

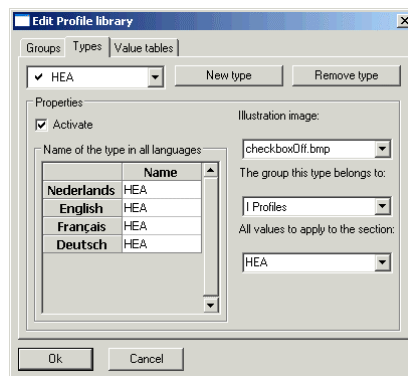
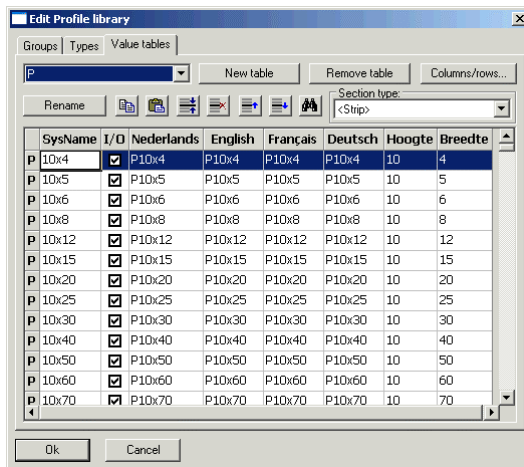
Parabuild training

New functionalities of Parabuild 1.0 and 1.1


Exercises

Editing the profiles library


- The profiles library is entirely adjustable
- In the spreadsheet we can modify each profile: name, dimensions, hide, delete, replace all, new spreadsheet.
- It is also possible to add new groups of profiles (tabs) and new types.
- Care should be taken not to accidentally delete a spreadsheet, because it would be permanently lost.



Exercise 1

1.  Open the profiles library
2. Activate the tab **Strips** (P) in the dialog window
3. Click on **Edit library**
4. In the spreadsheet, deactivate the checkboxes of strip *P10x4* and *P10x5*
5. Click on **Ok** and afterwards **Yes** to confirm the modifications
6. Remark that *P10x4* and *P10x5* are no longer visible in the list
7. Click again on **Edit library**
8. Click with the left mouse button on the file *P10x70* so that a frame appears
9. Now click on the right mouse button and select in the menu **Remove row**
10. Click on **Ok** and afterwards **yes** to confirm the modifications
11. Remark that *P10x70* no longer appears in the list and has been deleted permanently

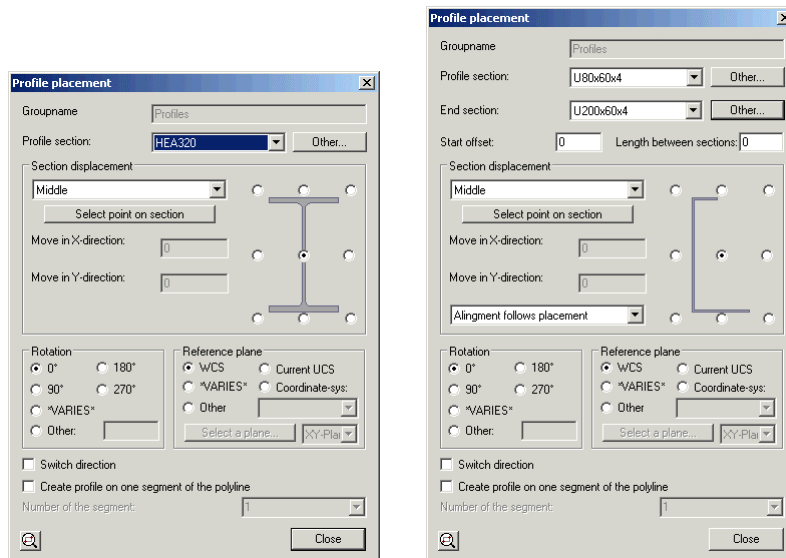
Exercise 2

1.  Open the profiles library
2. Click on **Edit library**

3. Activate the tab **Groups**
4. Click on **New group**
5. In the small spreadsheet, next to **English**, change the name to *My group*
6. Activate the tap **Types**
7. Click on **New type**
8. In the small spreadsheet, next to **English**, change the name to *My type*
9. Go to the option **The group this type belongs to**, and change its value to **My group**
10. Go to the option **All values to apply to the section**, and change its value to **HEX**
11. Click on **Ok**
12. Remark the new tab **My group** which contains **My type** as a profile type and **HEX** profiles in the list

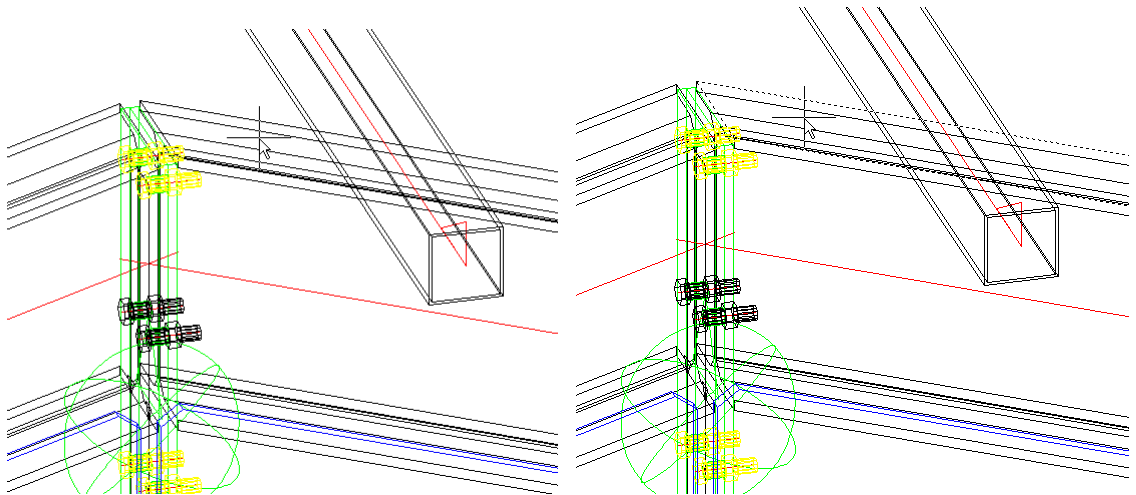
Drawing members

- The reference plane for the placement can be a plane of an existing object.
- Also profiles with two sections can be drawn using this command.
- Profile with two sections has a new option *Length between sections*.
- Cutting a Profile with two sections is only possible when using the command *Add cut to macro* (described in a later chapter).





Exercise 3

1. Open the drawing Exercise3.dwg
2. Open the profile library
3. Select the profile **TubeS80x80x3**
4. Click on **On modelline**
5. Select the green line and press **<Enter>**
6. In the dialog window at reference area, click on **Other**
7. Now we have to choose a plane of a profile, plate, polyline,.... We will select the top of one of the beams.
8. Zoom in on a beam using the wheel button of the mouse
9. Move with the cursor to an open area of the upper part of the beam
10. Click once on the left mouse button. The area you have just selected will highlighted with dotted lines. If the selection is not correct then move the cursor to the correct location and try again by pressing the left mouse button.



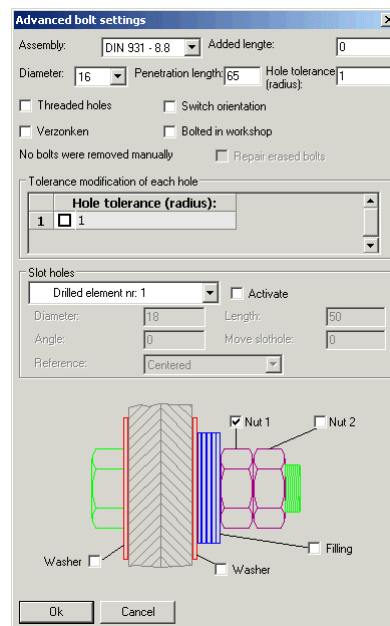
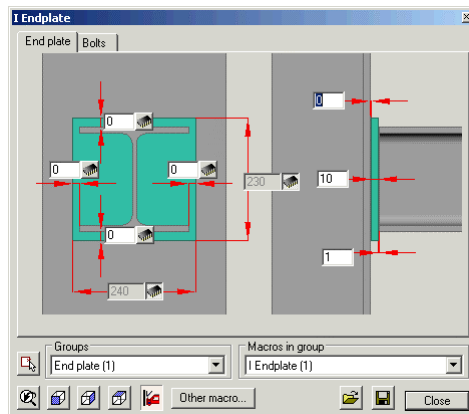
11. As soon as the top plane of the beam is drawn in dotted lines you should press the **<Enter>** button to confirm the selection.
12. Click on **Close**.

Exercise 4



1. Open the drawing  Exercise4.dwg
2.  Open the profiles library
3. Select the profile **U200x60x3**
4. Activate the option **Two sections**
5. Click on **On modelline**
6. Select the green line above the apex and **<Enter>**
7. For **Section displacement**, choose **Top**
8. At **End section**, click on **Other...**
9. Click on **Add quickly**
10. Enter the following dimensions: **Height: 500 Flange length 1: 60**
Flange length 1: 60 Thickness: 3
11. Click two times on **Ok**
12. We now see that the new profile runs through the beam. This is because the green line is 825mm long. The height of 500, which we have chosen for the end section, should be used in combination with a length of 1000, not 825 (for this slope). To solve this we will move the end section.
13. Change the option **Length between section** to 1000
14. Click on **Close**

Reviewing a macro



- This dialog window can remain open at all times during other commands. Multiple dialogs of this type can be active at the same time.
- You can decide whether you want to adapt the width/length of plates or not. When you changed the width/length, the computer will ensure that this dimension is always preserved.
- Distances between individual bolts are adjustable using the spacebar
- Slot holes, threaded holes and holes with varying diameters can be drawn with this dialog window.
- All user modifiable settings can be stored for later use. These stored settings will be checked for compatibility before loading it. Stored settings that are not 100% compatible will be hidden.
- Adapting several connections simultaneously is simple thanks to the grouping of connections according to name.








Exercise 5

1. Open the drawing  Exercise5.dwg
2. Start the command  **Review macro**
3. Select all elements in the drawing and press **<Enter>**
4. In the dialog window, choose in the lower left **Gusset Plate (12)** from the list
5. This group contains 2 different types of gusset plates. To make editing these different gusset plates easier, we will only modify gusset plates of the same type.
6. In the dialog window, choose in the lower right **Small pentagon gusset Plate (6)** from the list
7. Click on the button next to the dimension between the two bolts (the length of the plate will become adjustable)
8. Change the length of the plate to **180**
9. Remark that you have just modified the length of the 6 upper gusset plates of the 3 wind bracings.

Exercise 6

1. Open the drawing  Exercise6.dwg
10. Start the command  **Review macro**
2. Select the connection and press **<Enter>**
3. Activate the tab **Bolts**
4. Click on the **Advanced...** button
5. Activate the option **Threaded holes**
6. Below on the image, deactivate the option **Nut 1**
7. Click on **Ok**
8. Threaded holes were drawn in the last component (the blue plate)
9. Click again on the **Advanced...** button
10. In the middle of the dialog window, under **Hole tolerance**, activate the first item in the list and modify the tolerance to **2**
11. Click on **Ok**
12. Remark that the holes in the green endplate have become larger because this is the first 'drilled' component started from the head of the bolt.
13. Click again on the **Advanced...** button
14. In the dialog window at **Slot holes**, click on **Activate**
15. Below that modify **Length** to **15**
16. Click on **Ok**






Exercise 7

1. Open the drawing  Exercise7.dwg
2. Start the commands  **Review macro**
3. Select the second last haunch connection and press **<Enter>**
4. Activate the tab **Bolts**
5. Fill in the field for bolts in vertical direction: **80 280 80**
6. Click on  **Save settings...**
7. For name, type in **Exercise7**
8. Click on **Save**
9. Click in the dialog window on  **Selects other macros**
10. Select the last haunch connection and press **<Enter>**
11. Click on  **Load settings...**
12. The settings we stored just earlier are not immediately visible because this connection has other characterizations. To remedy this click on **Show files that are not entirely compatible.**
13. Select **Exercise7** from the list and click on **Open**
14. You will receive a warning that states that at least 7 settings were not loaded. This is due to the endplate of the last column, which has a horizontal orientation.
15. Click on **Ok**
16. Remark that the settings of the bolts were still copied over.

Manipulating macros



- A macro will adapt all of its components automatically after a modification. Manually adapting these components is not possible.
- When we remove the sphere of a connection, all intelligence will disappear and the components can be modified manually.
- Plates and profiles, which are components of a macro, can be removed with the erase command. One should not set the thickness value of a plate to 0.
- The sphere of a connection will become red when the connections cannot be calculated, for example removing or rotating some base profiles 90°.
- One can manually remove bolts. The macro will not restore erased bolts, unless you ask for it.
- Also bolt patterns can be removed
- Components of a macro can be welded, the macro will stay intact.
- If necessary you can remove the cut of a macro. This can be useful because a profile can't be prolonged when the end of the profile is determined by a cut of a macro.

Exercise 8

1. Open the drawing  Exercise8.dwg
2. Start the command  **Move**
3. Select the beam and **<Enter>**
4. Move the beam **300mm** in the **–Z** direction using ORTHO
5. The entire connection was adapted to this modification
6. Remove the macro (green sphere)
7. Move the beam **300mm** in the **+Z** direction using ORTHO
8. Not a single component of the connection was adapted.
9. Click twice on  **Undo** so that the macro is restored
10. Select the beam and press the **Delete** button of your keyboard
11. The macro becomes red because the connection can't be calculated without the beam. The dimensions of the plates can become unpredictable.
12. Click once on  **Undo** to restore the beam
13. Delete the endplate of the column
14. Start the command  **Review macro**
15. Select the macro and **<Enter>**
16. Activate the tab **Column endplate**
17. The settings of the endplate still exist, although the plate is gone. This is because the cut of the column is dependant on the deleted plate.

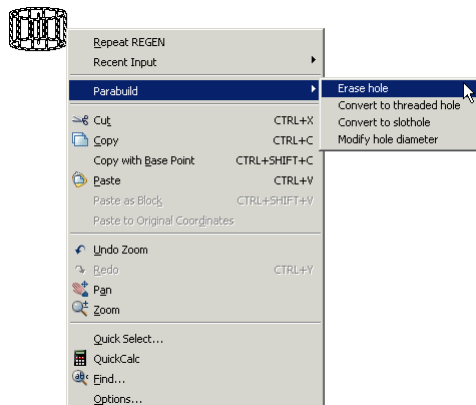
18. Modify the offset of the cut to - **10** (left in the middle of the image)
19. This way it is still possible to set the height of the column to the same level as the beam.

Exercise 9



1. Open the drawing  Exercise9.dwg
2. Remove all the stiffeners (4) of the haunch connection with the erase command
3. Start the command  **Review macro**
4. Select the haunch connection and **<Enter>**
5. Click in the dialog window on the button **Other macro...**
6. Choose in the list the same haunch connection, namely **Haunch plates**
7. Click on **Ok**
8. The stiffeners were restored because we have reloaded the original connection from the library.

Context menu

- Carrying out commands with the right mouse button without having to search for icons.
- Removing holes, modify diameter, convert to threaded hole, convert to slot hole.
- Removing cuts.
- Reviewing a macro.
- Copy settings of a macro to other macros.
- Selecting an element first and then clicking on the right mouse button works too, only pay attention that the menu will be at another location.
- The hole tolerances are also adjustable by means of the AutoCAD Properties of the bolt.





Exercise 10

1. Open the drawing  Exercise10.dwg
2. Remove the bolt that is located inside the tube
3. Move the cursor to above one of the holes stand now loose (a line of the hole must run through the small square of the crosshairs)
4. Click on the right mouse button
5. Move the mouse over the text **Parabuild** in the new menu (if this text is grey, then you hadn't placed the crosshairs on top of the hole)
6. Choose in the new menu **Erase holes**
7. Do the same for the other hole
8. The manually removed bolts were not automatically restored by the connection. We can however restore the removed bolts.
9. Start the command  **Review macro**
10. Select the connection of the tube and **<Enter>**
11. Open the tab **Bolts**
12. Click on **Advanced...**

13. Activate **Repair erased bolts**
14. Click on **Ok**




Exercise 11

1. Open the drawing  Exercise11.dwg
2. Start the command  **Review macro**
3. Select the first haunch connection and **<Enter>**
4. Activate the tab **Reinforcements**
5. Modify the length of the Reinforcement to **800**
6. Click on **Close**
7. Zoom in on the haunch connection you just modified with the mouse wheel button
8. Move the crosshairs to above the sphere of the connection
9. Click on the right mouse button
10. Move the mouse over **Parabuild** and click on **Copy all parameters to another macro**
11. Select the entire drawing and **<Enter>**
12. The computer has transferred the options to all the connections that are exactly the same. There is only one haunch connection in the back that did not receive the new settings, because the endplate of the column in that connection is different.



Drawing endplates and stiffeners


- When drawing an endplate, stiffener or reinforcement you must choose a reference that determines the placement. This reference has to be a plane or a point on a second object.
- By rotating or moving the second object you can modify the endplate afterwards.
- When drawing an endplate we need in fact a 2nd object. If we select a plane on the same profile, then a new object will be created automatically.

Exercise 12

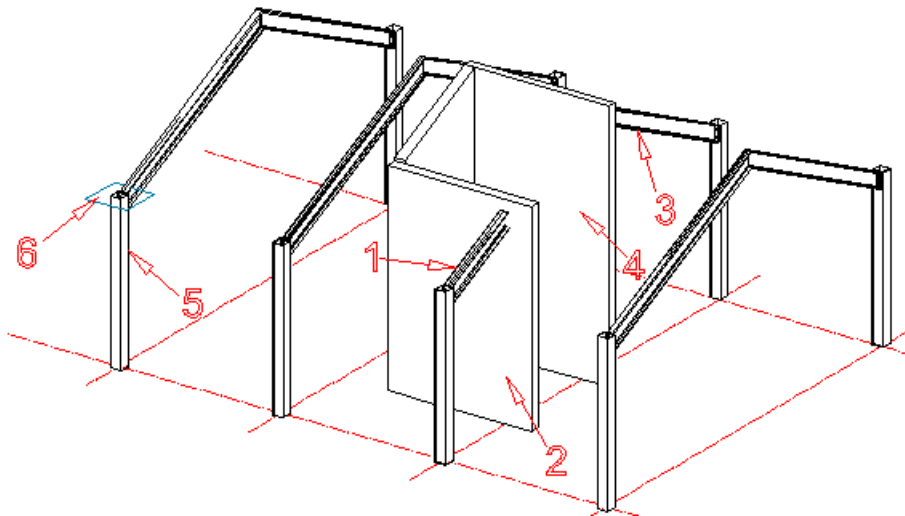
1. Open the drawing  Exercise12.dwg
2. Start the command  **Reinforcements**
3. Select the beam somewhere on the lower part and **<Enter>**
4. In the dialog window choose **Flangereinforcement** and click on **Ok**
5. Now move the crosshairs to above the area of the green plate and click on the left mouse button
6. Press **<Enter>** if the selection of the top plane is highlighted
7. Press **<Enter>** once again
8. Start the command  **Reinforcements** again
9. Select the beam somewhere on the upper part and **<Enter>**
10. In the dialog window choose **Flangereinforcement** and click on **Ok**
11. Now move the crosshairs to above the area of the green plate and click on the left mouse button
12. Press **<Enter>** if the selection of the top plane is highlighted
13. Press **<Enter>** once again
14. The location where we select the beam determines the placement of the reinforcement. This rule also applies to a lot of other connections.





Exercise 13


1. Open the drawing  Exercise13.dwg
2. Start the command  **Stiffeners**
3. Select the UPN member and press **<Enter>**
4. Double-click the connection **Stiffener to point**
5. Now move the crosshair to above the endpoint of the axis of the tube and click on the left mouse button
6. Press **<Enter>** if you can see the cross on top of the endpoint

7. Press **<Enter>** once again
8. Click on **Close**
9. Start the command  **Move**
10. Select the tube and press **<Enter>**
11. Move the tube 150mm in the +X direction
12. The stiffener moves along with the tube because it is dependant on the endpoint of the tube

Exercise 14



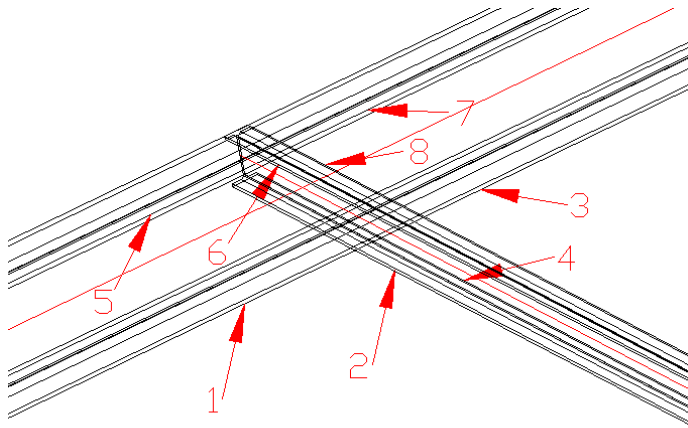
1. Open the drawing  Exercise14.dwg
2. Start the command  **End/Baseplates**
3. Select the beam (point 1)
4. Select **Endplate to plane** and click on **Ok**
5. Move the crosshairs to the open area of the wall (point 2), click on the left mouse button to confirm and then press **<Enter>**
6. Open the tab **Bolts** and click on **Advanced...**
7. Deactivate **Nut 1** and click on **Ok**
8. Start the command  **Automatically copy a macro**
9. Select the connection we just created and press **<Enter>**
10. Select the beam on the other side (point 3)
11. Select the wall in the middle (point 4)
12. Press **<Enter>**
13. Start the command  **End/Baseplates**
14. Select the last column (point 5)
15. Select **Endplate to plane** and click on **Ok**
16. Move the mouse to the open area of the polyline (point 6), click on the left mouse button to confirm and then press **<Enter>**



17. Repeat this for the other 7 columns by each time first selecting the next column and then always the same polyline (point 6)
18. Press **<Enter>** to end the command
19. Start the command  **Move**
20. Select the polyline (point 6) and **<Enter>**
21. Move the polyline 200mm in the +Z direction
22. All the column endplates move along with the polyline, also if you would rotate the polyline.

Adding connections from the library

- The location where we select the profiles can influence the placement of the connection.
- The icons of the connections serve as indicators only. There are more connections behind an icon than one would initially think. Moreover one can always search the entire library.
- To most of the connections the rule applies that one has to first select the continuing profile and then the ending profile.

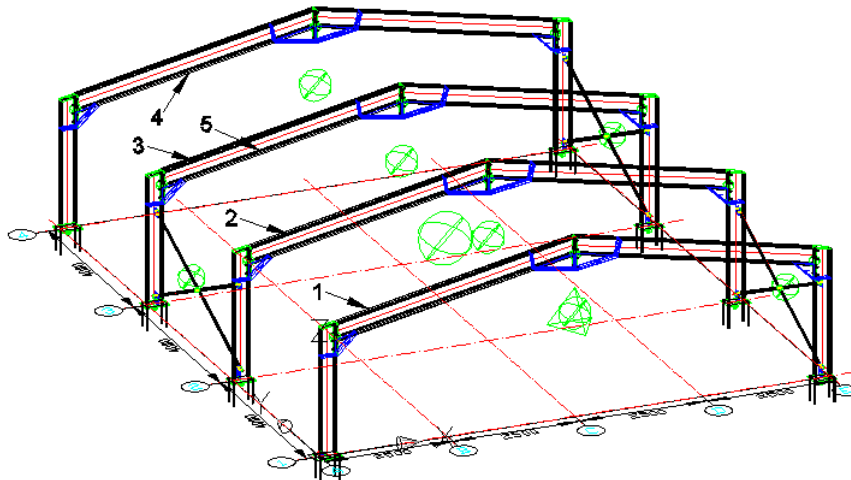
Exercise 15










1. Open the drawing  Exercise15.dwg
2. Start the command  **Beam vs beam**
3. Select the beam approximately on point 1
4. Select the first IPE120 approximately on point 2
5. Select the connection **Clipangle vs web** and click on **Ok**
6. Zoom in on the second IPE120
7. Select the beam approximately on point 3
8. Select the second IPE120 approximately on point 4
9. Zoom in on the third IPE120
10. Select the beam approximately on point 5
11. Select the third IPE120 approximately on point 6
12. Zoom in on the fourth IPE120
13. Select the beam approximately on point 7
14. Select the fourth IPE120 approximately on point 8
15. Press **<Enter>** and click on **Close**

16. Remark that the connection was mirrored four times, each time differently.

Exercise 16






1. Open the drawing  Exercise16.dwg
2. Start the command  **Wind bracing**
3. Select the first beam approximately on point 1
4. Select the second beam approximately on point 2
5. Select the bracing **Angles** and click on **Ok**
6. Press **<Enter>** and click on **Close**
7. Start the command  **Gusset 2 base**
8. Select the first beam
9. Select the first angle profile below (pay attention that you do not select the line)
10. Select the connection **Small pentagon gusset plate** and click on **Ok**
11. Select the second beam
12. Select the second angle profile below (pay attention that you do not select the line)
13. Select the first beam
14. Select the first angle profile at the top (pay attention that you do not select the line)
15. The command stops and the last connection were not created. This is due to the fact that the orientations of the angle profiles for the upper gusset plates require another connection. We will solve this by restarting the command again and Parabuild will search for the correct connection in the library.
16. Start the command  **Gusset 2 base**
17. Select the first beam
18. Select the first angle profile at the top (pay attention that you do not select the line)

19. Select the connection **Small pentagon gusset plate** and click on **Ok**
20. Select the second beam
21. Select the second angle profile at the top (pay attention that you do not select the line)
22. Press **<Enter>** and click on **Close**
23. Start the command  **Purlin on top of beam**
24. Select the first angle profile
25. Select the second angle profile (the order doesn't matter this time)
26. Select the bolt connection and click on **Ok**
27. Press **<Enter>** and **Close**
28. Start the command  **Review macro**
29. Select the large sphere in the middle of the wind bracing and press **<Enter>**
30. Change the distance **165** to **50** so that the bracing leaves the centre of the beams (you can do this by pressing the button next to the 165 distance)
31. Click on **Close**
32. Start the command  **Automatically copy a macro**
33. Select the large sphere in the middle of the wind bracing and press **<Enter>**
34. Select the second beam approximately on point 2
35. Select the third beam approximately on point 3
36. Select the last beam approximately on point 4
37. Select the third beam approximately on point 5
38. Press **<Enter>** and click on **Close**
39. While copying the wind bracing the 4 gusset plates and the centre bolt are taken along automatically because they depend on the wind bracing. Also selecting the beams in another order can change the reference side on the beam.

Drawing a profile in a macro

- While drawing a profile we can opt to keep the connection between the line and the profile preserved. We do this by putting the profile in a macro.
- In this case we can only move the profile by moving the line.
- The Rotation / Placement point / Reference plane are adjustable afterwards by means of the properties or by reviewing the macro.
- The profiles of a wind bracing can also be rotated with the properties dialog box. Pay attention when doing this because the gusset plate connections will become invalid.
- Also a profile with 2 sections can be drawn this way and therefore easily adapted afterwards.
- Both the new and the old way of drawing can be used in the same 3D drawing.

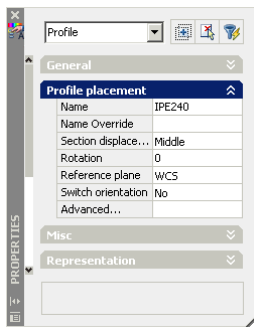
Exercise 17

1. Open the drawing  Exercise17.dwg
2.  Open the profiles library
3. Select **IPE200** and click on **On modelline**
4. Select the first vertical line and press **<Enter>**
5. Click on **Close**
6.  Open the profiles library
7. In the dialog window below at **In which macro?**, select there **In a new macro**
8. Click **On modelline**
9. Select the second vertical line and press **<Enter>**
10. Click on **Close**
11. Move the first column with the command move
12. Remark that you can only move the second column by moving the model line
13. Open the AutoCAD Properties by double-clicking on the first column
14. Open in Properties the beam **Profile placement**. These properties are not adaptable for this profile.
15. Press the **<ESC>** key
16. Select the second profile
17. Modify under **Profile placement** the property **Rotation** to 90
18. The rotation and the other settings are adaptable thanks to the fact that the link between the profile and the line is preserved by the macro







19. Click on the property **Advanced...** and click the ... button next to the property
20. In the Profile placement dialog window, activate the setting **Disconnect the link between member and line**
21. Click on **Close**
22. Open the Properties of the second profile again. The settings under **Profile placement** have become *Not available* for this profile.

Drawing profiles on grid lines

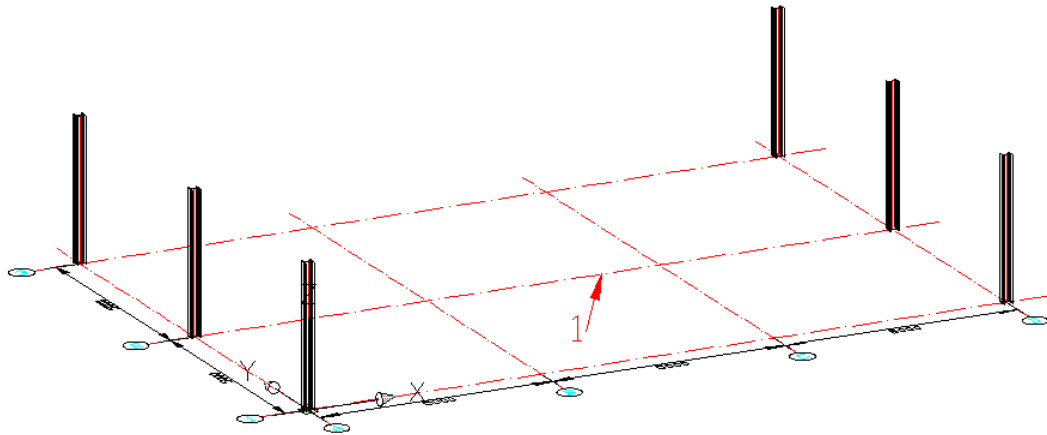
- Drawing columns and beams selecting grid lines.
- Drawing columns and beams by selecting points.
- Drawing apex and free profile (flexible in 3D all directions). A free profile can connect itself to ordinary lines...
- Columns, beams and apex can also be drawn at a certain level.
- Modifying profiles with the AutoCAD Properties.
- The connection between line and profile is interruptible by means of the Advanced button in the properties or by removing the macro
- Adding and modifying grid lines. Grid lines are lines that are on the layer Pb-GridLines.
- When moving grid lines all columns/beams that are connected to these grid lines will move along.






Exercise 18

1. Open the drawing  Exercise18.dwg
2. Start the command  **Grid**
3. Click on **Ok**
4. Start the command  **Modify levels**
5. Click on  **New...**
6. For the name enter **Roof**
7. Modify the height of the roof to **3000** and click on **Ok**
8. Start the command  **Columns with grid lines**
9. Change the **Top level** to Roof
10. Change **Offset top** to 0
11. Click on **Ok**
12. Select the grid lines A, D, 1, 2 and 3 and press **<Enter>**
13. Start the command  **Column with point**

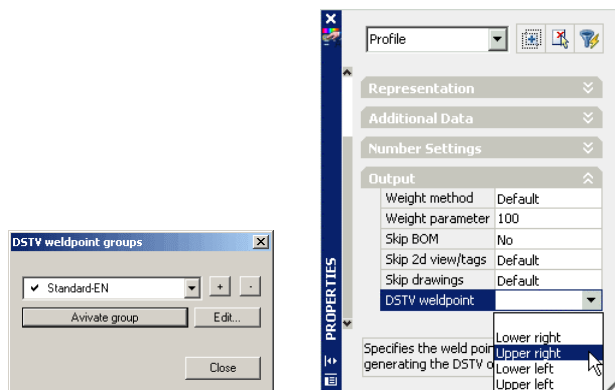
14. Change the **Top level** to Roof
15. Change **Offset top** to 0
16. Click on **Ok**



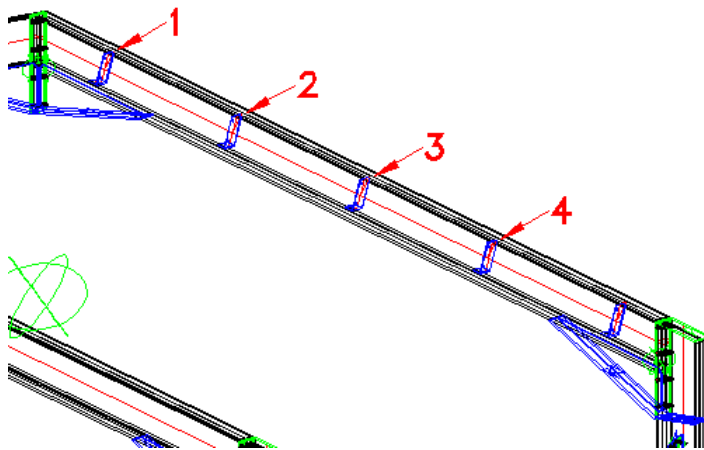
17. Click on point 1 with the help of **Osnap Nearest**
18. Press **<Enter>**
19. Start the command  **Beams with grid lines**
20. Change the **Level** to Roof
21. Change **Offset top** to 0
22. Click on **Ok**
23. Select the grid lines 1, 2 and 3 and press **<Enter>**
24. Remark that in the middle there were 2 beams drawn, a break at the location of the column.
25. Double-click on the column in the middle of the drawing
26. In the Properties dialog window, scroll down entirely below the list to **Member dimensions**
27. Modify the property **Offset X** to **500**
28. Because we did not draw this column using an intersection, the computer has chosen the distance to the nearest grid line (C). It is this distance that we've modified just now.
29.  Move grid line C 1000mm in the -X direction
30. Start the command  **Modify levels**
31. Select **Roof** from the list
32. Enter a new height of **4000**
33. Click on **Ok**

DSTV weld points



- In the general settings of Parabuild the desired weld points must be chosen in each 3D drawing.
- For each welded element that has to receive a weld point the correct weld point must be chosen in the Properties.
- The mark drawings will get a cross for each weld point. Of course the NC files will contain corresponding points.
- Several NC files will be made for one position number if a mark has weld points (for example both PR3.nc and PR3M6.nc can exist)
- More advanced settings are available ...



Exercise 19



1. Open the drawing Exercise19.dwg
2. Start the command **Settings**
3. Click in the dialog window on the button **DSTV weldpoints**
4. From the list, select **Standard-EN** and click on **Activate group**
5. Click on **Close** and afterwards **Ok**
6. Open the Properties dialog window of the first bracket (double-click point 1)

7. Scroll down the list to the very last property **DSTV weld point** (under output)
8. Select in the list **Lower right** for this bracket
9. Press **<ESC>** and select the second bracket (point 2)
10. Go to the property **DSTV weld point** and select in the list **Upper right** for this bracket
11. Press **<ESC>** and select the second bracket (point 3)
12. Go to the property **DSTV weld point** and select in the list **Lower left** for this bracket
13. Press **<ESC>** and select the second bracket (point 4)
14. Go to the property **DSTV weld point** and select in the list **Upper left** for this bracket
15. Start the command  **Workshop drawings**
16. Click on **Continue with temporary numbers**
17. Activate the option **Generate DSTV for plates** and **Generate DSTV for profiles**
18. Click on **Ok**
19. Open the drawing  Exercise19\M6.dwg
20. The bracket to the right has received a cross to the lower right
21. The second bracket to the right has received a cross to the upper right. This is because the software would never add a point "in the air".
22. The third bracket has received a cross to the lower left
23. The fourth bracket has received a cross to the upper left. This point is not entirely located at the top, but is 50mm (adjustable) removed from the flange.
24. Open Windows Explorer
25. Open the folder of the Workshop drawings: Exercise19\
26. Remark that there are two different DSTV files for the same position number PR3: PR3.nc and PR3M6.nc. The file PR3M6.nc contains weld points for the mark M6, PR3.nc does not contain any weld points.

Standard widths of plates

The triangle (= axis) of a plate used to determine the name of a plate.

Now the triangle only determines the direction of the plate.

The name of the plate (what is the width, what is the length) is now automatically decided by the software with the help of the axis and a standards file (standards file see S3d_Lib\PlateStdWidths.dat).

Revisions (recurrence)

Nothing has changed has changed to the numbering and revisions since Steel3d.

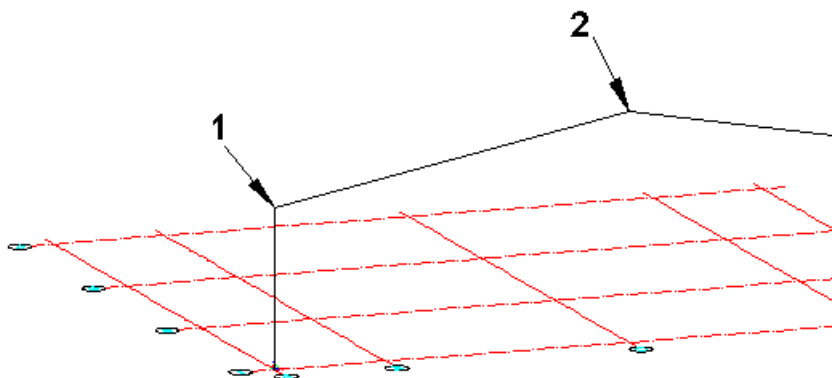
There is a new warning dialog box that will warn you concerning parts inside the drawing that haven't been numbered yet.

If a revision is not fixed, then the numbers in that revision can change without you asking for it.



Trusses

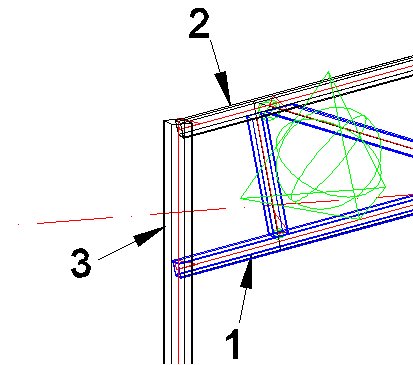
- A truss that we obtain from the library is always based on one line.
- This line determines the location, length and slope of the truss.
- Trusses can be connected using special connections.
- Trusses can be copied/mirrored when all parts are copied along with it, including the base line.


Exercise 20



1. Open the drawing  Exercise20.dwg

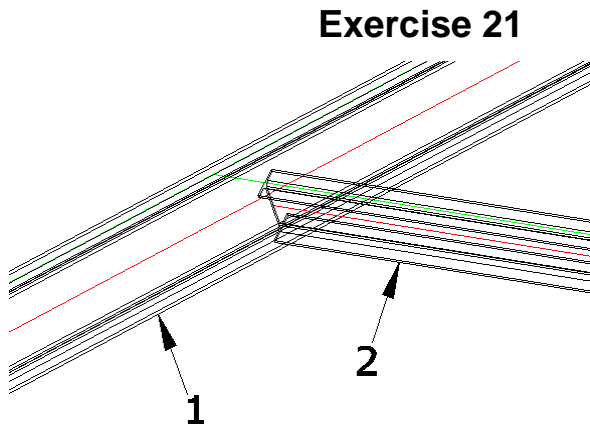
2. Start the command  **Truss**
3. Select **Truss perpendicular** and click on **Ok**
4. Select the endpoint point 1
5. Change the heights left and right to 700
6. Change the **number of struts** to 6
7. Click on **Close**
8. Zoom in on the end of the truss and select the upper line
9. Select the blue squares (grips) of the line and stretch it to apex (point 2). If this did not work, then you have probably moved the axis of the tube, so try again.
10.  Open the profiles library
11. Select the tube **TubeS100x100x3** and click on **On model line**
12. Select the left vertical line and press **<Enter>** and then click **Close**






13. Start the command  **Truss haunch connection**
14. Select the lower beam (point 1)
15. Select the upper beam (point 2)
16. Select the column (point 3)
17. Select **Truss Endplate** and click on **Ok** and then **Close**

Drawing a cut inside a macro

Also cuts can be placed inside a macro. This has the advantage that the cut will be adapted when one of the base profiles is modified.



1. Open the drawing  Exercise21.dwg
2. Start the command  **Beam vs beam**
3. Select the beam located to the left (**point 1**)
4. Select the beam located to the right (**point 2**)
5. Select the connection **Webplates**, click on **Ok** and then press **<Enter>**
6. The cut of the connection is at a 3D angle. We solve this by removing the cut and drawing a new one that is producible.
7. In the middle of the dialog window, select from the list **Perpendicular only in X-direction**
8. Select from the list **Perpendicular only in Y-direction**
9. Click on **Close**
10. Zoom in on the cut and move the crosshairs on top of a line of the cut
11. Click on the right mouse button
12. Move over the item **Parabuild** and click on **Erase cut** in the menu
13. Start the command  **Add cut to macro**
14. At perpendicular, select in the list **Perpendicular only in X-direction**
15. Activate the setting **Cut against the surrounding rectangle**
16. Click on **Ok**
17. Select the beam located to the right (**point 2**)
18. Select the beam located to the left (**point 1**)