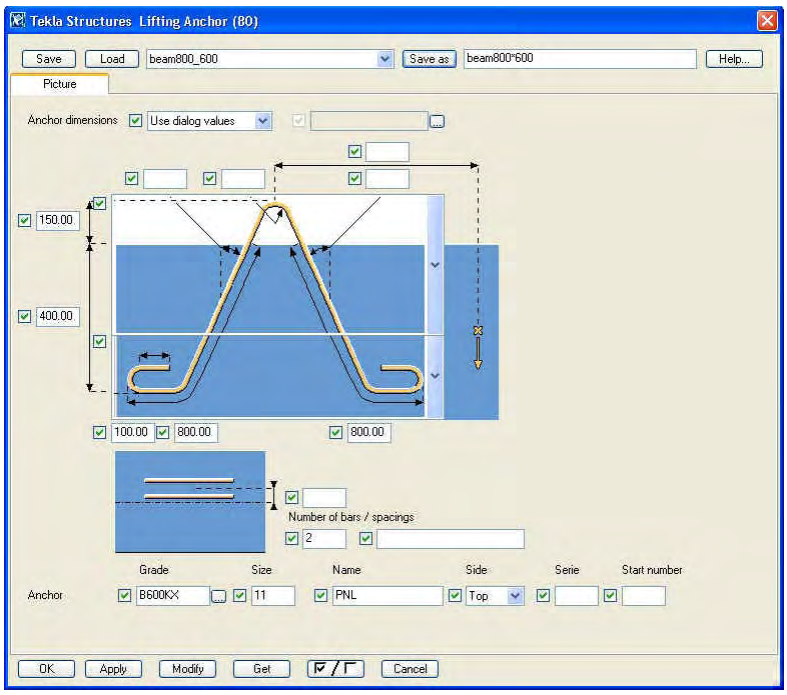
8. Using the **Lifting anchor (80)** macro, put anchors to columns sized 900*600 on grid lines **C** and **F**. This time use 2 times 9 mm bars in 50 mm intervals and dimension them to be 700 mms inside the column. Save the connection settings with name column900*600.



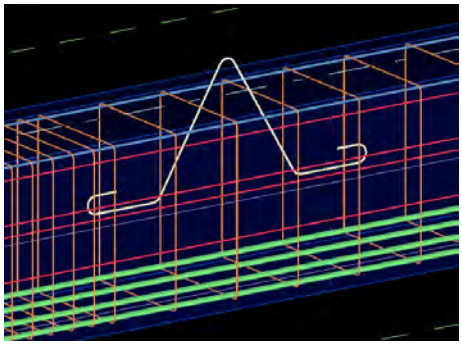
The macro generates lifting anchors in pairs inside the columns.



9. Still using the **Lifting anchor (80)** macro, put anchors to concrete beams between grid lines **C** and **F**. This time use one 11 mm bar and dimension them to be at 400 mms depth inside the beam, change the anchor shape as well. Save the connection settings with name beam800*600.



The macro generates lifting anchors inside the beams.



Create Rebars Interactively

Once you have created reinforcements to concrete parts with the help of macros, you may need to complete reinforcing interactively, i.e. manually bar by bar or in rebar groups. Rebar group consists of several identical reinforcing bars.

Every reinforcing bar has basic attributes such as grade, diameter, and bending radius. If a group of bars is to be created, the number of bars and creation method definition are also needed.

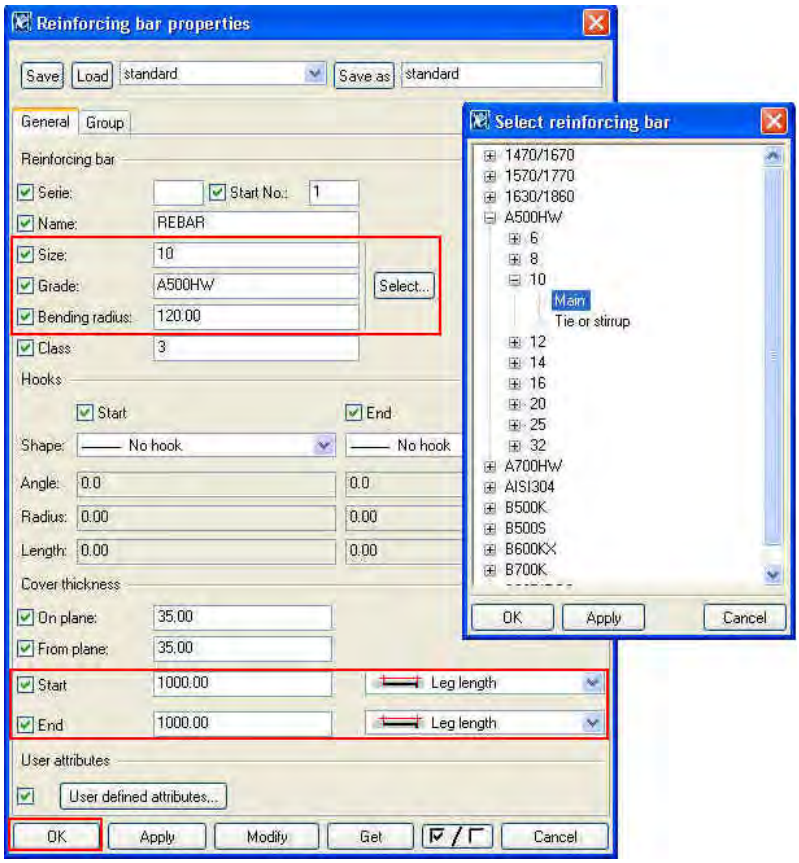
See more about reinforcements in Tekla Structures **Help: Detailing > Reinforcement > Reinforcement reference > Reinforcing bar** and **Help: Detailing > Reinforcement > Reinforcement reference > Reinforcing bar group**

Create single rebars

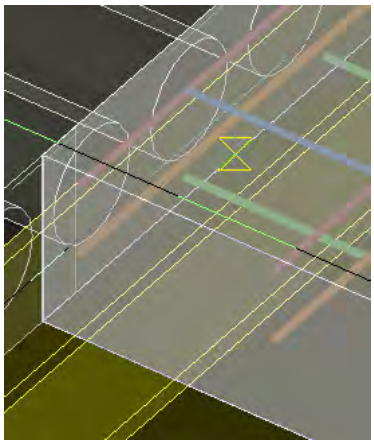
We will start interactive reinforcing modeling with single rebars and we will put side reinforcement to the in-situ slabs, which we modeled earlier in this lesson. For this, zoom in to first floor in grid lines **A** and **5 – 6** so that you can see the slab. **Create part basic views** (select part and right click, select command **Create view > Part basic views**) to help you in modeling.

1. To create a single reinforcing bar, double-click the **Create reinforcing bar** icon.

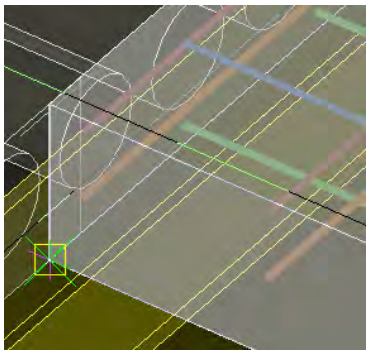
2. Select the needed rebar from dialog, which opens by pressing **Select…** button. Size, grade and bending radius are all defined together. Close the dialog with **OK**.
3. Use **No hook option** for both ends.
4. Set **start** segment and **end** segment to be defined by **Leg length**. Now give first and last segment length.



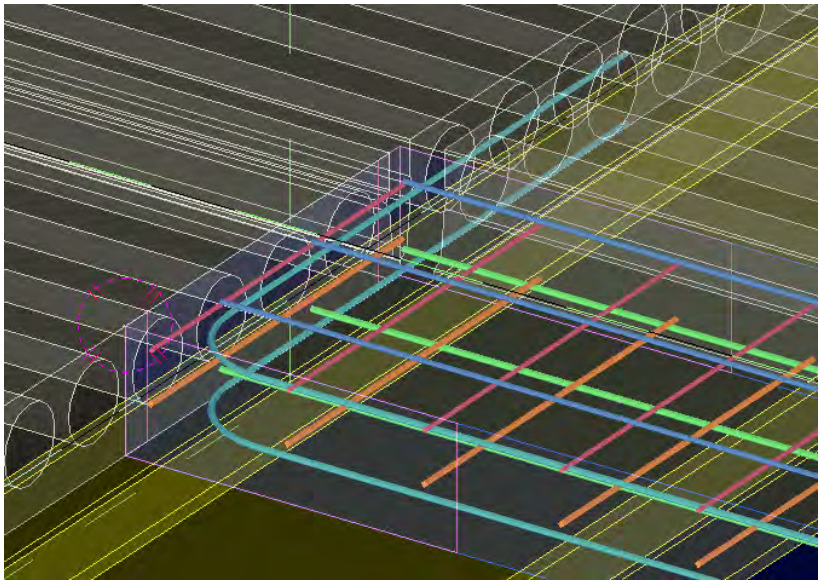
5. Close the dialog with **OK**.
6. Pick the lower slab.
7. Next define the rebar shape by picking a U-shaped polyline. You can start at any position at the short side of the slab, because the first segment will be 1000 mms.



8. Second and third positions are at the corners of the slab.



9. You can make the fourth pick the same way as the first one, at any position at the opposite short side of the slab, because the last segment will be 1000 mms.
10. End the polyline pick with middle button. A single reinforcing bar is created.
11. Next select the new reinforcing bar and **Copy > Translate…** it 175.00 mms in z-direction.
12. Double click the copied reinforcing bar and change the **From plane** cover thickness to –35 mms, then press **Modify**.

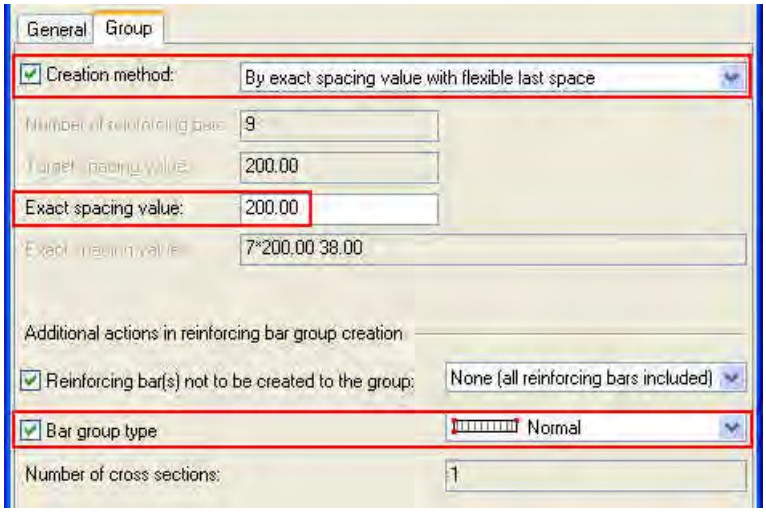


Create rebar group

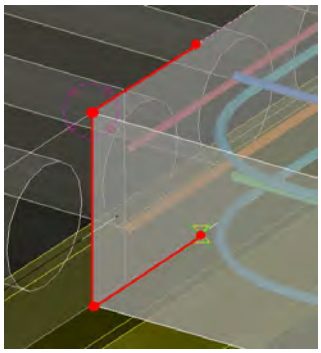
We will continue reinforcing the same in-situ slab and put a reinforcing group interactively to the outer side of it.

1. To create a rebar group, double-click the **Create reinforcing bar group** icon. The **Rebar attributes** dialog box appears. It is similar to the single bar case but there is the additional **Group** tab.

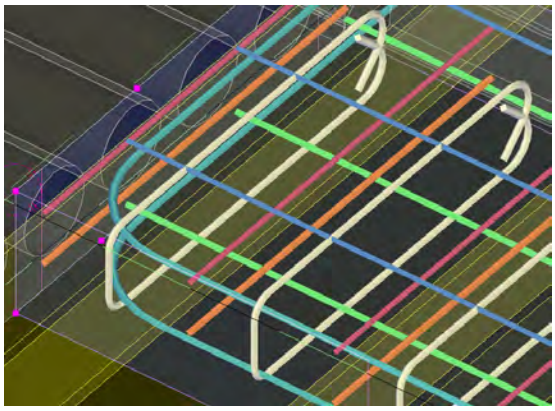
2. Define settings for 10 mms diameter stirrup in **General** tab page using standard 135-degree hook and 500mms leg length at both ends, with 25 mms cover thickness in on plane direction and 100 mms in from plane direction.
3. Open **Group** tab page.
4. Set the group creation method to exact spacing with 200 intervals using normal group type.



5. Close the dialog by pressing **OK**.
6. Pick the in-situ slab.
7. Now define the rebar shape by picking a polyline, same way as with single rebar. Again, start and end at any position at the short side of the slab, because the first and last segments will be 500 mms. End the polyline pick with middle button.



8. Next define the group location by picking start and end positions at long side corners. A rebar group is created.



Copy rebars

We will next copy the in-situ slab reinforcements to upper floor slab.

1. Select the single rebars and the rebar group, which we created in previous exercise.
2. Right click and select **Copy > Translate…** command.
3. Insert 3500 in z-direction.

4. Press **Copy** in the dialog. The selected rebars are copied and automatically attached to the upper floor slab.
5. Check the cast unit content by selecting the slab with **Alt** button down.

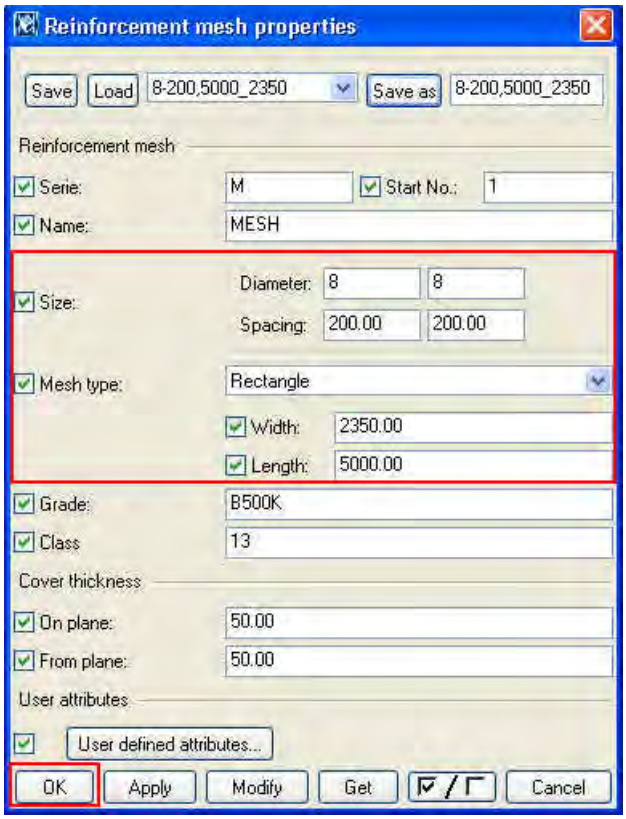
Mesh reinforcements

Now we will use a standard size mesh to reinforce the large cast in-situ slab, which is located in grid lines **C** to **F** and **1** to **7**.

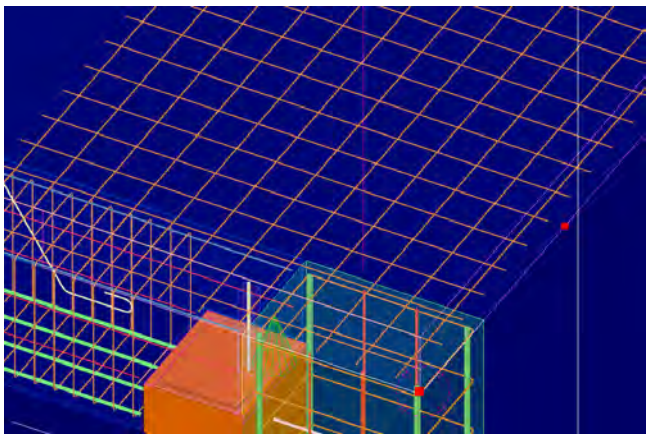
See more about mesh in Tekla Structures **Help: Detailing > Reinforcement > Reinforcement reference > Reinforcement mesh**.

1. To create a rebar mesh, double-click the **Create reinforcement mesh** icon. The **Reinforcement mesh properties** dialog box appears.

2. Define the mesh type to be rectangle.
3. Set size and spacing to be 8 mm diameter, with 200 mms spacings in both directions.
4. Define dimensions to 2350 * 5000 mms.
5. Define 50 mm cover thickness around the mesh.
6. Save the values using **Save as** function with name: 8-200,5000_2350.



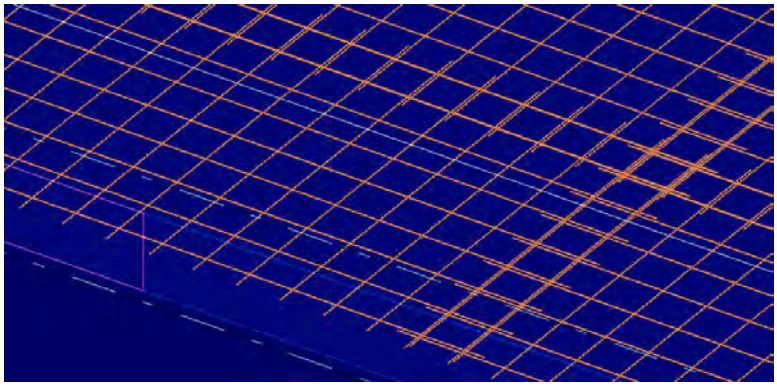
7. Close the dialog with **OK**.
8. Pick the slab.
9. Show position and creation direction by picking two points along grid **F**.



Copy Mesh Reinforcement

Now we will copy the standard mesh reinforcement to cover the whole slab area.

1. Select the mesh and right click > select **Copy > Translate…** command.
2. Put –2000 in dY direction and 16 in dZ direction and copy one time by pressing **Copy**.
3. Next select the both mesh reinforcements and repeat the copy, this time only in dY direction using value –4000, 5 copies.
4. Double click the last mesh near grid line **C** and change the width to 1800, then press **Modify**.
5. Now select all the mesh reinforcements and right click > select **Copy > Translate…**, this time one copy -4600.00 in dX direction and 16 in dZ direction.
6. Again select all the mesh reinforcements and copy 3 times -9200.00 in dX direction.
7. Double click one of the last mesh reinforcements near grid line 1 and change the length to 4350, then select all the mesh reinforcements near grid line 1 press **Modify**.



We have now completed the standard mesh reinforcements for the whole slab.

8.3 Add Components to Concrete Members

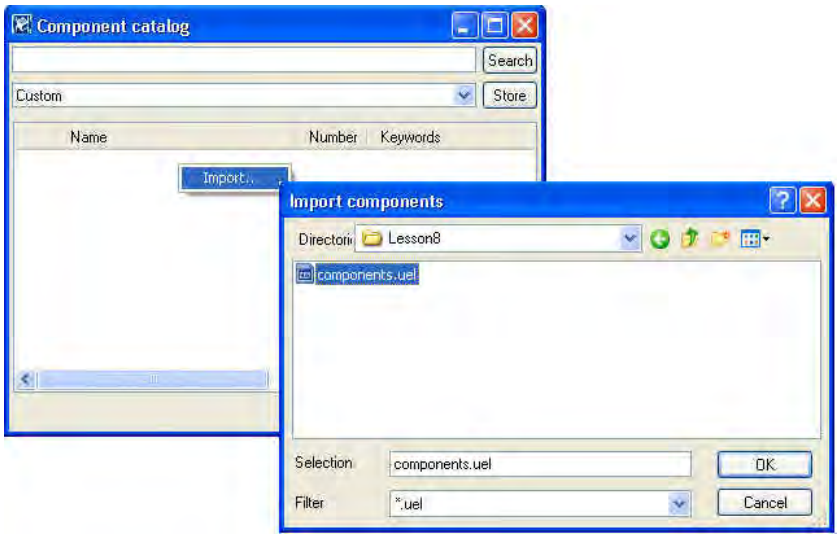
Column Shoe Connection

Next we will use components to connect the concrete columns to footings interactively. You can use ready-made components in Tekla Structures by importing them to the model folder. We will make a user defined connection, which has components inside of it. First we need to acquire the needed components.

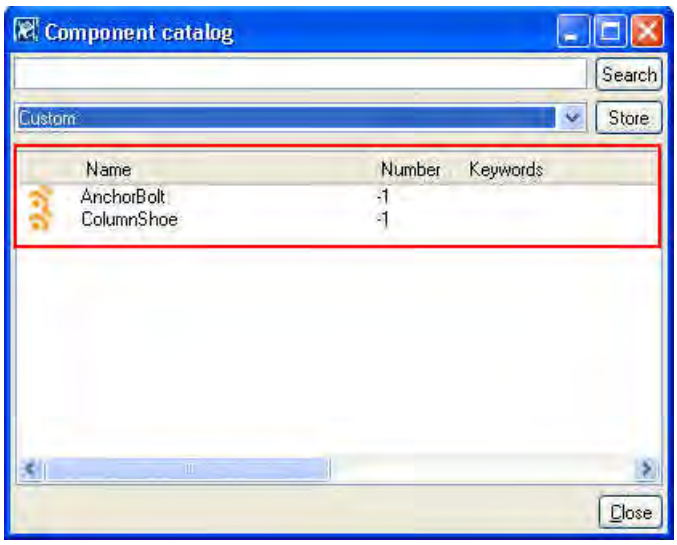
Import Component

1. To import custom components, open the **Component catalog** by pressing **Ctrl + F** or clicking the icon.

2. **Right click -> Import…** to open **Import components** dialog.



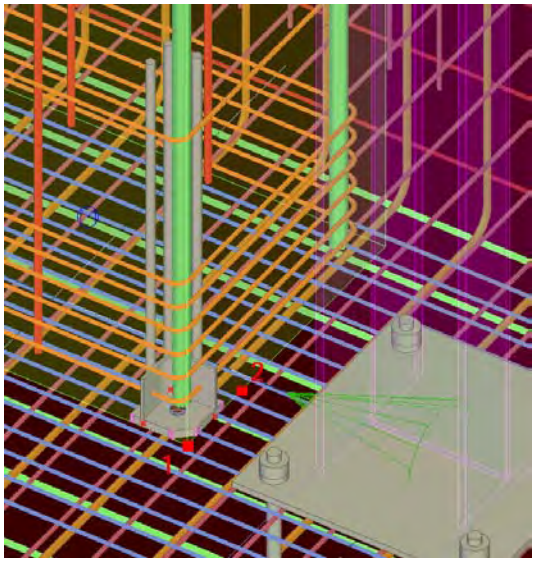
3. Select the components.uel file to be imported and press **OK** in the browser. New components AnchorBolt and ColumnShoe appear in the list and they are saved in the model folder.



Insert the ColumnShoe

Now we will insert the ColumnShoe component to concrete column.

1. Select ColumnShoe component from the list.
2. Close **ColumnShoe** dialog with **OK**.
3. Zoom in to bottom of column on grid lines **C-1** intersection.
4. Pick first column corner to define the column shoe position.



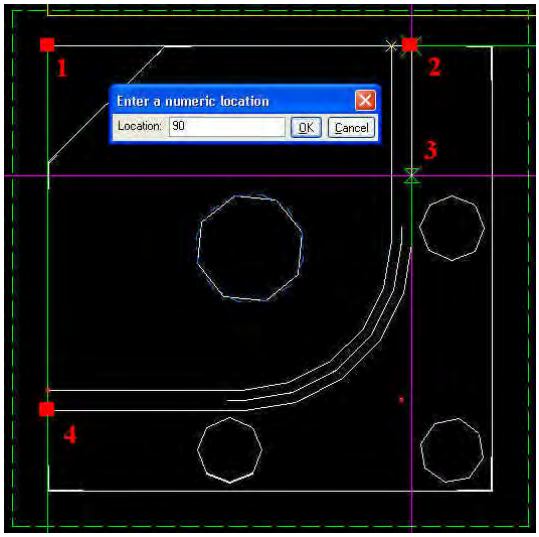
5. Next show direction of the column shoe by picking at any position at column shorter side. ColumnShoe component is created to the column corner.

Create part cut

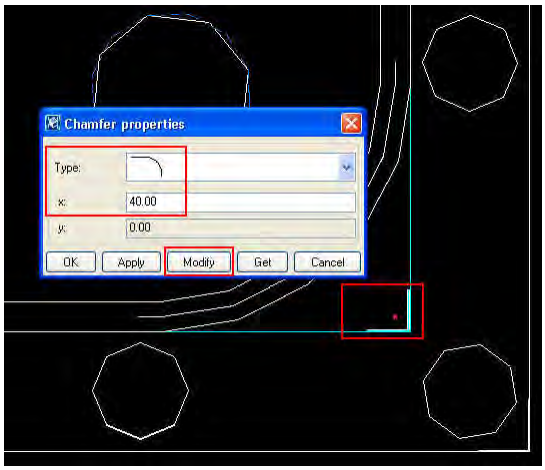
Next create a part cut to column corner. For this create first part basic views of the column shoe base plate. Check that select switch **Select objects in components** is pressed down. Then we will create the cutting part and cut the column corner with it.

1. Select the column shoe base plate and right click > Create view > Part basic views.
2. Double click the **Create concrete slab** icon.
3. Define part height to be 101 mms.
4. Close the dialog with **OK**.
5. On **Part top view** pick the polygon position with 4 points. First point is in the column corner, second point in the column shoe plate's outer corner, third point is given numerically: show direction at plate's side and type 90, then press **OK** on **Enter numeric**

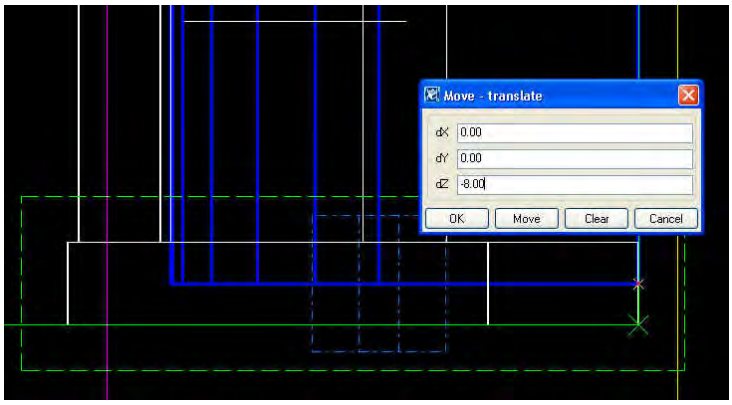
location dialog. Fourth point is in the column shoe plate's outer corner at opposite side, close polygon with middle button.



6. Now double click the chamfer at inner corner of the concrete slab you just created and set the type rounded with x-value 40 mms, then press **Modify**.

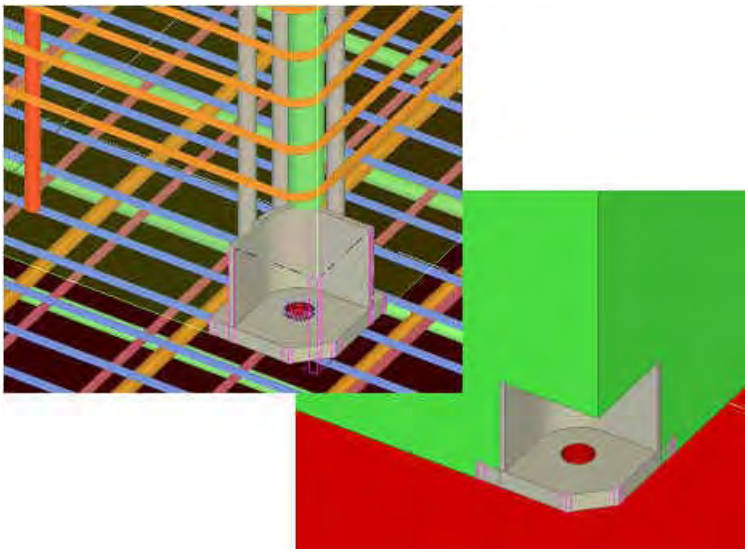


7. Select the slab and move it to correct position –8 mms in z-direction with right click command **Move > Translate…**



8. Now we are ready to make the part cut. Pick the **Create part cut** icon.

9. Select part to be cut, meaning the column.
10. Select cutting part, meaning the slab. The column corner is cut and the reinforcing macro reacts to the cut by shortening the corner bar and lifting up the lower stirups.



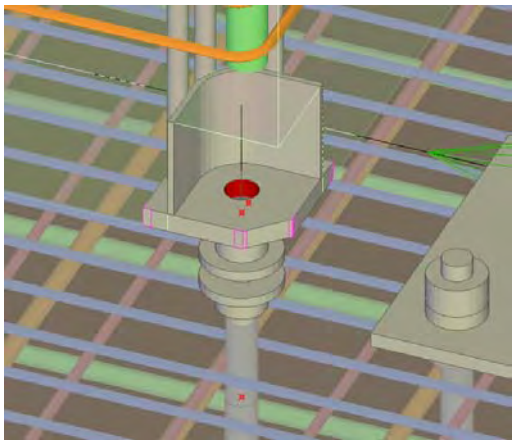
11. The cutting part is no longer needed and can now be deleted. Column corner is now completed.

Insert Anchorbar

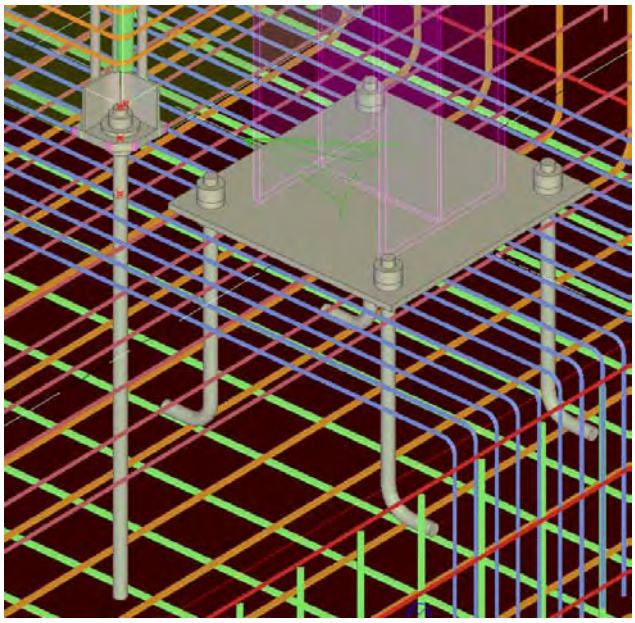
Next step is to add the anchor bolt to the footing.

1. First we need to create a point to the center of the hole in the column shoe. For this pick the **Create divided line points** icon.

2. Zoom in to the hole in **Part top view** window.
3. Pick at the middle of upper and lower segments of the hole, center point is created.
4. Next press **Ctrl + F** to open the **Component catalog** dialog.
5. Double click **AnchorBolt** from the list.
6. Pick the center point you just created.
7. Pick any position at the hole segments. Anchor bolt is created.



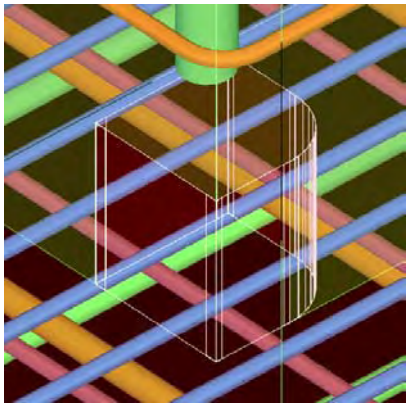
8. Select the anchor bolt and move it to correct position using right button command **Move > Translate...** Set 61.02 in dZ-direction and press **Move**.
9. The anchor bolt is too long and goes 409 mms below the footing bottom. To modify the bolt length, double click the anchor bolt having select switch **Select component** pressed down.
10. In **AnchorBolt** dialog, set **Total length** value to 750 mms and press **Modify**. The first column footing corner is now completed.



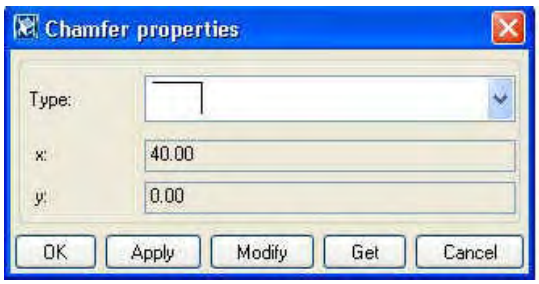
Copy the objects

When one corner has been completed we will copy the components and the cut to the remaining corners of the column. We need to do the copying in two steps, first the cut and then the components.

1. Select the cut in the column corner.
2. **Right click > Copy > Translate**. Set 510.00 in dX direction and press **Copy**. The column corner is cut, but the shape needs modification.



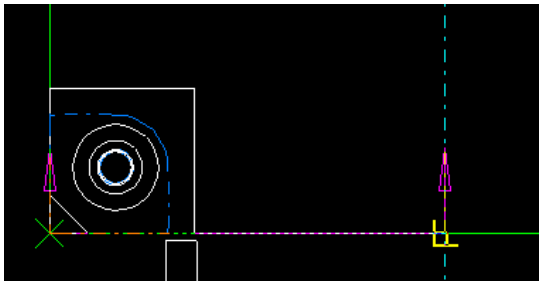
3. Double click the rounded chamfer of the cut to open the **Chamfer properties** dialog.
4. Select the inner sharp corner and press **Modify** in the **Chamfer properties** dialog.
5. Select again the first rounded corner and change the chamfer type to sharp and press **Modify**. The cut has now correct form.



We will next copy the components to the same corner. For the copying we need to insert a center point of rotation. Create part basic views of the footing to help modeling.

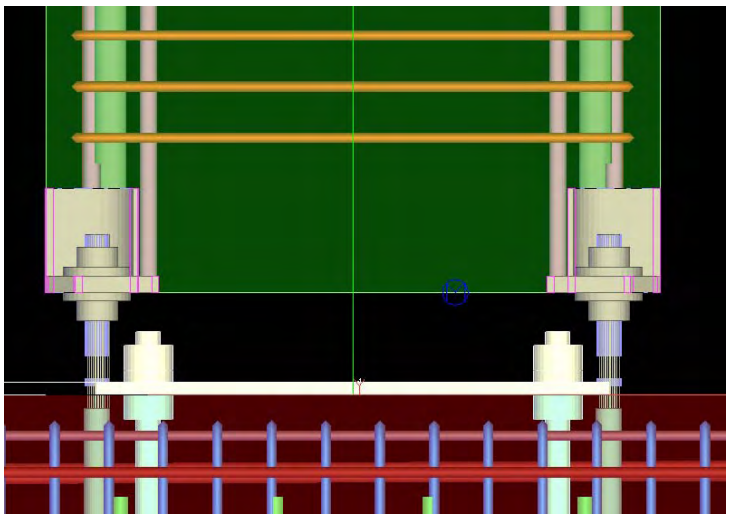
1. Double click **Create parallel point** icon and set distances to 300 mms and press **OK**.

2. On **Part top view**, pick first the corner, then the intersection of grid line at shorter side of the column.



3. Select the components **AnchorBolt** and **ColumnShoe** with select switch **Select parent** pressed down.
4. **Right click > Copy > Rotate…**
5. Set rotation angle to 90 degrees around Z. Note that pick mode must be set to View plane.

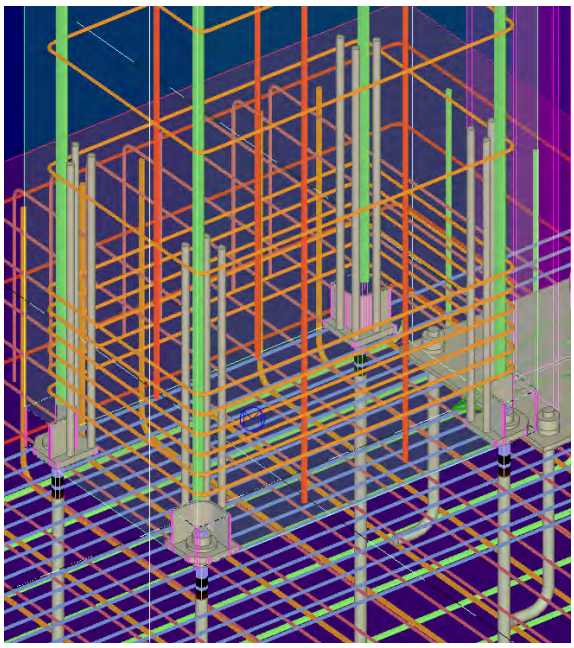
6. Pick center point of rotation.
7. Press **Copy**. We now have completed two corners.



We will now complete the opposite side of the column.

1. Copy first both of the corner cuts to the opposite side with **Copy > Translate…** using 810 mms in dY-direction and then modify the cut shapes as in described in previous part cut copy.
2. Insert new center of rotation point using **Create parallel point** tool and set distances to 450 mms, pick first corner, and then the intersection of grid line at shorter side of the column.
3. Select the **AnchorBolt** and **ColumnShoe** components and **right click > Copy > Rotate**, pick the new origin and set angle to –180 in Z-direction. Press **Copy**.

We now have components and cuts around the column.

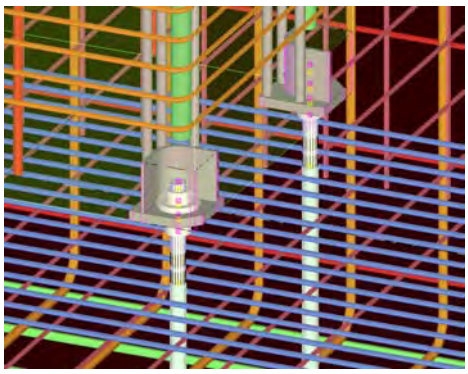


Define Cast Units

After all needed parts have been completed we will define cast units, meaning that the steel parts will be attached to correct concrete member. See more about cast units in Tekla Structures **Help: Modeling > Parts > Cast units and assemblies**.

We will first define cast unit of the footer.

1. Select all parts of Anchor bolt components having select switch **Objects in components** pressed down. Check that no parts, which belong to column shoe, are included.



2. Then add the footer to the selection and **right click > Cast unit > Create cast unit**. Anchor bolts will be now automatically become attached to the footer when we in next chapter create a connection of them.
3. Next select the anchor bolts under the steel column and **right click > Cast unit > Add to**. Then pick the footer. Cast unit for footer is now complete.
4. Repeat step 3 to all footers in grid line C.
5. Now select the parts, which belong to column shoes and add the concrete column to the selection.
6. **Right click > Cast unit > Create cast unit**. Column shoes will be now automatically become attached to the concrete column when we in next chapter create a connection of them.
7. Check the cast unit contents by selecting footing or column and **right click > Inquire > Assembly**.

Create connection

When corners have been completed we will create a user define connection, which has all the components and cuts in it.

1. Open **Custom element Wizard** from **Detailing > Define custom element**.
2. Select type to be **Connection** and give a name for it. Add description text. Then press **Next>**.
3. Select the column, footing and all the components (Note that you have select switch **Select parent** pressed down) and cuts, then press **Next>**.
4. Pick the footing to be the main part. Continue by pressing **Next>**.
5. Pick the column to be the secondary part and press **Finish**. New connection with name Column. Footing is created and the name appears in **Custom elements** dialog. We will not parameterize it in this exercise, but use it as it is.

Use the connection

We will now use the Column-Footer connection to all concrete columns. For different size than 900*600 columns, some adjustments are needed.

1. On a connections toolbar, double click the **Create custom element** icon.
2. Select the Column-Footer connection and press **OK**.
3. Use first the connection to columns in grid lines **C** and **F**. Pick first the footer, then the column.

Now create the connection to one 600*600 column. The components and cuts appear in wrong position and the connection needs modifications.

1. Select the connection and **right click > Edit custom element**. An editor opens.
2. In editor windows, select the components and cuts and **Move > Translate** them to correct position 150 mms in dY direction, at both side of the column.
3. In Custom element editor toolbar, select **Copy element to library with new name** and give a name Column-Footer-600 to it.

4. Close the editor and save it. We have now new connection, which can be used for 600*600 columns.
5. Use the Column-Footer-600 connection to columns in grid lines **D** and **E**. Pick first the footer, then the column.