# Parabuild manual

# Table of contents

# New features

This chapter contains the new functions of the major Parabuild releases.

## Version 3.0

- Visibility manager
- Context Modeling
- Drawing a range of bolts (on a plane)
- Drawing holes for galvanisation
- Adding annotations automatically
- Reusing an expired workshop drawing

## Version 2.1

- Show only a selection
- Isolate objects based on a grid line
- Showing / hiding camera's
- Countersunk holes
- Blind holes
- BIM: Ifc file import
- BIM: Ifc file export
- The format, scale and page arrangement of workshop drawings are now determined by the AI
- Export new workshop drawings as PDF file while generating drawings

## Version 2.0

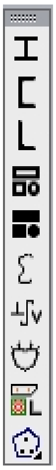
- Renewed Workshop drawings
- Renewed General Arrangement drawings
- Camera's for General Arrangement views
- Automatic dimensions are now drawn with AutoCAD objects
- New annotation objects
- Weld contours can now be added automatically to Dstv files

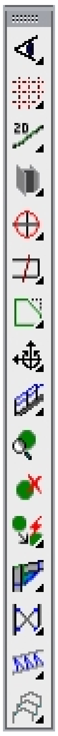
## Version 1.0

- The profiles library was improved. Now you can easily modify it yourself.

- All connections are entirely renewed. Some of the most important renewals are:
  - More connections
  - Images for all connections which illustrate the usefulness of settings
  - Connections adapt themselves automatically to changed situations (section modified, base profiles moved,…)
  - Several connections are adaptable simultaneously with one operation
- Advanced users can produce connections themselves, without the need for programming.
- A project can be built intelligently. For example profiles which remain dependant on their model lines.
- Automatic drawing of a cage ladder, frame and trusses have been renewed.
- Weld points specially created for DSTV files. Parabuild will create a weld point for each welded element in the DSTV file and the CNC machine adds the point automatically so that the welders do not have to measure over the length.

## Base AutoCAD knowledge

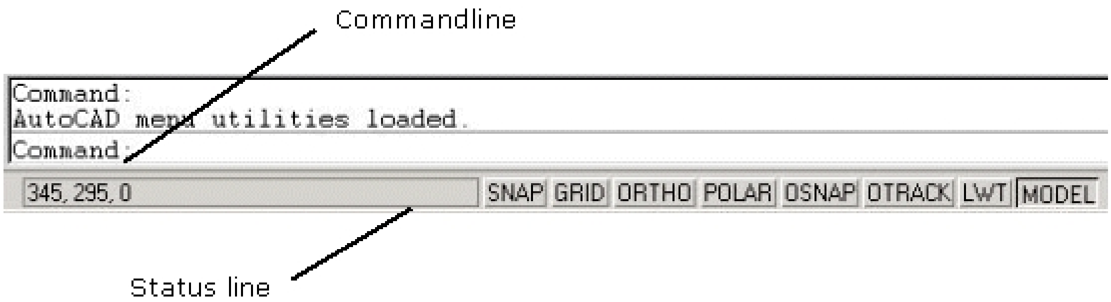




The Parabuild toolbars are positioned on the right hand side of the screen. The lower toolbar contains all of the Parabuild functions.

You can move this toolbar to a different position in the display by dragging the **grab bar**.

The command line and status line are at the bottom of the screen. You can activate or deactivate a command in de status line by clicking once with the left mouse button. The settings for these commands can be modified using the right mouse button.




In this tutorial you will notice text in boxes, this text consists of command lines that are used with certain commands for demonstration purposes. In these boxes comments are displayed in **Bold**.

## Some rules about 3D

Some general rules about 3D coordinates and work planes.

Before we can set the properties of a work plane, we have to explain the AutoCAD co-ordinate system (UCS, WCS, etc... ).

**Absolute coordinates:** e.g.: 20,40,50

Values:

X = 20

Y = 40

Z = 50

The coordinates 20, 40, 50 are the absolute coordinates from the origin of the current WCS(UCS).

**Relative coordinates.** e.g.: @37,-25,50

@-sign indicates relative values from the last point.

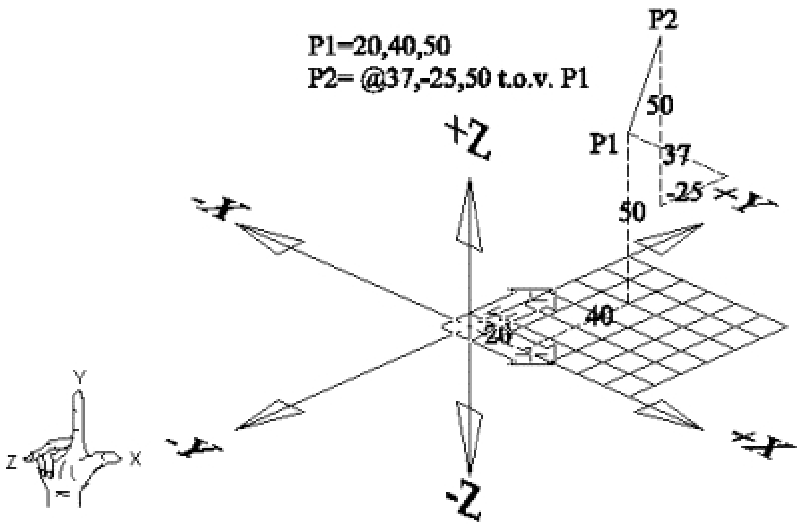
Values:

X = 37

Y = -25

Z = 50

The relative coordinates @37,-25,50 define a coordinate at a distance of X = 37, Y = -25, en Z = 50 from the last point entered (20,40,50).




To determine the positive direction of the Z-axis you can use the right-hand rule:

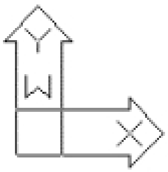
- Thumb = direction of the positive X-axis
- Index = points to the positive Y-axis
- The remaining fingers bent inward give you the direction of the positive Z-axis.

When you start a new drawing in a WCS coordinate system, the positive Z-axis points towards you.

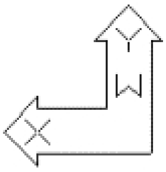
## There are three different coordinate systems:

1°-WCS World coordinate system.

Standard coordinate system (origin at the bottom left).



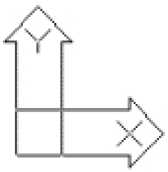

WCS TOP view



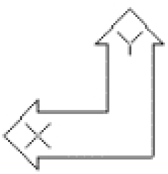

WCS BOTTOM view

2°-UCS User coordinate system.

User defined coordinate system (For each UCS you set a different origin, x-axis, y-axis and z-axis).

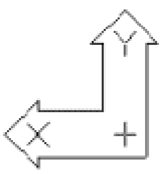



UCS TOP view
not linked to origin




UCS BOTTOM view

not linked to origin



UCS BOTTOM view

linked to origin(+)

3°-OCS Object coordinate system.

This coordinate system cannot be viewed or modified. It is embedded in each object and is stored in the AutoCAD database. This OCS has direction and origin from the UCS in use at the moment that the object is created.

You can make the UCS coincide with the OCS by placing the work plane according to an object.

Correctly setting the UCS-planes is a determining factor for good results in 3D-drawing. Once you have set the UCS wrong, the drawing is lost because you will repeatedly encounter objects with wrong co-ordinates.

With the UCS you can define a co-ordinate system in space. This allows us to use all 2D commands in 3D.

In a 3D space we can only draw lines, 3D polylines and 3D objects. All other commands are executed in the current UCS.

You have to avoid reusing too many times the current UCS to create a new UCS: otherwise this will multiply previous errors. In other words: always return to the WCS before creating a new UCS!

### Importance of Osnap when setting the UCS.

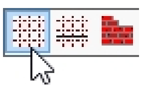
When you use Osnap options (see Basic Training), AutoCAD will ignore all other selection methods. This means that whatever the settings are for UCS, elevation, etc. you will select the correct X, Y, Z-coordinate when you use the Intersection-command.

Without Osnap options, any point selected randomly on the screen will always be on the current XY-plane or UCS unless Elevation is set. Beware of the Osnap Nearest Option! Also keep in mind the size of the selection square.

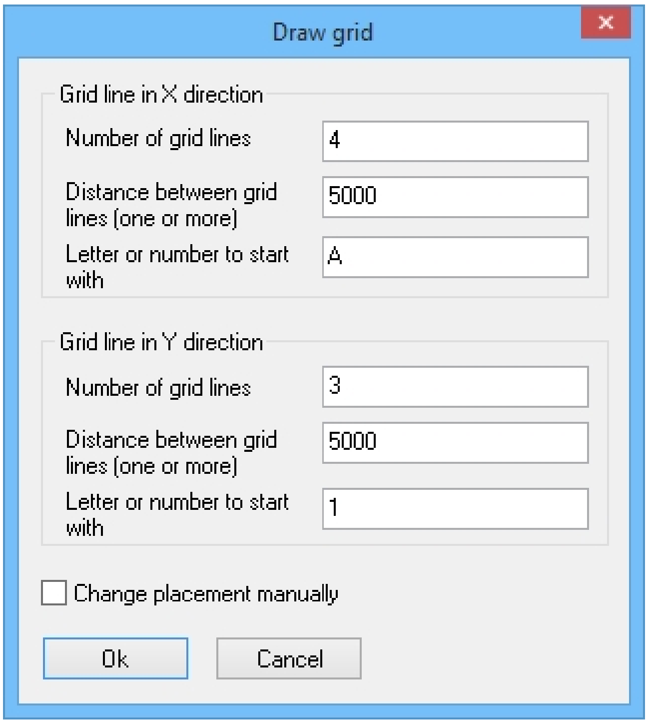
What AutoCAD defines as the nearest point is not always the same as the nearest point we perceive ourselves.

## Grid lines

Command : **S3d_DrawGrid**



When you click the Grid icon, the dialog with the grid options is displayed:

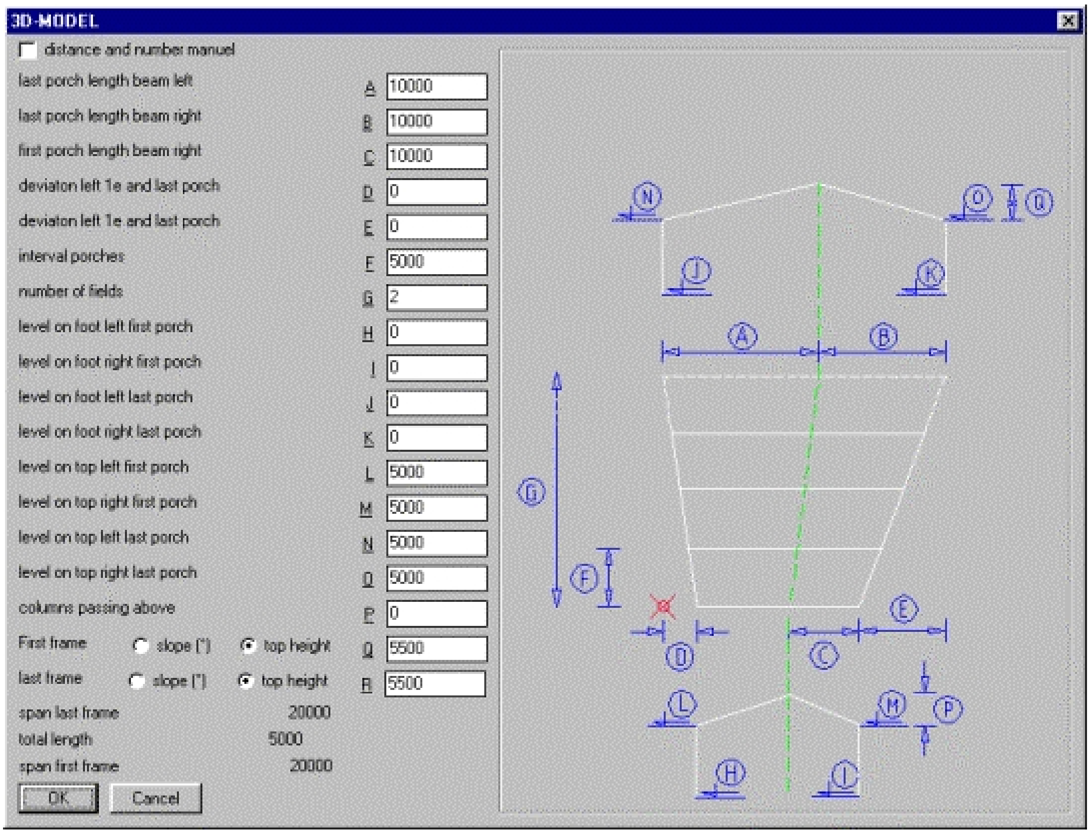



If you need to draw grids that are unequally spaced then you should enter in the **Distance between grid lines** field something like this for 4 grid lines : "5000 4000 6000"

When you click **OK**, the grid is drawn on the origin.
You can move/rotate/stretch these entities; these are regular lines that have their grid name attached to them.

## Frames

 When selecting this command, the following dialog box is displayed.

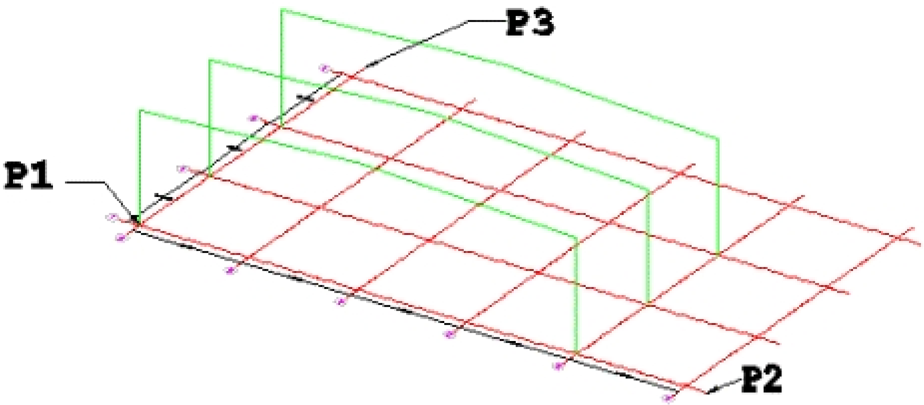



*Filling in the fields of this dialog is self-explanatory.*

Command: the lower-left corner of the matrix: **P1**

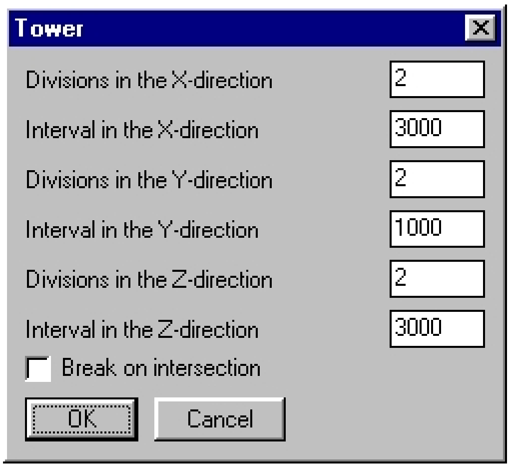
Command: the second point, to the right (X-direction): **P2**

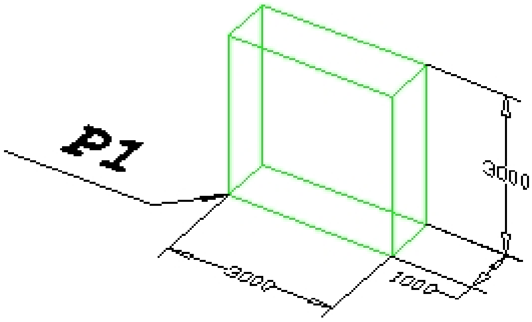
Command: the third point, for the Y-direction: **P3**



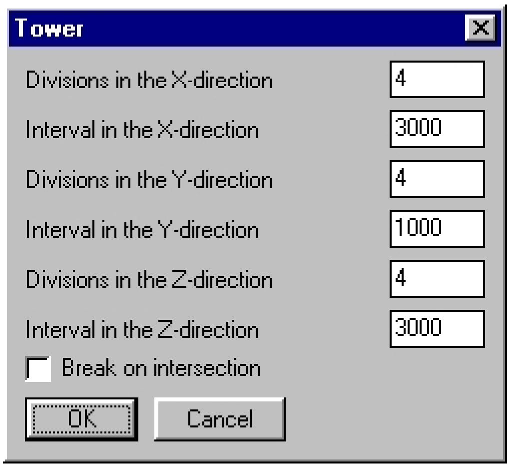

## Tower

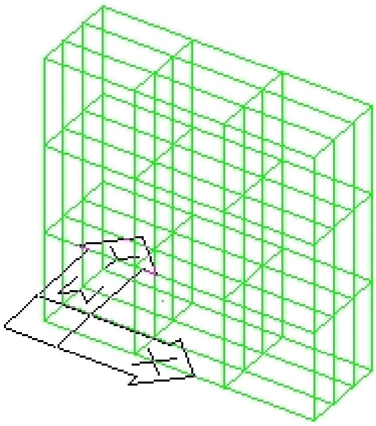
When you click the Tower command button, the following dialog is displayed:



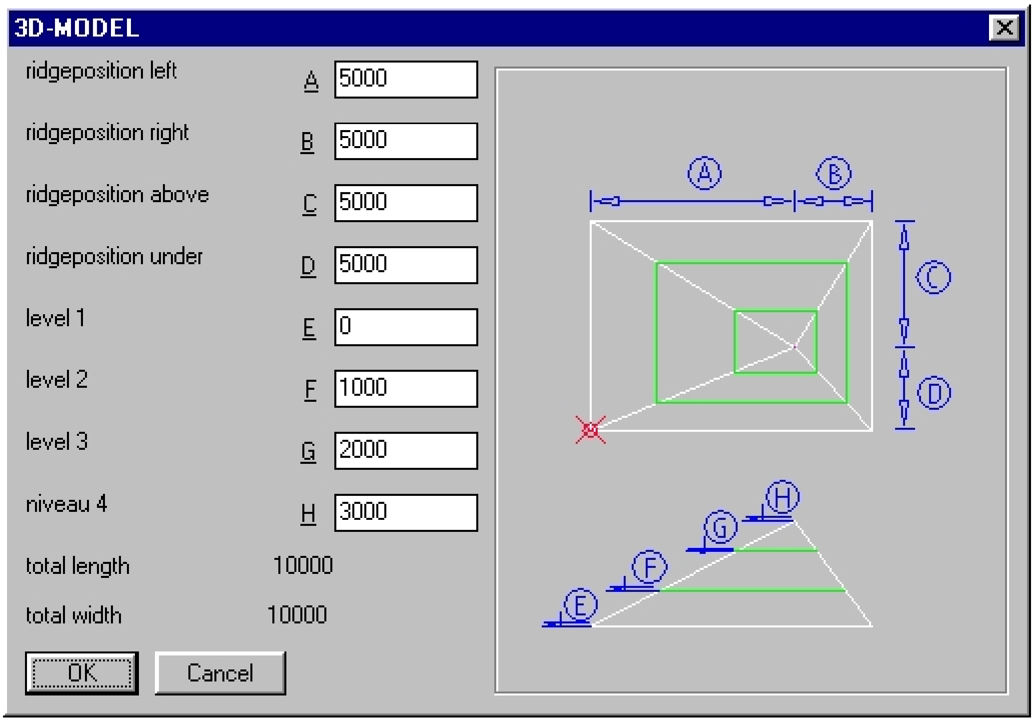




Command: Select the lower-left corner of the structure: **P1**

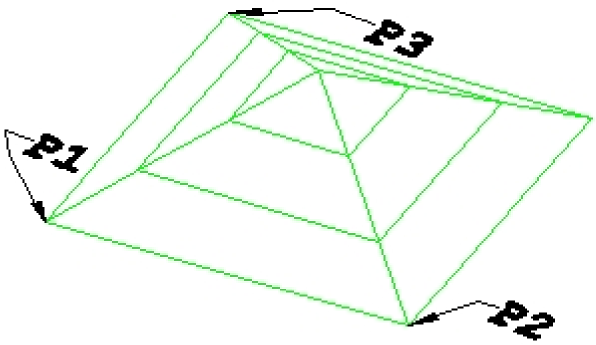


## Pyramid




*Filling in the fields of this dialog is self-explanatory.*

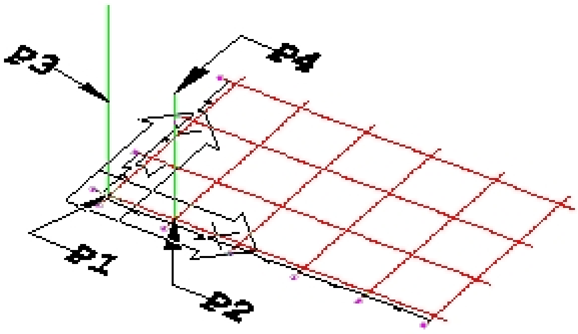



Command: the lower-left corner of the matrix: **P1**

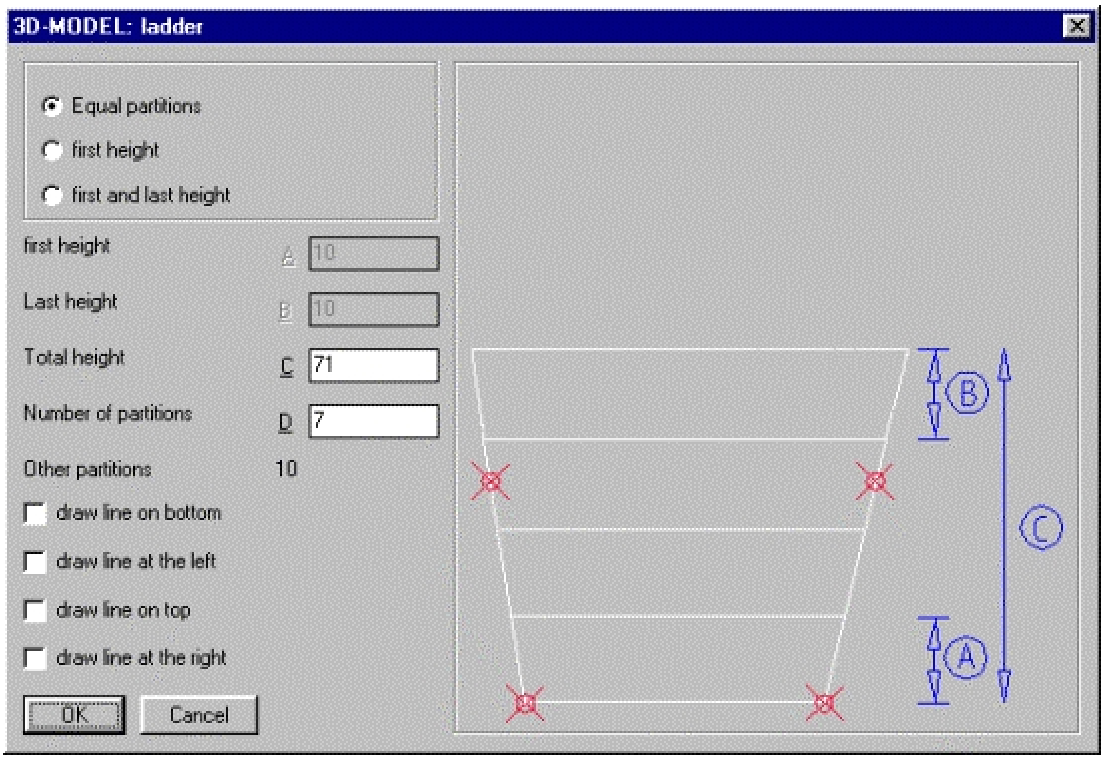
Command: the second point, to the right (X-direction): **P2**

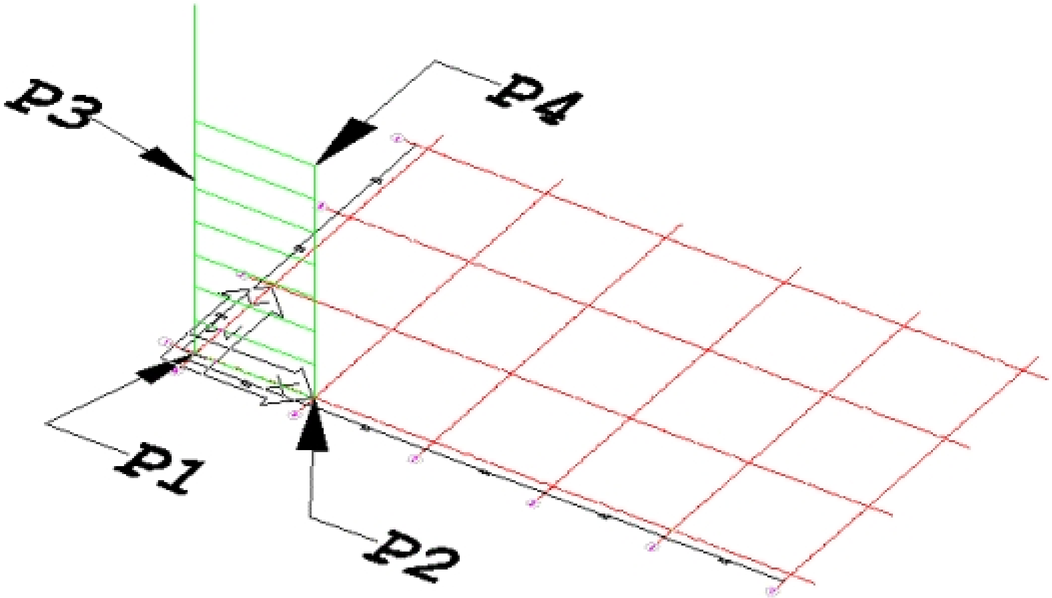
Command: the third point, for the Y-direction: **P3**

## Ladder




Command: the lower-left corner: or **P1**
Command: the lower-right corner: or **P2**
Command: the third point, somewhere on the left beam: to **P3**
Command: the endpoint on the right beam: **P4**





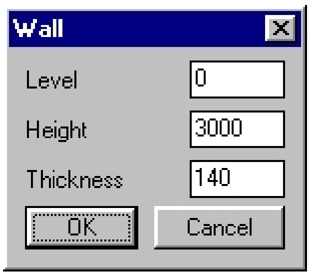

*Result*

## Extra framework

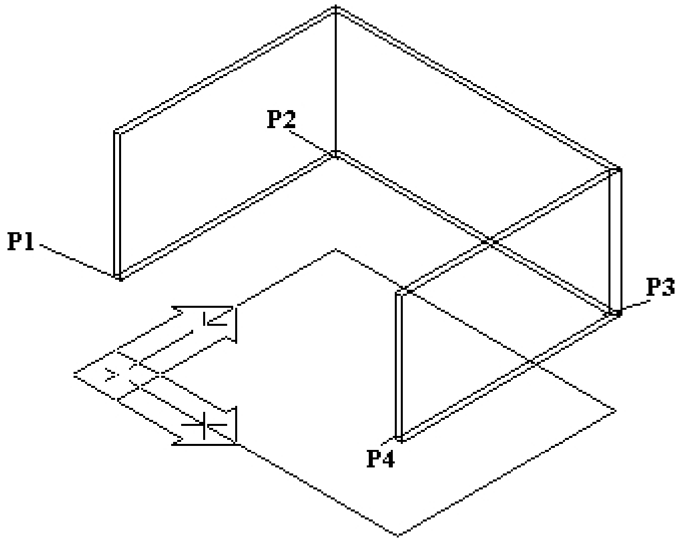
### Wall

When you select the Wall command you can quickly draw a wall.

The input dialog allows you to enter the parameters for the wall.




The wall will always be drawn perpendicular to the active work plane. If you specify a value for elevation, then the base of the wall will be positioned at that height from the work plane.




Elevation Wall base = 2000.

The first part of the wall is only displayed after you have selected the third point. The last part is displayed when you close the command.

Command: Specify start point: **P1**

Command: Specify next point: **P2**

Command: Specify next point: **P3**

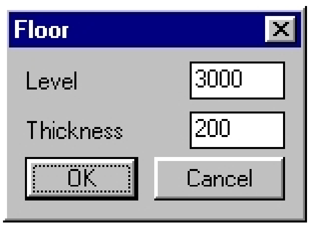
Command: Specify next point: **P4**

Command: Specify next point: **Enter**

## Floor

 After selecting the command `Floor`, the following dialog appears:

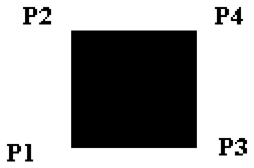
In this dialog we enter the elevation and thickness of the floor.

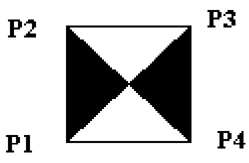


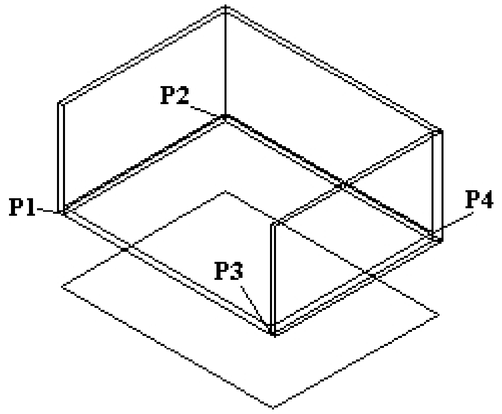

The elevation is the underside of the floor.

When you select points with Osnap (e.g. Endpoint) then the floor will be positioned at the elevation of the points selected.

The sequence in which you select is equally important:



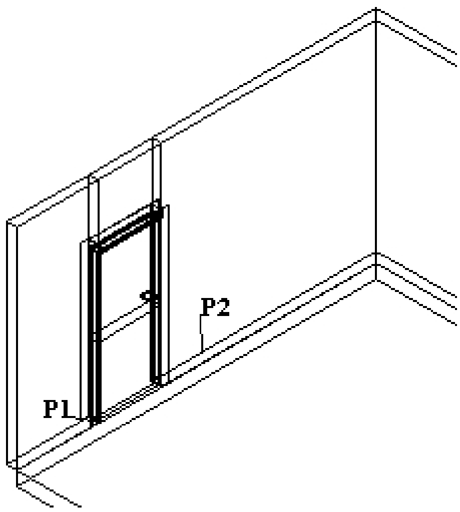

*Result*

Command: Specify first point: _endp or **P1**
Command: Specify second point: _endp or **P2**
Command: Specify third point: _endp or **P3**
Command: Specify Fourth point: _endp or **P4**
Command: Specify third point: **Enter**

## Door

To insert a door you have to enter the parameters in the door-dialog box.

The height and width of the door, whether it is a single or double door, the positioning of the hinges on the wall and the direction of the placement all need to be specified.



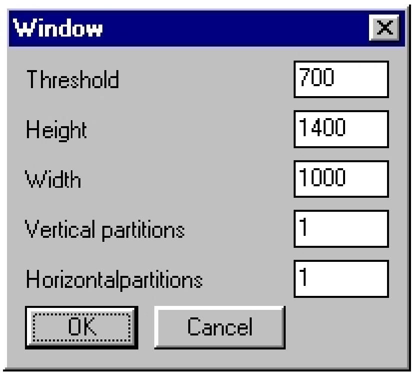

Command: Select the wall to place the door, at the point where you want the hinge: to **P1**
Command: in which direction: **P2**

## Window

Placing a window is almost identical to placing a door.

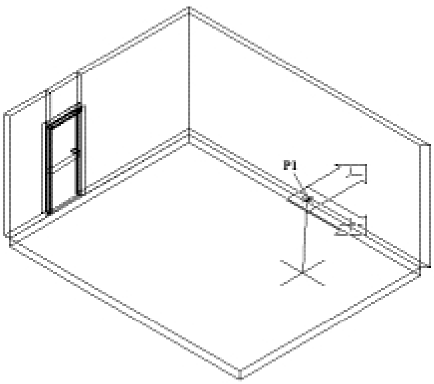
The parameters for inserting a window are the height of the ledge, height and width of the windowpane and the number of horizontal and vertical divisions.

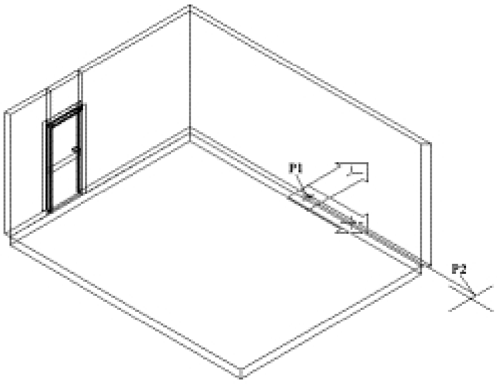
To insert the window you select a point that indicates the beginning of the window. For the second point you give the direction into which the window will be placed.

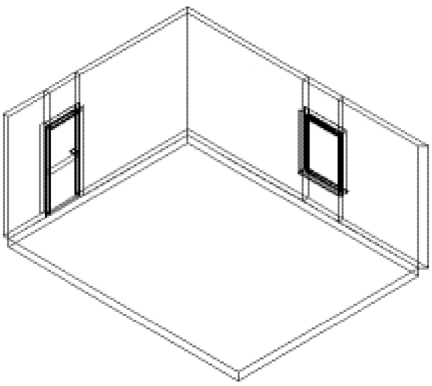



Command: Position the central point of the column: or **P1**

Command: in wich direction: <Ortho-on> **P2**



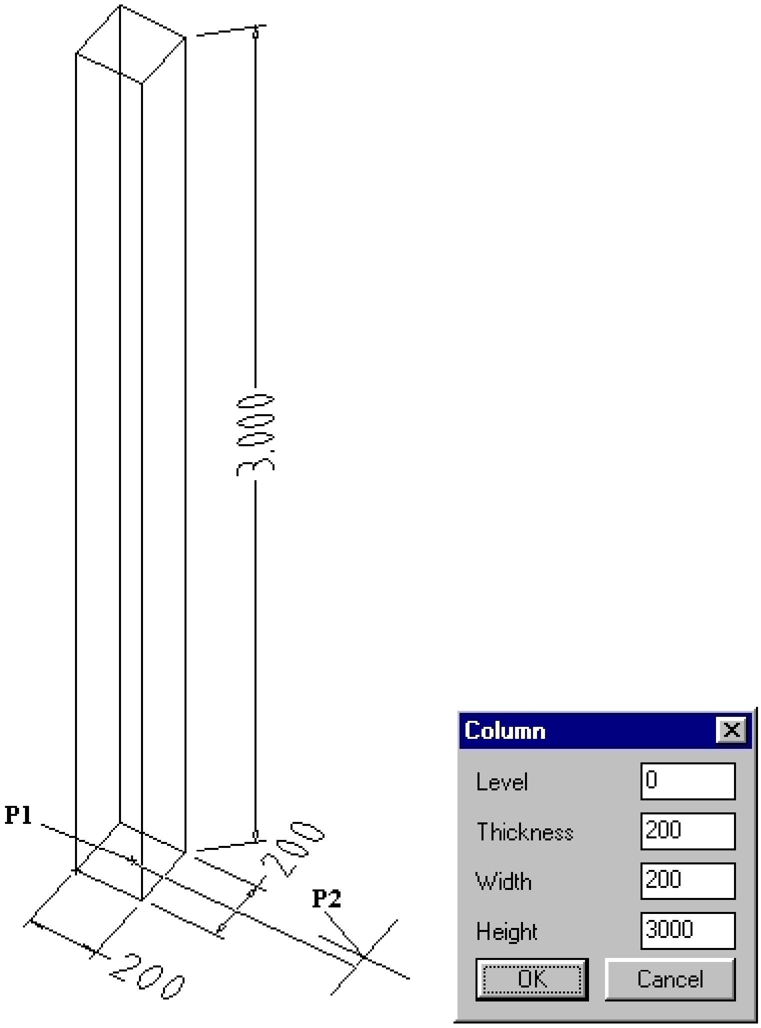



## Column

If you want to draw a column, click the corresponding command-button and set the parameters in the dialog box.

When you specify the elevation then the column starts at that distance from the work plane. A negative value positions the column below the work plane.




Command: Position the central point of the column: **P1**

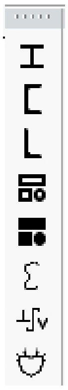
Command: and the insertion angle: <Ortho-on> **P2**

## Drawing profiles

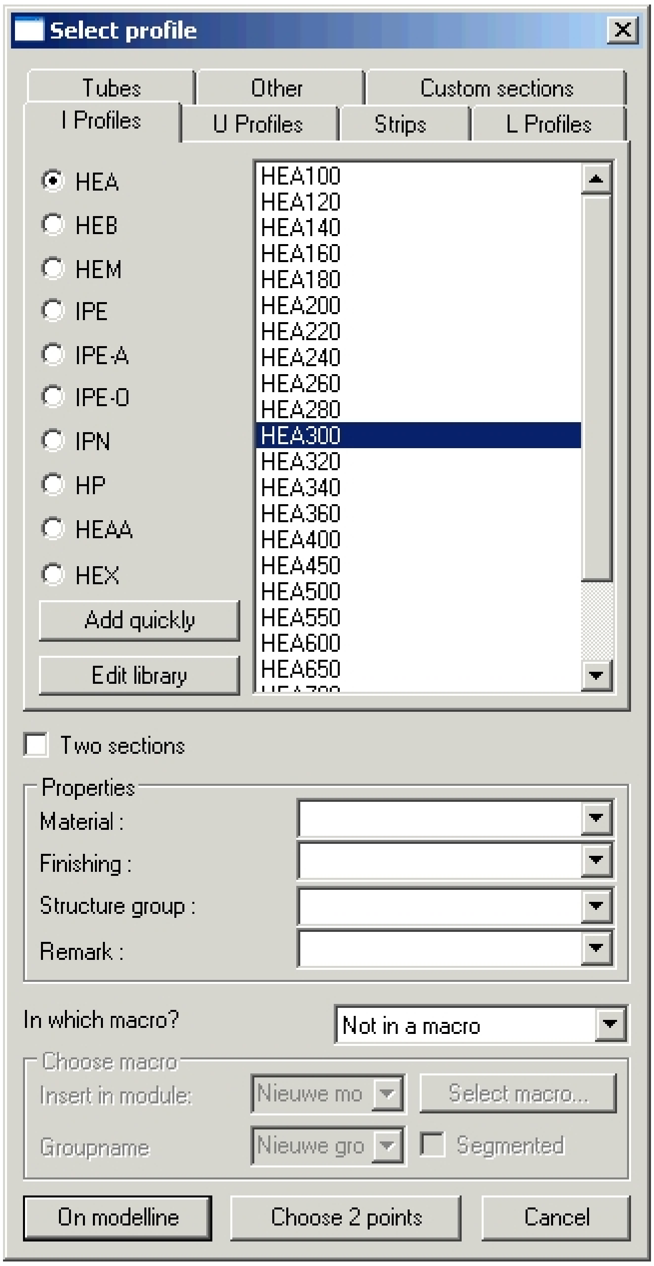
Profiles are based on model lines.

You can place profiles on the following line types:

- Line
- Arc
- 2D Polyline
- 3D Polyline
- Spline



Each of these 8 icons will start the profile-selection dialog box with the corresponding tab as standard selection.




Make your profile choice from the lists.

It is possible to edit the library from here by clicking on the button edit library. See chapter **Editing the profile library** for further information.

Below the profile selection you can enter 4 properties that the new profiles will receive. For each of these 4 properties you have the possibility to make a list with standards, so that you do not have to retype the property each time. You can produce the lists in the dialog box: **Parabuild Settings** > tab **Global** > button **Advanced**.

The bottom settings are only relevant if you wish to add the new profiles to a macro. This is useful if you wish that the profile makes itself dependant on the model line with which it was drawn. If the model line changes, the profile will modify automatically. It is the macro that keeps the link between the line and the profile intact.

This link works only in one direction: the profile is dependent of the line and not reversed.

By way of illustration:

Profile is moved. **Consequence**: Profile automatically moves back to the original placement on the line.

Line is moved: **Consequence**: Profile moves with the line.

If you add a profile to a macro, you also have the advantage that you can for example later modify the rotation of the profile with one click on the button by editing the relevant macro.

The link between the profile and the line is removed if the line, the profile or the macro is removed.

You have the choice to make a new macro or to insert the profiles in an existing macro.

If you choose to add the profiles to a macro, then you must still choose in which module the profile must be saved and also a group name for the profiles. You can choose an existing profile module from the list. If you type a profile module that does not exist, a new one will be created.

The group name allows you to add several profiles under one name, with the goal to be able to modify the placement of these profiles simultaneously.

For example all columns at the left-hand side of the building have the same rotation and the same reference plane, so it is useful to give them one group name e.g. columns-left.

Entirely below there is one remaining option: "segmented". If you activate this option, you will be able to use the macro to break the profile in segments on distances you choose. This option was made for drawing handrails.

For this you must have drawn one long polyline as a base line for the handrail.

After you have modified all options to your wish you must click "on model line" or "indicate 2 points".

With the first button you must select one or more lines.

The last button will draw a new line after you have indicated 2 points.

Finally we see the **Profile Placement** dialog box.

With this dialog box we stipulate the correct position of the profile on the model line on which it is based.

The new profiles were already drawn at this moment. When you modify an option you can directly see the result on the screen.

# Profile placement

Beside the 9 placements on the image there still are 3 other possibilities available in the list:

**Manually**: You can indicate a point yourself on the section or type in 2 coordinates.

**Start point of polyline**: We mean the beginning point of the polyline of the section.

**Neutral axis**: The balance point of the section.

**Rotation**:

The rotation of the section around the model line.

**Reference plane**:

- **WCS**: Bases the placement according the World coordinates of the drawing.
- **Current UCS**: Bases the placement according the current UCS coordinates.
- **Coordinate system**: If you have created coordinate systems in the drawing, and you give these a name, then you are able to choose in the list one of the coordinate systems as a basis.
- **Other**: You must manually select the reference plane on which the rotation of the section will be based. You can indicate as a reference plane the surface of a polyline or one of the planes of a profile that already exists.

**Create profile on one segment of the polyline:**

When you have chosen as model line a polyline that contains several lines (=segments), then you can choose here to place the profile on all lines or on one of the lines. For the last option you must fill in the number of the segment (the first segment has number 0).

## Customised sections

Customised sections are sections the dimensions of which can be determined by the user. If the normal profile-section library does not support a desired shape or section, this can be drawn by the user, given a name and added to the library for general use in all projects.

# Creating a new custom section.

Command : **S3d_NewCustomProf**

First, draw the cross section with polylines. Multiple of polylines may be drawn if necessary! For example, a round tube requires two polylines because the two circles do no touch.

- Select a frontal view of the cross section, and zoom in on it. A small preview will be captured using this view.
- Start the command  to add a customised section.
- On the left-hand side, first select the desired location for the new customised profile, to be more precise, the folder location. This dialog window enables the user to create folders for organising the profiles into groups.
- Enter the name of the new customised profile then click on Ok. This name will be used in the library and as the profile name for all Parabuild applications (part lists, workshop

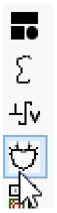
drawings...) when the drawing of the profile is completed.

If folders are to be created or deleted, first open Windows Explorer and go to the directory c:\Parabuildv1\Pb_Lib\User Sections\ (if c:\Parabuildv2 is the installation directory).

This folder contains the contents and representation of the dialog window in Parabuild. Any folder created within this folder in Windows Explorer, will subsequently appear in the dialog window.

# Drawing a custom profile from the custom profiles library.

Command : **S3d_CustomProf**



The new section should appear in this dialog box. Select the section and click on Ok. The rest of the procedure is the same as when drawing normal profiles.

## Editing the profile library

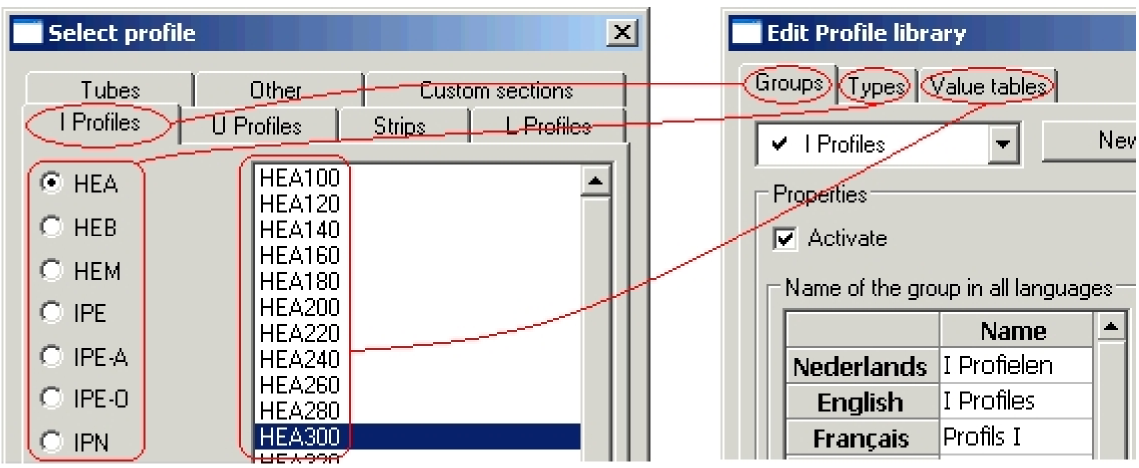
You can open the dialog box for the editing of the profile library by means of the dialog box **Placing Profiles**, with the button **Edit library**.

To make the choice from the large number of profiles more accessible, all profiles were split up in 3 levels.

Profile **groups**: we put together similar sections, like for example IPE, HEA and IPN under the group “I profiles”.

Profile **types**: we put together in one type profiles that have the same characterizations but other dimensions. Example: HEB200, HEB300, HEB400 is all profiles of the type HEB

**Value tables**: contains the table with each individual profile of one type. This table contains all dimensions that determine the size of each profile. These dimensions together with the type are used to draw the profile.



In this dialog box we can edit, deactivate and remove these groups, types and tables.

## Groups

We will stipulate each tab contained in the **Select profile** dialog box.

The first tab contains multiple groups of profile types.

On the upper left you can select from the list one of the existing groups. If you click on the buttons "new group" or "remove group", this list is automatically adapted.

If you select a group from the list then the options of that group are reflected below:

**Activate**: You can deactivate the group (becomes invisibly in the **Select profile** dialog box) without having to remove it.

**Name of the group in all languages**: Give a name for each language.

**Illustration image**: this small image is placed before the name of the tab so that the group can be recognized rapidly. (.bmp files are being searched in Parabuild\Pb_lib\Prof\)

If you yourself have modified or changed groups, then you can also create a new icon for each group that directly starts the profiles dialog box with the group as standard active.

This is possible using the next line as a command of the icon:

(S3d_CreateProfDlg "I profiles")

This is a command that receives a text (in this case: "I profiles"). You must replace the text by the name of the group that you want activated as default. The name must stand between the quotation marks and must correspond exactly to the name of a group that exists. Opening and closing brackets in the before and after the command are also required!

## Types

Each group can contain several types of profiles. This list contains all types and we indicate for each type in which group they belong.

For each profile type the following options exist:

- **Activate**: make the type invisibly without having to remove it.
- The name of the profile type in **4 languages**.
- A small **image** as illustration (.bmp files are begin searched in Parabuild\Pb_lib\Prof \).
- The **group** to which this profile type belongs. The number of types that you can place in one group is in itself infinite, but keep into account that the profiles dialog box can only show a maximum of 30 types, and only 10 types will fit on a dialog box with a reasonable size.
- The **value table** that represents this profile type.

## Value tables

At the top we have a list of value tables.

The name that is granted to the table must be unique (two tables with the same name is impossible).

A value table is always associated with a section type. This section determines the form of the profile. In the list of sections the most occurring section are available as standard (I profile, U profile, …). Below these there are also custom made sections, each contained in a .dwg drawing. You can read a bit further in this manual about how to add to these custom sections yourself.

The value table itself contains the data of each profile. Each row represents a profile. Each column is a property of the profile. We will look at each column of the table:

**Sysname**: The system name of the profile that Parabuild needs internally for the unique recognition of the profile. You must fill in here a text that is not already used by another profile in the table. The prefix (name of the table) is put before this text to obtain the complete system name. This system name is not used in the bills of materials or dialog boxes: Parabuild uses this name internally only. This system name is required because Parabuild needs to have a unique name that is independent of the name of the profile in other languages.

**1/O**: You can deactivate the individual profile so that it becomes invisible in the **Select profile** dialog box without having to remove it. Parabuild will nevertheless recognize the profile if it was drawn in a drawing. Therefore if you deactivate the profile here it only has influence on the **Select profile** dialog box.

**Nederlands, English, Français, Deutsch**: The name of the profile that Parabuild uses everywhere for identification (dialog boxes, bill of materials, shop drawings,…).

All columns that follow hereafter are columns that define the dimensions of the profile (height, width, thickness,…). These columns can differ depending on the type of section that was chosen for the table.

The table has the following functionalities that are useful during input/editing:

- **Columns/rows**: You can modify the number of rows and columns. To modify the number of rows is of course important for adding more profiles. Also the column names can be modified here (only usefully if you yourself have added custom section types, see further in the manual).
- **Mouse Right click somewhere in the table:** opens a context menu which offers several functionalities such as insert row, insert column, …
- **CTRL+C or copy**: You can copy one or more fields to the Clipboard. You can select several fields by keeping the left mouse button pressed and by dragging the mouse.
- **CTRL+V or paste**: You can paste one or more fields of the clipboard in the table. This is also possible with tables that you have copied to the clipboard from other programs (for example Microsoft Excel).
- **TAB** : to go easily to the next field in the table.
- 4 arrows ↓ → ←: for moving to other fields.
- Function key **F3**: it opens the search/replace dialog box with which you can look up texts in the table. It will search for the next occurrence if the dialog box is already open.
- Function key **F5**: Quick replace button for replacing the next occurrence of text.

## Producing intelligent sections

You can produce (intelligent) sections yourself that can be reused for other dimensions.

This is possibly if we draw a section by means of geometrical rules.

You must therefore have knowledge of the part of Parabuild that allows producing intelligent connections.

The logic of a section is drawn in the same way as a connection in 3D.

However now we do this just in 2D (we draw each polyline flat on the world coordinates).

The values of the value tables are used on the section to calculate the real coordinates.

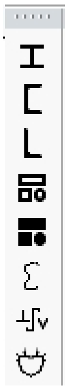
This is the working method:

- You start with an empty drawing (or you can reuse one of the pre-made sections).
- You must save the drawing in the following folder: Parabuild\Pb_lib\Prof\
- The section can contain only polylines (no lines or anything else).
- All the polylines must be closed.
- Absolutely no polyline can cross another polyline.
- There is no restriction in the number of polylines in one section. The polylines can stand separate from each other (multi-beam).
- You must add one macro in the drawing that defines all the polylines. It is best to create one module for each polyline. Now you define the form of the polyline(s) by adding geometrical rules to the module. If you do not know how to do this, then you must read the manual for producing connections.
- You must create a new value table that will be used in combination with the custom section that you just produced.
- In the value table you must make one column for each adjustable dimension contained in the macro of the drawing. If the name of the column corresponds to the name of a dimension in the macro, then the value from the table will be used as a value for that dimension when calculating the real section.

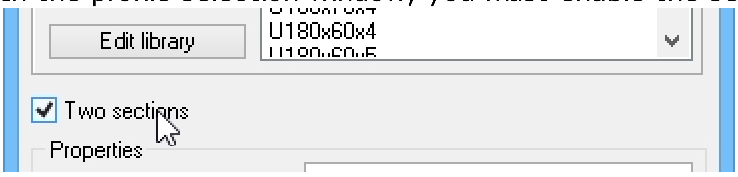
## Drawing a morphed profile

It is possible to draw profiles with 2 different sections. The transition between the two sections is drawn.

To draw this type of profile you should start one of the regular commands to draw a profile:

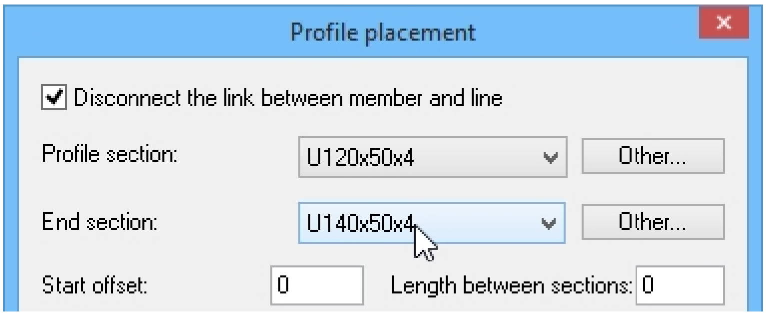


In the profile selection window, you must enable the setting **Two sections**.




Then you draw the profile the normal way.
In the last window where you can change the location of the profile it is now possible to

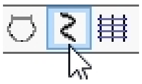
enter two different sections.




A prerequisite for this function is that the two sections have the same number of segments. Otherwise, a logical sequence cannot be created; The transition is being made from segment to segment (a segment is a piece of straight line or a curve piece of a polyline).

Due to limitations in AutoCAD Parabuild can't convert morphed profiles to 3D Solids. The command for exporting to 3D Solids will not export these profiles.

## Drawing a Helix



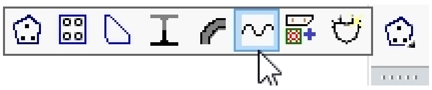
This command allows you to draw a profile on a 3D spiral axis. The axis of the profile is drawn using straight segments, as an approximation of a spiral. The accuracy of this approximation is set by the user.

This command will draw the following entities :

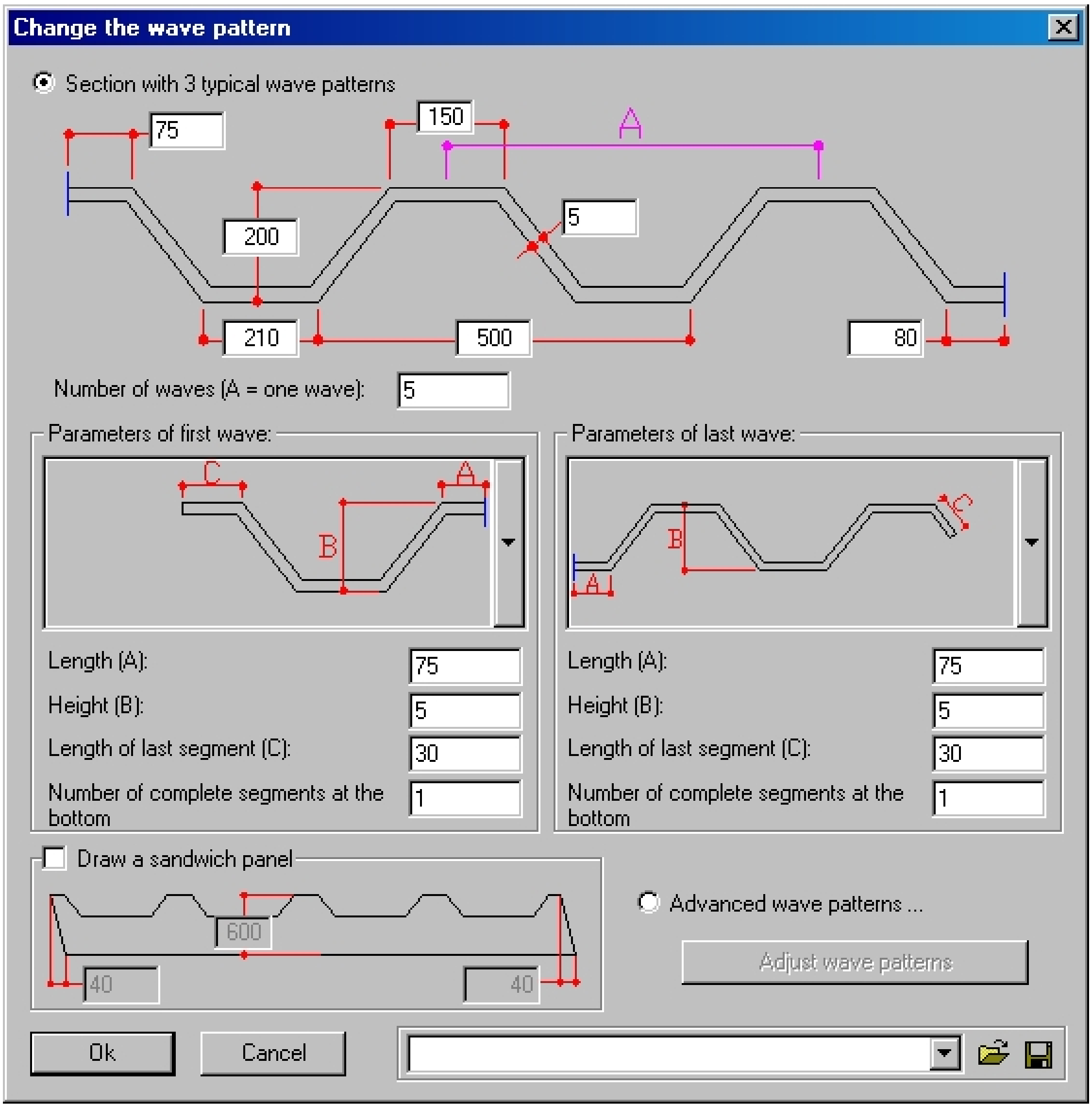
- The spiral model line
- The spiral profile
- A macro that connects the spiral profile to the model line, and calculates the twist angle of the profile.

To change the location and rotation of the spiral you need only move/rotate the model line. To change the dimensions of the spiral itself, you need to use the Properties of the model line.

## Sandwich panels / corrugated sheet metal sections

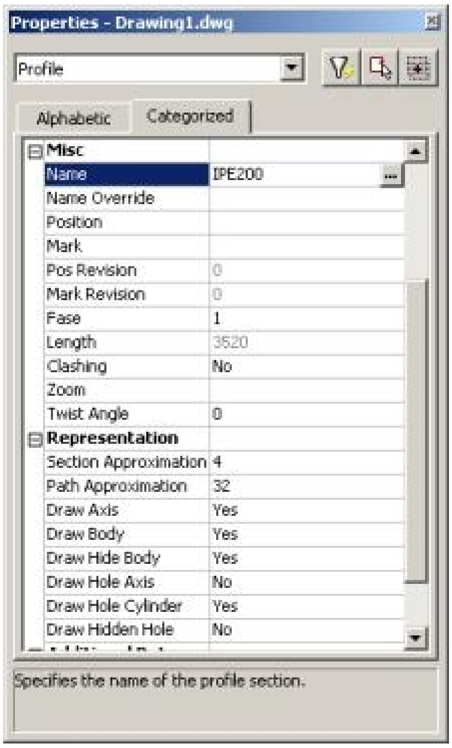


This command will draw the section of andwich panels or corrugated sheet metal sections as one polyline. This polyline can be used as a section to draw a profile. You can choose between 3 types of sections with help from the illustrations. The first wave, the last wave and the repeating wave in the middle. If you want to create more waves, you have to use the option **Advanced wave patterns**.




## Manipulating elements

### AutoCAD Properties




The properties of all objects drawn using AutoCAD can be changed using the Properties dialog window (lines, polylines, dimensions, solids,...).

The properties dialog window can be started in four ways:

1) The command **properties** on the command line

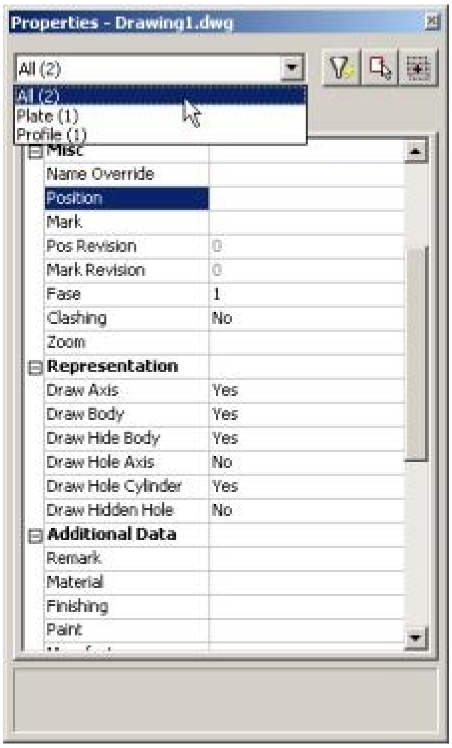
2) At the top in the toolbar **Tools** -> **Properties**

3) Select an object, using the right-hand mouse button, click on the object, and then select **properties** in the drop-down menu

4) Using the left-hand mouse button double click on an object.

The properties dialog window is no ordinary dialog window. This dialog window can be retained on the screen. When the dialog window is on the screen, all other commands can be carried out without interfering with the properties dialog window. This dialog window can also be integrated into the icon menus by moving it over the menus.

Just as with AutoCAD objects, Parabuild objects can also be edited in the same dialog window. When the dialog window is displayed, one or more objects can be selected and the common properties of the selected objects will be displayed immediately. These properties can then be changed as required.




Imagine that a plate and a profile are selected at the same time. Some of the plate properties are not found in the profile, and vice versa (a plate has a thickness, a profile does not). Only the properties that the plate and the profile have in common will then be displayed. The property ‘thickness' is not displayed, but the position number, for example is. This is however simple to bypass: At the top left of the dialog window a number of objects of the same type can be changed. In this case one plate or one profile (see illustration). Using this method, all objects of the same type can be taken from the selection in order to edit the special properties.

The properties dialog window contains another button that offers a great number of possibilities: **Quick Select**, the button with the funnel.

Quick Select allows the conditional selection of objects. For example, all HEA200 profiles can be selected from a drawing with a few mouse clicks, or all plates/bolts/profiles from revision 0.

Once all elements have been selected, the properties of the selected objects can be changed using the properties function.

This dialog window can also be used to view and change dynamic properties. Dynamic properties are properties applied by the user. For further explanation see the chapter Dynamic properties.

# All properties explained one by one

## 1) Profiles

**Name**: The name of the profile (E.g.: HEA200). Edited by clicking on the button.

**Name override**: The replacement of the normal profile name. This replacement name will be used in all part lists and workshop drawings. An application example is an handrail connector:

instead of using the name 'BR33.7x2.65' this can be replaced with: 'Handrail connector 33/90°'

**Position**: The number automatically given to elements by Parabuild. Elements that are the same, (same dimensions, holes, sections) are given the same position number. These positions are partially adaptable with prefixes and similar, see below.

**Mark**: The number that Parabuild gives to every mark. When two marks in one drawing are the same, Parabuild gives them the same mark number (To be the same, the marks must contain the same position numbers and should be welded at the same locations). Partially adaptable with prefixes and similar, see below.

**Pos Revision**: Not directly adaptable. Can only be edited via the Revisions) system. The element will be given the current revision, not only on being created, but also when changed. Example: when a hole is added to a profile, the profile is automatically updated to the current revision.

**Mark Revision**: The same as pos revision. Example: when a welded profile is moved, all parts of the same mark will be given the current revision.

**Phase**: Adaptable. When a profile/plate is created an element will be given the current phase. The phase then remains unchanged (unless changed here manually). The current phase is located in the Global settings) dialog window.

**Length**: The length of the profile. Not adaptable here.

**Clashing**: Adaptable. This is set to yes or no and refers to whether the profile is currently clashing with other elements or not. This property is automatically set by the clash control, and is only included in properties for search objectives. For more information read the chapter Clash control.

**Zoom**: This is not really a property but an action: by repeatedly clicking on the button, every element in the selection is zoomed in on one by one.

**Twist Angle**: Adaptable: determines the angle that the profile has to turn over the length of the profile. This should only be used for profiles on a spiral path or other special 3D-forms.

**Section Approximation**: Determines how accurately the cross section of a profile should be drawn. This has a particular influence over curves in a cross section, for example when the rounding between a beam web and flange is applied.

**Path Approximation**: Determines how accurately the profile is drawn over its path. This has a particular influence over curved profiles.

**Draw Axis**: Draws the axis of the profile.

**Draw Body**: Draws the complete 3D-model of the profile.

**Draw Hide Body**: Draws the complete 3D-model of the profile during the HIDE command.

**Draw Hole Axis**: Draws the axis of every hole in the profile.

**Draw Hole**: Draws the cylindrical path of all holes in the profile.

**Draw Hidden Hole**: Draws the cylindrical path of all holes in the profile during the HIDE command.

**Remark**: Adaptable. Can be used in a variety of ways. This field is maintained for every element. It has its own column in the part lists, and can be used for sorting part lists and workshop drawings. This property has no further influence.

**Material**: Adaptable. This field also has its own column in the part lists and can be used for sorting, but this has a direct influence on the position number (and consequently the mark number). Two elements that are identical, but have another material assigned to, will be given another position number. This enables a total categorisation of different materials in part lists and workshop drawings. Example: two profiles are exactly the same except for their material property. If they were given the same position number, only one workshop drawing would be made for both profiles, and it would never be known how many, and in what material these 2 would have to be manufactured. Therefore, different materials should always have a different position number.

The weight factor of every material can be changed in the Global settings) (See advanced). Parabuild will use this weight factor to calculate the weight in the part lists.

**Finishing**: Adaptable. Reacts the same as 'Comment'.

**Paint**: Adaptable. Reacts the same as 'Comment'.

**Manufacturer**: Adaptable. Reacts the same as 'Comment'.

**Struct group**: The same as Comment. Structure groups can also be displayed in the balloon captions of 3D captions and 2D views.

**Pos Prefix**: Determines the prefix of the position number. For further explanation, see Numbering of elements.

**Pos Suffix**: Determines the suffix of the position number.

**Pos Startnumber**: Determines the start number of the position number.

**Mark Prefix**: Determines the prefix of the mark number.

**Mark Suffix**: Determines the suffix of the mark number.

**Mark Startnumber**: Determines the start number of the mark number.

**Output**

- **Weight method**:
    - **Default** (%) : The weight of the complete length of the profile will be calculated, without subtracting the holes and cuts. In the next property you can adjust the percentage of this value that should be used.
    - **Cutted**: The cuts in the profiles are subtracted to calculate the weight.
    - **Cutted and drilled**: The cuts and and the holes are subtracted to calculate the weight
    - **Fixed value** : The value you enter in the next property will be used as the weight for this element in kg (This value will be taken over in the BOMs without any adjustments).
- **Weight parameter** : This property is being used in combination with the above properties "Default" and "Fixed value".
- **Skip BOM** : If you set this property to "Yes", this element will not appear in the BOMs.
- **Skip 2D view/3D tags** : Adjust this to skip this element for the 3D tags or the 3D tags + 2D view.
- **Skip drawings** : You can skip only the Pos drawings, only the mark drawings, or both the Pos and Mark drawings.

**User properties**: All properties applied by the user in Dynamic properties) can be edited here.

## 2) Plates

Plate-specific properties.

**Thickness**: Changes the thickness of the plate.

**Trac Plate**: Changes the plate to normal or to end/baseplate. Endplates/baseplates are green, the normal plates are blue. End/baseplates are given an extra plate view in the assembly drawings.

## 3) Bolts

**Orientation**: The orientation of the bolt can be changed by clicking on the button.

**Normal Display**: The detail in which the bolt is drawn.

**Hide Display**: The detail in which the bolt is drawn in the command HIDE.

**Display Axis**: Draws the axis of the bolt.

**Added length**: Minimum length of the bolt = net length (material length) + nut thickness + thickness of all washers + added length. This minimum length will be used to extract the actual bolt length from the part database. The actual bolt length shall be equal to, or longer than the minimum length of the bolt.

**Hole tolerance**: The diameter of the bolt's hole = hole tolerance + diameter bolt.

**Bolt length**: The effective length of the bolt taken from the Bolt parts database). The length of the bolt can be changed temporarily for drilling extra holes. Method: Bolt penetrates one flange of a tube; it needs to penetrate both tube flanges. Increase the bolt length enough to pass completely through the tube. Now check for new holes. The bolt will be given the correct length and the second hole will be made.

**Assembly**: Use this to select one of the Bolt Assemblies). A great number of properties are dependent on the bolt's assembly.

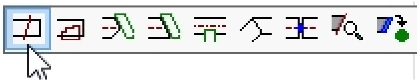
**Diameter**: Select one of the available diameters from the standard bolt (the standard is extracted from the assembly).

**Washer 1/2, Nut 1/2**: Each part can be turned on or off separately. The standards used for these parts are recorded in the bolt assembly.

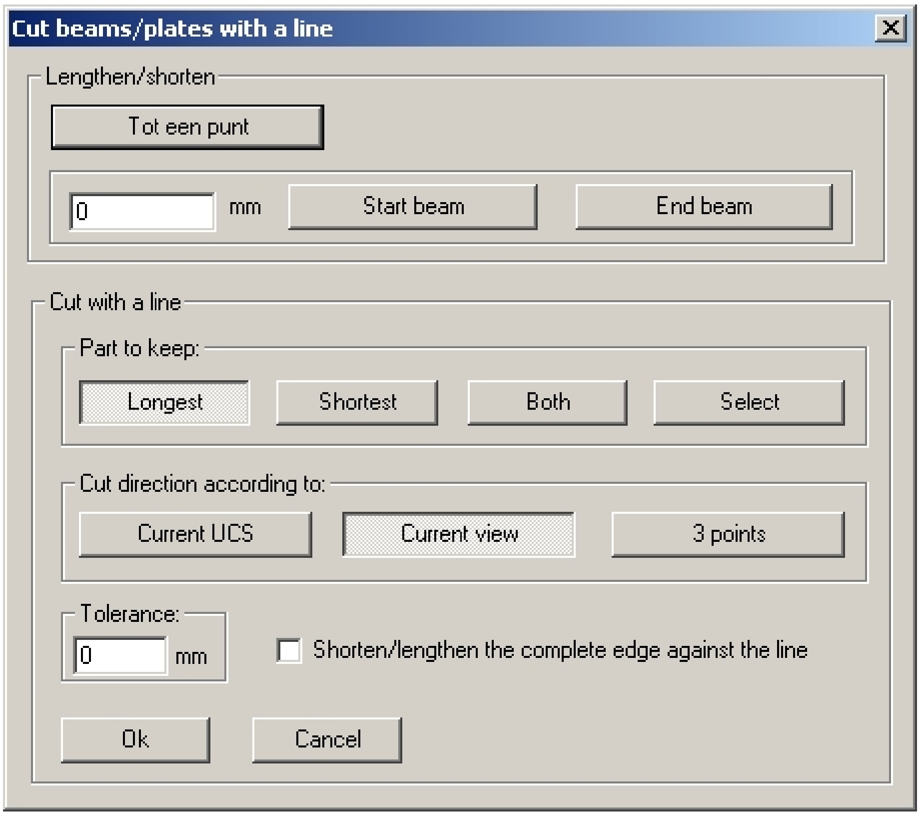
**Filling washers**: If the length of the bolt thread is not long enough (according to the standard) then it will not be possible to tighten the nut onto the work. This will also be visible in your drawing. Turning on filling washers will add sufficient washers enabling the bolt to be completely tightened.

## Cutting with a line

Command : **S3d_LineCut**



The following dialog window will be displayed after starting the command.




**Shorten/Lengthen**:

First you enter the distance to be shortened or extended, then you choose from **To Point**, **Begin beam** and **End beam**.

When you select To point, you have to set the reference point and the beam will be cut to that point.

**Start beam** will shorten/lengthen the beam in function of the value you entered at the origin of the beam (this is the side where the little triangle is drawn).

**End beam** does the same as Begin beam but at the opposite end of the beam.

**Cut using a line**:

There are several possibilities:

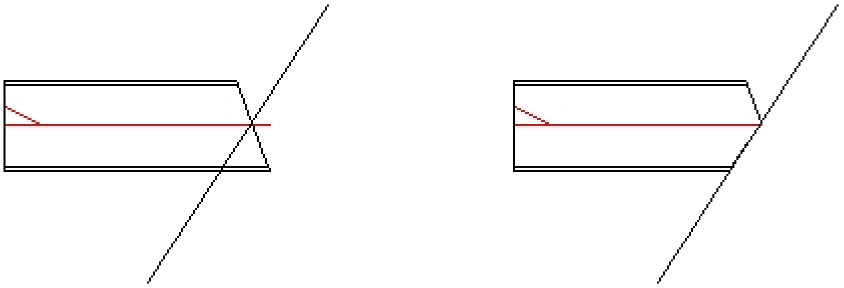
Part to be kept: Longest, Shortest or Both. Longest and Shortest are self explanatory. Both means that Parabuild makes two beams out of the original beam.

**Orientation cutting plane**:

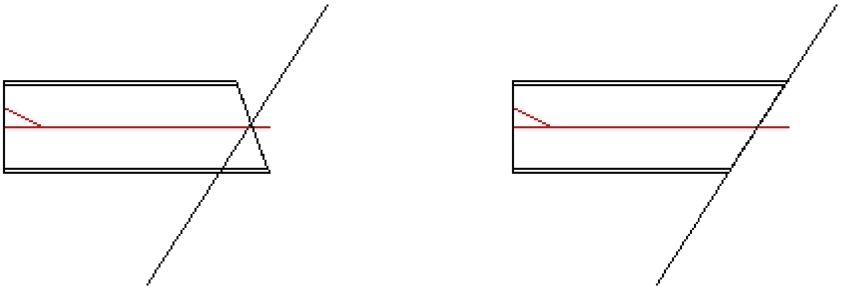
- **Current UCS**: Cut along the current work plane (UCS).
- **Current view**: Cut according to the current view.
- **3 points**. You are requested to set/select 3 points. The cut will be according to the plane defined by these points.

**Tolerance**: You can also set the tolerance.

**Shorten/lengthen the entire edge against the line**:



*An example of cutting with a line using the setting **Complete edge** off.*

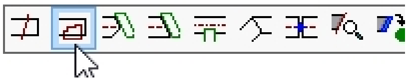


*An example of cutting with a line using the setting **Complete edge** on.*

It will be noticed that when the setting **Complete *edge*** is off, any existing cuts remain. When the setting is on, the complete end is cut off up to the line and any existing cuts are erased. This setting does not have any influence on plates as plates do not have an end.

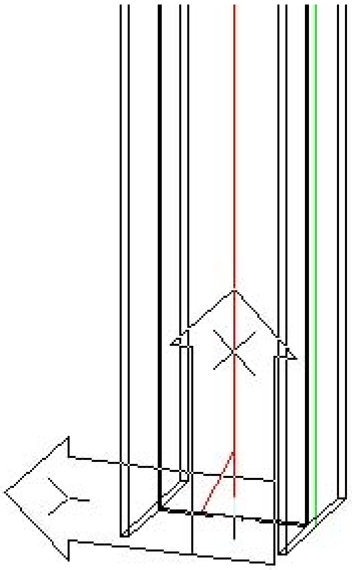
## Cut with polyline

Command : **S3d_CutByPoly**

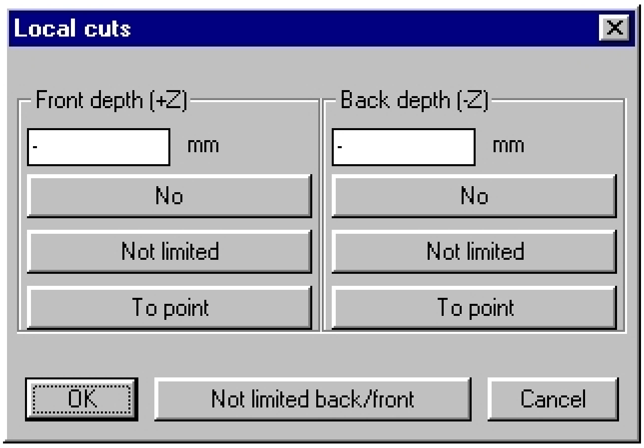


With this command you can make cut-outs of beams or strips using a polyline.

Cutting with a polyline always uses the current UCS. Before you initiate the command, you have to set the UCS (in this example UCS of the beam):



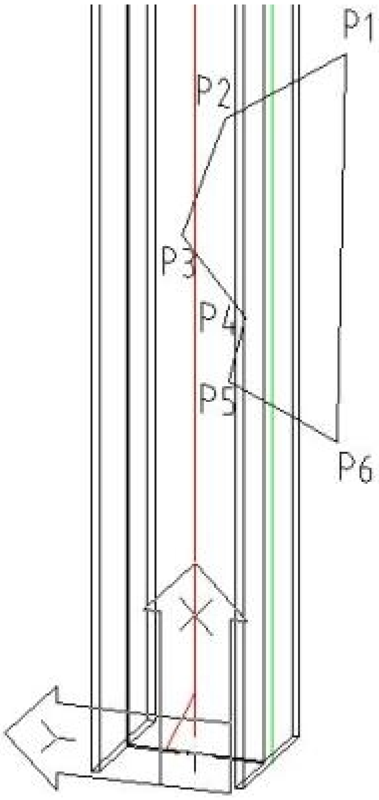
When you activate the command, you have to select the beam (or strip) to be cut. The following dialog is displayed:



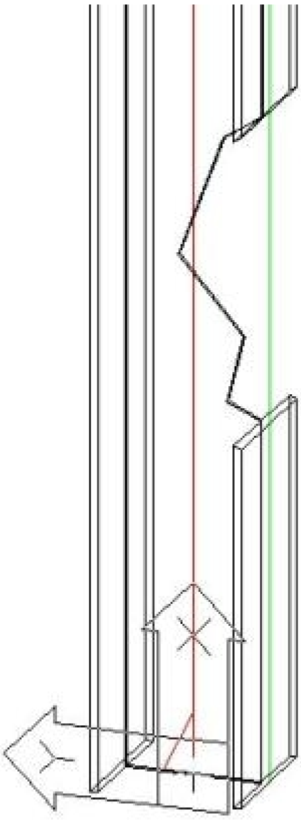

In this dialog the depth of the cut along the Z-axis needs to be entered. At first, we do not take this into account and select **Unlimited front/back.**

Now you have to draw the polyline.

You enter the points like in the following example and press **Enter**:



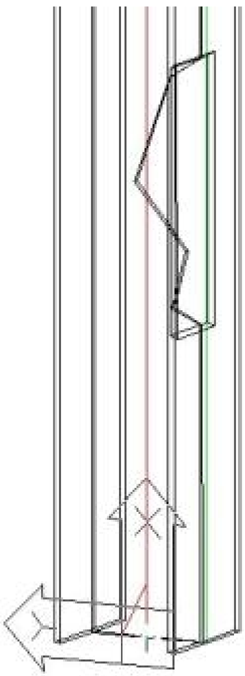

*Drawing Polyline*



*Result*

The cut is made following the positioning of the UCS, even if the view was different while drawing the polyline. To cut at an angle, we set the UCS in the required angled plane before making the cut.

Now we'll take a closer look at the **depth front/back** in the dialog:

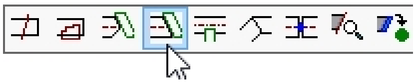


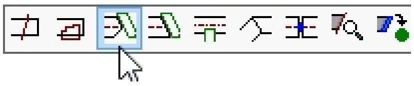
In this example we used a depth front (+Z) 120 and a depth back (-Z) 30.  We draw the same polyline. In the result you can see that from the UCS in direction +Z 120 mm was cut, while 30 mm in the -Z direction.

You can set both depths independently.

## Cut against element

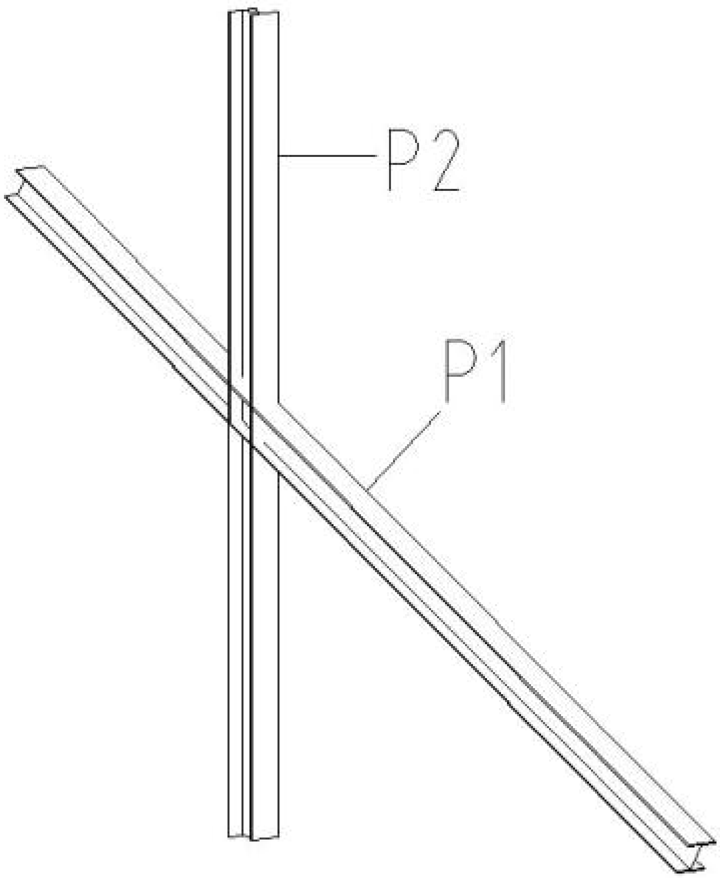
Command : **S3d_PrPlCut** and **S3d_AddPrPlCut**



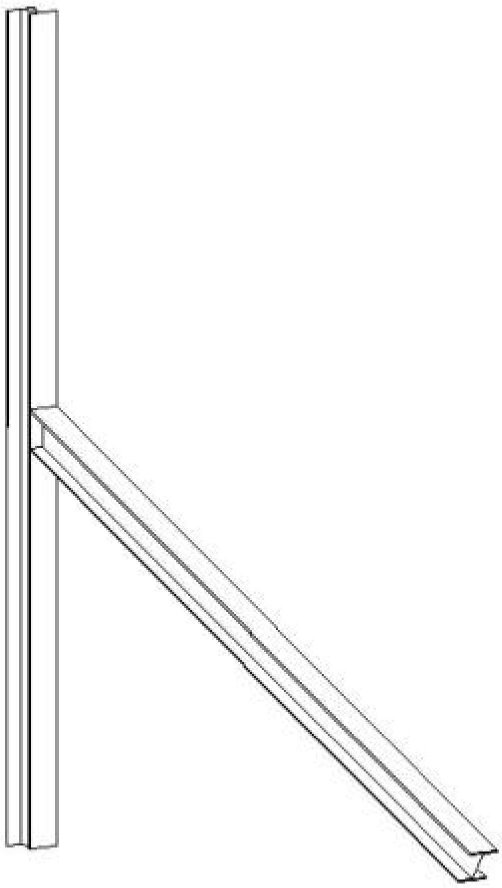


We can cut one or more beams or strips along a plate or another beam. This command is similar to Cutting with a line). In this command, another element replaces the line.

When you activate the command, you need to select the beam to be cut:  Select P1. The plate or beam that determines the cutting line: Select P2.




*Beam to be cut*

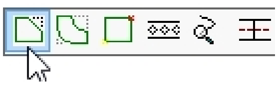


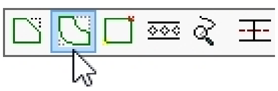
*Result*

The beam (P1) to be cut is automatically extended instead of shortened when it does not reach the other beam.

## Chamfer and Fillet

Commands : **S3d_Chamfer** and **S3d_Fillet**





# Chamfer

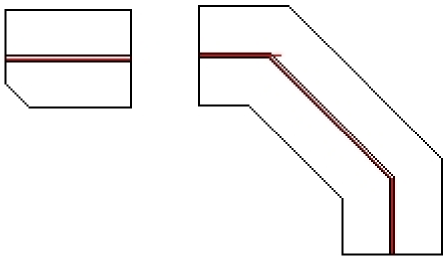
Using the chamfer command  corners of plates and profiles can be cut off.

When you start the command, 2 distances are requested. These are the cutting distances from the corner. Then 2 lines are asked, these must be 2 lines from profiles or plates from which you want to cut off the corner.

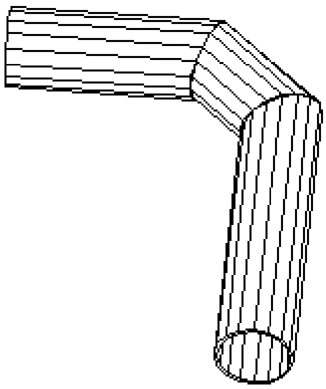
You can also apply this command several times to the same corner: suppose you have cut off a corner of a plate with distances 20,20. If you subsequently want to make a corner of 10-10 you start the command again and indicate the same lines.

With this command you can also 'stick corners on' instead of cutting them off: for internal corners instead of external corners.

This can also be used for changing the axis of a profile or placing a (welded) connection piece between 2 profiles. Examples of this function are shown below.



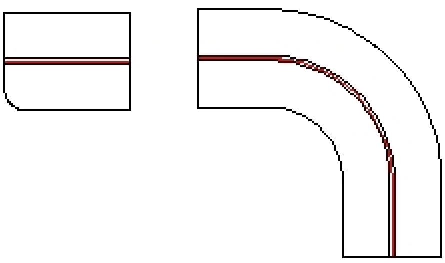
*The illustration on the left shows the chamfered corner of a profile, and on the right an example of a chamfered axis.*



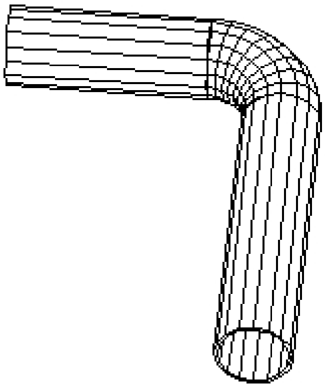
*An example of a connection piece between two tubes using the Parabuild chamfer command (A third tube in the middle was created after two tubes were selected).*

# Fillet

The fillet command can be used to round the corners of plates and profiles. This can also be used to apply a radius to an axis, or insert a connection piece between two profiles. Examples of this function are shown below.



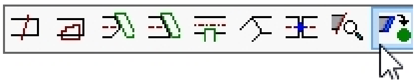
*The illustration on the left shows the rounded corner of a profile, and on the right an example of a rounded axis.*



*An example of a connection piece between two tubes using the Parabuild chamfer command (A third rounded tube in the middle was created after two tubes were selected).*

## Adding an intelligent cut to a macro

Command : **S3d_AddMacroCut**



Just like we can add profiles to a macro, we are also able to add cuts to a macro. The advantage of this is that the cut will adapt automatically if one of the base profiles of the cut changes. Also the options of the cut can always be modified after the cut has been drawn, without the cut needing to be drawn again.

 After you start the command you get a dialog box on the screen.

First of all at the top you must choose a type of cut because below are some options that change dependent on which type of cut you choose.

**Profile against profile/plate/plane**:

This is a straight cut against a profile or a plate.

**Equal angled cuts**:

With this cut both profiles are extended or are shortened so that they connect to each other at the same angle.

**Cutout profile/plate**:

This cut cuts out the form of one profile in another profile or plate.

**Against plane with 3 segments**:

This cut is used for connections that have clearance cuts.

**Chamfer**:

With this cut we cut a chamfer from a profile using two distances. You must select two planes as a basis.

**Fillet:**

With this cut we cut a fillet from a profile using a radius. You must select two planes as a basis.

At the bottom there are some options that you must use to choose in which macro and in which module the cut must be added to.

As soon as you click on OK you are asked to select the components or planes.

Afterwards the cut is produced and stored in a macro. You can revise the macro to modify for example the offset. If you remove the macro, the cut will remain but it will no longer be intelligent.

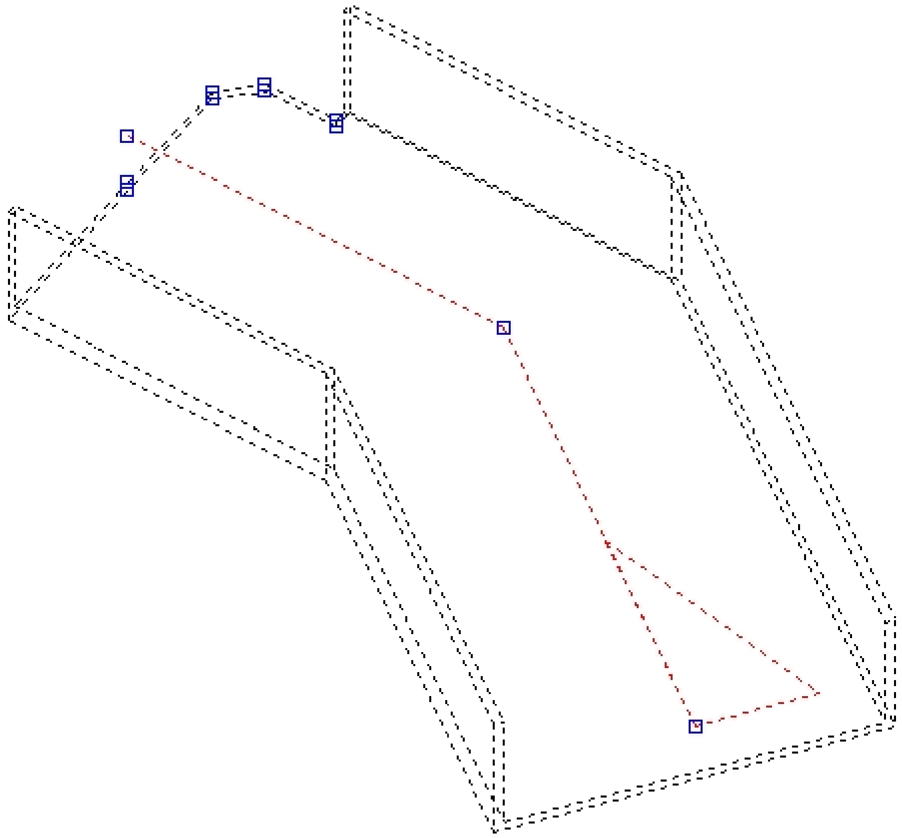
## Stretch

Parabuild objects can be stretched in two ways, both of which are described below.

### 1) the AutoCAD Grips

If the GRIPS variable is set to '1' and an element is selected a number of small squares will appear on the element.

Bolts only have two small squares and will only allow the bolt to be moved. Plates have a small square on each of its corners. Profiles always have a small square at the start and at the end of the axis allowing the profile to be lengthened. In addition, profiles also have small squares at certain corners of cuts enabling the cuts to be edited. Curved profiles also have small squares on the axis at the place of bending and curving to alter the curve or the bend of a profile. Using the left-hand mouse button, click on a square and move the mouse, this will stretch the element. Click again on the left-hand mouse button and the modification will be applied.



*The profile illustrated has small squares or grips at the start, at the end and in the middle at the place of its bend. It has various others at the cuts.*

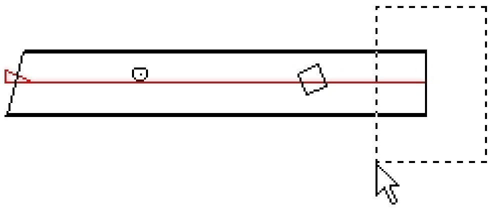
## 2) The Stretch command

This command can be started by typing **Stretch** on the command line or by selecting **Modify** > **Stretch** in the toolbar at the top.

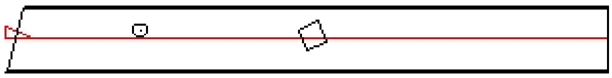
This command allows multiple profiles to be stretched at the same time.

When the command is started, you first have to select all the elements you want to stretch in a window. This is the most important step, because it not only determines what has to be stretched, but also the location where all the elements will be stretched.

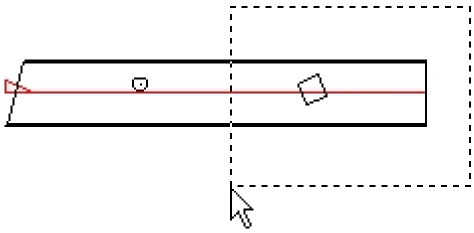
This is clarified in the following illustrations:



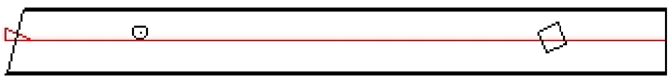
*Selection of only the right-hand end of the profile.*



*Result after stretching with the selection of only the right-hand end of the profile.*



*Selection of the right-hand end including the square cut in the profile.*



*Result after stretching the right-hand end including the square cut in the profile.*

The two examples show that the lines of the window determine the stretching line. This is the point at which the profile will be stretched. The holes and cut-outs within the window are also moved during the stretching procedure, the holes and the cut-outs outside of the window remain unchanged.

Slanting profiles can also be stretched using this command, or profiles that are vertical can be stretched while remaining vertical. This is controlled by the two points that are requested after the selection of the profiles.

Stretching occurs in relation to the current UCS. This is only effective when working on a sloping UCS: Stretching with the UCS in World often gives different results for the end cuts of profiles compared with stretching using a sloping UCS.

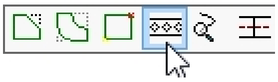
Remember in certain situations (especially with a sloping stretch) after stretching, the connections may no longer be correct, because the current connections do not anticipate being stretched. This can be solved by editing the connections afterwards and clicking on the button 'Reset'.

## Mirror

The mirroring of Parabuild elements can be carried out in the same way as the mirroring of AutoCAD elements.
The command **Mirror** is available on the command line and can also be found under the **Modify** toolbar of AutoCAD.
The method is simple: First select all elements to be mirrored, this may be lines and profiles at the same time, then add 2 points to determine the mirror line.
Plates, profiles, bolts and connections can be mirrored. If a mirrored connection has all of its main elements then these can be edited.

## Drawing hole patterns in a profile

Commando : **S3d_Hex**

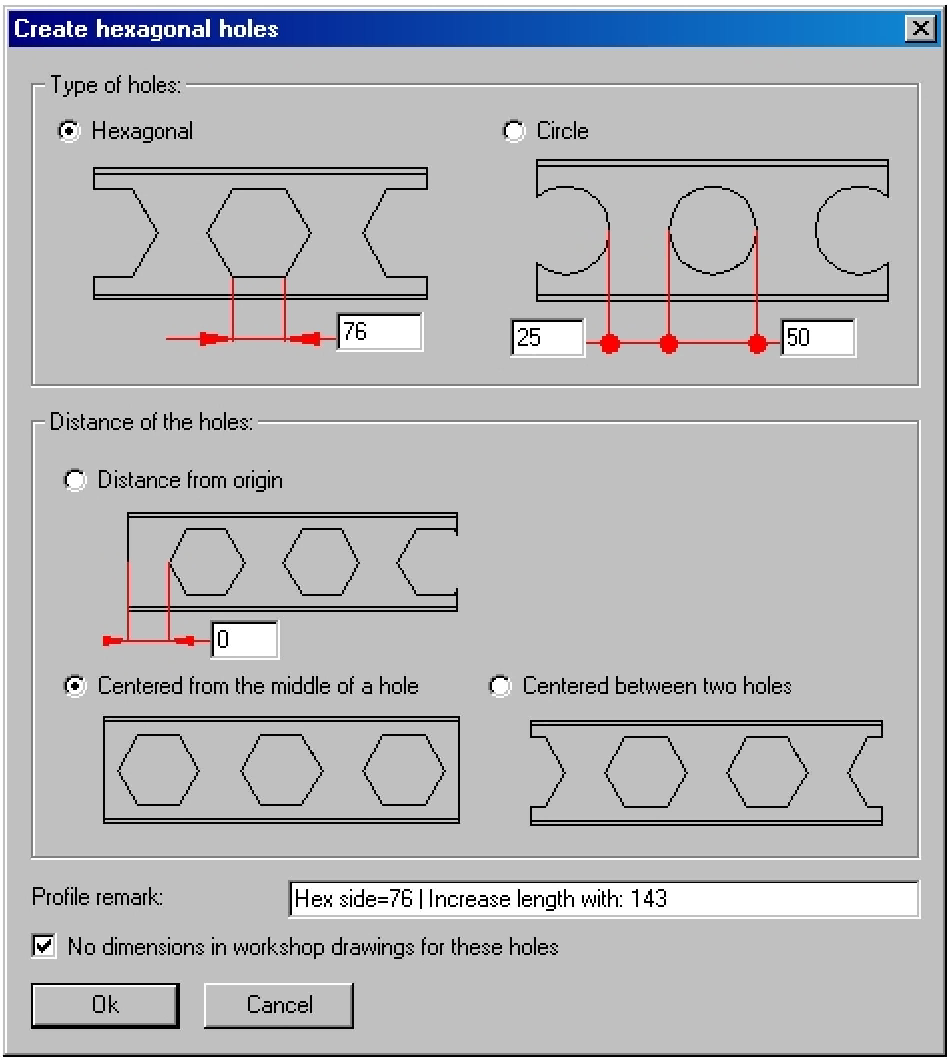


You have to use this command on HEX profiles from the section library.

This command will draw hexagonal (or round) holes. The IPE profiles are cut transverse in half according to the pattern (half hexagonal) to achieve a gain in height. The HEX profiles from the library are IPE-profiles with the gained height. For example the profile HEX300-200-115 is an IPE200 with new height 300 made using hexagonal holes with sides dimensioned 115.

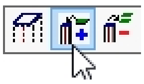
By cutting transverse there is always a loss in length because the two sides are shifted before they are welded together again. This loss in length will be calculated by this command and will be added as a remark to the profile.

You will have to manually add the lost length to your ordering lists! (because in 3D we drew a profile that is too short)

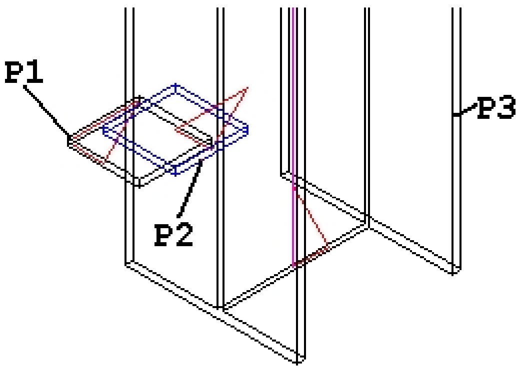



## Welding parts (assembly creation)

Command : **S3d_Attach**



This command allows you to weld one or more parts to a main part.




To add parts to an assembly you first need to select the sub parts (P1 and P2) and then the main part (P3).
Welded sub parts will receive a blue or green color by default (in *Sub types* visibility mode).
This allows you to see immediately which parts are welded.

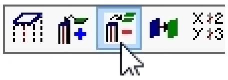
All parts of each welded assembly are merged together into one selection group.

This means that if you select one part then all parts of that assembly will automatically be selected.
This behaviour can be disabled (so you can switch between part selection and assembly selection).
To learn more about this see Assembly/Part selection switch.

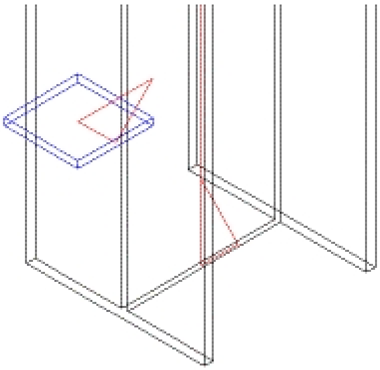
Note: A plate cannot be used as a main part to weld other parts to it. Only profiles can be main parts.

## Detach elements from an assembly

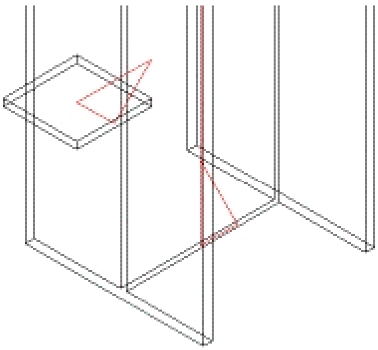
Command : **S3d_Detach**



This command allows you to detach a part from an assembly.



*Connected to a beam (Blue plate)*



*Detached from assembly (White plate)*

## Assembly/Part selection switch

Command : **S3d_AssemblySelection**

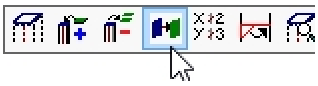
This command functions like a 'switch'. Every time you click on it the assembly selection is turned on or off.

All components of one assembly are grouped together. If you turn off assembly selection, then you can for example move a welded plate without moving all of its assembly members at the same time. The groups always stays intact, i.e. when you turn assembly selection back on all components of that mark will 'stick together again'.

To make things clear: assembly selection only changes something about the way in which elements are selected while drawing, not which element is welded against which. So the welding data remains intact after the assembly selection has been turned on or off.

### Creating an endplate

Command : **S3d_SetTracPlate**



With this command you can convert an ordinary (blue) plate to an endplate (green).

The only difference between the two is that the green plate gets an additional front view in the assembly workshop drawing.

### Elements library

This system strongly resembles that of the Customised sections system.

There is a command to add elements to the library, and a command to extract an element from the library and add it to the current drawing.

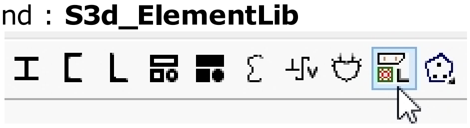
## Creating a new library element

Command : **S3d_NewElementLib**

- First take a good (3D) view, and zoom in, A small preview will be captured using this view.
- Start the command  to create a new library element.
- Select the location in the dialog window and the name of the dialog window and click on Ok. For further information about this system, see Customised sections.
- Now give the insertion point that will be used later to insert the profile into the drawing.
- The elements to be added to the library may now be selected. These can be any types of elements: plates, profiles, bolts, structures, lines, texts, dimensions, solids, ...

## Adding a library element to a drawing

Command : **S3d_ElementLib**



- Start the command 
- Select the element to be inserted.
- If the elements are to be inserted into the drawing as separate normal elements, then

select **Insert as regular elements** at the bottom.

- If the elements are to be added to the drawing as one structure, then select **Insert as one structure**. The structure will be given the name of the library-element. You can find more information about Structures in the Structures chapters.

## Structures

The name given to structures, is self-explanatory: totally flexible and can be used to achieve many objectives. Objectives that can be invented by the user.

### An example of the application of structures are treads

The problem presented by stairs for Parabuild is the following: if a tread has to be drawn, various strips have to be drawn. The problem first appears in the part lists and workshop drawings, where the strips that make up the tread have been included. This was never the intention as the treads are bought in as one unit.
An alternative is to draw the treads, not with Parabuild but with lines/solids. In this way, the treads will not be added to the part list, but a variety of advantages will be lost: no bolt check, no clash control. ...
These problems can be solved by using a "structure" in Parabuild.
A structure is a group of elements that are grouped into one element by Parabuild and will therefore appear as one element in the part lists and workshop drawings.

### A structure is created in the following way:

- First draw the tread (This drawing may include a mixture of a variety of objects: plates, strips, bolts, lines, 3D-Solids, ...).
- Deposit all of these elements together in the elements-library. Choose the name in the library that will later appear in the part lists.
- Now open the elements-library. To insert the new structure just click the option located on the left-hand side at the bottom "Insert as one structure".

The stairs will appear on the screen, but when selected all elements will form one group.
One element for the tread will now be added to the part list (under the name selected by the user).
If this element is selected, and the properties are requested, the element will be recognised as a structure with its own properties.
The structure also has a position number, a mark number, a phase and a revision. It can also be welded to Parabuild elements which means that it will be added to the mark drawing as an auxiliary element!
Parabuild can calculate the weight of the structure. This weight is of course dependent on the material assigned to the structure.
Multiple treads can be added to the drawing by simply using the copy command. The actual number of treads used will appear automatically in the part list.

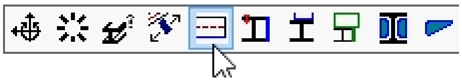
### Structures: under the hood (what does Parabuild actually do with Structures).

When making insertions from the elements-library a block with the selected name is added to the drawing.
A "Block Reference" is then created so that the block appears once on the screen.
This Block Reference is labelled as a Structure type by Parabuild and is expanded so that the following features can be used with structures: properties, welding, clash control, numbering, part lists and workshop drawings.
When the structure is copied, only the block reference is copied, and not the actual block.
This means that the block only appears once in the drawing, but has a variety of reference points, allowing several to be created.

## Miscellaneous commands

# Applying a line to a profile edge

Command : **S3d_SubGeomPoly**

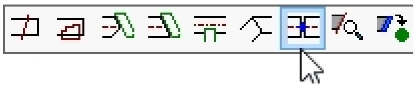


After this command is started an edge or a plane of a profile or plate should be selected. A line will be drawn along the edge. The line will not be visible at first as it is lying along the profile line.

This function can also be used to select the edges of cuts.

# Profile breaking

Command : **S3d_BreakProf**

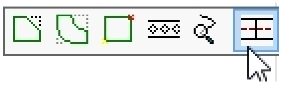


Just as with breaking a line when using the AutoCAD command, this command can be used to break a profile.

The profile will be broken at a pre-selected breaking point and this will result in two profiles being created.

# Changing the axis of a profile

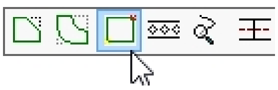
Command : **S3d_EditProfPath**



This command can be used to add or remove bends to the axis of a profile.

# Changing the corners of plates

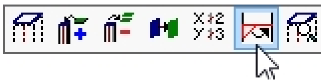
Command : **S3d_EditPlate**



This command can be used to add or remove corners of plates.

# Rotating the triangle of a profile

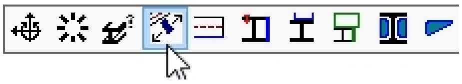
Command : **S3d_SwitchProfEcs**



The triangle of a profile determines the profile's orientation on the workshop drawings (the triangle is always placed on the left-hand side of the page). This allows total rotation of the orientation.

# Move along a line

Command : **S3d_LineMove**

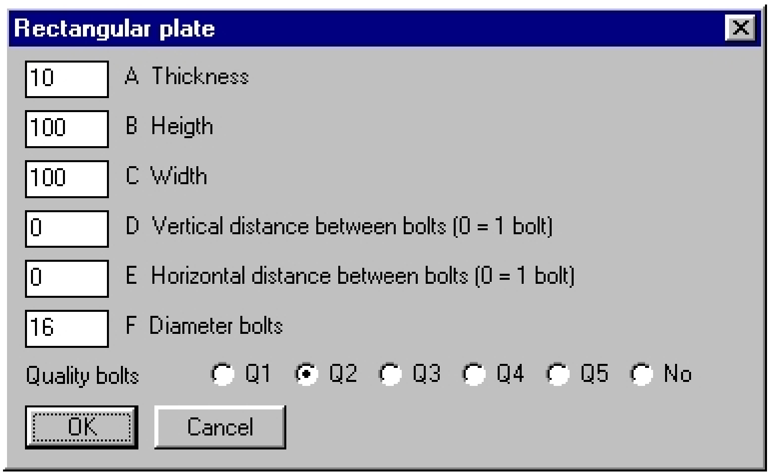


With this command you can move elements along a line. This line can be any line from a profile or plate.

## Rectangular plates

 Rectangular or square plates can be drawn very easily with this command.

After selecting the command, the following dialog appears:



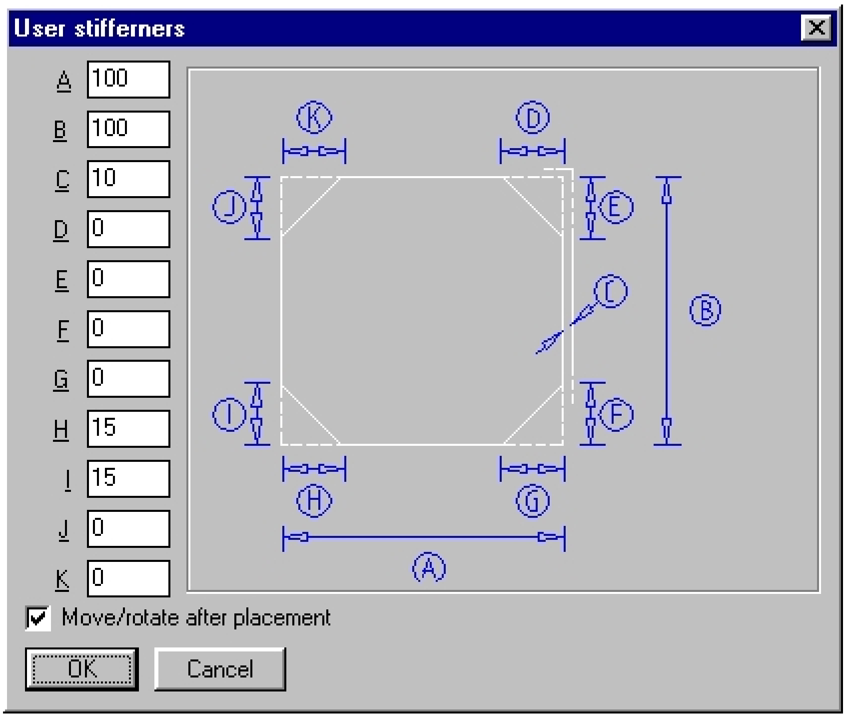

The options are self-explanatory.

When you click **OK** you are requested to select the positioning point. The middle of the plate will be positioned at this spot.

## Plate with chamfer

 Chamfered plates like Ribs can be created very easily with this command.

After selecting the command, the following dialog appears:




All corners of the plate can be set to your requirements.

C is the thickness of the plate.

The rotate/move dialog is displayed after you have selected the positioning point. This dialog is explained in detail in the move/rotate section.

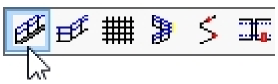
## Plate with polyline

Command : **S3d_PolyPlate**

If you could not draw the required plate with any of the previous commands, you can always use the `plate with polyline` command.

## Stair macro

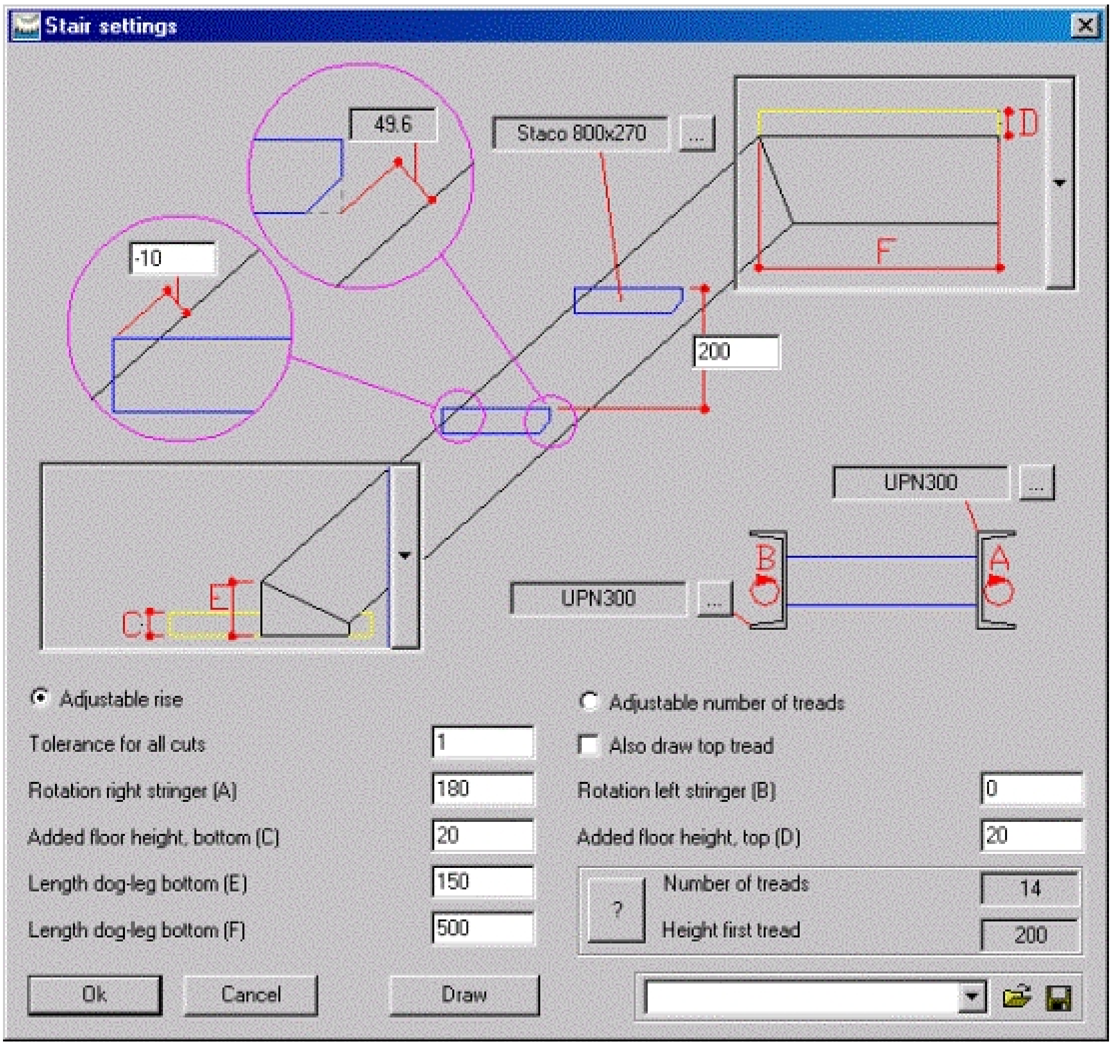
Command : **S3d_DrawStairDlg**



Before you start this command, you have to draw a line that will be the basis for the stair.

The line defines the begin at the bottom and the end at the top (not necessarily the floor height), the slope and the middle of the stair..

After starting the command and selecting the line, you get the following dialog box:




Most options are well illustrated, but some still need to be clarified:

**Adjustable rise**: When enabled, you can adjust the distance between treads in the image. The program will calculate the required number of treads. The remaining distance will be used for the first tread. If you would like to know the height of the first tread for the number of treads you entered, click on the button "?".

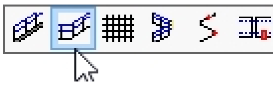
**Adjustable number of treads**: When enabled, you cannot adjust the distance between treads, but you can adjust the number of treads. The distances between the treads will be made equal. If you click on the button "?", you can see the distance between the treads in the image (this option is only available when you are drawing one stair at a time – you selected only one line).

**Added floor height**: This option has no influence on the stringer, only on the calculation of the treads: the height of the floor bottom and top will be subtracted from the total height of the stair used to calculate the number of treads. The first tread will be placed above the bottom floor height.

**Rotation stringer** : The rotation of the stringer (options A and B) allow us to rotate the flange of the U-beams to the inside (to the treads).

## Railing macro

Commando : **S3d_PolyRailDlg**

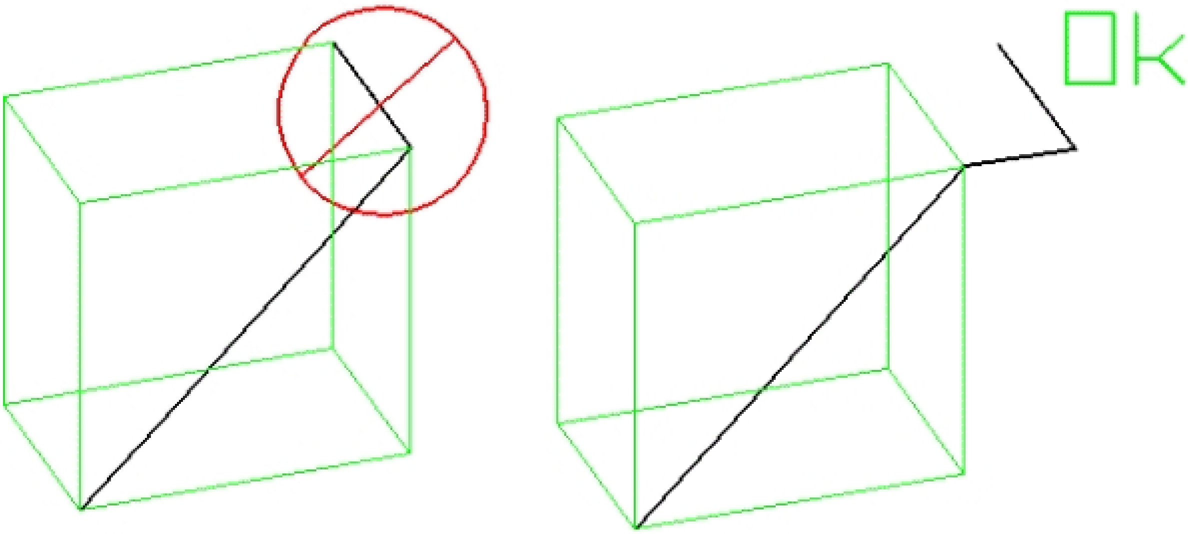


Before you start this command, you have to draw a line.

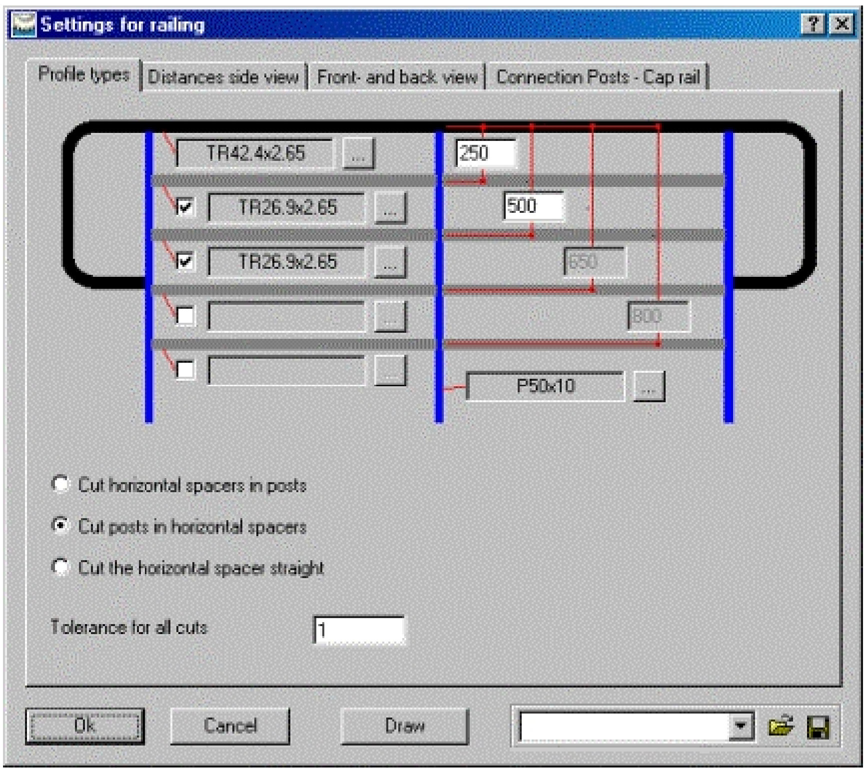
This line could be just a regular AutoCAD line, a 2D polyline or a 3D polyline.

The line determines the 'path' the railing will follow.

For drawing a 3D polyline there is one limitation: The transition of 2 segments can only be in one direction at a time. Look at the illustration beneath for clarification.




The options in this dialog box are illustrated.

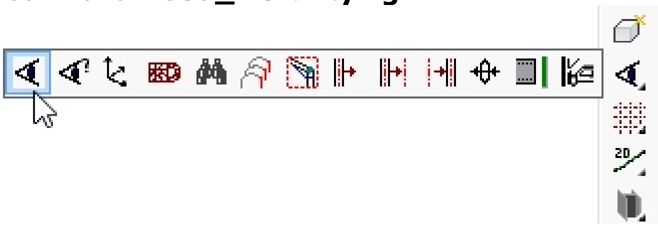


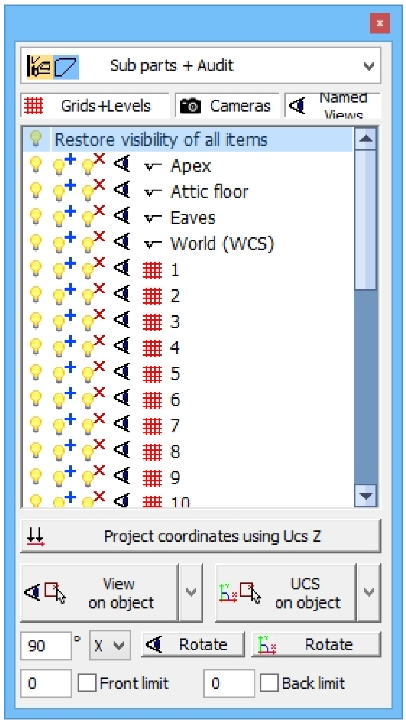

## Visualisation of 3D parts

The commands in this chapter are tools for showing and hiding 3D parts.

## Visibility manager

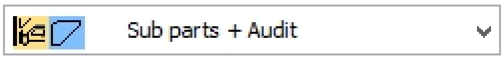
Command : **S3d_VisibilityMgr**





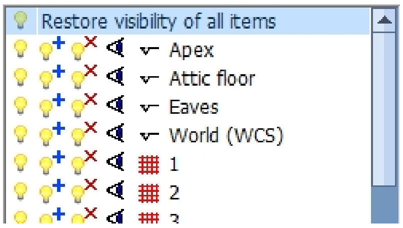

This command opens a window that can remain open.
The window contains a set of tools that help you to :
- change the color of all elements
- hiding elements
- change the view
- change the Ucs

We review the purpose of each item in the window :




This list shows the available color styles for Parabuild objects. Change the style if you want the elements to follow the layers, or if you want the welded elements in a striking color (green and blue) or if you want the clashing elements in a striking color (yellow).


Only those object types that have been activated here will be shown in the list below.




In this list the levels, grid lines and cameras that exist in the drawing are shown.

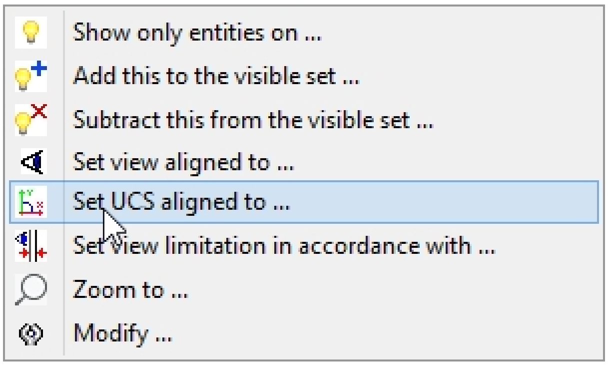
 If you click on the lamp next to fe *grid A*, then all elements in the drawing will be hidden except the elements that are close to *grid A*.

 This lamp can be used to restore visibility of all elements close to fe *grid A* in case they were rendered invisible by a previous operation.

 With this lamp, you can hide all elements close to fe *grid A*.

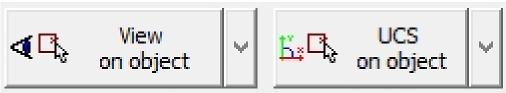
 Click on the eye next to an item to align the view with the level / grid / camera.

When you right-click an item in the list then you also have the possibility to change the Ucs and the view limitation according to the item, as well as to zoom in on this element in the drawing.

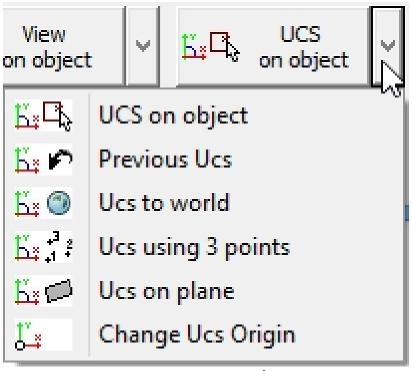




If you activate this button, all Z coordinates will be projected to the XY plane. The Z coordinates will thus always be 0. Note: This button affects only the Object Snap tool of AutoCAD / BricsCAD.
(This tool works by modifying the AutoCAD variables OSNAPZ and ELEVATION)

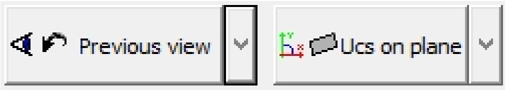


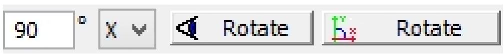

These buttons give access to a range View and Ucs tools.



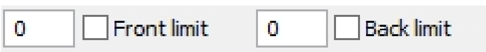

Press the button  to see all the available tools.

The last tool that you used will be shown so that you can quickly repeat the last tool.




With these buttons it is possible to rotate the view or Ucs.




These check boxes allow you to activate or deactivate the view limitation at any time.

## Hiding volumes

Command : **S3d_HideObjectsNoAxis**

This command allows you to completely hide a selection of objects, if they obstruct the view.

## Volumes -> Axes

Command : **S3d_HideObjects**

This command allows you to hide a selection of objects, if they obstruct the view.

However, the axes of profiles and plates remain visible.

## Axes-> Volumes

Command : **S3d_ShowObjects**

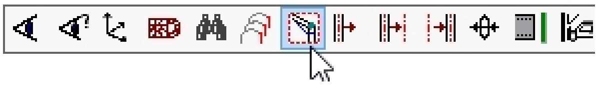
This command can restore the visibility of a selection of hidden objects.

The volumes of which only the axes were visible can be selected and their full visibility restored.

To restore the visibility of all objects in the drawing, press **<Enter>**.

## Showing a selection

Command : **S3d_IsolateSelection**



When you run this command, you can select several objects from the drawing.

These objects will still be visible, while all the other objects in the drawing will become invisible, regardless of the status of each object's layer.

Also, all AutoCAD objects will be hidden such as lines, dimensions, and blocks.

This way you can very easily continue to work on one part of the drawing without other objects obstructing the view because they are located in front of the objects of interest.

To restore the visibility of all objects in the drawing, you can at any time activate the  *S3d_ShowObjects* command and then press **<Enter>** to display all objects.

## Showing/Hiding camera's

Command : **S3d_SwitchViewdefVisibility**

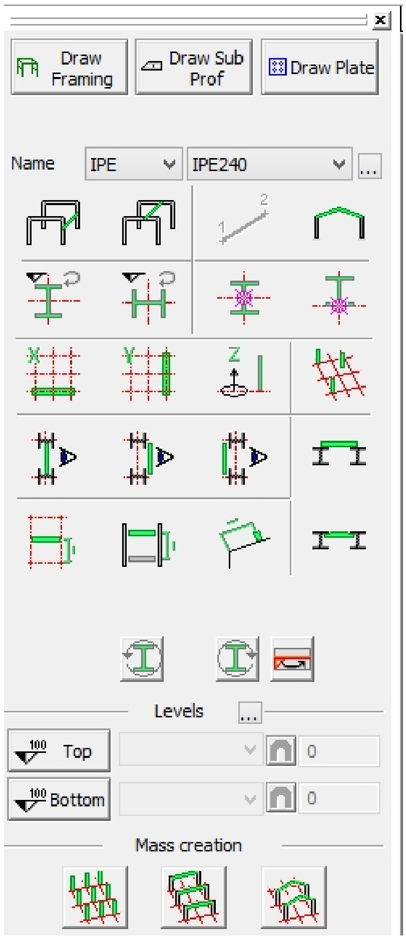
This command works as a switch.

It will hide all the cameras in the drawing if they are currently visible or show them if they are currently invisible.

# Context Modeling

Command : **S3d_ContextModeler**

This command opens a window that can always remain open.



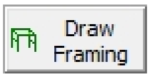

This window has three important buttons at the top.

When you press one of these buttons then the options underneath (context buttons) will change.

After clicking one of the main buttons you can already start to draw the requested element type : Move the cursor over the drawing and Parabuild immediately shows you the element that it can draw on that location.

Click the left mouse button to draw the element that is currently visible.

## We will first go into detail about the available types of objects

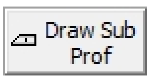



With this action button you can draw columns, beams and rafters.

But also side rails, purlins, joists and all longer profiles of your structure can be drawn with this button.

The profiles that you draw are based on :

- grid lines
- levels
- other profiles that already exist in the drawing
- The World coordinate system(in case there are no grid lines)



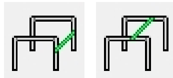

With this button you can draw smaller profiles. The goal of these profiles will rather be to connect profiles with each other.
The profiles that you draw with this tool are based on one or more profiles or plates that already exist in the drawing.


With this button you can draw plates.

The plates that you draw with this tool are based on one or more profiles or plates which already exist in the drawing.

## More about the context buttons (restrictions)

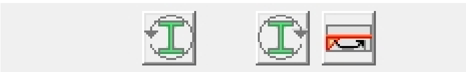
When you press one of the action buttons, then you will see the available options for that action underneath.
If you press one of these context buttons, then you put a restriction on Parabuild.
This is sometimes very useful, because in some situations, there are many different possible solutions for Parabuild.
Parabuild can not always show all the possible solutions at once. Then the context buttons offer a solution.

For example, if you want to draw a beam, you can press one or both buttons .
Then Parabuild will not propose to draw a column or rafters.

## Various functions are available while drawing elements

While drawing elements, so while there is an element hanging on your cursor, you can use the following manipulation buttons:

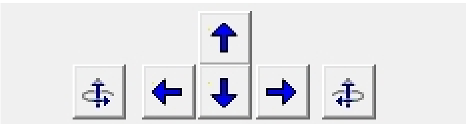
**While drawing structures**:



These buttons can rotate the section of a profile per 90 °. You can also switch the start and end of the profile.

TIP: Use the arrow keys on the keyboard ← →, and the ⇓ *<Page Down>* key to quickly access these functions without having to move the cursor to the window!

**While drawing smaller profiles and plates**:



With these buttons you can rotate the element in all directions.

The most important of these buttons are  and , because they rotate the element on it's base surface (the base surface is the plane to which the element was oriented)
TIP : Use the arrows and *<Page Up/Down>* keys on the keyboard ← → ↑ ↓ ⇑ ⇓ to quickly access these functions without having to move the cursor to the window!

**Some other keys that are available** :
*<Space>*: When pressed, searching for solutions is temporarily suspended. The last element will remain on the screen. You can now freely move the cursor without changing the element, and you can also adjust the element using the arrow keys. Then press *<Enter>* to draw the element, or return to finding other solutions by pressing *<spacebar>* again.

*Left Mouse Button*: Draw the element that is currently visible

*Right Mouse Button* or *<Escape>*: Cancels the command without drawing the last element

# Applying connections

In this chapter we learn the application of a connection from the library.

The connections from the library were subdivided in groups so that choosing the correct connection from the large offer can be chosen rapidly. There are also some filter capabilities to reduce the number of connections to choose from.

The division in groups ensures the first disposal of irrelevant connections. Some examples of groups are Haunch connections, apex connections, end plates, base plates, gusset plates, etc…

Each group has its own icon. After you click on the icon you will be asked to select the base profiles. A logical and intentional consequence is therefore that all connections of the same group must have the same amount of base profiles.

The order in which you select the profiles is important! For a haunch connection you are asked to first select the column and then the beam. If you select in the wrong order, the orientation of the resulting connection will be wrong.

The base profiles that you select are checked with the purpose of knowing which connections are applicable and which not. This is a filter that happens automatically for you. The connections that do not apply to the chosen profiles will not be visible.

**Extra information:**

This automatic filter compares the profiles that you have selected against the base profiles of each macro. If the section type differs from the base profiles (U <-> Tube), the macro cannot be applied (for each type of section there exists a separate connection).

Also the orientation of the base profiles plays a role.

For example a haunch connection with reinforcement: For the traditional application it is expected that the beam is oriented with the flange horizontal so that the reinforcement can be welded against the flange. In the event that the beam that was selected doesn't have this orientation, then you are not able to use this haunch connection.

After the selection of the base profiles a dialog box appears in which you must select a connection (one of the images).

On top of the dialog box there is a table with which you can filter the offered connections.

With this you can reduce the number of available connections by clicking the properties you need.

The properties have been split up by module. One row represents a module. The first field in a row is the name of the module itself. All the following properties are the properties of that module.

Some typical examples of properties:

Module: Bolts   Properties: One row of bolts, several rows

Module: Reinforcement   Properties: Reinforcement plates, Reinforcement profile

Module: Column endplate   Properties: Inclined placement, Horizontal placement

If you want to see only the haunch connections that have inclining column endplates, then activate the check box for the property **Inclined placement**. The connections that do not have this property will disappear from the list.

You can activate as many properties as you wish.

After you have chosen a connection and clicked on OK the connection is drawn and a dialog box will be opened that allows you to adapt the dimensions of the connection. The next chapter discusses this dialog box.

You are free to remove a part of the connection (components such as plates, profiles or bolts). The macro will never redraw the elements that you have removed: they are permanently gone. All other elements of the macro will possibly continue to work normally after this, depending on the situation:

For example you can remove the stiffeners of an apex connection without any impact for the end plates or the reinforcement.

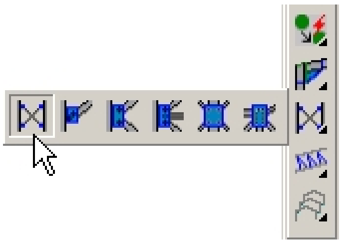
But if you would remove an end plate, then this has consequences on other components in the connection. As it happens, the reinforcement and the cut of the beam are based on the end plate of the beam.

The consequence would be that these components can only be adapted partially or can no longer be adapted at all. This can cause undesirable results.

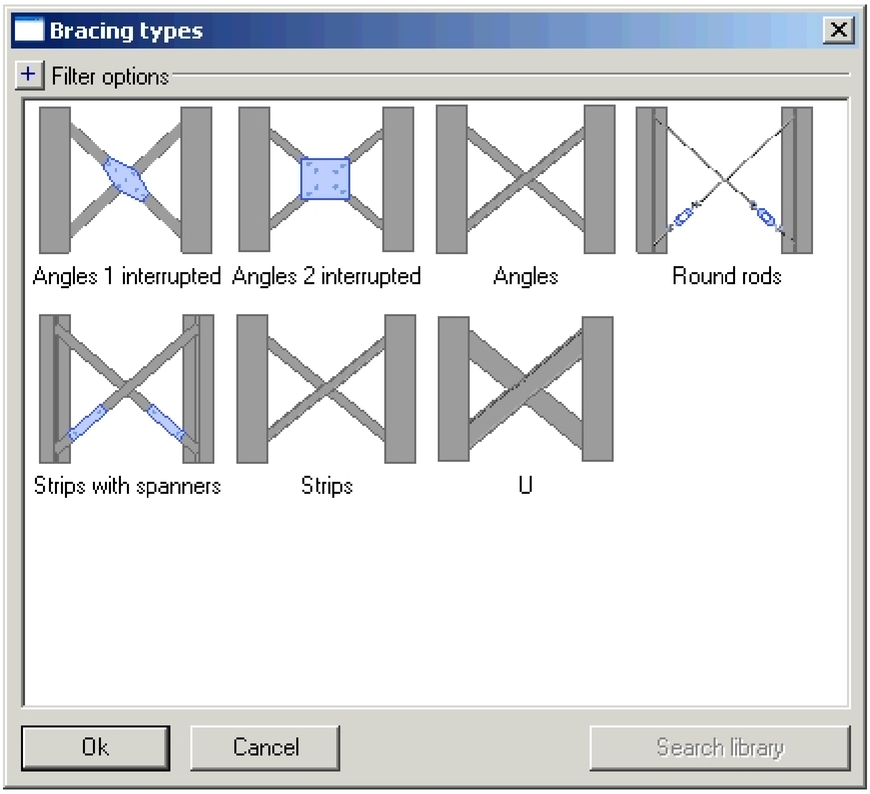
You are free to remove just the macro sphere. The components of that macro will stay intact, but they will no longer be automatically adapted because the macro sphere takes care of that. Also the dimensions of the connection can no longer be adapted. The connection's elements become loose elements as if they were drawn manually.

## Drawing bracings

To draw a wind bracing, we activate the icon **Wind bracing** in the Parabuild toolbar.

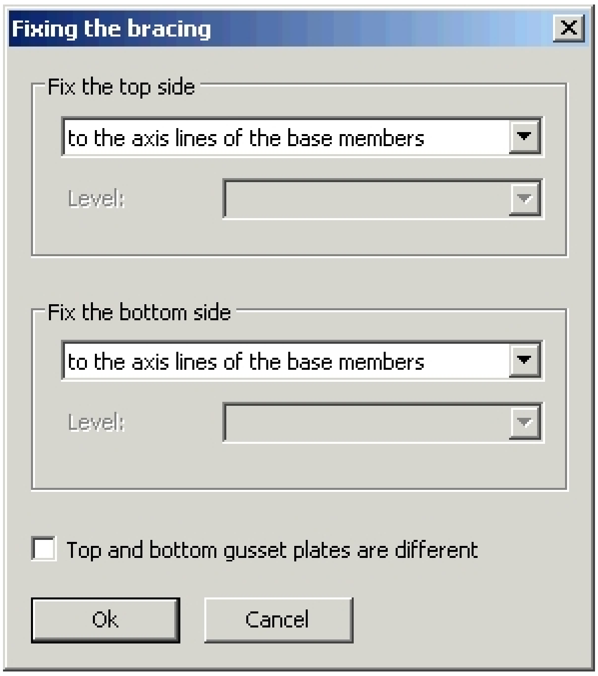


A dialog box appears which allows us to choose one of the wind bracing types:

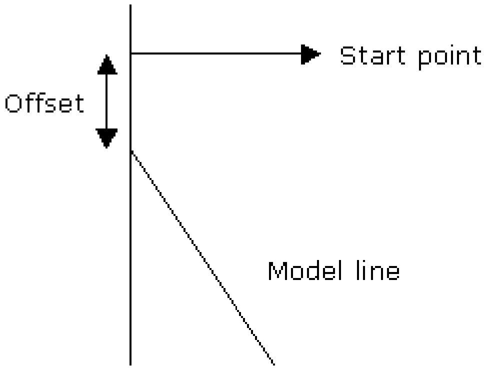



Select the desired bracing type and click on **Ok**.

Now a dialog box appears with which we will determine the position of the wind bracing:

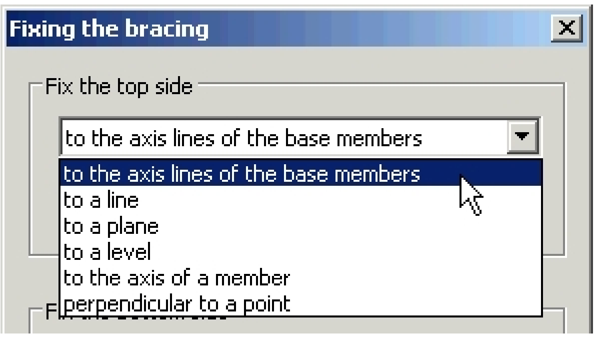



In order to draw a wind bracing, Parabuild needs two model lines on which to draw the two members. These are two intersecting lines that are drawn in the plane determined by the axis lines of the two base members. The start point of these lines is determined by a start point on the line and an offset. The end point of these lines is determined the same way.




The standard value for the offset is 500. This value can be changed in the **Bracing** dialog box that is shown at the end of the command.

The start point at the top and the end point at the bottom can be determined as follows:




**to the axis lines of the base members**

This means on the start and end of the axis lines.

**to a line**

The line has to intersect with the base members.

- For a symmetric bracing, the start point is the intersection of the line with the axis of the first base member. This intersection point is used for both left and right.

- For an asymmetric bracing, the left start point is the intersection to the left, the right start point is the intersection to the right.

Attention!

- In case the line extends beyond the length of the base member and there is no real intersection, then Parabuild will use the apparent intersection of the line with the prolonged member axis. In this case, the gusset plate connection(s) might be drawn outside of the members in the air.

- In case the line is not drawn in the same plane as the plane of the bracing and thus a there is no real intersection, then Parabuild will project the line to the plane of the bracing to obtain the intersections.

**to a plane**

The same procedure as with lines, but in this case the intersection of the plane with the members axis is used.

**to a level**

The same procedure as with planes, but the difference is that these planes are always horizontal. This option therefore cannot be used for bracings in the roof.

**to the axis of a member**

The same procedure as with lines, but in this case the axis of the member you will select will be used.

**perpendicular to a point**

We can select a point (for example a point of a plate). The start point is then determined by projecting the point perpendicular to the member axis.

**Top and bottom gusset plates are different**

When you activate this option, Parabuild will show you the gusset plate selection dialog box twice. The first time for the top gusset plates and the second time for the bottom gusset plates

If this option is deactivated, the gusset plate dialog box will be shown just once and all four gusset plates will be the same.

When you click on **Ok**, the following prompts will show on the command line:

**Select the left profile on the side where the bracing should be placed:**

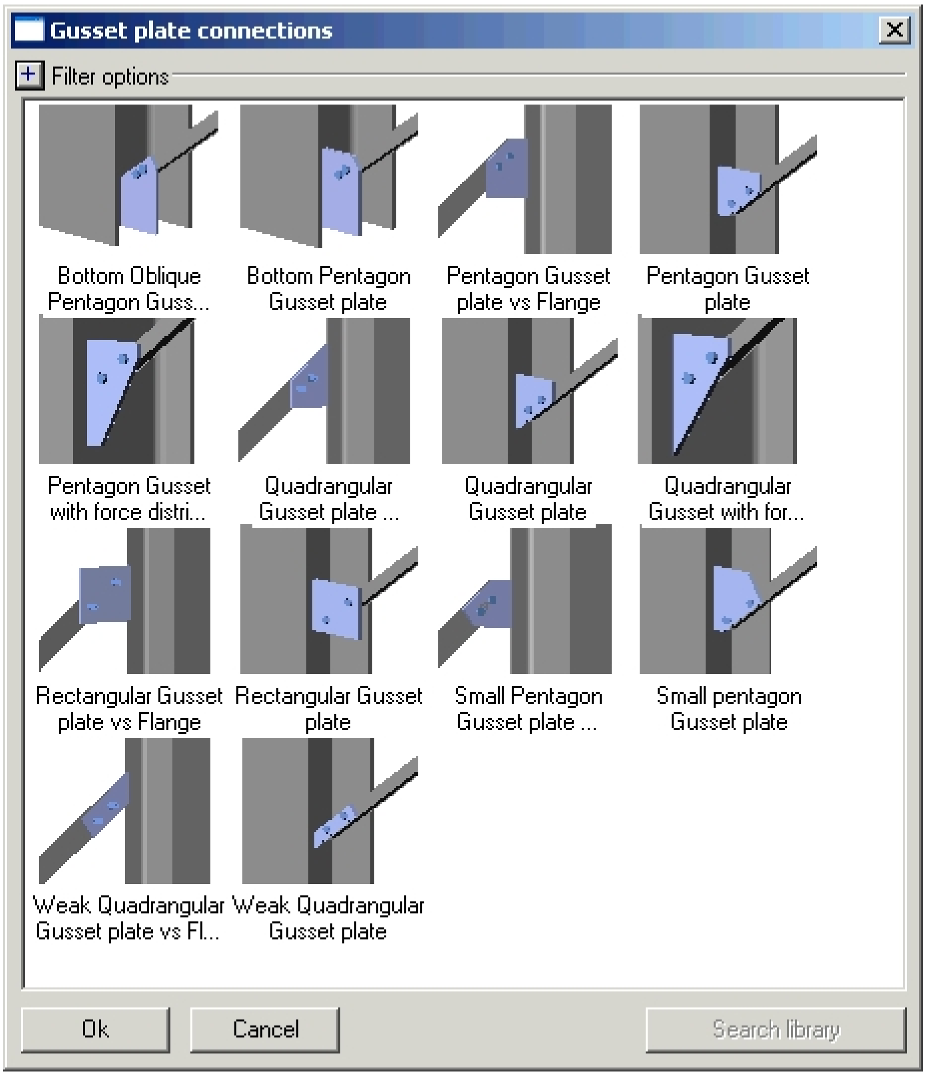
and

**Select the right profile:**

We select the backside of the flange of the member, on the side of the bracing

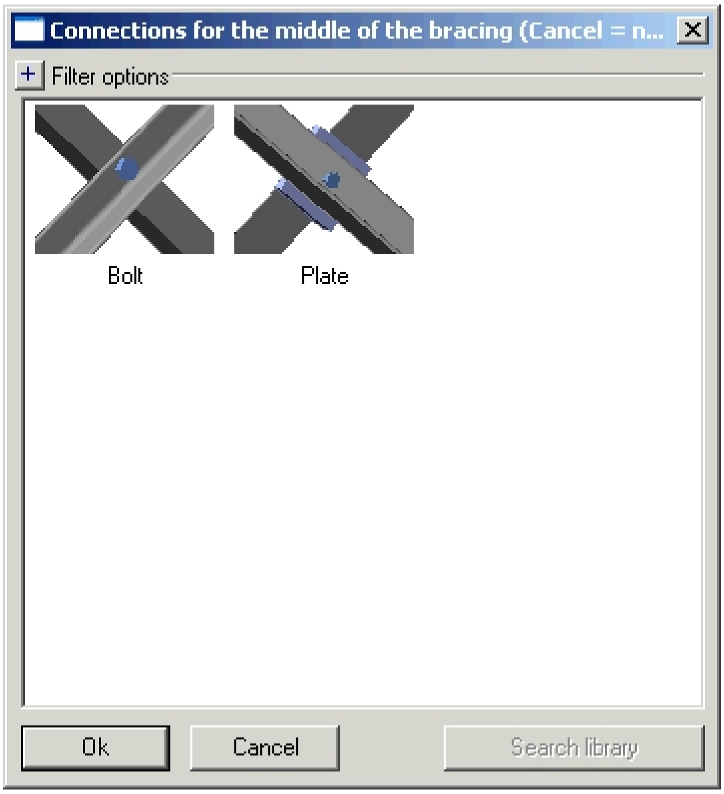
Depending on the selected options, we are also asked to select line(s), member(s), plane(s) or point(s).

Hereafter the **Gusset plate connections** dialog box appears.




We choose the desired gusset plate and click on **Ok.**

Now the dialog box **Connections for the middle of the bracing** appears:

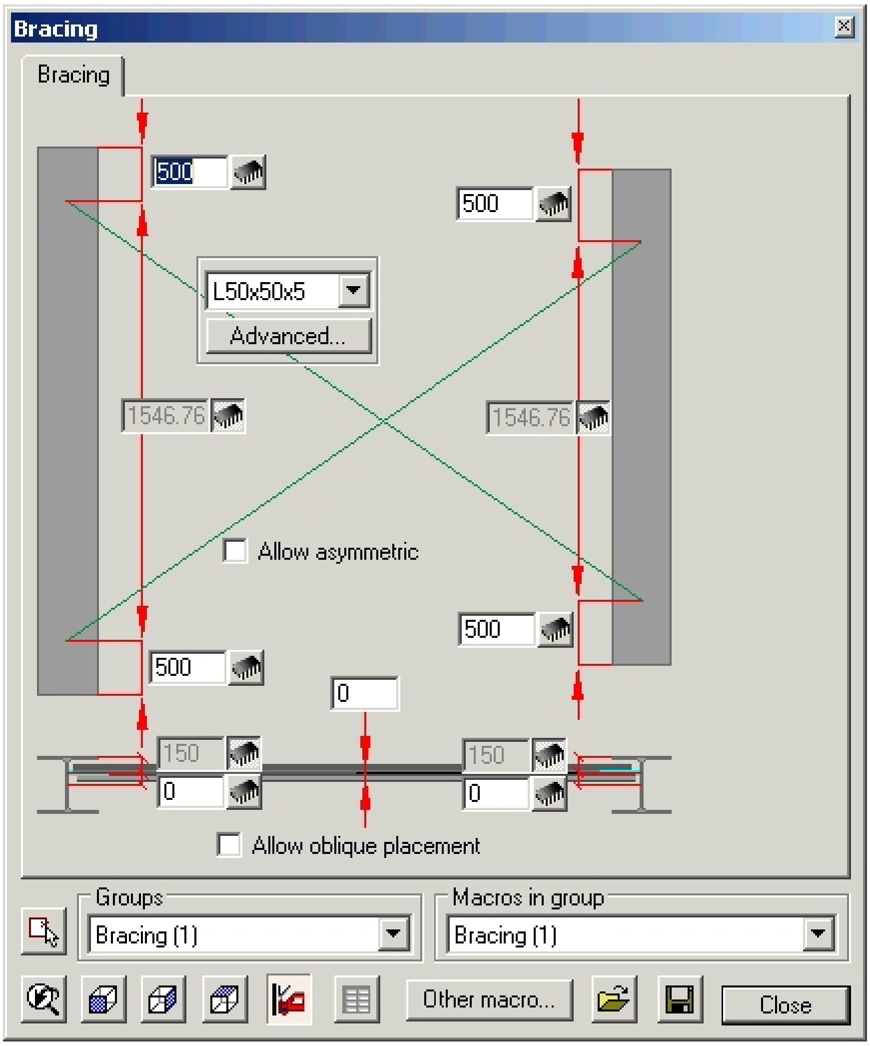



We choose an option and click on **Ok**.

Now Parabuild draws the wind bracing.

The previous prompts are repeated so that you can draw the same bracing on other locations. Press **<Enter>** when you have drawn enough bracings.

The **Bracing** dialog box now appears




We can:

- Change the offset at top and at bottom. Only the values to the left apply because Parabuild has drawn a symmetric bracing. If we activate the **Allow asymmetric** checkbox, the values to the right will apply too.
- Change the member section by using the drop down tool.
- Choose another section type by using the **Advanced...** button. Warning: if you change something using this tool, then the gusset plate connections will likely become corrupted. In this case you should redraw them.
- Change the horizontal position of the wind bracing. This is possible by changing the position of the bracing axis relative to the member axis or the member edge. Only the left values apply. If you activate the **Allow oblique placement** checkbox, then the right values also apply.

Remark

This last setting gives us the possibility to draw a bracing between members whose axes are not aligned.

Attention

The gusset plates that Parabuild has drawn are only suggestions and should be reviewed to make sure the dimensions adhere to the load calculations.

The gusset plates can be modified individually with the **Review macro** command on the 4 smaller spheres.

## Adapting dimensions of connections

A "sphere" represents a connection. For this sphere you must imagine yourself a collection of data that holds the intelligence of the elements of the connection.

When a base profile of a connection is modified, then the sphere will ensure that the elements in that connection adapt automatically to that modification.

We call this sphere a macro because this sphere can contain not only connections but also larger elements such as a tread, a cage ladder, the wire frame of a building,….

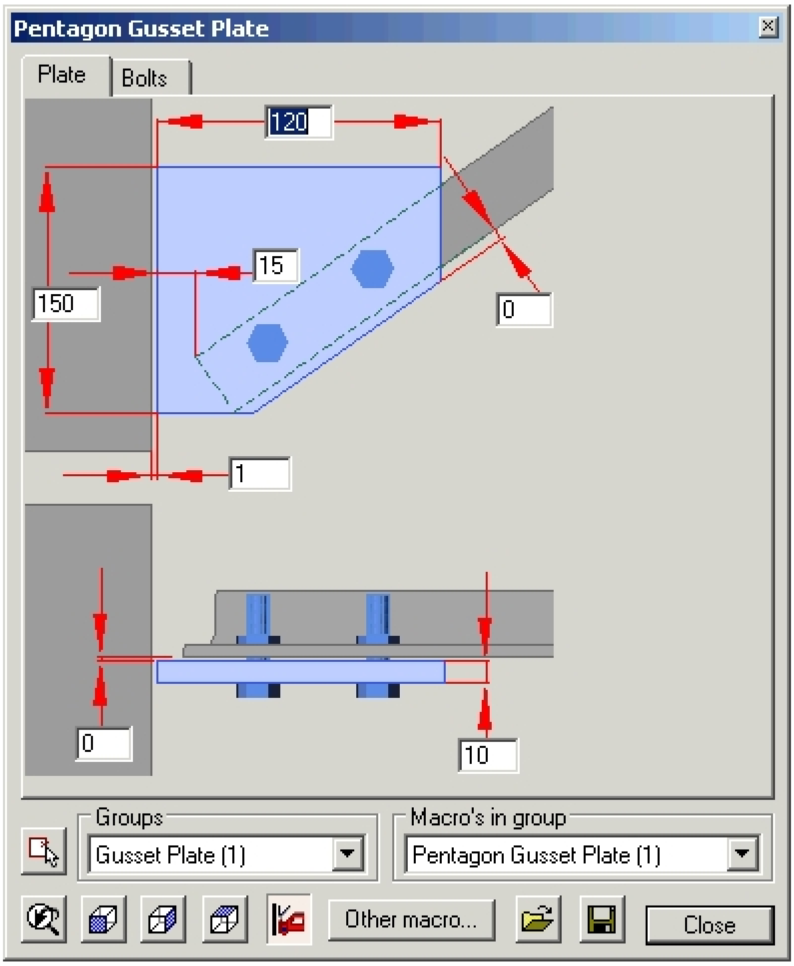
To modify the dimensions of a connection you use the command **Review macro** .

Afterwards you select the macro (sphere) that you wish to modify.

(it is also possible to adapt several macros at the same time, see further in the manual)

The dialog box that follows has a functionality that most dialog boxes do not have:

The dialog box is not fixed to a command. This means that while this dialog box is active, you can start any other command. The dialog box is always open until you close it. You can even open several of these dialog boxes at the same time that for example each adapt other macros.




It is possible in this dialog box to modify the dimensions of several macros at the same time.

At the bottom of this dialog box there are some options that are to be used to select which macro(`s) must be edited.

You can do this by selecting several macros while starting this dialog box or afterwards by pressing the button  Select **other macros**.

During the selection process you can select other elements except macros (for example the complete project). The elements that are not relevant will be ignored.

To prevent the editing of several macros becoming chaotic each macro belongs to a group.

You can only edit one group of macros at the same time.

Select in the first list the group that you wish to edit.

In the second list you can choose to edit all macros in the group, or only one of the macros. This second list is therefore dependent on the group that you have selected: the list will be updated each time you select another group.

When you edit several macros at the same time, the tabs and dimensions of all these macros are added in this dialog box.

The dimensions with the same name are reflected as one dimension and get behind the name between brackets a number that indicate at how many macros the dimension represents. The value of the dimension becomes *VARIES* if the values from the different macros differ. You can simply modify this value: the dimension will be applied to all macros that have the dimension.

 Use these icons to change the view to one of the side views of the connection.

 Whenever you modify a dimension of the connection, then the complete connection will be checked for collisions. This icon works as a switch to enable or disable this automatic clash control. It can be useful to turn it off when you edit a large connection or many connections simultaneously and the clash control demands too much calculation time.

With this button you can choose from the library another connection for the current connection (s). This works more rapidly than to remove the macro and to apply a new one. The dimension values of the current connection will be copied to the new connection where possible.

 With these icons you can save all values of the current selected connection (s) under a name and recall them at a later time. A value list can only be loaded if the connection is compatible with the connection with which the value list was made (this means the name of the macros must be identical).

On top of this dialog box there are a number of tab pages.

Each tab contains dimensions and/or options of a certain component of the macro.

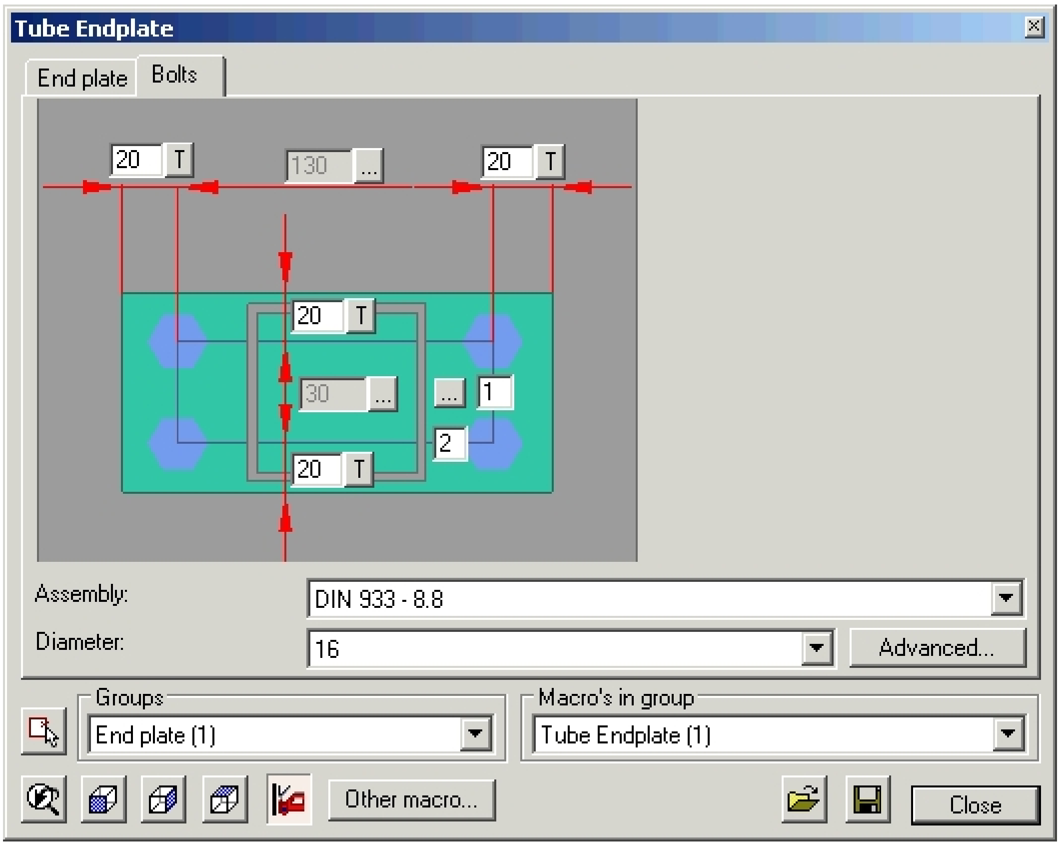
The number of tabs and the contents of it are therefore dependant on the macro. There are 3 types of tabs:

# Ordinary tab

This tab only contains dimensions.

# Bolts tab

This tab contains dimensions such as the ordinary tab but also contains special options for bolts.

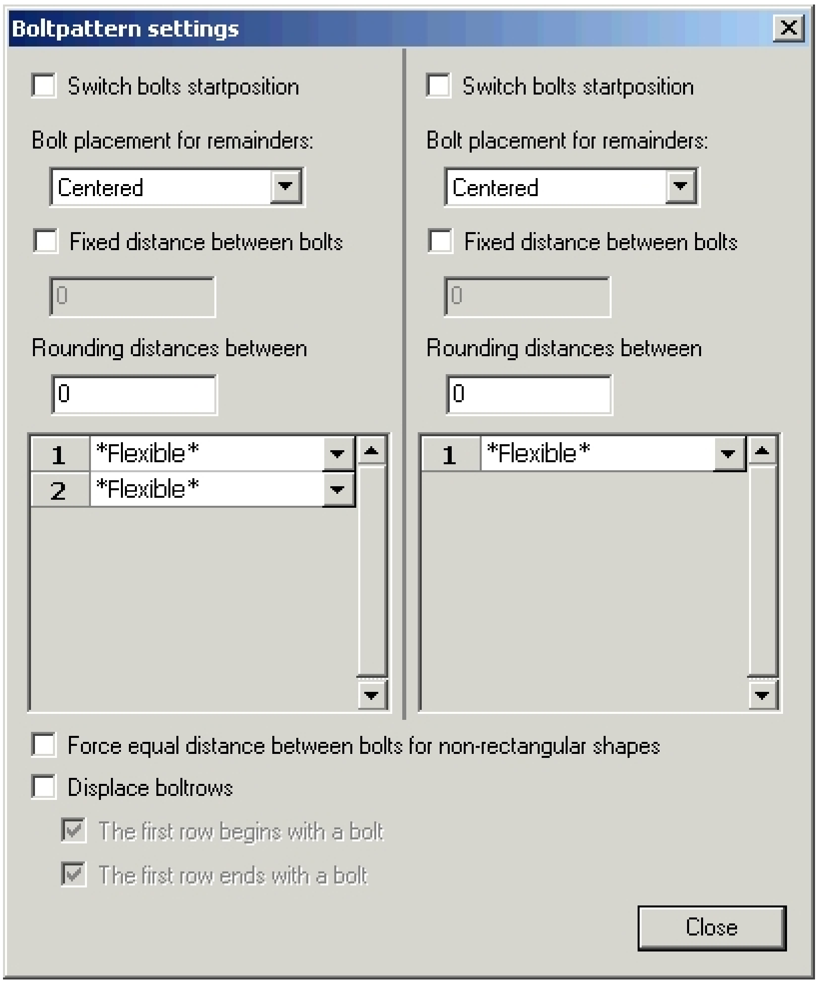



In the above example we have one bolt pattern. Each bolts pattern has two adjustable values that indicate the number of bolts vertically and horizontally.

Keep into account that not all bolt patterns have two numbers. There are also line and circle patterns that have but one number.

Besides the fields of the number of bolts there is always a button '...'.

If you click on this button then you can change advanced settings of the pattern:




Some of these options need some more explanation.

Columns horizontally/vertically: All options are double to enable us to modify all the options separately horizontally and vertically (if the pattern is rectangular).

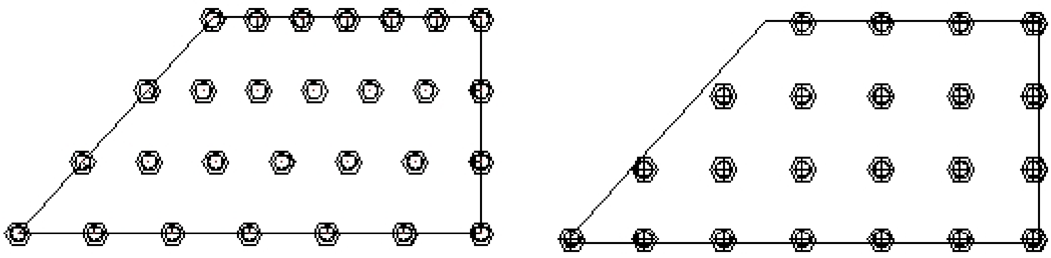
**Fixed distance between bolts**: The distance that you enter will always be respected. The option **Bolt placement for remainders** is important in combination with this option because with it you stipulate at which side the remaining distance will be placed.

**Rounding distances between**: This is an important option to avoid undesirable distances. Imagine that we have a pattern with a length of 100 mm. You choose 3 bolts for this pattern. The bolts are divided equal and as a result we get a distance between the bolts of 33.33 mm. Later that distance (rounded to 33 or 33.5) will be put on the workshop drawings, which is of course undesirable!

We avoid this by setting the rounding to 1 (or 5 or 10?). The distance between bolts is taken never smaller than the number that you enter here. Therefore with this option the bolts will never be drawn at distances of 33.33 but at distances of 33, or 35 or 30, depending on the rounding you choose.

**Force equal distances between bolts for non-rectangular shapes**:

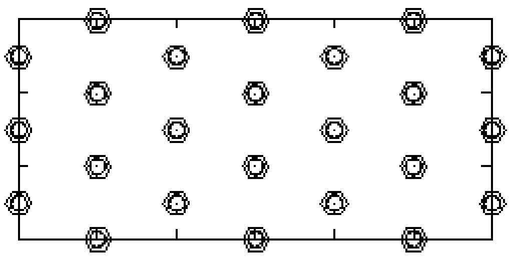
In the next image the option is off and in the second image it is on:



If you enable this option, the bolts that fall outside of the pattern will be removed.

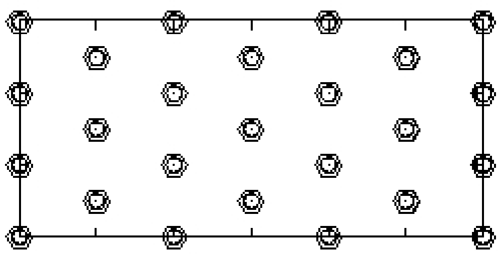
**Displace bolt rows:**

With this option enabled the bolt rows will alternate with an open spot:



**The first row begins with a bolt:**

With this option enabled the arrangement of the bolts changes:



At the lower part of the tab there are some general options that have influence on all of the bolts of all patterns in this tab.

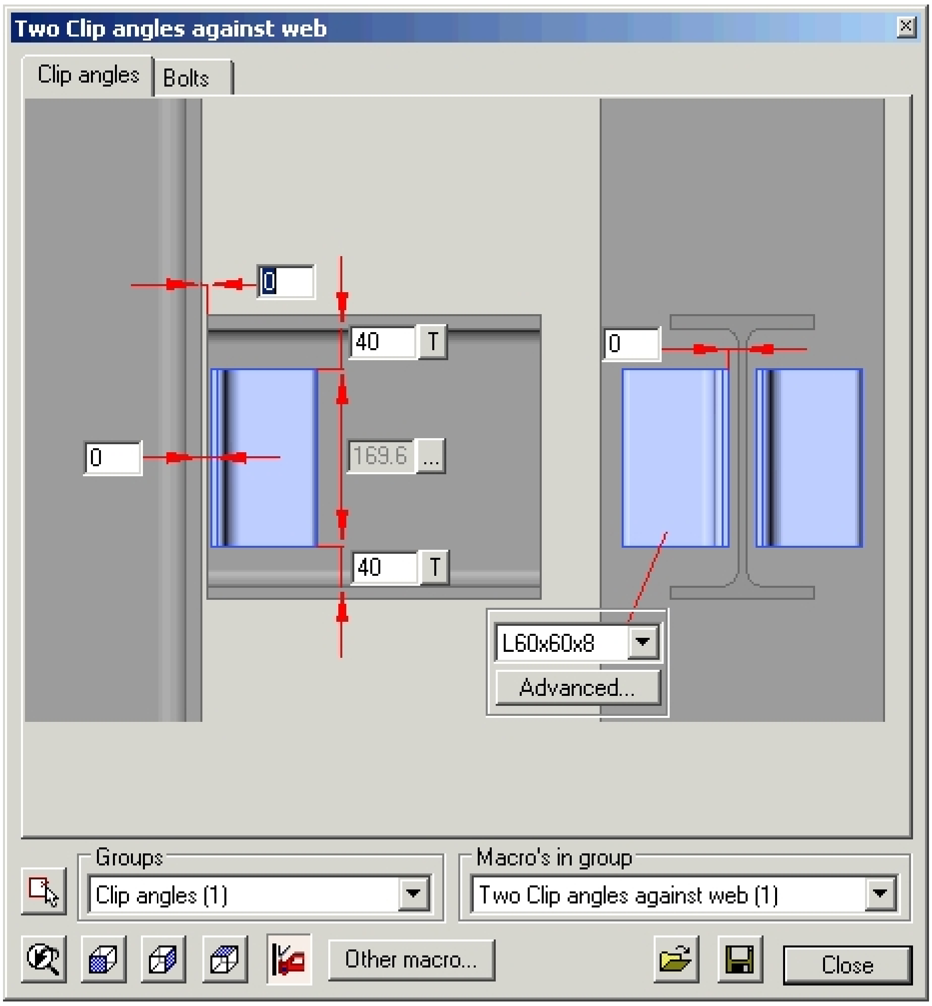
The options explain themselves except in the dialog box that you get if you click on Advanced.

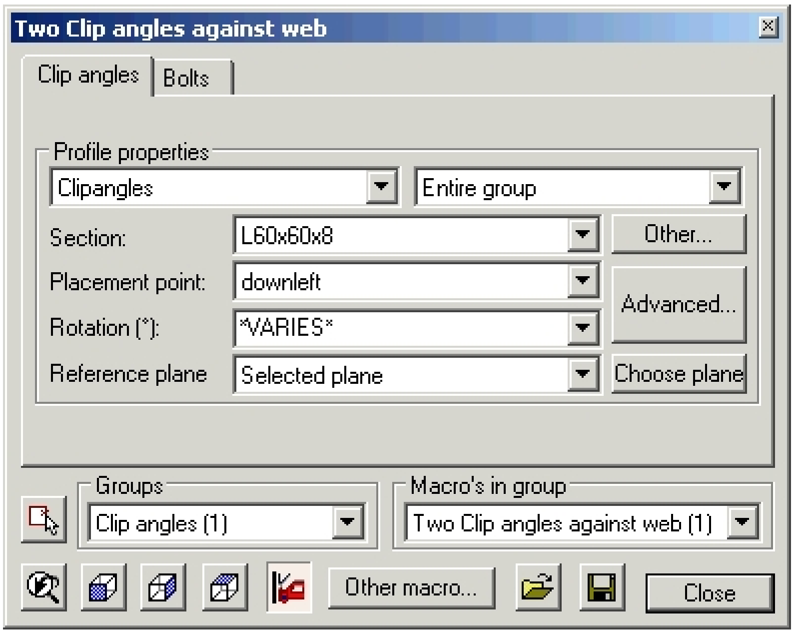
Slot holes.

You can activate the slot holes for each drilled element. Each drilled element receives a number. The first drilled element is the element that lies the closest against the head of the bolt. The numbers of the following elements continue to go upwards from there.

# Profiles tab

This tab contains several properties that have influence on the profiles in the tab.




In the lower part of the tab the placement and the section of the profiles can be modified.
Because this tab can contain several profiles we work with groups of profiles.
With the upper 2 selections you choose which profiles you want to modify.
The first selection is the group.

With the second selection you choose which elements from the group you wish to edit.

In the above example the complete group was chosen. Each profile has its own, unique number like for example 1074631000.

You have the possibility to edit each profile separately by selecting the number in the list.

The options that we see on the tab itself are a limited selection. You obtain a collection of all possible options with the button **Advanced**.

That dialog box is explained in the **Place profile** chapter.

Below we have a button **Segmented profiles**.

Some profiles can be segmented.

The intention is to treat the profile as one profile on a polyline, but to split this profile on chosen distances.

In the dialog box you choose on top one of the segmented profiles.

At this moment you can modify the options of that profile:

**New name**: Modify the unique name of this profile (name for in selection list).

**Offset**: The offset that must be used for each break.

**List with segmentation points**: The list gives the absolute distance of each segmentation point on the profile. You can remove some or add new with the buttons underneath.

The distance starts from the first point of the line (this is the first point which you indicate during drawing the polyline).

The button **Indicate new point on screen** is a tool to add new segmentation points. You can add a segmentation point by selecting a point on the line of the profile. The correct absolute distance of the point that you have selected will be calculated and added to the list.

### Smart copy for macros

# Applying a macro automatically

With this command you can copy an existing macro to other base profiles in the same drawing. First you must select the macro that you want to copy.

Afterwards you must select the base profiles where the macro and all of its elements must be copied to. The new components are just like the original adaptable by means of the new macro “sphere”. These macros will however not be connected with each other:

When you modify dimensions or remove components of the new macro then the original macro will not modify.

### Apply macro manually

# Apply macro manually

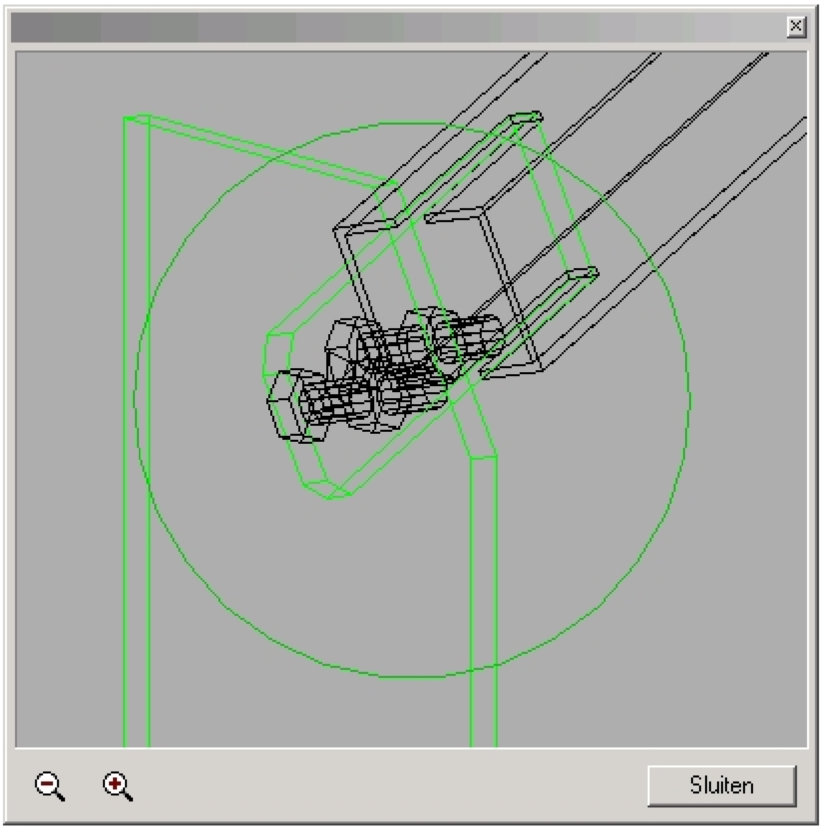
This command does approximately the same such as **Applying a macro automatically**, only you have more control concerning the way the macro is applied.

Normally you can only copy a macro to base profiles of the same type.

Normally, if the base profiles of the macro have both I-sections, then the destination profiles also have to be I-sections.

However for some connections the section does not matter that much, and for this reason you can remedy this restriction with this command.

You are not being asked to select destination base profiles, but some you are asked to select sub geometries of the new base profiles.




After you have selected a macro, a special dialog box appears. In this dialog box you see a new, small drawing. This is the drawing of the source macro.

You can zoom in/out in this window.

In this window 3D-orbit is active: you can modify the view by clicking and dragging the left mouse button.

In the window you should see flickering sub geometry in thick red lines.

You must select the equivalent sub geometries of the destination base profiles.

If you move away from the window with the mouse, you can select the sub geometry in the real drawing (the window remains visible).

After you have selected the sub geometry you should click on the right mouse button, immediately in the window the following sub geometry is indicated of which you must indicate the equivalent again.

The number of sub geometries that you must select depends on the macro.

**Merge macros**

# Merge macros

With this command you merge two macros into one macro. There will be no loss of intelligence.

You need this command if you are composing a new macro by means of components from the library.

# Producing intelligent elements

It is possible to modify the connections in Parabuild, or to develop your own connections.

You do not have to learn programming for this. You must however learn this chapter so that you can draw intelligent connections by means of geometrical rules.

If you want to draw intelligent elements, you must think in a different way concerning the drafting of the 3D-Model.

The traditional manner of drawing each element has been based on coordinates in 3D.

Now those coordinates are replaced by geometrical rules and as a result of this we obtain intelligent elements.

The advantage of intelligent elements is therefore that we draw the intention of elements, not simply the final result. For example we hang the upper part of a base plate fixed to the lower part of a column with a geometrical rule. The consequence is that the base plate automatically moves if the column moves/is extended.

Using geometrical rules it is also possible to draw other things than traditional connections, for example the wire frame of a building.

This can offer large advantages, especially if the drawing must be adapted afterwards.

To learn to draw with geometrical rules we must adapt our way of thinking. At present we have been tuned to coordinate-based drafting.

The fundamental difference between the two manners of drafting is the following:

Coordination-based drafting: The draughtsman puts logic into coordinates and enters the coordinates in the computer. The computer itself on the basis of other coordinates calculates a part of the coordinates. These coordinates are fixed; they modify only if the draughtsman modifies them.

Geometric rules-based drafting: The draughtsman enters the logic of the elements in the computer. The computer calculates the coordinates using these rules. The coordinates can at every time modify because the computer uses the rules each time to (re) calculate the coordinates.

## Geometrical rules

Most rules are based on two geometries. First we will look at all possible geometries that one can use in rules.

### Geometries

These are all possible geometries that one can use in rules:

- point
- line
- circle
- plane

- spline
- ellipse
- cylinder
- helix (diagrid)
- cone
- torus
- sphere

Each element that we draw consists of a collection of geometries.

A rectangular plate consists of the following geometries (we call these sub-geometries of the plate):

- 8 points
- 12 lines
- 6 planes

If the plate has holes or saw patterns, then of course the plate has more sub-geometry. One can also use these sub geometries in rules.

## Rules

A rule defines the relation between two (sub) geometries.

A rule can be set both between 2 sub-geometries of the same element or between 2 different elements.

We look at each type of rule and give an example.

### Parallel

Example:

- 2 parallel planes: 2 sides of a plate parallel to obtain a rectangular plate.
- A plane parallel to a line

### Coincident

Example:

- A plane flat on another plane: The bottom plane of an end plate on the upper cut plane of a column

This rule can also been used between a cylinder and a plane, a cylinder and a point,… Cylinders appear when we use rules on round tubes or bended profiles.

### Perpendicular

Example:

- Perpendicular planes: to obtain a rectangular plate

### Concentric

Because the centre point of a circle or the centreline of a cylinder are not visible, this rule exists to put circles or cylinders on each other.

**Tangent**

This rule can be used only in the following combinations:

Line or plane tangent with circle or cylinder

**Equal radius**

This is a useful rule to reduce the number of dimensions. For example a plate with fillets. One needs to set the radius of only one circle or cylinder, and with this rule all radius can be set equal.

**The following rules are dimensions. Further in the manual it is explained how and when the values can be adapted.**

**Distance**

This dimension is placed between two (sub) geometries.

Pay attention that this dimension is automatically also a parallel rule: we cannot, as it happens, define the distance between two geometries if they are not parallel. It would be therefore superfluous to make two planes or a plane and a line parallel when there is already a distance between them.

**Angle**

This dimension is placed between two (sub) geometries.

To place an angle can be time-consuming, for this reason we recommend to add only an angle dimension if there is no other possibility: If you can obtain the same result without angles, then execute it without angles.

**Radius**

With this dimension you can set the radius of a circle or a cylinder.

## Degrees of freedom

The restrictions that the rules impose on a 3D-model can be seen as removing degrees of freedom of the 3D-Model.

A degree of freedom is the way an element can move in 3D.

Each rule puts restrictions on the 3D-Model. The one rule imposes more restrictions than the other. Also the type of geometry plays a role: two planes that lies on each other put many more restrictions on than two lines which lie on each other. A plane means in that respect more than one line, and a line more than one point. You must therefore use as many planes as possible, because then you have to produce fewer rules and you will reach your expected result more rapidly.

Example: 2 planes on each other say as much as 2 time 2 lines on each other

An intelligent 3D-Model should ideally be seen as having absolutely no degrees of freedom. It is possible to use a 3D Model with degrees of freedom, but this can end in unexpected results.

Imagine yourself the following scenario as an example:

You put a plane of a plate on a plane of a profile. The thickness of the plate was not defined (degree of freedom!).

When you would move the profile, the plate must move too. While calculating your macro Parabuild has a problem: does the plate have to become thicker in the distance that was moved, or will the plate be moved entirely and preserve the same thickness? In this case Parabuild will preserve the thickness. Parabuild will always try to preserve the original form of the plate, but it is not a perfect solution: Parabuild sometimes has to "gamble", or sometimes will not be able to find a solution. To receive results that are always correct you should therefore always add rules until your 3D-model has 0 degrees of freedom.

## Macros and modules

Parabuild uses geometrical rules to calculate a solution. The solution is the coordinates-based 3D-model. The geometrical rules are at all times kept inside a macro, and as a result the 3D-model can be calculated again if data changes. An example of such a change is the modification of the slope of a roof plane.

The following chapters explain how we must draw these geometrical rules in modules.

The commands that are discussed in this chapter are in a hidden toolbar that you must activate first. To activate the toolbar do the following:

- Go to the top of the AutoCAD menu **Tools**, choose **Customize**, and then **Toolbars**
- Select the menu group in the dialog box: **Parabuild-EN**
- Now activate the toolbar **Geometrical rules**
- Click on **Close**

# Macros

A macro is a collection of modules.

The macro itself contains no geometrical rules; it only contains modules.

The macro offers the user functionalities that allow us to reuse the macro in several situations.

As an example we take a haunch connection:

The haunch connection has been based on 2 profiles: the column and the beam.

It is possible to reuse this angle connection in another situation: a case where the beam lies under another slope, or where the column and beam are other profiles (HEA200 versus HEA220).

It is however not possible, apply the haunch connection to U-profiles. The stiffeners are foreseen on the geometry of an I shaped profile. The geometries on which the stiffeners are based are lacking in the U-shape and as a result the macro cannot be applied.

 You can create a new macro using this icon. You are being asked to give the name of the first module. A macro without modules is not useful.

# Modules

A module is a collection of geometrical rules: if you produce a geometrical rule it will always be added to a module.

The purpose of modules is to split up a connection in several logical parts.

There is nothing that obliges you to create a macro with several modules. You can draw everything in one module if you wish.

However working with several modules offers several advantages:

- The calculation work that is necessary for Parabuild to solve the macro is strongly reduced if you work with several modules.
- Modules can be reused in new macros (for example the stiffeners of an angle connection can be reused in a new haunch connection).
- The logical division of the connection in modules ensures a more synoptic macro during the design, especially when it contains many components.

When applying a macro one gets to see the modifiable values grouped by modules.

To prevent modules from contradicting each other, there is a list for each module that keeps which elements the module is based on and which elements it adapts/create.

If a module needs a certain element (or a piece of it), it is possible with one of the 3 “possession degrees”:

- Only based (= fixed). This means that the module **uses** the element **as a basis** and the module cannot adapt or move the element.
- To move (=rigid). The module cannot adapt the element itself, but can **move** it.
- Flexible. The module can **adapt and move** the element.

With these possession degrees it is ensured that modules will never contradict each other. There is, as it happens, but one module that can have an element in a flexible way (the same applies to move). On the other hand there is no restriction in the number of modules that has an element as a fixed element. The “move” possession degree is only needed if an element must never be adapted intelligently but should be moved.

All elements that are being used are added to the element list of the modules with one of these three possession degrees. One exception exists: profiles. A profile can be split up by means of its cuts. It is therefore possible that a module does not posses flexible the profile itself, but it does a cut of the profile. Thus several macros can shorten/extend or cut out a profile without having to posses the complete profile (in a flexible way). Modules can posses both line cuts and polyline cuts separately.

Parabuild calculates modules in serial. If a macro is calculated, then each module is calculated one after the other. Parabuild will choose automatically which module must be calculated first. This depends on which modules are dependant on which modules.

As simple example we take a plate with bolts: we have a module “plate” that entirely defines the plate (width, thickness,…). Then we have a second module “bolts” that defines a bolts pattern on top of the plate.

The bolts module will become dependant on the plate module because the bolts module uses the plate as “fixed”.

Parabuild will calculate the plate first, afterwards the bolts module.

Due to this serial manner of calculating the bolts module can never adapt an element of which the plate module is dependant. This would be a circle of dependence, which can never be calculated. Parabuild does not allow making such a circle of dependence. When you try to make such a circle then  Parabuild simply refuses to create the geometrical rule. Keep this into account when Parabuild does not create your geometrical rule.

## Set macro as current

Because a geometrical rule always must be placed in a module, you must indicate which macro and the module are current/active. All geometrical rules that you create will be placed in that current module. This command is carried out automatically for you when the **Edit macro** dialog box is active. Therefore you only need this command if you use one of the 10 icons of the geometrical rules in the toolbar.

This command also regulates simultaneously the automatic calculation of macros. When you have set a module as current, then no single macro will be recalculated. As you know macros always ensure that its components automatically adapt when a basic fact is adapted. That automatic adaptation is suspended for all macros in the drawing. You can recalculate the macros with the command **Recalculate all** (see further in the manual). Now you decide the calculation of the macros.

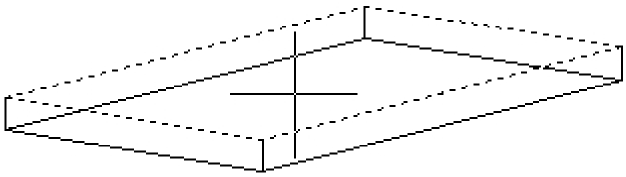
To incite the automatic recalculation, you set no module as current by starting this command and immediately pressing **<Enter>**.

## Creating geometrical rules

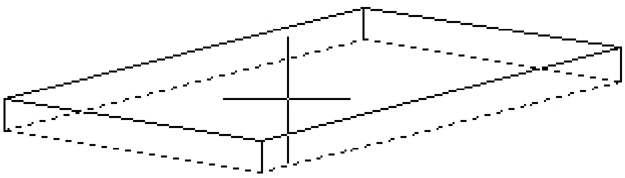
For creating geometrical rules there are icons on two locations. The first range icons are in the hidden toolbar **Geometrical rules**. The second range has been processed in the dialog box **Edit macro** (see further in the manual).

The icons are all the same: they were added twice for the convenience of the user.

While creating a rule you are being asked to select 1 or more geometries. Because we must be able to select sub geometries, the selection works differently than what wee are used to in AutoCAD. On your screen you see a cross as the cursor of the mouse. If you wish to select a plane, then you should move the cross in the plane and click once on the left mouse button. Take care that you crosses' intersection not too close to a line because then you could select that line. Now you will immediately see on screen what you have selected: the lines of the selected plane will become dashed. There are however always several planes behind each other. For this reason, if the desired plane is not selected you should push the left mouse button a second time. You can continue clicking the left button - without moving the mouse – to cover all available planes.

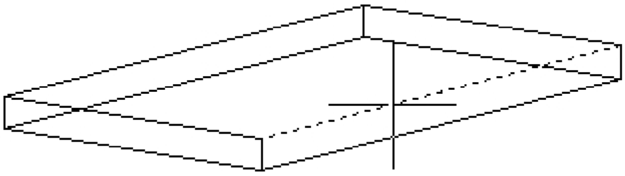


*Selection of the upper plane*



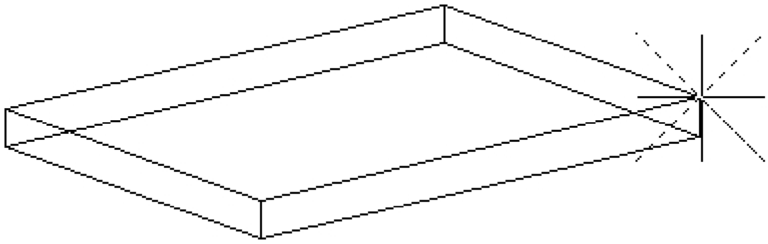
*Selection of the bottom plane by clicking a second time on the left without moving the mouse.*

To select a line you put the intersection of the cursor on a line and click on the left mouse button. On the screen you see immediately which line you have selected as a dashed line. If the selection is wrong you can try simply again with the left mouse button.



*Selection of a line*

To select a point you move the intersection point of the cursor to an endpoint. On the screen you see that a cross appears on the location of the selected point.



*Selection of a point*

If you are satisfied with the selection on the screen – whether this is a point, a line or a plane - then push on the right mouse button or <Enter> to go to the next selection/question.

After you have selected all sub geometries the dialog box Edit macro will be opened. The macro and module in which you have created the rule is active immediately. The geometrical rule that you have just created will also be selected so that you can immediately modify the options for that rule. More information about this see further in the manual.

## Calculate all macros

 You need this command while you are creating geometrical rules.

It calculates all macros that were modified in the entire drawing (geometrical rules added, position changed,…)

After you have created or adapted a geometrical rule, the macro will not be automatically recalculated.

**This is useful because this way you can recalculate the macros only after a large part or all geometrical rules of an element were created.**

If an element was not defined entirely with geometrical rules, then you leave some variables to the computer concerning the placement/size of an element.

It is the current placement of the elements that the computer will try to preserve (as far as possible).

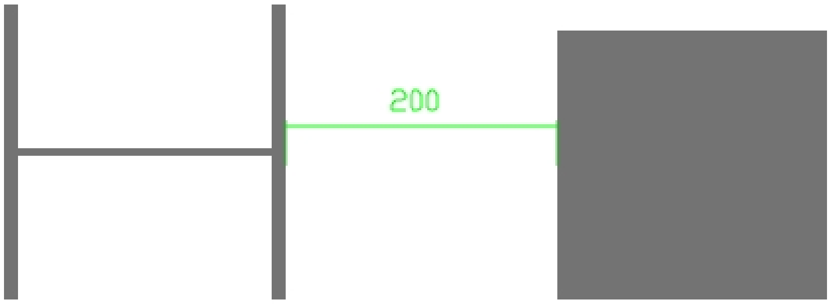
It can however happen that when your elements were not defined entirely and you recalculate, that something unexpected happens with your element (ex. the element is placed some meters further away so that it disappears of your screen).

For this reason it is useful that you decide yourself when the elements should be calculated, and if necessary make the calculation undone with UNDO so that your elements move back to their original place.

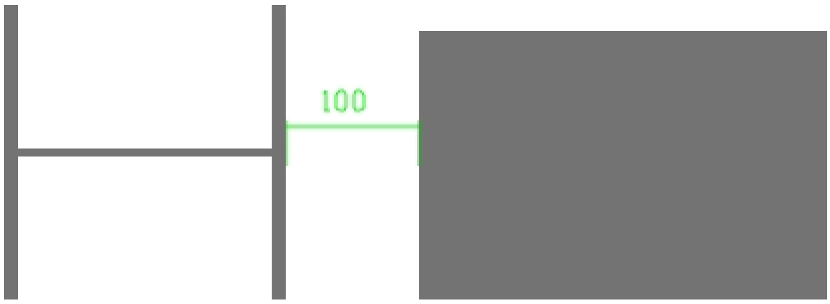
We explain this problem what more closely by means of a concrete example:

In this example we have a plate next to a profile.

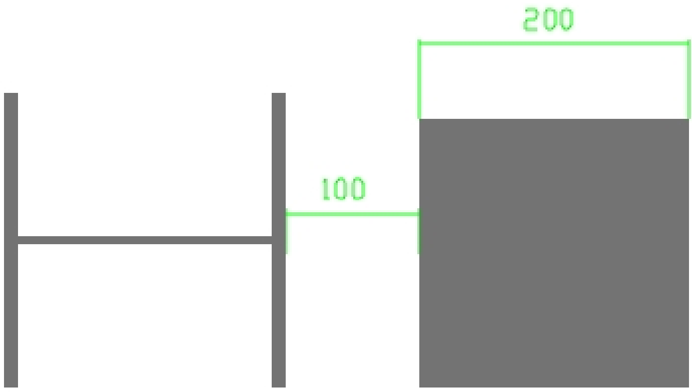
The profile is fixed (we do not adapt). The plate is flexible (move + adapt).




*In this image only one geometrical rule was added and that is the dimension that you see.*




*In this image the value of the dimension was changed and the macro was recalculated. Because the plate was far from defined we let the computer gamble to find a good solution. In this case the plate was made longer, it is also possible that the plate would be moved.*




*The solution: better define the plate.*

Pay attention that it is also possible that the plate in this example could move in altitude or along the profile. It did not happen in this case because the computer tries to preserve the original placement. This system will not work in every situation the way you expect it to. You cannot count on Parabuild always preserving the correct placement.

**This calculation command in combination with one UNDO command is a skilful manner to test your solution while you are placing geometrical rules.**

**Take care that the elements in your macros were entirely defined!**

## Edit macro

You will need this extensive dialog box for the editing of a macro.

This dialog box can only edit one macro at the same time.

Just like most dialog boxes the upper options have influence on the bottom options. If you modify something at the upper options, the options below it will modify. We will first discuss the 4 buttons and the list on top of the dialog box:


**Other macro**: Click on this if you want to edit another macro.

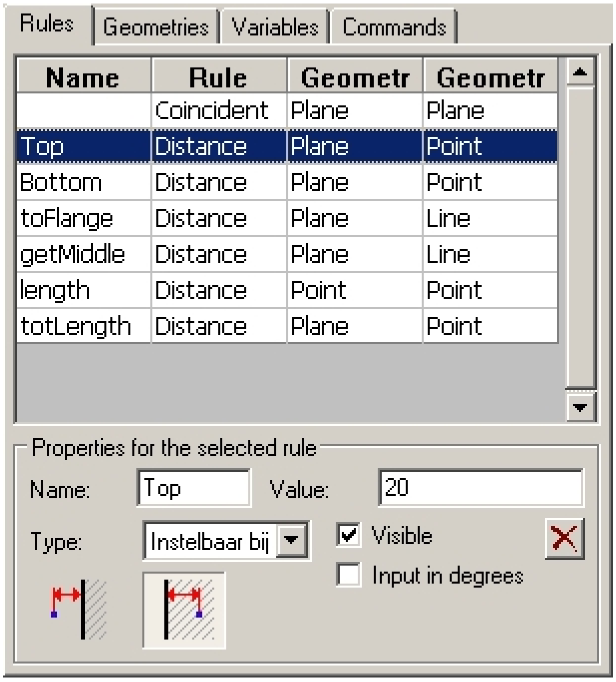
**List**: This is the list with all the modules in the macro. Select the module that you wish to edit. The options in the 4 tabs below this dialog box are the options of the module that you choose here.

**+:** Adds a new module.

**Rename**: Modifies the name of the module that was selected.

**X**: Removes the module that was selected.

## The tab rules

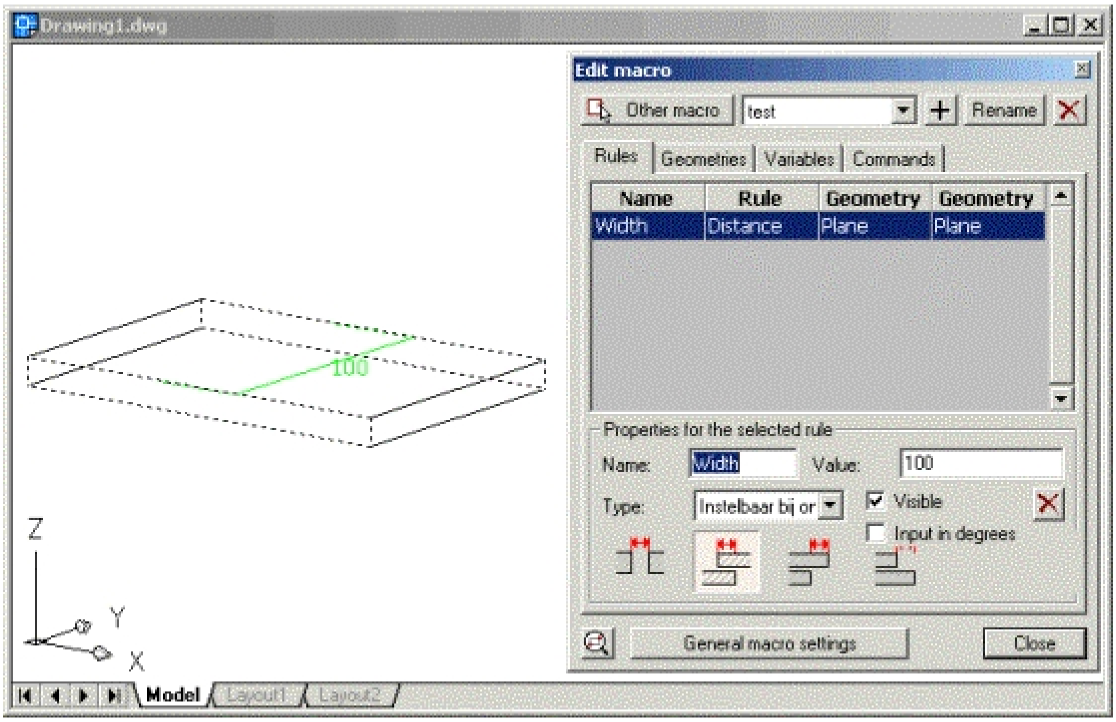



This tab contains a list with an overview of all geometrical rules in the chosen module.

One line in the list presents one rule. The list contains 3 columns. The first column is the type of rule. The second and third column describes the sub geometries of that rule.

If you select a rule in the list two things will happen:

- Below the list, all options of the rule are shown such as dimension name, value,…
- In the drawing itself the geometries of the rule will be indicated with dashed lines. In the example shown below we see that two planes of the plate in a rule defines the width of the plate.




# The properties of rules

The first four properties are only available for dimensions.

**Dimension name**: We use the name as a unique recognition of the dimension in the module. The name must contain at least one character. The name can contain numbers as long as the first character is not a number. A name can only be used once by one dimension within the module.

**Visible**: This makes the dimension visible in the **Revise macro** dialog box. You can hide dimensions that contain comparisons and thus cannot be modified. It can also be useful that one can see the value of a dimension without it being adaptable. You can create a dimension with the only purpose of it to view the value of that dimension: for value of the dimension you give a number (no comparison) and you set the type as flexible.

**X**: With this button you can remove the selected rule.

**Value**: This can be an ordinary number, a dimension name or a comparison. You can enter a comparison, for example “Length1+Length2”.

You can use the following symbols to make comparisons:

- **+-/***: To add up, deduct, divide and multiply.
- **sin () cos () tan ()**: Calculates the sine/cos/tangent of the value that stands between the brackets. The value between the brackets can itself be a variable or comparison.
- **() ^2**: Power of 2.
- **sqrt ()**: The square root.

**Type**: The type determines the flexibility of the value of the dimension:

- **Adjustable at design**: The value of the dimension is a number and can be modified during the use of the macro (the dimension will be adjustable in the **Revise macro** dialog box!).
- **Fixed value**: the value must be a number. This value is not adjustable afterwards during the use of the macro.
- **Flexible value**: You must use this if other dimensions can influence the value of the

dimension. If you enter for example an equation then this option must be used (this is generally already done automatically for you).

# Directions of rules

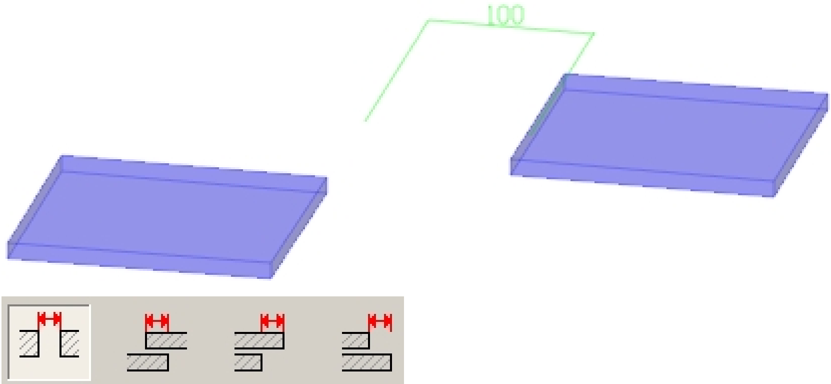
As last property you must select one of the images mentioned below.

The images depend on both the type of rule and on the sub geometries.

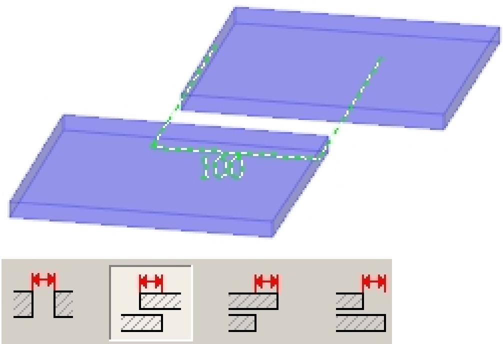
The image that you select indicates the context in which the rule must be interpreted.

We clarify using some examples:

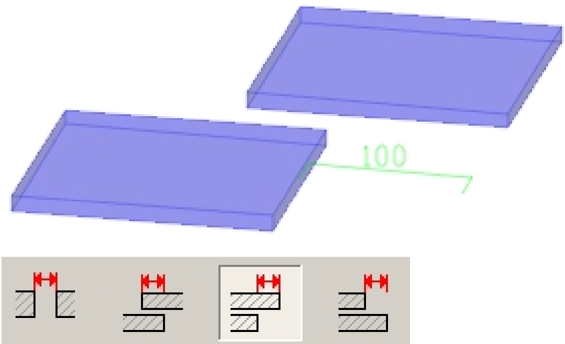
We will see some scenarios of a dimension between two planes of different plates:



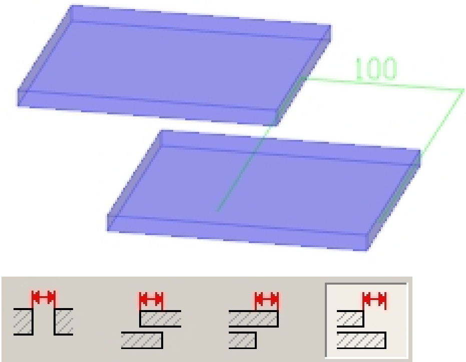

*Here we've chosen an empty space between the plates.*



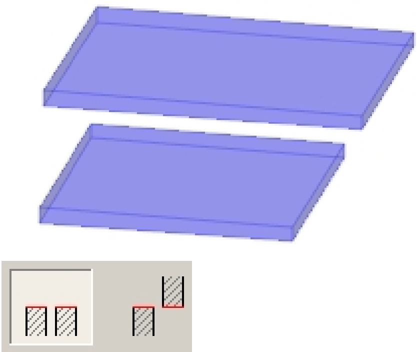

*Here we let the elements flow in each other over a distance.*



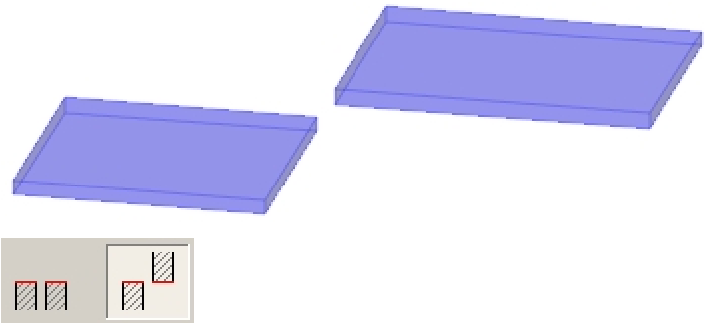
*The one plate is put here beyond the other. Which plate is put beyond which plate depends on the order in which the sub geometries were selected during creation of the rule.*




*Again one plate is put beyond the other but the roles are switched.*



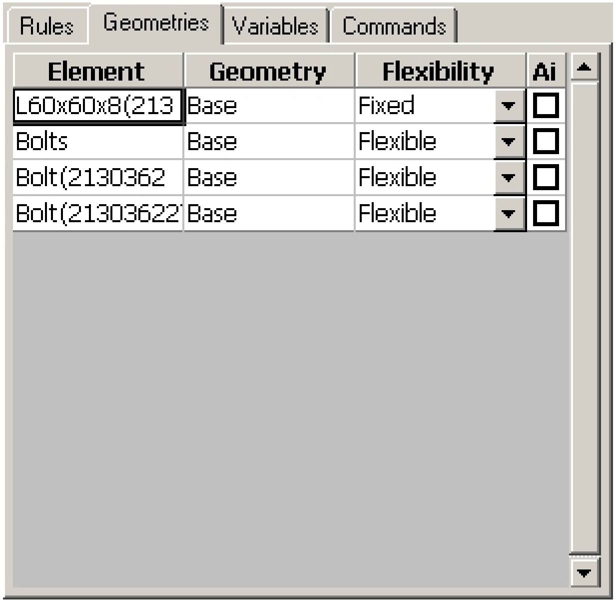
*This is the coincident rule. In this case we put the two right planes on each other with the "body" of the plates in the same direction.*



*The planes in the middle become placed on each other with their body in opposed direction*.

There are other situations possible such as distance between plane and line, cylinder and line,… but the functioning is always similar.

## The tab geometries




This list contains all (sub) geometries of which the module is dependant or which the module defines (=flexible).

Each line is one geometry.

This list is created automatically when you add geometrical rules to the module.

The purpose of the 4 columns:

**Element**:

The name of the element. The unique number of this element is also visible between brackets. This number serves to be able to recognize in this list several elements with the same name.

**Geometry**:

The component of the element that it refers to. If this is Base, then it refers to the entire element. For a profile this geometry can be something other than Base, namely Cut x. It refers to only the cut of the profile with that number.

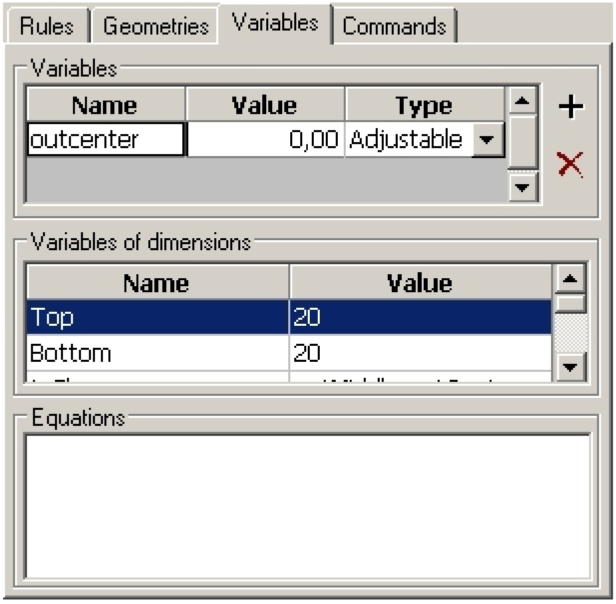
**Flexibility:**

- Fixed: The element is only used, cannot be adapted/moved.
- Flexible: The element can be adapted and moved.
- Rigid: The element cannot be adapted, only moved.

**Aid geometry:**

If you activate this checkbox, then the geometry will become invisible when the macro is not being adapted. This is intended for geometry that is only used to calculate a certain position. An example of its use is the apex connection: this contains a hidden coordinate system (what is this? see further in the manual) that determines the intersection point between the two beams.

## The tab variables

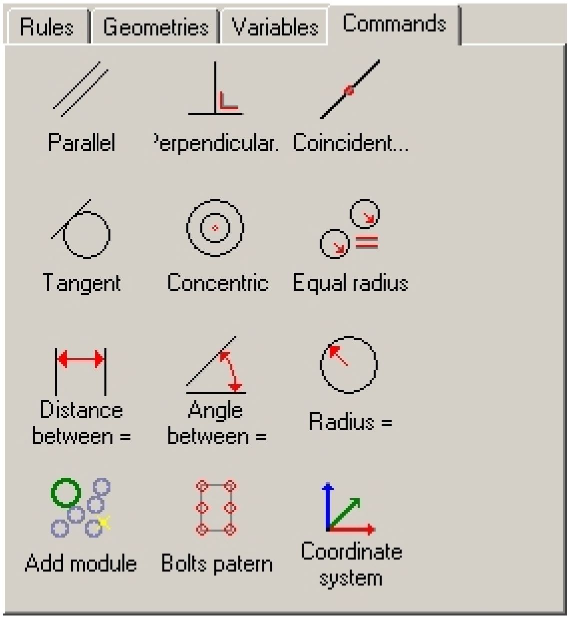



At producing a rule we already use automatically variables and comparisons. The name of each dimension is, as it happens, a variable that you can reuse.

In the value of a dimension you can enter a comparison (enter for dimension A the value B: you have made the comparison A=B).

You only need the lower part of this tab if you cannot add enough comparisons in the values of dimension names (that will seldom occur).

## The tab commands




With these commands you add a new geometrical rule or something else to the current the module.

All these icons are the same as the icons that you can find in the toolbar "Geometrical rules".

The 3 icons on the bottom are discussed in order hereafter.

# Add module

This command adds a new module to the current macro.

The number of modules that a macro can contain is indefinite.

As you already know the primary use for a module is to add geometrical rules and for the organization: - which element depends on which element -

Beside this functionality there are still other uses for modules, namely adding bolts and profiles.

There are 3 types of modules:

- The **ordinary module** contains only geometrical rules
- The **bolts module** contains geometrical rules and can also contain bolt patterns. The intention is that the bolt pattern determines the placement of several bolts. For more explanation: see the command **Bolts pattern**.
- The **profile module** contains geometrical rules and can also contain profiles that are based on a base line. To add a profile to the module is only possible using the **Place profile** dialog box. The correct orientation of the profile is modifiable in the **Revise macro** dialog box, in the tab of this profile module.

## Bolts pattern

You can only add bolt patterns to a bolts module.

The problem with bolts is that we need a range of bolts and the number of bolts must be flexible. You cannot draw a flexible number of bolts just like that with geometrical rules.

For this reason the bolt pattern was made. We produce a pattern that can be determined with geometrical rules. The placement length and width of this pattern will determine later where the bolts must be placed. At any moment the end-user can adapt the number of bolts in the **Revise macro** dialog box.

There are 3 types patterns:

- **Line**: Allows us to create one row of bolts.
- **Rectangle**: Allows us to create several rows of bolts in the rectangle.
- **Circle**: Allows us to create a circle of bolts (think of pipe connections)

When starting this command, you are asked to indicate the type, name and placement of the new pattern.

The name of the pattern will later serve to recognize the pattern in the **Revise macro** dialog box. A bolts module can contain several bolt patterns.

The placement that is asked is only for your convenience and is not the definitive placement. You should define the pattern entirely with geometrical rules. In the event that you draw a rectangular pattern you constrain the plane of the rectangle on another plane (probably the plane of a plate) and fix the four lines with 4 dimensions (this is a typical example, it can also be done differently as long as the rectangle has no degrees of freedom).

## Coordinate system

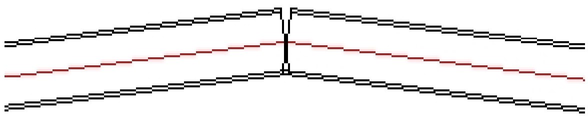
A coordinate system is a type of UCS object in the drawing. This object has no practical use for the output of the drawing itself (this object will never appear in a bill of material or workshop drawing).

This object serves only as aid object for the placement of intelligent elements in certain geometrical situations.

This object has 3 lines, 3 planes and a point, each in its own colour. You can use all of them to add geometrical rules.

As you have already seen, each drawing has such a coordinate system that is fixed on the World of the drawing and that you cannot remove. We can also use this special World coordinate system in geometrical rules (that is it's sole purpose). It is useful for connections that are based on the “world plane” of the drawing (for example a base plate).

### We take the apex connection as an illustration example:

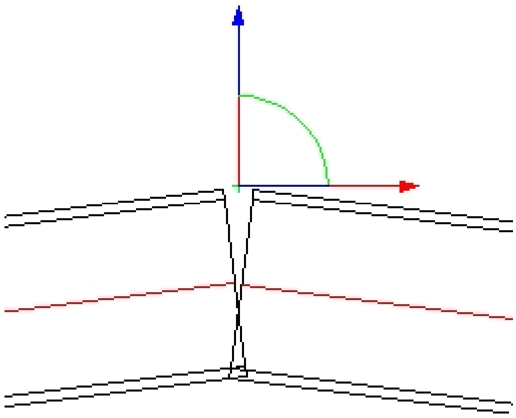


The problem: 2 beams at an opposite inclination. We must create 2 end plates on the intersection point.

There are ways to solve this without using the coordinate system by directly adding geometrical rules on the end plates. However to achieve it that way would require many (difficult) rules and therefore also a lot of thinking work, both for you while adding the rules and also for the computer when solving these rules.

If we want to solve a difficult problem in an easy way, we should do it by splitting the problem up in pieces, and by solving those pieces individually.

Here the coordinate system and the modules come into the picture. We use the coordinate system just to calculate the intersection point of the profiles. As soon as this is done, we use this coordinate system as a basis for the end plates. We solved the problem in two pieces.



We create the geometrical rules so that the coordinate system lies with its point on the intersection of the profiles. The correct orientation of the coordinate system must then still be defined, in this example a plane was made parallel with the World coordinate system.

Then we have to base the end plates on this coordinate system. It is a good idea to create the end plates in a separate module. This way you will see the rules for the coordinate system and the rules for the end plates each grouped separately.

## General macro settings

You can start this dialog box with a button at the bottom of the **Edit macro** dialog box.

This dialog box contains 5 tabs that do not all have a connection to each other.

We look at all options of each tab.

The two tabs **Filters on angles** and **Filters on distances** contain options that are used during the application of the macro from the library. With these options becomes decided if the macro is appropriate for the base profiles which the user select or not.

# Filters on angles

It is not obligated to fill any of the filters. You fill in filters only if you do not want a connection to be applicable in certain orientations.

For each filter type we will find a list with combinations. Each element in the list represents a filter. As soon as you select a filter in the list, all its options become visible. Only the filters that you activate will later be used.

### The same orientation at the smallest angle is obligatorily

This is a useful option for base profiles with non-symmetrical sections. Imagine you a gusset plate connection for an angle profile. For such a connection two different macros must be made. With this filter activated only the applicable connection will be allowed. The illustrations clarify this.

### Filters on angles of the axis

You can produce filters on the angle between the axis of the base profiles and another line/plane. The angle between the elements must lie somewhere between the minimum and maximum values that you specify, otherwise the macro is considered as not appropriate.

In the list there are also filters that are in fact ordinary filters on the orientation of the base profiles versus the World of the drawing. Entirely below there is also a filter that filters on the orientation (angle) between the base profiles themselves.

### Filters on angles between axes (A) and sections (X)

If we look at the section of a profile, then we can obtain a line from that section (x). Here we add filters on the angle of this line and the axis of another profile.

# Filters on distances

### Filters on the distances between axes

Here we can add filters that require that the base profiles should not be positioned too far from each other or just the opposite.

A apex connection or a haunch connection can use these filter well: it is not desired, as it happens, that the base profiles are positioned 1 meter besides each other. It can however occur that the profiles are not exactly positioned on each other. In the case of a haunch or apex connection we can use as example a minimum of 0 and a maximum of 100.

(0 = the axis cross each other!)

This filter can therefore also be used for connections where we do not want that the profiles intersect each other. An example of this is a connection where one profile lies on top of the other. In such a case we would take for example a minimum of 100 and a maximum of 1000.

### Filters on the size of the section

You can limit the size of each section. It can be useful not to allow certain small sections or not to allow certain large sections.

## Module properties

When we obtain a macro from the library and apply it automatically in the drawing, then we always see a dialog box with a choice of several macros. At the top of this dialog box are the properties of all the macros in the list.

With this tab you can enter all the properties of this macro, so that this macro appears or disappears from the list according to the properties that the user enabled.

The properties are kept for each module. Therefore first select the module at the top and then modify the properties for that module.

**Image of this module (optional)**: Select from the list a dialog box image. An image dialog box is an image on which all modifiable dimensions are placed. These dimensions are coupled with a dimension name. More information about this: see the chapter **Dialog box design**.

## Variables

The general variables that you enter here will be available in the value of dimensions or in equations in all modules of this macro.

This is very useful for situations where a distance needs to be set just one time, but it has to be used in several modules. Example: the thickness of 4 stiffeners in 4 separate modules that of course always must be the same.

The general variables of a macro are always adjustable in the **Revise macro** dialog box.

There are also general variables that are stored in the drawing instead of in the macro. This means therefore that these variables are established one time for all macros in the whole drawing. A practical example of this is the variable for the welding offset between welded elements.

You can create these drawing variables in the **Parabuild Settings** dialog box > tab **Global** > button **Variables of macros**.

You do this in the drawing that contains the macro and you use the names of the general variables in the macro. As soon as a macro that contains drawing variables is copied to a new drawing, and this drawing does not have general variable, the variable in the drawing will be copied.

All these general variables always start with **gen_** (gen comes of general = commonly used).

If you do not enter this gen_ prefix of the name, then Parabuild will automatically add the gen_ prefix.

## Groups

First of all you must enter a short and a long name for the macro.

Example **short**: Haunch **long**: Haunch with end plates, reinforcements and stiffeners

These two names are used in the **Revise macro** dialog box when several macros are selected: it allows editing these macros simultaneously (those with the same short name).

Below you can enter a group name for each module that exists in the macro.

Each module gets a tab in the **Revise macro** dialog box.

In case you have many modules and you want fewer tabs (many modules is the best manner of working) then you should give the modules that must be merged a common group name.

For example two stiffeners that were each placed in another module: "Stiffener-left" and "Stiffener-right". You give the group name "Stiffeners" to both modules and the options of both modules become merged in one tab called "Stiffeners".

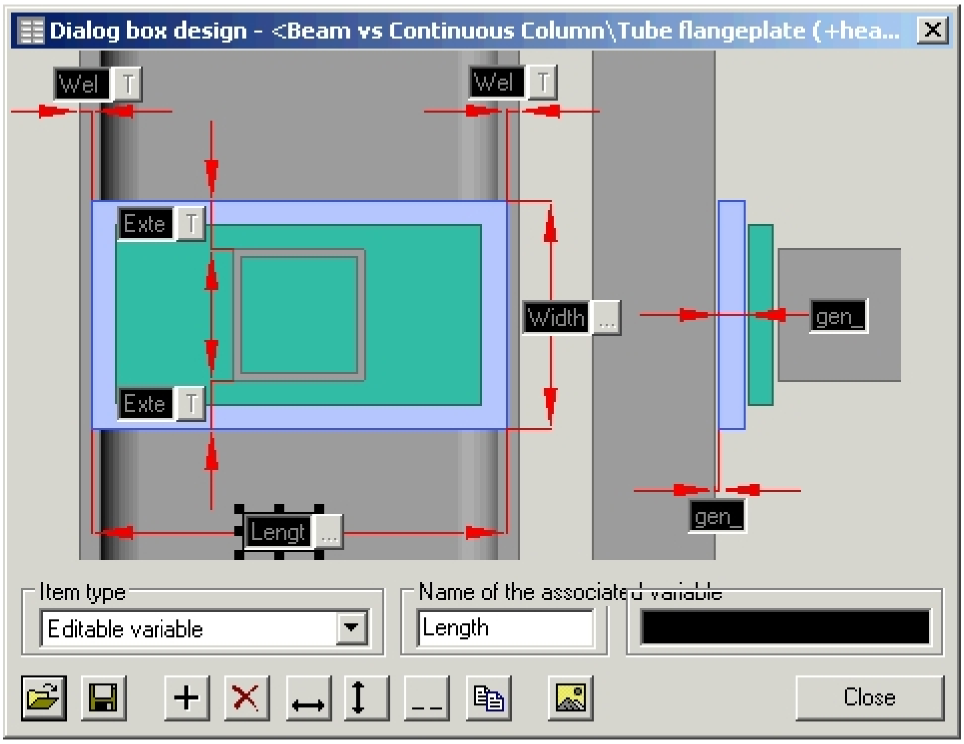
Comparing the names of the adjustable dimensions will merge the tabs.

In case they have the same name they will be merged into one dimension. When the user modifies this option then that modification will be made for both dimensions in both modules.

**Sorting number**: Using this number you can determine the order in which all modules appear in the **Revise macro** dialog box.

## Dialog box design

 With this command you can create or edit a dialog box. Each dialog box will appear as one tab in the **Revise macro** dialog box if the dialog box was chosen in a module.




First of all one must create a bitmap file (with extension .bmp).

With this button  you can load the bitmap.

 With these save/open buttons you store the complete dialog box as a certain filename.

Use these buttons to add or to remove a dimension control.

Use these buttons to change the size of the selected dimension controls.

When you select a dimension control, then you will see 4 black squares. With these squares you can change the size of the control. You can also move the control by dragging it.

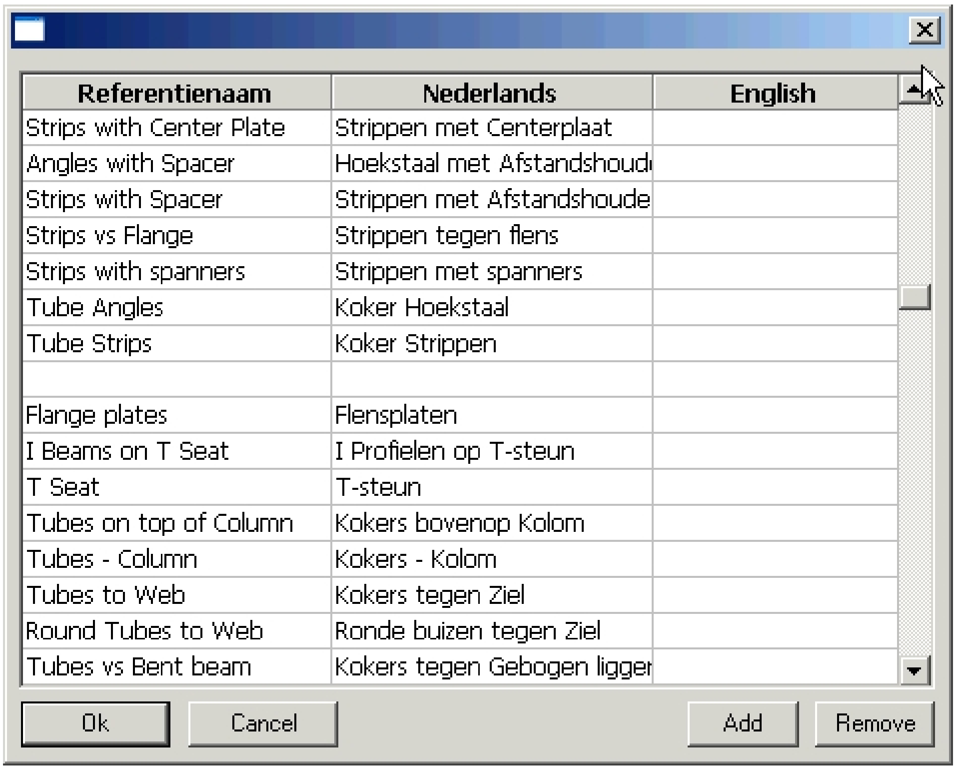
Also the properties of the control are adjustable below.

Enter a type (is it an ordinary dimension? A number of bolts? A profile's section?).

In the second property you should enter the name of the dimension name, the name of the bolts pattern or a profile group.

## Automatic text translations

With this command you will se a table with translated texts. Such as you have noticed all names of dimensions and macros are placed in English. These English texts will be translated to other languages in the **Revise macro** dialog box using this table.



| Referentienaam | Nederlands | English |
|---|---|---|
| Strips with Center Plate | Strippen met Centerplaat | |
| Angles with Spacer | Hoekstaal met Afstandshoud | |
| Strips with Spacer | Strippen met Afstandshoude | |
| Strips vs Flange | Strippen tegen flens | |
| Strips with spanners | Strippen met spanners | |
| Tube Angles | Koker Hoekstaal | |
| Tube Strips | Koker Strippen | |
| | | |
| Flange plates | Flensplaten | |
| I Beams on T Seat | I Profielen op T-steun | |
| T Seat | T-steun | |
| Tubes on top of Column | Kokers bovenop Kolom | |
| Tubes - Column | Kokers - Kolom | |
| Tubes to Web | Kokers tegen Ziel | |
| Round Tubes to Web | Ronde buizen tegen Ziel | |
| Tubes vs Bent beam | Kokers tegen Gebogen ligger | |



In the very first column you fill in the reference name. This is the name of a macro or a dimension.

In the following columns you enter the translated text for that reference name, for each language.

When Parabuild encounters a text, it will search for an exact text match in the reference name column. If a matching text was found, then the corresponding text of the current language will be used.

## Edit macro groups

As you already know we have access to the macros library by means of several icons.

Each icon contains a “group” of similar macros.

With this command you can edit these groups and produce new groups.

Each macro in such a group must have the same number of base profiles.

Furthermore there are no obligations that macros can be put in which group, but it is of course useful that we subdivide the macros in logical groups such as haunch connections, apex connections, end plates,…

As soon as you start this command you will get the **Add groups of macros** dialog box with on top a list of all groups.

Choose the group from the list that you want to modify: the options of that group will become visible underneath the list.

At the bottom of the dialog box a list with all drawings that belong to the group appears (a drawing can contain only one macro that will be applied automatically).

Besides drawings you van also add folders to this list.

A folder means that all drawings in that folder belong to the group.

This is very useful because if you save all macros of a group together in one folder then you only have to set the options to this group once and when you add a macro, this macro will automatically belong to the group.

After you have produced a new group you also have to create an icon that gives direct access to that group. You do this by producing a new icon that starts the following command:

(S3d_MacroGroup "Haunch connections")

The brackets and quotation marks are obliged!

Between the quotations mark you must fill in the unique name of the group (the unique name is the very first option of a group in the **Add groups of macros** dialog box).

## Macro apply settings

With this command you define how the software that automatically applies macros must react to new situations. The automatic applying is among other things started when we obtain a macro from the library. You must define these options for the macro source (the macro that resides in the drawing in the library).

The new, unconnected base profiles that the draftsman selects are compared to the base profiles of the macro from the library. With the aid of the options set with this command it will be decided firstly if the automatic applying is possible. If applying is possible, then the order and the location of the points that you selects in this command determines how the macro will be applied in all situations.

When starting this command you are first asked to select the macro of which you want to adapt the options. These options are stored inside each macro separately.

Afterwards you are asked to select the base profiles of the macro one for one.

The order in which you select the base profiles will also later determine the way the macro will be applied: for example an haunch connection: you select first the column and then the beam. When this macro is then being applied somewhere else then one must also select first the column and then the beam so that the connection will be oriented correctly.

Sometimes not only the order in which you select the base profiles matters, but also the position on the profile that you indicate can play a role for some connections. A bit further we explain this with some examples.

After you have selected all base profiles there will appear a dialog box with some options:

**Does the orientation of the base profiles have to be the same?**

If you enable this option, then the orientation of the base profiles of this macro will be compared with the orientation of the new base profiles and applying will be refused if the orientation differs too much.

This option must always be enabled except for some special macros that allow an orientation difference.

**Don't allow mirrored situations**

The macro will never be applied if the new situation would be a mirrored macro.

Example: A connection with a U profile as base profile.

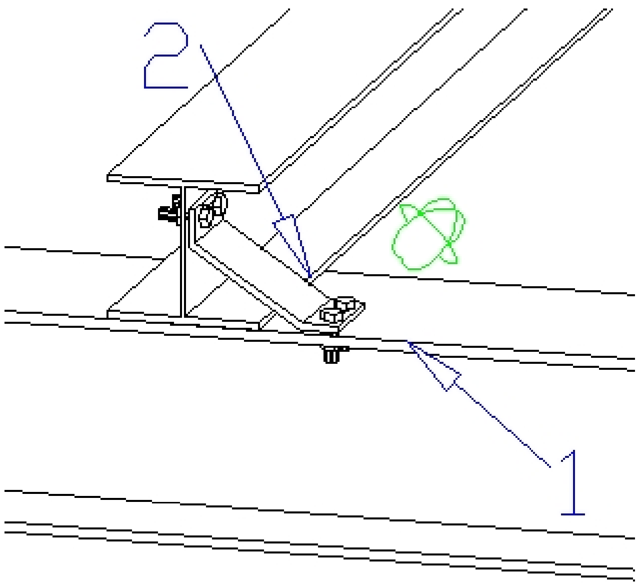
**Allow but avoid mirrored situations**

Avoid mirrored situations if possible, but mirror if it cannot be applied differently.

**Allow mirrored situations**

Always apply mirrored situations.

The following connection is an example where mirrored situations play a key role and it also explains the importance of where you select a base profile.

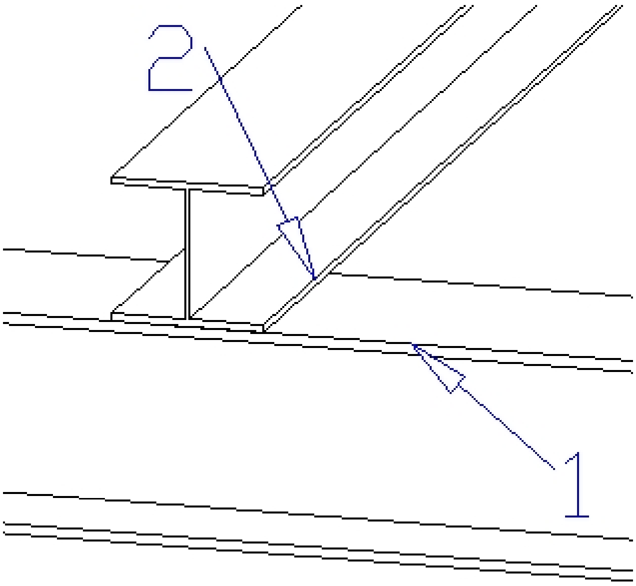


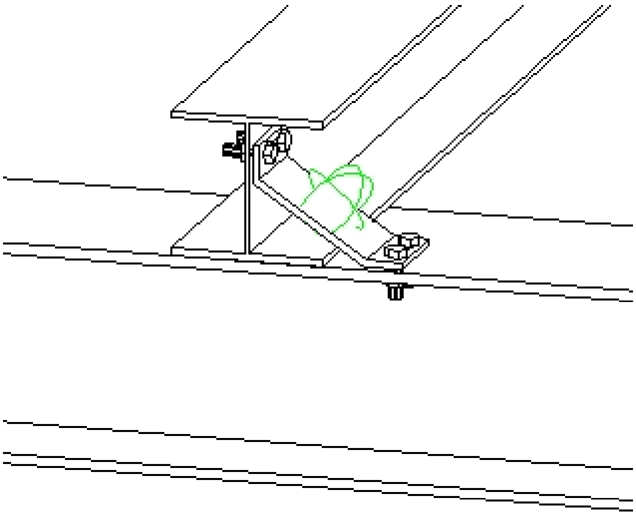

The above image is the image of the macro source itself (the drawing that resides in the macro library). The indicated points 1 and 2 were clicked in the **Macro apply settings** command. The bottom profile is therefore the first; the upper is the second base profile. The above image only illustrates the settings of the source macro, because it influences the way the macro will be applied.

With the following four scenarios we illustrate what happens if during the application of the above connection, the base profiles are indicated on other locations.

For each scenario the left image shows the selection that was made for applying of the macro. The right image is the result after the macro was applied with those selections.

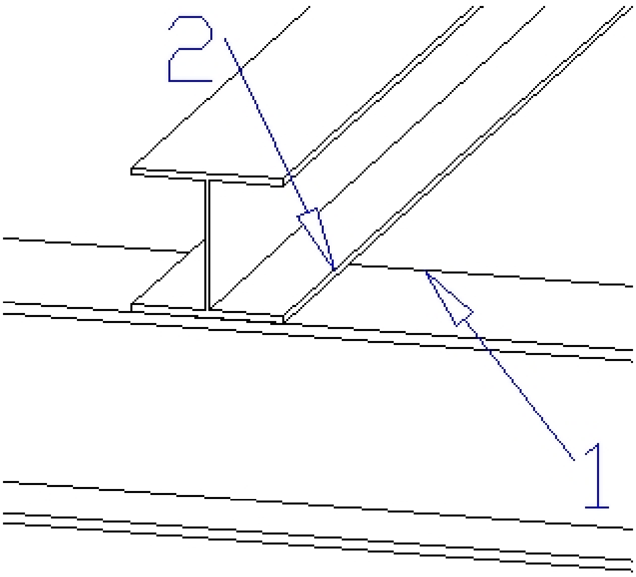
Scenario 1:

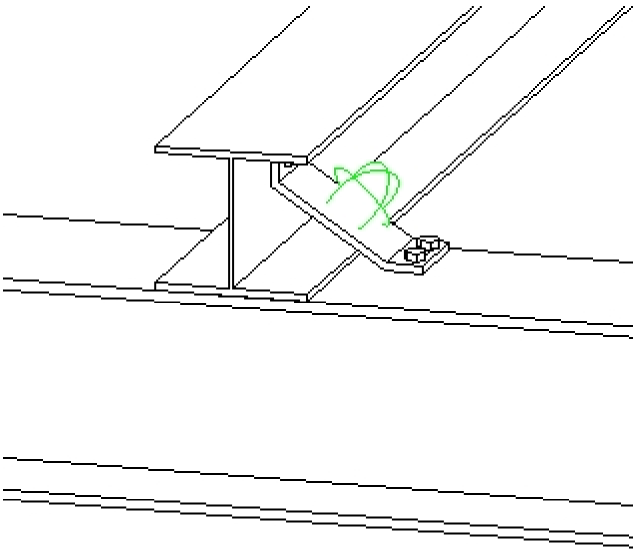


The result of this scenario is no surprise: the base profiles were selected on exactly the same spots as those of the macro source. The macro is copied to exactly the same place as the source macro.

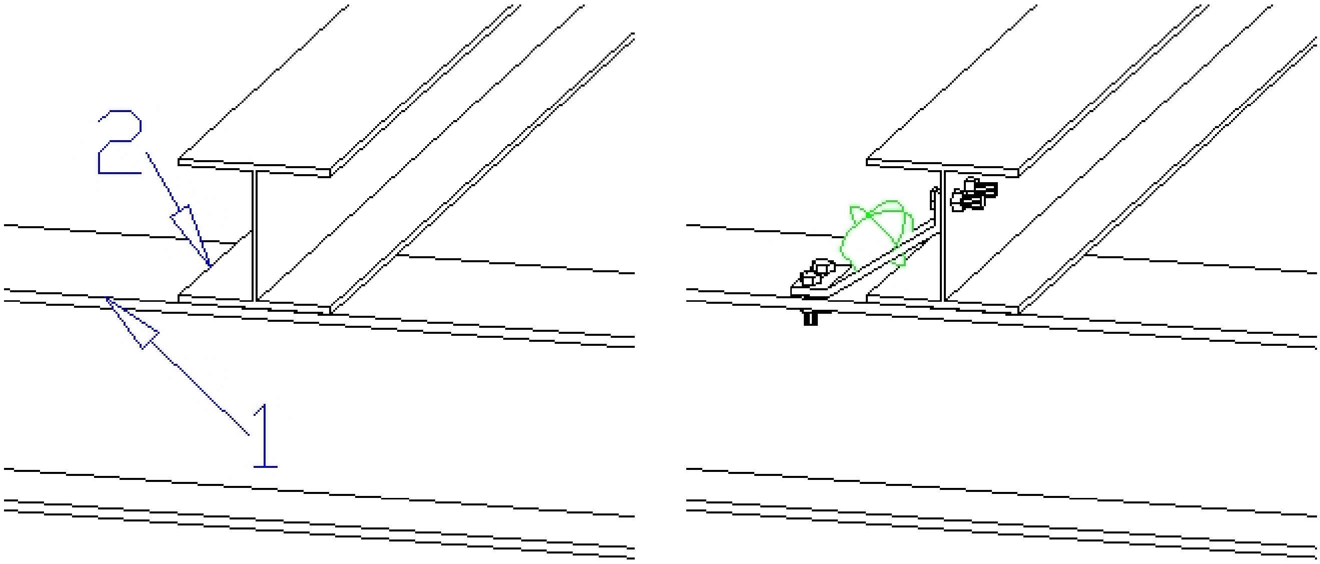
Scenario 2:



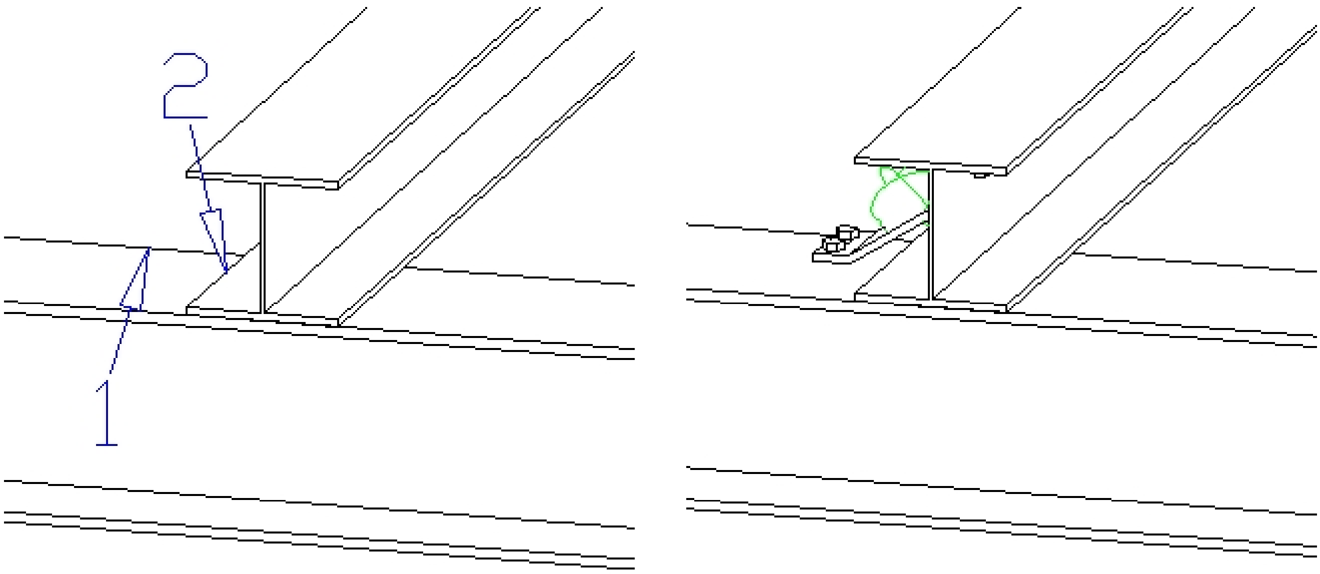
In this scenario the first base profile was selected on another spot, on the other side of the upper flange. The result is that the bent plate is also placed on that side of the flange. The bent plate was mirrored over 1 axis.

Scenario 3:




In this scenario the second base profile was selected on another spot, on the other side. Again we get a mirrored plate but at the opposite side of the second profile.

Scenario 4:



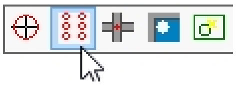

In this last scenario both the first and the last base profile were selected on another spot. The result is a double-mirrored plate.

As you can see, some macros can be copied in a lot of ways. With other macros, such as a haunch or an apex connection, only the order of the selection matters.

## Drawing bolts

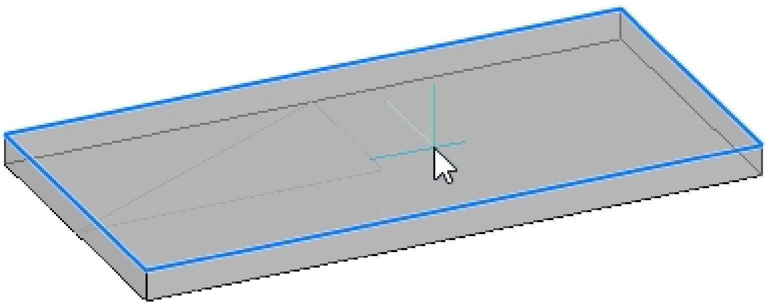
# Draw a range of bolts

Commando : **S3d_BoltsOnPlane**



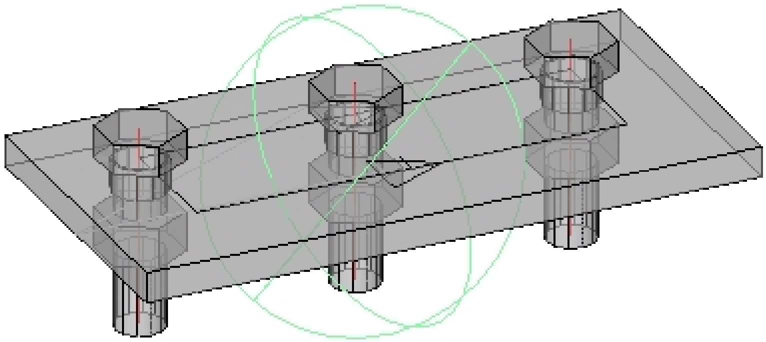
With this tool you can quickly draw a range of bolts.
When you start the command, you first have to select a plane on a plate or profile.
You can select this plane by moving the cursor inside a plane, and then clicking the left mouse button.

The circumference of the plane will be drawn thicker so that you can see the selected plane :



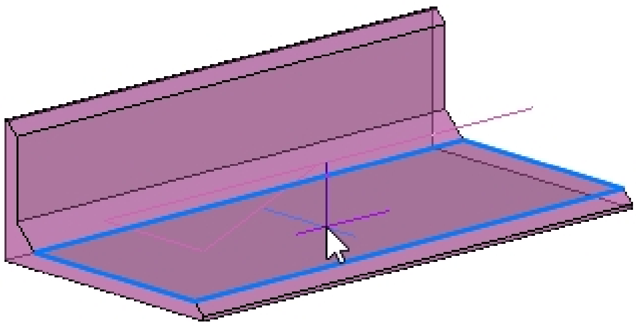
To select a different plane, you need to move the cursor and the left-click the mouse again.

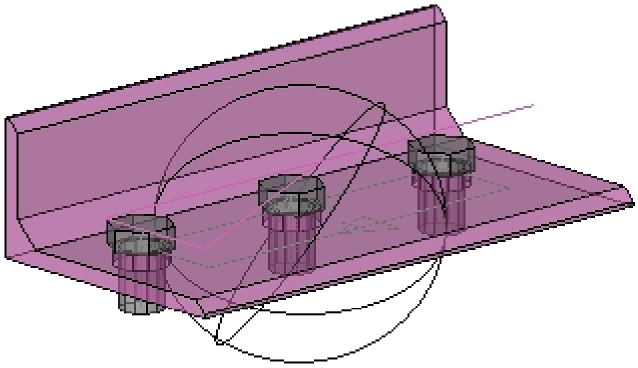
To accept the plane selection, press the **<Enter>** key.



*The result when the top plane of a plate was selected*

This is an example of a profile :

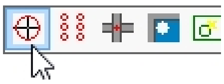




The new bolts are added to a macro.
This means you can adapt the location and amount of bolts later at any time, using the command Review macro.

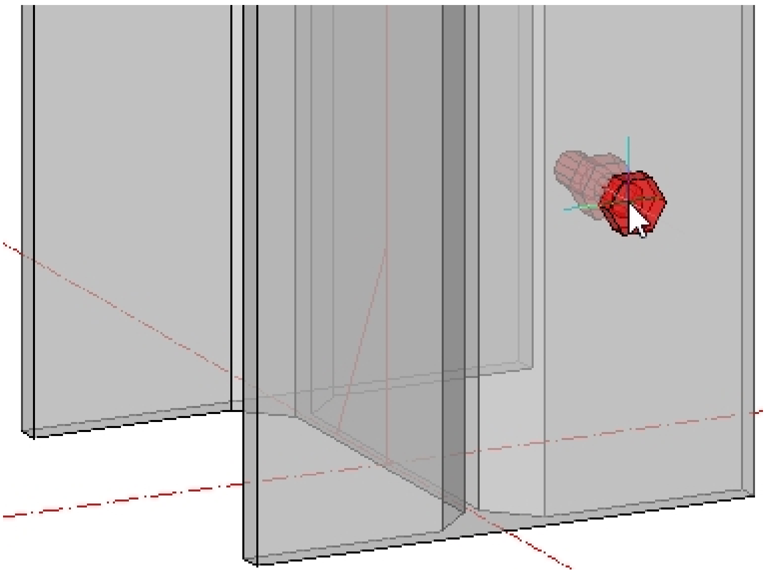
# Draw a single bolt

Command : **S3d_ArxBolt**



To freely draw a single bolt, you first need to select a part that should receive the bolt.

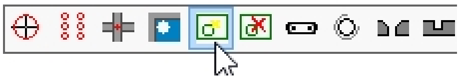
Then use the cursor to position the bolt. A single click on the left mouse button draws a single bolt.



In the above example you cannot place the bolt in the web. To position the bolt in the web, you would need to rotate the view to the web before positioning the bolt.

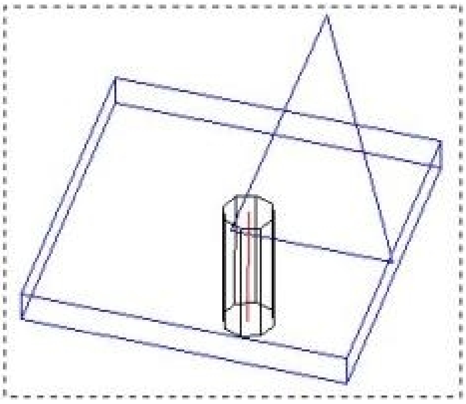
## Verifying new holes

Command : **S3d_HoleCheck**

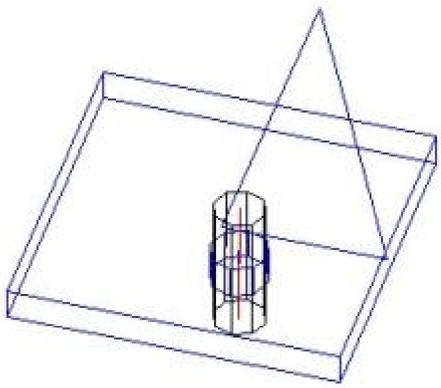


The command Verifying new holes allows Parabuild to check if there is a plate or beam with a bolt inserted but without a hole. In this case Parabuild will draw the hole.

You activate the command and select by crossing the beams, plates and bolts:



The result:



Warning: If the bolt is too close to the edge of a beam or plate or if it is too close to another bolt, it will be coloured red and no hole will be drawn. If this is the case, you need to move the bolt or change the settings: reduce the **bolt to edge** or **bolt to bolt** distance in the the Global settings) dialog box.

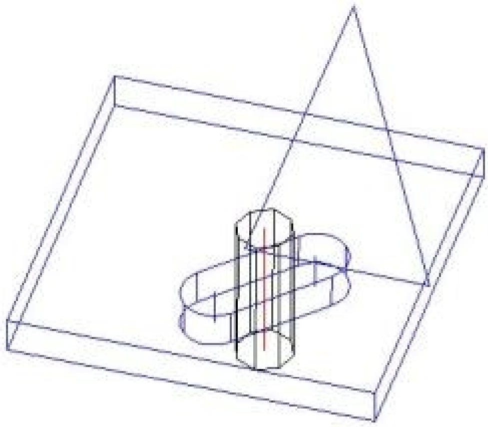
## Slot holes

Command : **S3d_SlotHoleDlg**

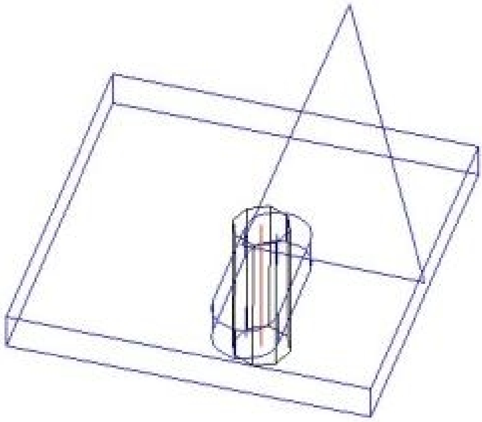
At first, Parabuild will ask you to select the hole you want to convert into a slotted hole. In the dialog, you can select a specific measurement or with **Other** specify your own measurements.

After that, you need to specify the rotation angle.

Slot hole rotated 45°:



Slot hole rotated 90°:

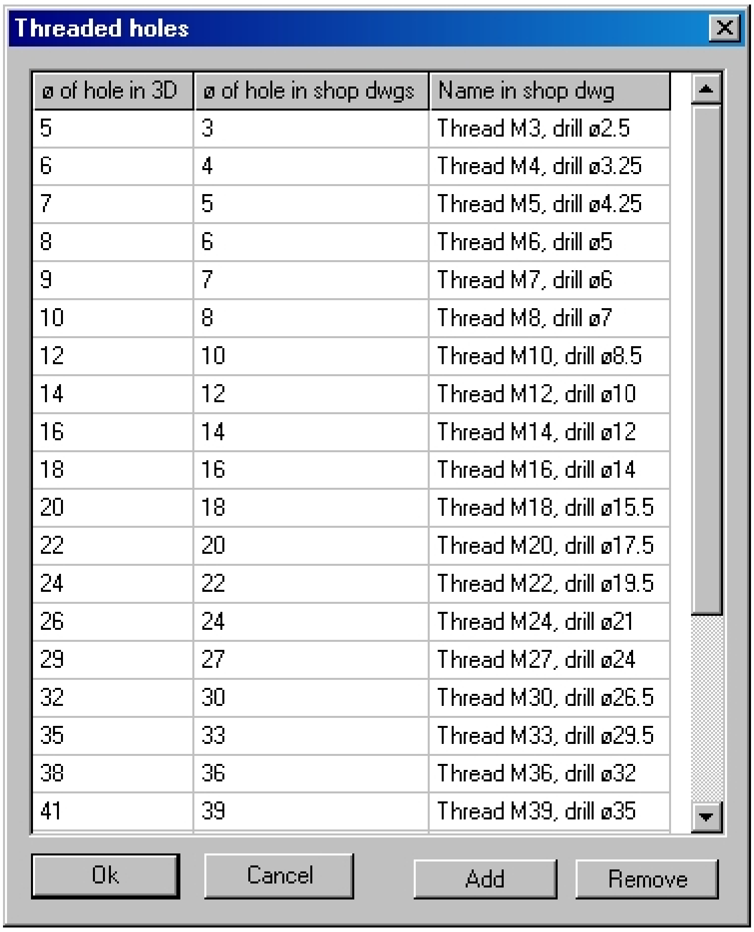


## Threaded holes

Command : **S3d_ChangeHoleThread**

With this command you can convert a regular hole into a threaded hole and vice versa. For illustration purpose threaded holes have some extra lines similarly to the thread of a real hole.

If you convert a regular hole into a threaded hole, the diameter of that hole in 3D will stay the same. This diameter is therefore not used in the workshop drawings: you can adjust yourself which diameter and which special comment should be added on the drawings for the hole. You can do this in the Global settings (button 'Threaded holes').



Threaded holes

| ø of hole in 3D | ø of hole in shop dwgs | Name in shop dwg |
|---|---|---|
| 5 | 3 | Thread M3, drill ø2.5 |
| 6 | 4 | Thread M4, drill ø3.25 |
| 7 | 5 | Thread M5, drill ø4.25 |
| 8 | 6 | Thread M6, drill ø5 |
| 9 | 7 | Thread M7, drill ø6 |
| 10 | 8 | Thread M8, drill ø7 |
| 12 | 10 | Thread M10, drill ø8.5 |
| 14 | 12 | Thread M12, drill ø10 |
| 16 | 14 | Thread M14, drill ø12 |
| 18 | 16 | Thread M16, drill ø14 |
| 20 | 18 | Thread M18, drill ø15.5 |
| 22 | 20 | Thread M20, drill ø17.5 |
| 24 | 22 | Thread M22, drill ø19.5 |
| 26 | 24 | Thread M24, drill ø21 |
| 29 | 27 | Thread M27, drill ø24 |
| 32 | 30 | Thread M30, drill ø26.5 |
| 35 | 33 | Thread M33, drill ø29.5 |
| 38 | 36 | Thread M36, drill ø32 |
| 41 | 39 | Thread M39, drill ø35 |

Ok Cancel Add Remove

Here you can adjust for each diameter in 3D the diameter and comment to use in the workshop drawings.

A threaded hole is different from a regular hole, even when Parabuild numbers your elements.

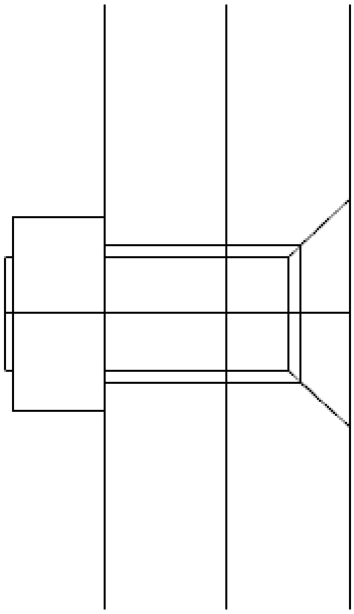
Imagine yourself two identical plates with the same number of holes at the same location, but one plate has one of its holes threaded and the other does not. Parabuild would assign these two plates a different position number.

## Countersunk holes

Command : **S3d_ChangeHoleCountersunk**

Countersunk holes can be activated using the following methods :

- From within each macro that contains a bolt pattern. The holes are adjustable using the button **Advanced** in the bolts tab.
- By right-clicking on a bolt. Then you will see a context menu with the available functions you can perform on the bolt.
-  With this command you can convert a regular hole to a countersunk hole and vice versa.



The countersunk hole is shown in 3D just as the real hole would look like.

The depth and diameter of the countersunk part are determined by the head of the bolt.
If the hole contains a bolt with a hexagonal head, then the countersunk won't be drawn oblique (a counterbore hole is drawn).
If the bolt's head is oblique, then the hole will be oblique too at the same inclination as the bolt head.

The bolt heads of all the bolts are adjustable in the bolt parts database.

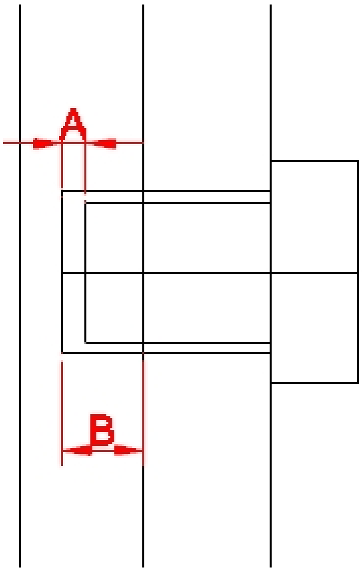
## Blind holes

Command : **S3d_ChangeBlindHole**

Blind holes are incompletely drilled holes.

Blind holes can be activated using the following methods :

- From within each macro that contains a bolt pattern. The holes are adjustable using the button **Advanced** in the bolts tab.
- By right-clicking on a bolt. Then you will see a context menu with the available functions you can perform on the bolt.
-  With this command you can convert a regular hole to a blind hole and vice versa.




A blind hole can only be activated for the last hole of a bolt.

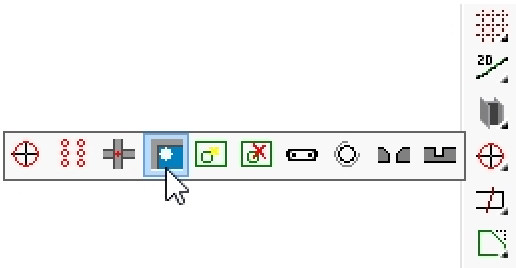
The depth of the blind hole is determined by the ending of the bolt. If preferred the hole can be drilled deeper than the bolt (A on the illustration).

The minimum and maximum depth of the hole can be adjusted (B on the illustration).

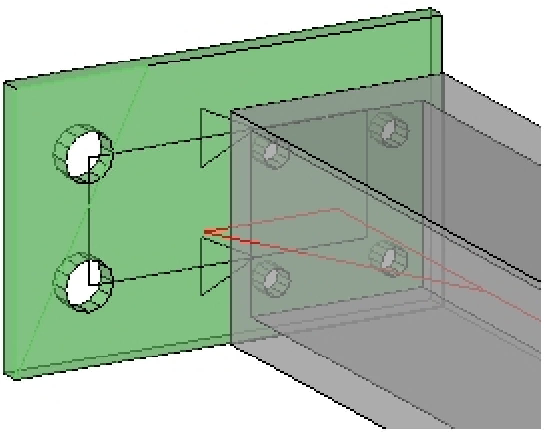
If the bolt length would cause the hole depth to be outside of the minimum and maximum, the hole depth will be adjusted automatically to adhere to the minimum and maximum.

## Holes for galvanisation

Command : **S3d_AutoGalvaHoles**



After starting this command you need to select at least 2 parts.
You could also select the entire drawing.



Parabuild will search for all the 'corners' in which pockets of liquid are possible.

Parabuild will zoom in on each group of galvanisation holes. You then have the choice to either draw the new holes or not. After that the next group of galva holes will be zoomed in to.

Parabuild uses bolts with the name *Helper* to draw these galva holes. However these bolts are not visible and they will not be shown in the bill of materials.

## Bolt standards

When a bolt is drawn in a connection or manually a bolt assembly should be assigned. These Bolt Assemblies contain the description of the standard for the bolt, nut and washers.

If bolt DIN 931 is adopted within the assembly, then Parabuild reads the entire Bolt parts database. Parabuild will then select a bolt with the DIN931 standard, the selected diameter and one of adequate length. Depending on the ones selected, a number of washers and nuts will be added to the bolt. The standard of the nuts and washers are also determined by the assembly and are selected from the part list.

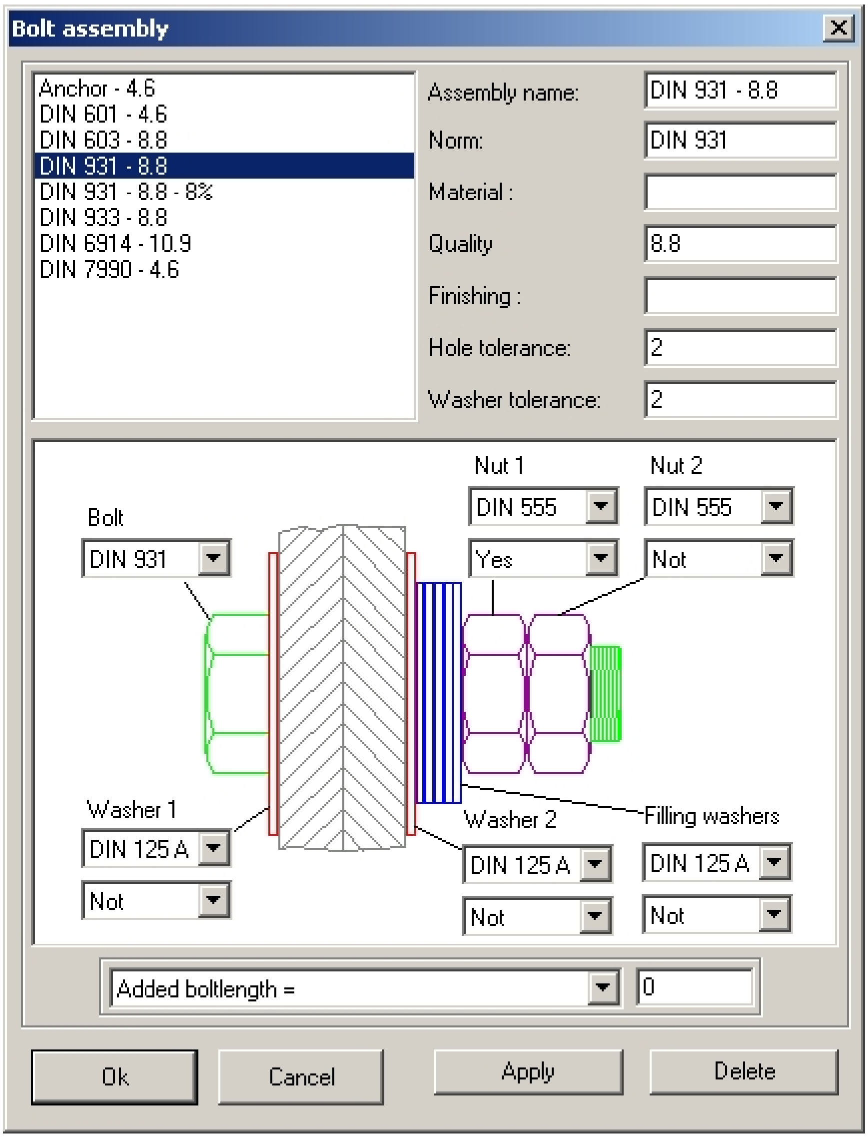
There are two types of lists. The list of assemblies and the list of bolts, nuts and washers. These lists can be added to in the Global settings. The part lists contain the sizes of each part that Parabuild is able to use.

See the AutoCAD Properties for a detailed explanation of the bolt properties that can be changed.

### Bolt Assemblies

The bolt assemblies can be changed within the Global settings.

The assembly of a bolt determines the standards of the bolt, washers and nuts. It also determines the tolerance of the hole and the like.




A summary of all existing assemblies is shown in the top left-hand side of the dialog window. Clicking on one of the assemblies will display all of its settings. These settings can be changed and then saved by clicking on **Apply**. If another assembly name is assigned to the assembly, click on **Apply**, and a new assembly with that name will be created.

The standards of the bolts, nuts and washers can be set within the dialog window. The standards selected will be taken directly from the Bolt parts database.

With the nuts and washers a selection may be made between **not**, **yes**, **never** or **always**. When you choose yes or not, you will be able to enable/disable them while drawing, when you choose always or never, you will not be able to disable/enable them while drawing.

The added length can be made diameter dependent. The added length is to make the bolt longer than is actually necessary. A good example of this is the anchor bolt: which only needs to pass through a footplate, but must actually be much longer.

The added length and the hole-tolerance can still be changed while drawing, as can the turning on and off of washers and nuts.

This is the sequence for drawing a new bolt:

1) A new bolt is drawn, but it has to be assigned a assembly and a diameter.

2) Parabuild searches within the selected assembly for the standard for the bolt, washers and nuts that should be used.
3) Using this standard, Parabuild consults the bolt parts database, and selects a bolt using the following details: Standard, diameter and length (length = penetration length + washers + nuts + added length).

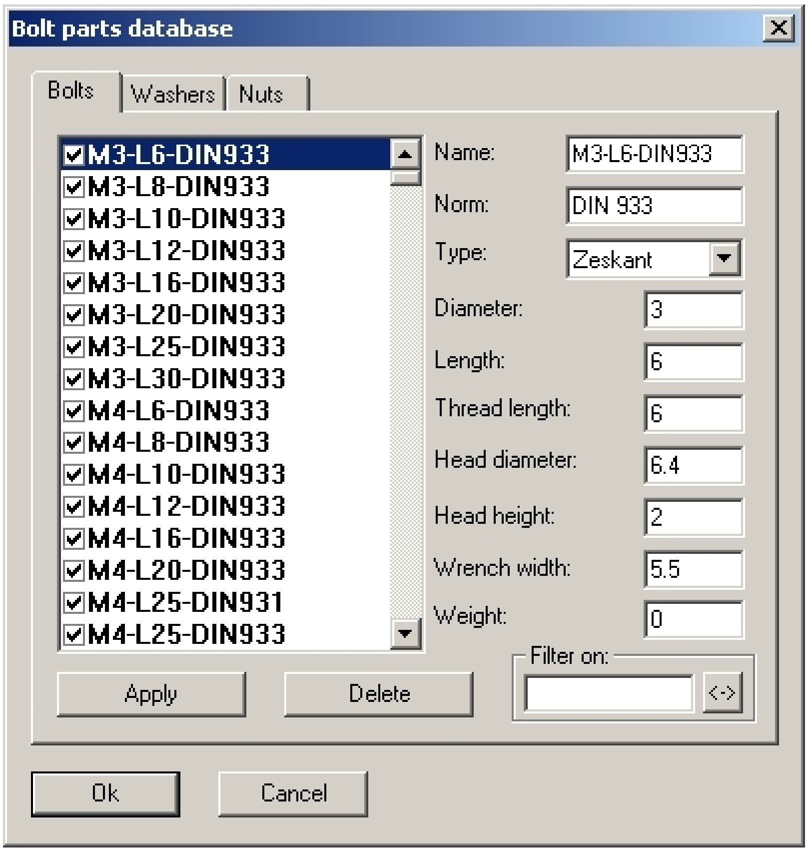
## Bolt parts database

The bolt parts-database can be changed using the Global settings.

All bolts, nuts and washers to be drawn come directly from this database.

This database contains every bolt with its exact dimensions (diameter, length, thread length, ...).

This enables Parabuild to select a bolt from the bolts that the user makes available, and draws them exactly.




This dialog box works in exactly the same way as that of the assemblies: a list of parts on the left, with the properties on the right.

Every part has a checkbox in front of its name. If the box is not checked, Parabuild will never use the part. This allows parts to be turned on without having to delete them. This feature can prove very useful if a part with a certain diameter, length and standard is never used or is not in stock, in which case it can be turned off.

All parts in the available lengths and diameters for the respective standards have already been entered.

Because the number of parts total more than 1000, a filter can be applied. Type in DIN933 in the box in the bottom right-hand corner, and only the parts with DIN933 will be displayed. Click on the adjacent button and all except those with DIN933 will be displayed.

## Clash control

Commando : **S3d_FullClash**

You can use this command to check for clashes between elements.

If a plate clashes with a beam, both will be coloured yellow. This will immediately show errors in the design or drawing.

Bolts are coloured yellow when they are too close to a plate or beam. Bolts will be coloured red if they are drawn in the air or if they are too close to the edge of a beam or plate that they are piercing. The maximum distance between a bolt and the edge of a beam or plate can be set in the Global settings dialog box.

## Numbering of elements

The position and assembly numbers can be given a suffix using AutoCAD Properties.

Example:

The following values of an element are entered:

Prefix: K

Suffix: Z

This will result in the final number on part lists and workshop drawings: K9Z

The number 9 is assigned automatically by Parabuild, depending on what numbers are available.

Two elements that are exactly the same (geometrically), but have a different prefix or suffix will be given a different position number.

The start number of an element means some numbers will be ignored.

If an element is given the start number 100, then Parabuild will try to assign the element K100Z, if this number is already being used by another element then Parabuild will try to assign the next number, and so on.

## Revisions

Command: **S3d_RevisionManager**

Working within a revision is compulsory.

Anything drawn - a bolt, a plate or a profile - the new element will be given the current revision. With a new blank drawing, the current revision will be 0 unless this is edited.

The revision of an element can not be edited. The only influence that the user has over the revision of an element is at the creation or the adaptation of an element: in these cases, the element will be given the current or 'work' revision.

If a new revision is required, the previous revision with which the user had worked will be locked.

Once a revision has been locked it can no longer be edited. When the revision is locked, all position numbers are automatically assigned their permanent numbers (any 'gaps' between numbers will be filled).

The purpose of the revision system is to enable the user to easily identify the differences between two revisions. Part lists can be made for two revisions which will clarify the differences between the two revisions.

Example:

revision 0: Pos Pr5 total:5

revision 1: Pos Pr5 total:3

In revision 0 there were five elements with Pr5 in the drawing. After revision 1 only three of Pr5 remain. This can happen due to Pr5 in revision 1 deleted or adapted.

These details can be used therefore to edit workshop drawings made using revision 0 (e.g. toals) so that they comply to revision 1. However, this does mean that the new position numbers of revision 1 should be added to the drawings.

It is possible to create workshop drawings and part lists from a revision that is still “open” i.e. not yet locked. Care should be taken when doing this that the revision lists system is not used. This is because the revision lists only work with revisions that have been locked.

**If the revision is not locked, the position numbers may be altered.** If a part list is created using elements from revision 0, and revision 0 is not locked, when a change is made or profiles are added can follow that not all elements will be assigned the same position number. This happens because the whole project is assigned different numbers and every time a part list is generated, the numbers are reorganised geometrically. Conversely, when a revision is locked the position numbers are retained. This means that when permanent part lists, drawings or 3D captions are to be created the revision should first be locked before each operation.

The only way to delete a locked revision is by deleting all revisions.

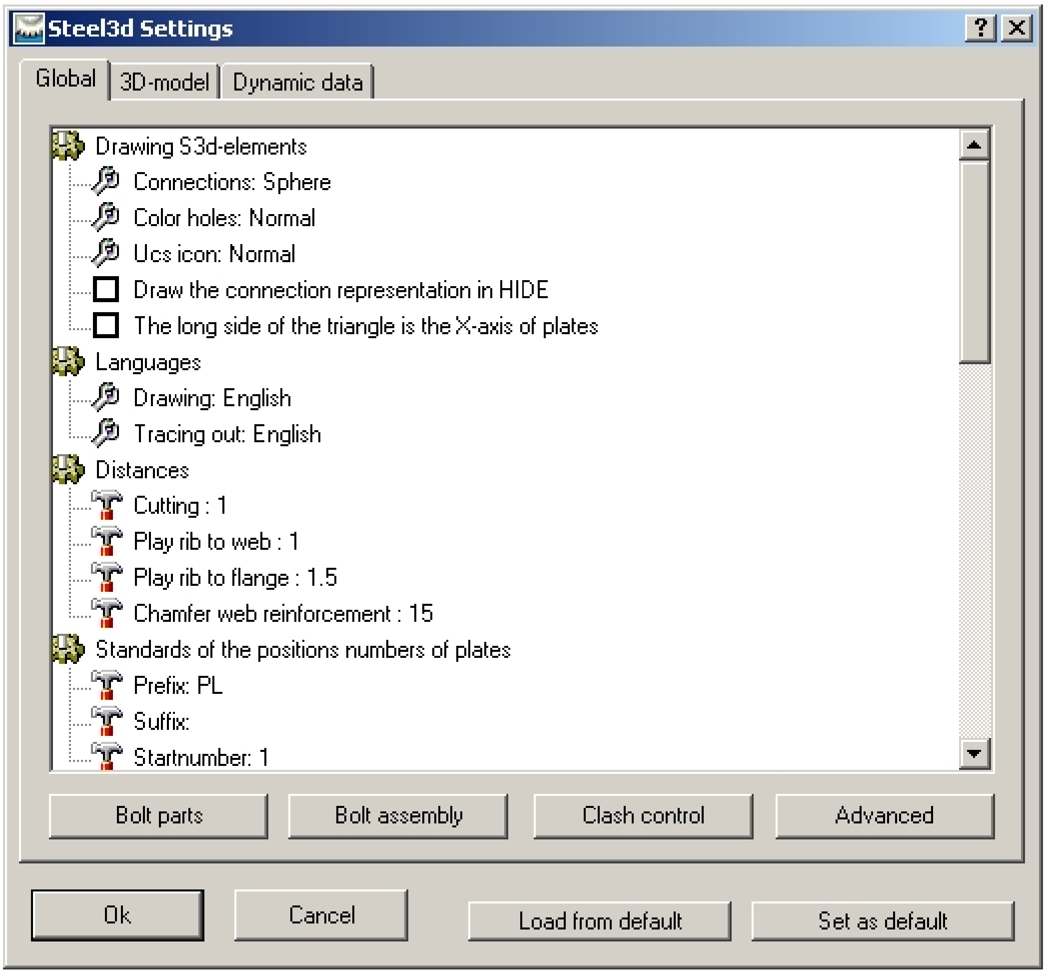
At the very bottom of the Selecting part lists chapter is an exercise that clarifies the creation of a revision-part list.

## Global settings

Commande : **S3d_Settings**


This dialog window can be opened by clicking on the 'SET' icon.

The following is an explanation of the settings .




## 1) Global

**Drawing elements:**

**Connections:** When drawing connections you can choose to represent it by a ball, a triangle or nothing.

**Hole colour**: By default the colour of the holes is drawn in the colour of the beam or plate. You can set the colour to, for example, red so those holes are always drawn in red.

**Languages:**

**Drawing**: The dialogs and prompts on the command line are displayed in this language.

**Plotting**: If you want a different language for the text of the bill of materials and the plots (title area, …) you can select it here.

**Distances:**

**Cutting:** When cutting a beam to a plate, the distance between these two remains open. A distance of 0 is not allowed, but you can enter 0.1 or 0.01 instead of 0.

**Tolerance rib to web**: When Parabuild places a Rib during one of its automated routines, this tolerance is applied to the distance between the Rib and the web of the beam. We recommend a minimum of 0,01mm.

**Tolerance rib to flange**: This is the tolerance between Rib and flange of the beam for similar Ribs. Same minimum of 0,01mm applies.

**Chamfering web rib**: The flange reinforcements at the end of a beam towards the head plate have a triangular body plate in the extension of the web of the beam. This plate can be chamfered at the corners using the measurement indicated here.

**Standards of position numbers, mark numbers....**

All position numbers can be given a prefix or a suffix. The prefixes or suffixes that are to be given to new elements can be entered here. See Numbering of elements for further explanation.

**Verifying new holes after moving them:** When a bolt is moved, the corresponding holes are moved with it. You can switch this off.

**Weight for bill of materials**: Here you fill in the weight to be used for 1 m3 of volume (mostly 8000 or 7860 kg)

**Tolerance for holes**: You set the tolerance for the placement of the holes that Parabuild has to allow when determining the position numbers of beams and plates.

**Tolerance for plates**: The tolerance in measurements of plates that Parabuild has to allow when assigning positions.

**Tolerance for beams**: The tolerance in lengths and cuts of beams that Parabuild can use to identify positions.

**Phase**:

If you want to split your project in several phases (to adapt to splits in productions or to distribute among different designers in a network) then you can set the phase using a number (1, 2, 3...).

**Save extra data so that this drawing is visible in plain AutoCAD**

This option allows the 'Proxy' details of a 3D-drawing to be saved. Parabuild creates its own objects (profiles, plates, bolts ...) which means that the without Parabuild, AutoCAD will not recognise these objects and therefore will not display the profiles. This is solved by the 'proxy' details, details on the appearance of the objects, which is saved within the drawing. One disadvantage of this is that the drawing becomes 5 to 7 times larger. However, this does not result in any great delay when opening or editing the drawing, as these details are not actively used, and is therefore not loaded into the memory when working with Parabuild. Remember that when this option is turned on within an existing drawing and then saved normally, the details are not yet saved. This can be solved by either setting the variable 'ISAVEPERCENT' to 0, or by saving the drawing under another name.

One final requirement is that the proxy details on the computer without Parabuild is set to display. This can be set up in AutoCAD as follows: *Tools > Options > Open And Save > Proxy Images for custom objects* should be set to *Show proxy graphics*.

**Standard values for material, Comment, ...**

The standard values for newly created elements can be entered here.

**Bolt parts**

Use this button to view/change the bolt parts.

**Bolt assembly**

Use this button to view/change the bolt assemblies.

**Clash control**

This button allows you to configure the clash control in a self explanatory dialog window.

**Advanced**

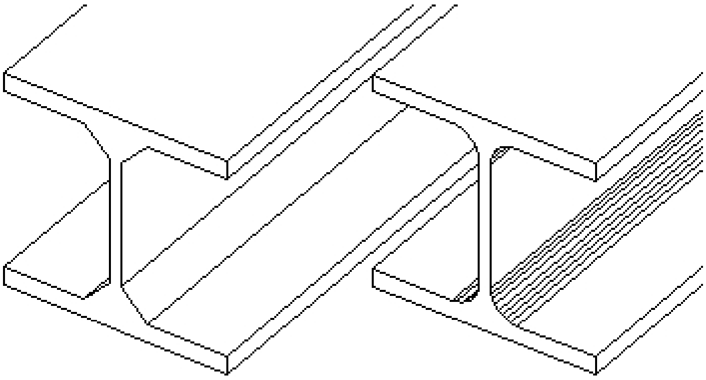
This button allows you to set the possible choices of materials and such like. For example when a new profile is being drawn, a material for the profile can be selected from a list. This dialog window allows the changing of this and other lists.

**Set As Default**: After setting your preferences you click on **Set As Default**. These settings will now be used for every new drawing.

**Load from Default**: When clicked, the settings previously saved with the button **Set As Default** will be loaded.

## 2) 3D-model

This tab determines the graphical representation (detail, ...) of profiles, plates, structures and bolts. **Caution**: These settings will only be applied to newly created elements, and not to existing. To edit these settings to existing elements the properties must be edited per element in the AutoCAD Properties.



## 3) Dynamic data

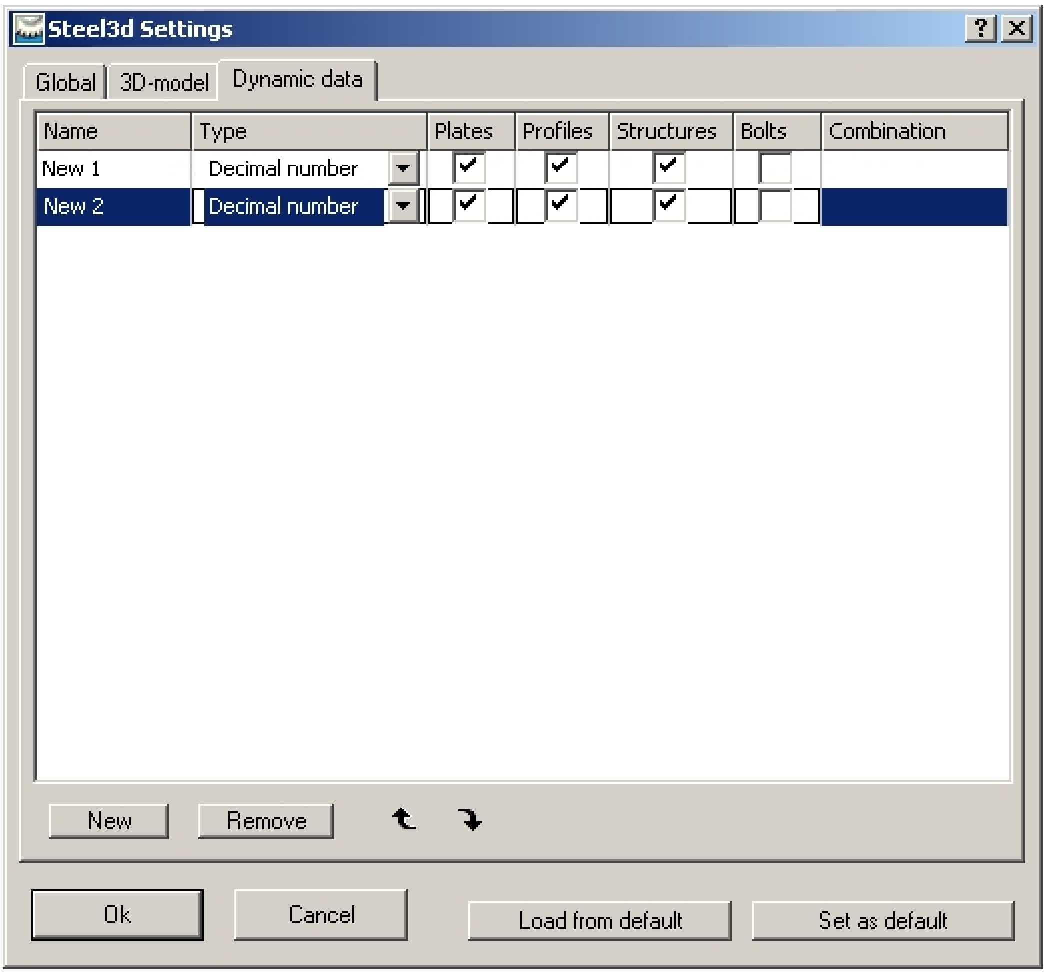
This allows you to add your own properties to plates, profiles, structures or bolts. The complete explanation can be found in the Dynamic properties chapter.

### Dynamic properties

Dynamic properties are properties that the user can assign to Parabuild objects.

This allows additional properties to be created to supplement the properties already offered by Parabuild.

Additional properties can be created using the Global settings dialog window . All properties shown here are also available and adaptable in the AutoCAD Properties. Every dynamic property also has a column in the part list (also on workshop drawings) that can be enabled/ disabled.




Every row represents a property.

The first column contains the name of the property. This name is used everywhere as a reference.

The second column, type, can contain the determined value of the property:

- *Decimal number*: e.g. 23
- *Comma-number*: e.g. 25.6
- *Text*: e.g. stock nr 5
- *Combination text*: This can be used to contain a fixed number of predetermined texts. These texts should be entered in the last column 'combination'. The texts should be separated with a comma. E.g. "steel,concrete,aluminium" will allow the user to make a selection of these three materials.

The remaining four columns, all of which are ticked, determine for which elements the property shall be used for.

After setting the properties and restarting AutoCAD, all properties will be automatically added to all drawings to be opened or to those already open.

## Standards for connections

The standards system is a collection of rules that can automate filling the values of dimensions and components in connections.

Below are some examples of things that can be automated:

- Plates shouldn't be too thick or too thin.
- Plates can not have a width of 137.4 but should be 140, or 150 …

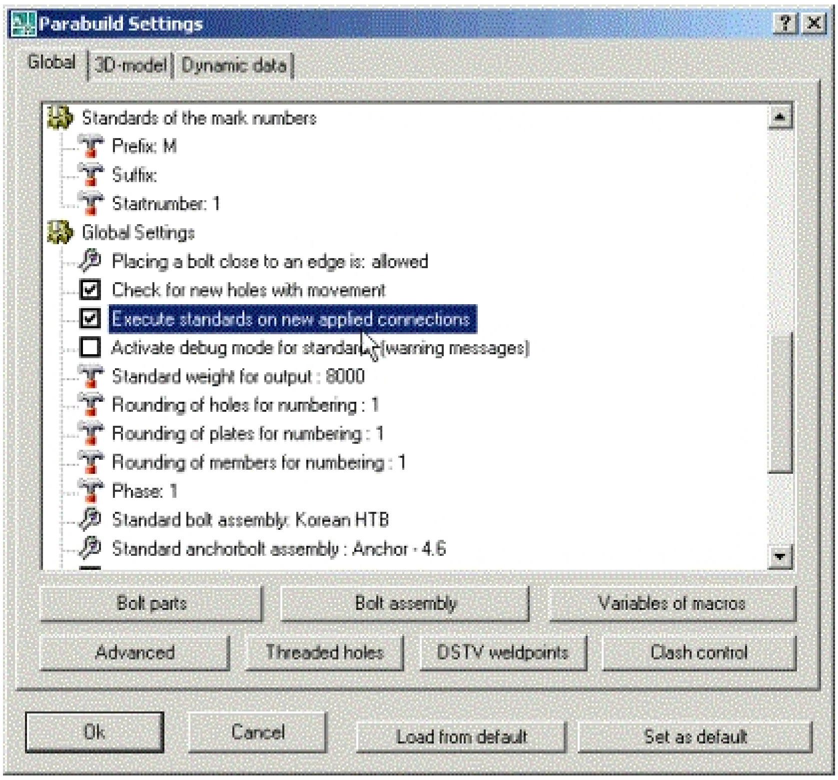
- Bolts need to have a certain diameter, and need to be at predetermined distances from the edge and from each other, depending on the situation.

These are common modifications, which we can easily write in rules.

It is important to know that the standard system does not carry out an analysis of the structure. It is only a tool to automate repetitive modifications.

To use the standards you do not need to do anything; initially they are activated for you and changes are applied to new connections that you apply.

Deactivating the standards is possible in the general **Parabuild Settings** dialog box using the checkbox: **Execute standards on new applied connections**:

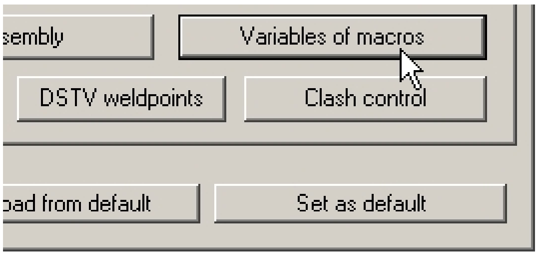



## Configuring the standards

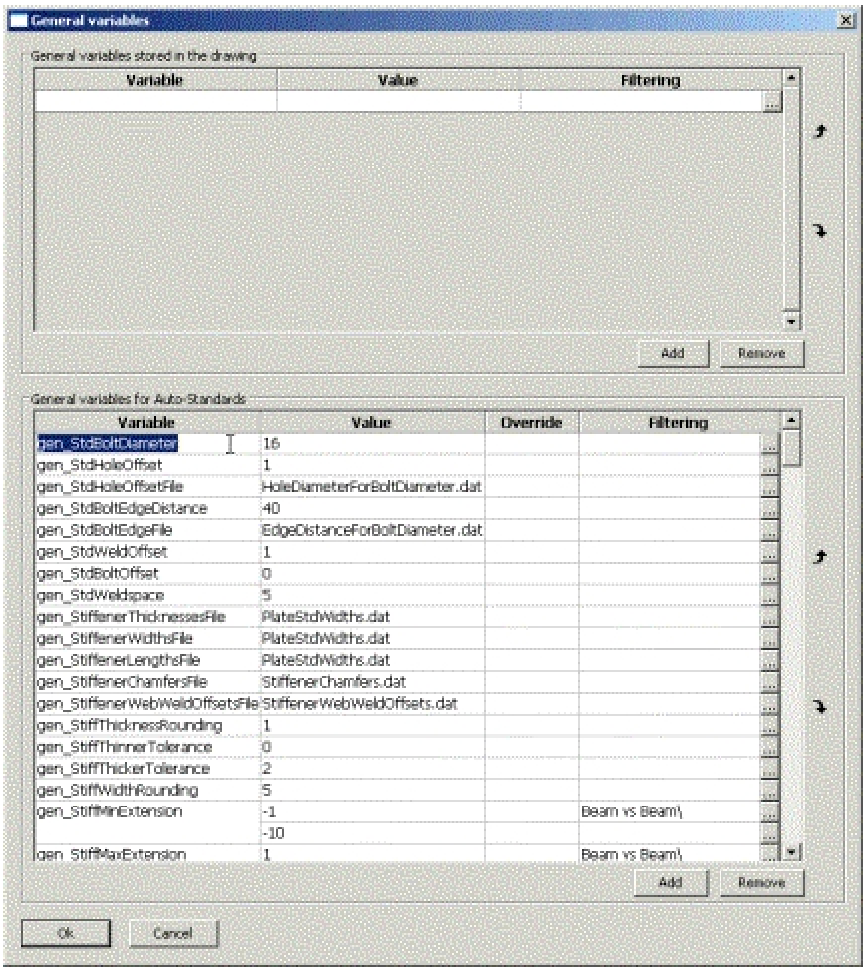
If you want to configure the standards, then you need to make changes in two different locations:

### General variables

In the **Parabuild Settings** dialog box you should click on the button **Variables of macros**.




The following dialog box appears, with below a list of existing variables:




You can influence the standards by changing the value of a variable. The changes you make to the lower list of variables apply to the system, so they apply to all projects you draw.

However do not remove variables; this is preserved for people who extend the standards.

Each variable points to a specific setting in one or more connections. It is possible that a variable has an effect on one connection, on a group of connections or on all connections.

We cannot explain all 200 variables in this manual, but we will explain a select few as examples:

**gen_StdWeldOffset** : Determines the distance that is kept between two welded objects. This setting applies to all connections.

**gen_StdBoltOffset** : Determines the distance that is kept between two bolted objects. This setting applies to all connections that contain bolts.

**gen_StdHoleOffsetFile** : This variable contains a filename as its value. In the file itself you can make changes to influence the hole tolerance. See the next chapter for more information.

**gen_StdHoleOffset** : De (radius) tolerance of the hole of a bolt. This setting has an effect on all connections that contain bolts. However it only applies when the variable gen_StdHoleOffsetFile does not contain hole tolerance information for a certain diameter.

**gen_StiffenerThicknessesFile** : The variable refers to a special file that contains multiple plate thicknesses and widths. From this file a plate thickness will be chosen for the stiffener.

The file is located in the library of Parabuild : S3d_Lib\PlateStdWidths.dat

If you change this file, then you can determine the available plate thicknesses and widths.

This will not only be used by the standards, but also to assign names to plates, for example:

If P10x160 is a standard size, then Parabuild will automatically choose P10x160-143 for the name of a plate that measures 143 to 160.

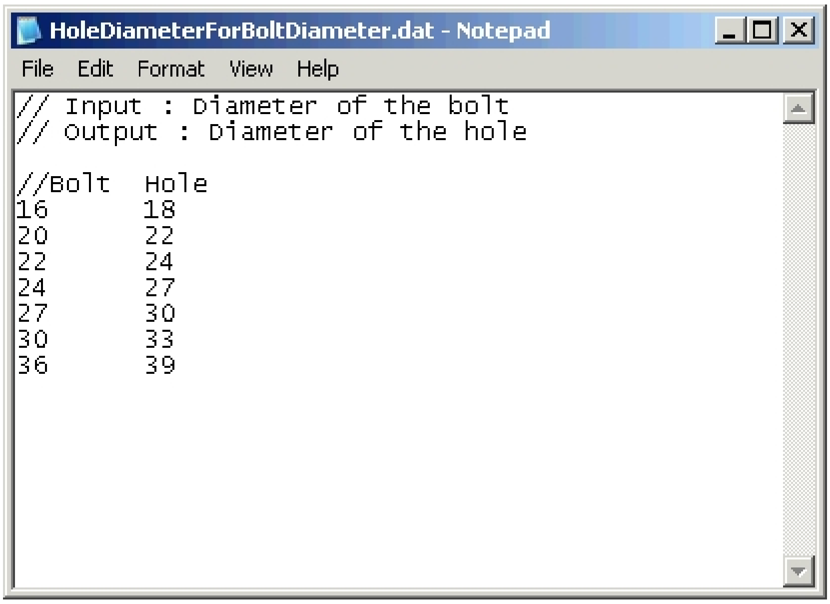
### Files

A part of the standards variables refer to a file. You will find these files in the following location on your computer:

S3d_Lib\Connection Standards\Data\

The .dat files you find here can be modified using Notepad.

The contents of the file HoleDiameterForBoltDiameter.dat looks like this:



HoleDiameterForBoltDiameter.dat - Notepad

File Edit Format View Help

```
// Input : Diameter of the bolt
// Output : Diameter of the hole

//Bolt  Hole
16      18
20      22
22      24
24      27
27      30
30      33
36      39
```

The lines that start with "//" are comments. They have no effect on the workings of Parabuild

The purpose of each file is explained in these comment lines.

The purpose of this file is to choose a hole diameter for each bolt diameter.

If you want to add a new diameter M12, then you would add the following line:

12      14

Between the numbers 12 and 14 you should type a TAB character. Columns are separated this way.

## BIM : importing files

Command: **S3d_Import**


With this command you can read files of types that are not recognised by AutoCAD.

BIM stands for Building Information Modeling. What does that mean?

In short: BIM is keeping the important information of a 3D Model, and not only the geometry. For example a beam is not only a volume existing of planes and lines, but it has a name, material, welding data, position number, etc... These data are what we call BIM data. Parabuild saves this data in the drawing together with the geometric data.

BIM data can be exchanged between different application with the help of the IFC format.

Only the Ifc format supports exchanging of BIM data.

The other formats support only the exchanging of geometry.

The following file types can be read:

- **Ifc 2x3** : Allows you to read files coming from for example Revit or ArchiCAD, with BIM data.
- **Measuring points** : Allows you to read a simple text file that contains the measuring points that were obtained on site. For each measure point a point object is created.

## BIM : exporting files

Command: **S3d_Export**


With this command you export the current Parabuild drawing to a range of file types that AutoCAD cannot write directly.

BIM stands for Building Information Modeling. What does that mean?

In short: BIM is keeping the important information of a 3D Model, and not only the geometry. For example a beam is not only a volume existing of planes and lines, but it has a name, material, welding data, position number, etc... These data are what we call BIM data. Parabuild saves this data in the drawing together with the geometric data.

BIM data can be exchanged between different application with the help of the IFC format.

Only the Ifc format supports exchanging of BIM data.

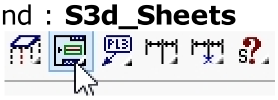
The other formats support only the exchanging of geometry.

The following file types can be written:

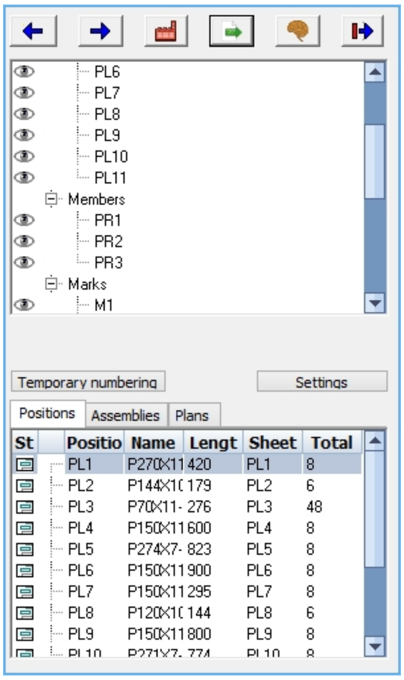
- **AutoCAD 3D-Solids drawing** : Exporting to a 3D-Solids file implies that each Parabuild part will become a 3D Solid. A consequence is that the Solids in the new drawing do not contain any intelligence.
- **Acis .SAT file** : This export method is comparable to the 3D-Solids drawing, but the .dwg file is not written. This facilitates the compatibility with software that does not recognise the .dwg format.
- **Ifc 2x3** : Allows you to deliver the Parabuild drawing with BIM data to the client for checking and planning purposes.
- **Steel3D drawing for versions 6.0 until 6.2** : This export allows you to save the drawing to an older version of Parabuild, but with the loss of some data.
- **Steel3D drawing for versions 7.0 until 7.2** : This export allows you to save the drawing to an older version of Parabuild, but with the loss of some data.

## 2D Sheets Manager

Command : **S3d_Sheets**



With the command **Sheets manager**  you can open a dialog box that can stay opened permanently. It can also be docked to the edge of your CAD frame.




## Two lists

The top list of this dialog box contains all the 2D sheets that you already created, and also all available bill of materials.
The bottom list reflects all 3D parts in the drawing. Each member or plate is listed here according to it's position or assembly number.

## Right-clicking

By right-clicking on a 2D sheet you can perform a range of actions on the sheet.
By right-clicking on a position number, assembly number or camera you can perform an action on the 3D part.

## This dialog box allows you to do the following general actions

**With the top half of the dialog :**

- Generating all workshop drawings.
- Generating all DXF files.
- Generating all DSTV files.
- Generating all bills of materials.
- Creating new General Arrangemant views.
- Starting a slide show of all 2D sheets.
- Printing all 2D sheets.
- Exporting all 2D sheets.
- Learning from all 2D sheets .

**With the bottom half of the dialog :**

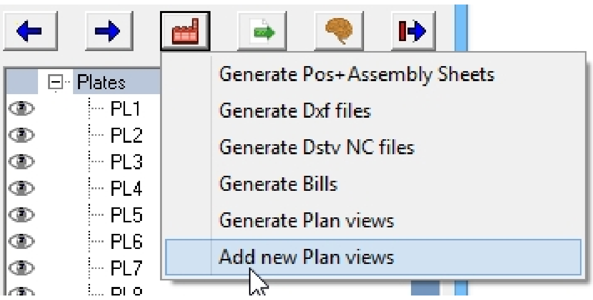
- Settings. For the generation of bills of materials, position and assembly workshop drawings and General Arrangement views.
- **Temporary numbering**. Temporarily assigns position and assembly numbers to all 3D parts.

- An overview of all position numbers, assembly numbers and camera's that exist in the 3D drawing.
- Generating workshop drawings one by one.

## Creating a General Arrangement view

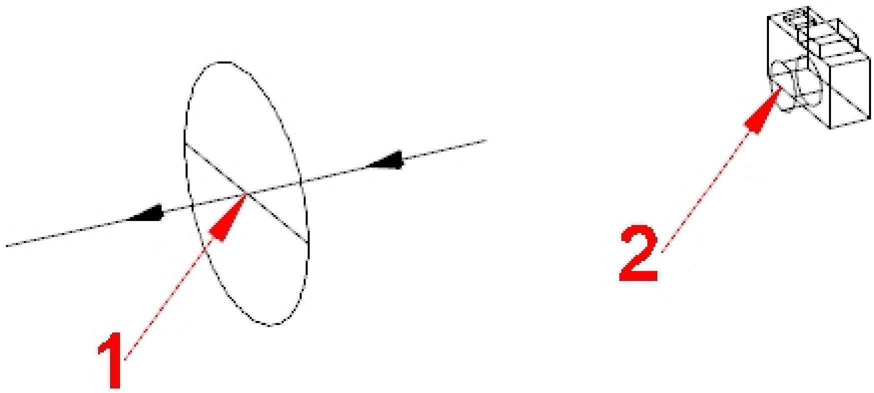
There are 3 methods for creating a new General Arrangement view.

- The first method is from within the  **Sheets manager**, click on the button  and then **Add new Plan views**.



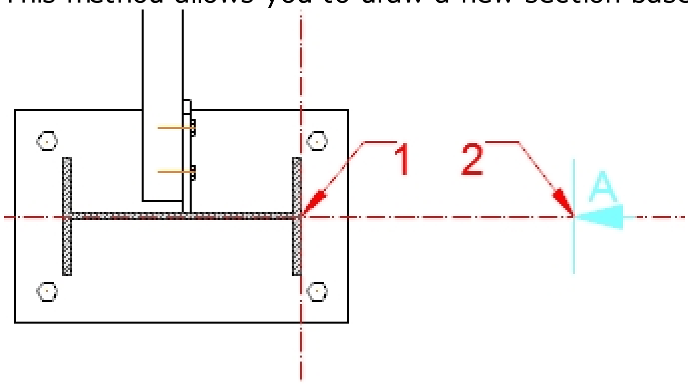

In the dialog box that follows you can create a new view based on a grid, a level or an ISO view.
The new view will be drawn on a new 2D sheet if no sheet is currently active.
If a 2D sheet is currently active, the new view will be added to that sheet.

- The second method is using the command  **Draw section** while model space is active.

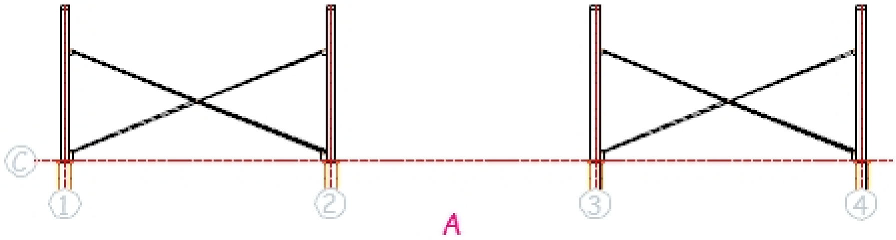



For the first point you select the middle of the camera object, this will later become the middle of the view limitation.
For the second point you choose the location of the camera, this point also influences the front view limitation of the new camera.

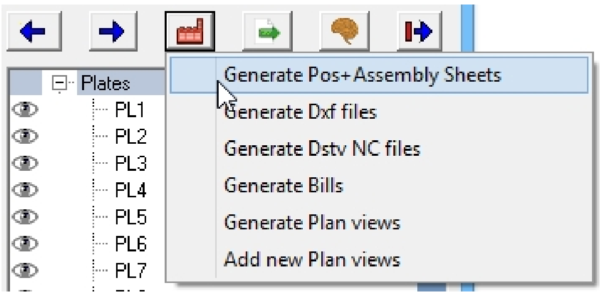
- The third method is with the command  **Draw section** while a 2D sheet is active.
This method allows you to draw a new section based on an existing view.

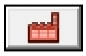


For the first point you give the center for the new camera.
For the second point you give the camera location, this point also influences the front view limitation of the new camera.
The new view will always be drawn upright when seen from the 3D drawing. This is the result of the section on the anchor plan :




## Generating all position and assembly workshop drawings



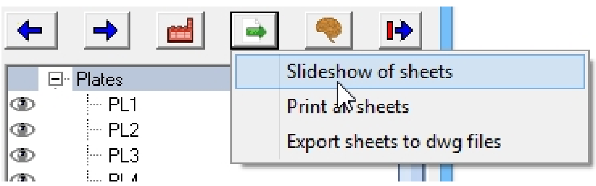

From within the sheets manager , click on  to start the function **Generate Pos +Assembly drawings**.

Here you have the choice to process a part of the 3D drawing, like for example a particular phase or revision.
The phase or revision choice you make here will be permanently stored with the 2D sheet.
The number of part that will be shown on the sheet is only the number of parts counted in the phase/revision selection.
If you've changed the 3D drawing which would cause the number of parts on the sheet to change, then you can start the function Refresh views on the sheet(s) to update the bill on the sheet(s).

This dialog box also has the ability to show the sheets that were already generated. This can be useful when you are generating a large amount of sheets, and you already want to check the sheets during the generation process.

You can also store each finished sheet to a PDF file. This way you can also open and review the sheets during the generation process with the help of a PDF Reader (not available in BricsCAD).

## Slideshow of all 2D sheets




From within the sheets manager , click on  to start the function **Slideshow of sheets**.

You only have to enter the time to wait between each sheet.
The slideshow will start from the currently active sheet, or from the first sheet if none is

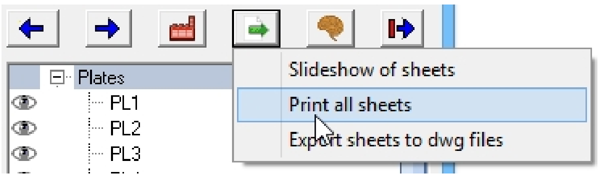
active.

During the slideshow you are free to perform changes to the 2D sheet.
But when the waiting time has elapsed, the 2D sheet is stored and the next sheet will be shown on screen without your intervention.

If you want to interrupt the slideshow, then click on the button 

in sheets manager.

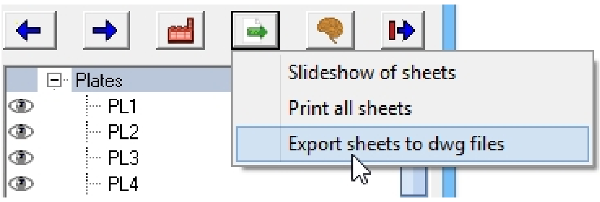
## Printing all 2D sheets




From within the sheets manager , click on  to start the function **Print all sheets**.

You will be prompted for the printer and printer format for each page format.
But, if for example you already printed an A4 page and you are again printing an A4 page, then Parabuild has remembered the printer and printer format from the previous print operation.

## Exporting all 2D sheets



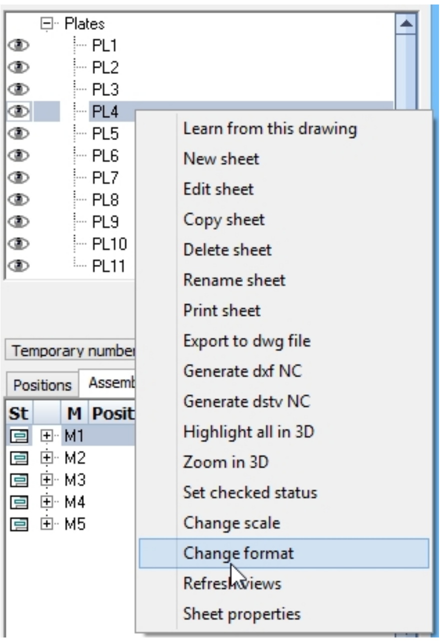

From within the sheets manager , click on  to start the function **Export sheets to dwg files**.

Parabuild stores all 2D sheets in one .dwg file, together with all the 3D parts.
Use this function if you want to generate a separate dwg file for each 2D sheet.
This is necessary when you want to deliver the sheets to someone who doesn't have access to a Parabuild license.

The files will be stored in the same folder as the locations of the 3D dwg file.
Do note that the exported 2D sheets will not have a connection with the 3D parts.

## Right-clicking on a 2D sheet

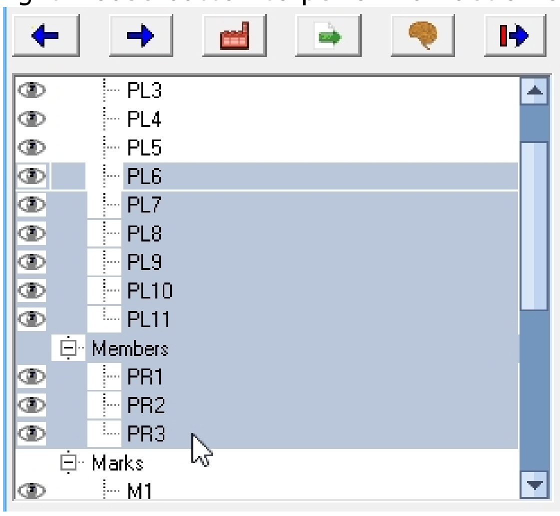
When you right-click on a 2D sheet from within the sheets manager , then you can perform a range of actions on the sheet.




- **Learn from this sheet**. Parabuild will learn from the dimension of this sheet. The learned data will be forgotten when you close Parabuild. To learn in a permanent manner, you need to store the entire drawing in the library using the button .
- **New sheet**. A new empty sheet will be created.
- **Edit sheet**. The sheet will be opened for editing (this is the same as double-clicking on the sheet in the list).
- **Copy sheet**. Allows you to copy the entire sheet.
- **Delete sheet**. De sheet will be removed.
- **Rename sheet**. Use this to change the name of the sheet. The name must be unique and should not contain any of the following symbols : <>?;/\.
- **Print sheet**. Prints the sheet. You will be prompted for the printer and printer format for each page format.
  But, if for example you already printed an A4 page and you are again printing an A4 page, then Parabuild has remembered the printer and printer format from the previous print operation.
- **Print as PDF file**. Will export the 2D sheet directly to a PDF file. The file will be located in the same folder as the location of the 3D drawing (not available in BricsCAD).
- **Export to dwg files**. Exports the sheet to a dwg file. The file will be located in the same folder as the location of the 3D drawing.
- **Generate dxf NC**. Generates a dxf file for machine cutting if the sheet contains a plate. The file will be located in the same folder as the location of the 3D drawing.
- **Generate dstv NC**. Generates a dstv NC file for machine cutting/drilling if the sheet contains a member or a plate. The file will be located in the same folder as the location of the 3D drawing.
- **Highlight all in 3D**. If you click this function then a sample 3D model will be shown and selected of the plate, member or assembly that is drawn on the sheet.
- **Set checked status**. Will set the status to checked if you checked the sheet and it is ready for printing. When something has changed to the 3D model which caused this sheet to be changed, then the status will automatically be reset so you know that the sheet needs to be checked.
- **Change scale**. Allows you to change the scale of the sheet. The frame of the sheet will be rescaled giving you more or less drawing space.
- **Change format**. Allows you to change the format of the sheet. The frame of the sheet will be adjusted giving you more or less drawing space.

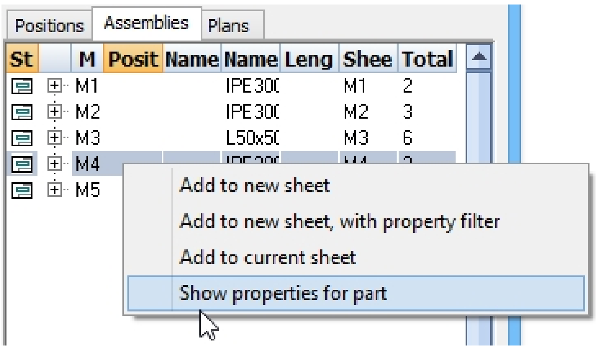
- **Refresh views**.
- **Sheet properties**.

Many of these function can also be executed on multiple sheets at once.
To do this you first need to select the first sheet, press and hold the SHIFT key, and then select the last sheet : all sheets within the chosen sheets will be selected. Now click the right mouse button to perform an action on the sheets.




## Right-clicking on a position/assembly number

When you right-click on a position number, assembly number or camera from within the sheets manager, you can perform a range of actions on the part.




- **Add to new sheet**. With this function you create a new sheet with the 2D views of the 3D part, with or without automatic dimensions.
- **Add to new sheet, with property filter**. With this function you create a new sheet with the 2D views of the 3D part, with ot without automatic dimensions. You can also enter a filter on for example a particular phase so that only the 3D parts of that phase are taken into account when the amount of parts are counted in the bill of the sheet.
- **Add to current sheet**. Allows you to add the 2D views to the current sheet. A sheets needs to be active in order for this function to succeed.
- **Show properties for part**. When you activate this function Parabuild will show and select the 3D part.

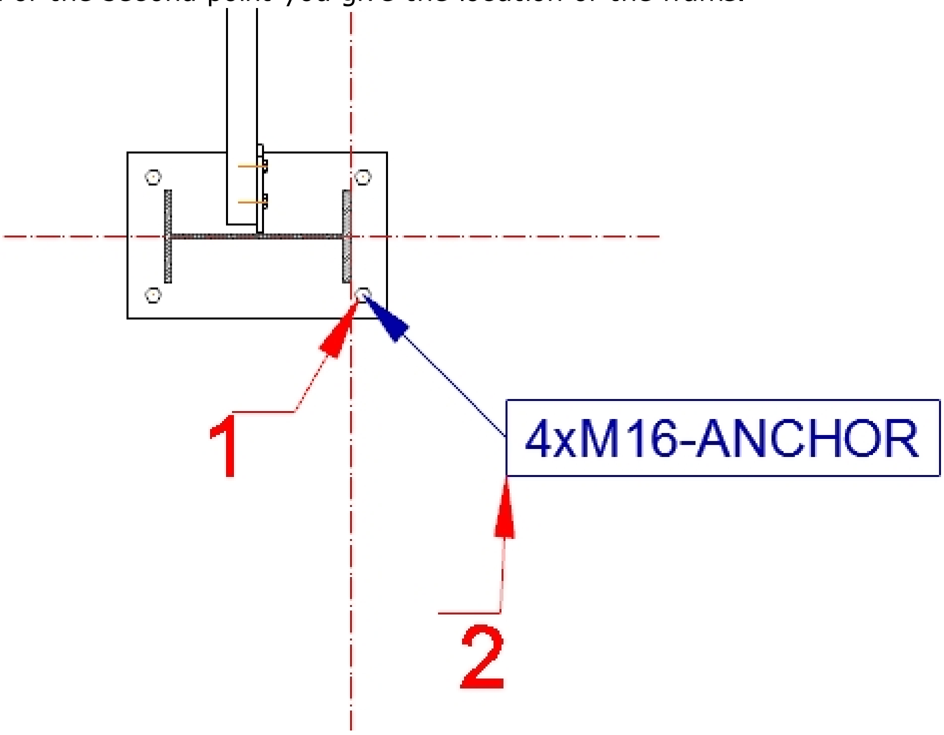
## Annotations

The annotations are meant to show extra information about a part on the 2D sheet, or for showing the level at a certain point.
You can also draw annotations in model space between the 3D parts, but drawing them this

way is less advantageous.

The different types of annotations can be drawn with the following commands :

For the first point you indicate the part that needs to be annotated.
For the second point you give the location of the frame.




We will go more into detail for each command :

Will display the position number + name of the member or plate, the name+standard for bolts and the diameter for a hole.

Will display the assembly number+name for a member or plate, the name+standard for a bolt and the diameter for a hole.

Will only display the name of a member or platen, the name of a bolt and the diameter for a hole. This is useful for the plans that need to be submitted for approval.

Allows you to add a comment to any object.

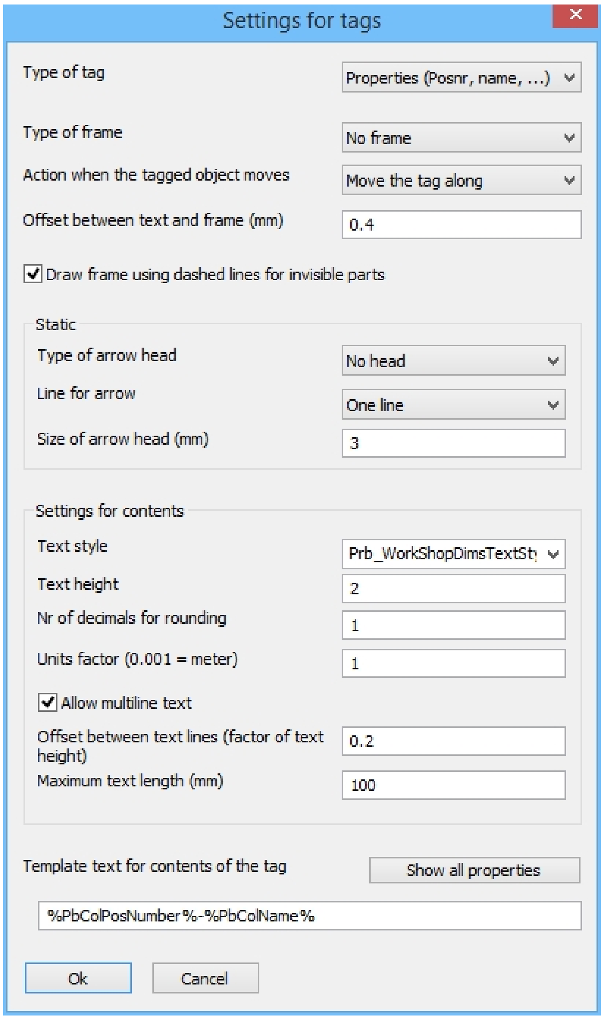
Allows you to measure the height of the object in 3D.

The appearance of the annotation can be adjusted in the AutoCAD Properties dialog box.

## Changing the standard settings of annotations

You can modify the standard settings by starting one of the above commands and you immediately press **S** and **<Enter>**.
In the list of the dialog box **Managing annotation styles** you can change the style of each type of annotation.
There is a separate style for each type of annotated object so that we can choose a different frame, text style and text height for each type of object.

We will go into more detail on some of the settings of the annotation style :




- **Type of annotation**. Choose the type of annotation that needs to be drawn.
- **Template text for contents of the annotation**. Choose the text that the annotation should show. You can combine regular text with properties. To enter a property of the element you enter a variable name between %. So the position number will be %PbColPosNumber%.
  Click on **Show all properties** to find out which variables exist..

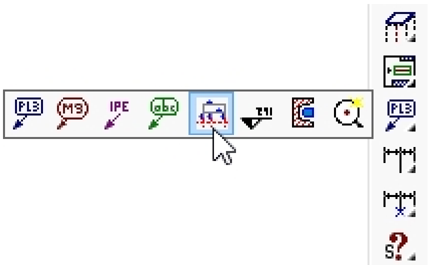
## Creating your own annotation type with your own icon

It is also possible to create your own icon that will draw new annotations. To do this, proceed as follows :

- Create one or more styles in the **Managing annotation styles**. Open this dialog box by starting one of the annotation commands  and then pressing **S** and **<Enter>**.
- Give all the new styles the same style groupname, for example *My group*.
- Create a new icon with the help of the CUI command.
- For the *Macro* field of the new icon your enter the following command :
  ^C^C(S3d_TagGroup "*My group*")

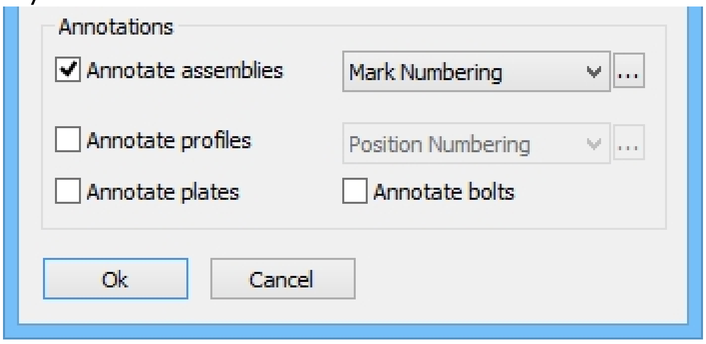
## Adding annotations automatically

Command : **S3d_DrawTagsOnView**

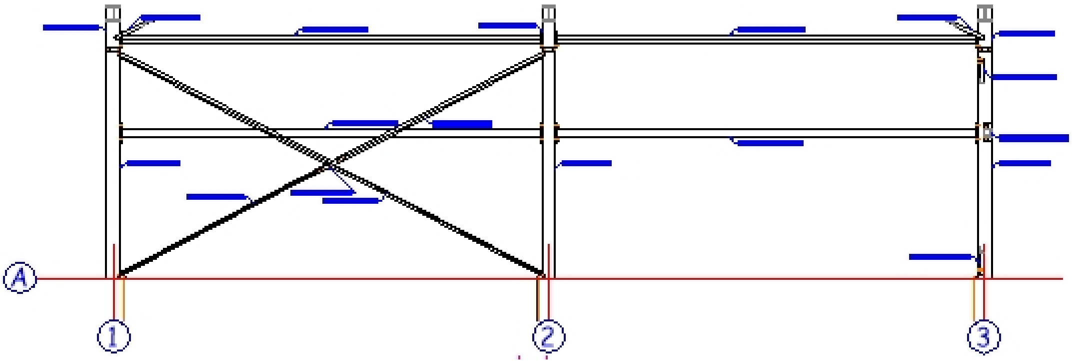


When you start this command, you will first need to select a view.

Then you can choose the part types that need to be annotated, and also the annotation style :



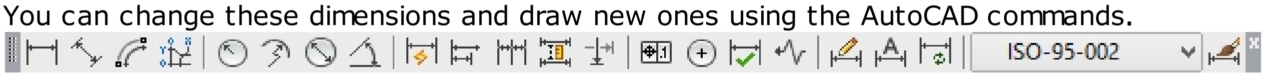

You can just leave the default styles in case you didn't create your own annotation styles.



*The result after annotation of all assemblies*

## Dimensions

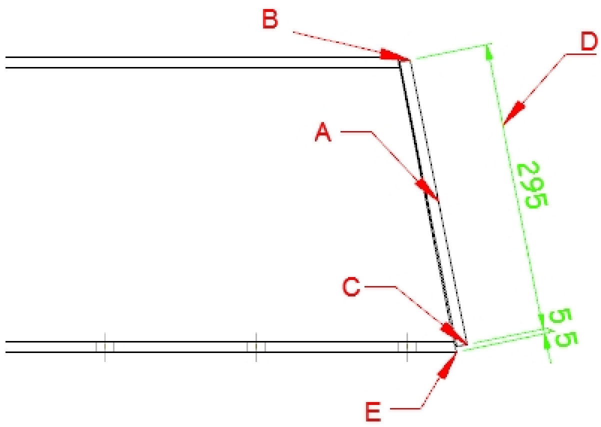
All dimensions that are drawn automatically by Parabuild are regular AutoCAD dimensions.
You can change these dimensions and draw new ones using the AutoCAD commands.




In some cases the AutoCAD tools are impracticable.
Especially for drawing chain dimensions, ordinate dimensions and oblique dimensions the AutoCAD tools are slow in usage.

That is why some commands were added to Parabuild to intercept these shortcomings.
These commands will however draw AutoCAD dimensions, so you can still use the properties and dimension styles of AutoCAD on these dimensions.

 With this command you draw a new chain dimension.
This command works almost exactly the same as the **Linear dimension** command of AutoCAD.




- (Optional) If you want to draw an oblique dimension, then you need to press **<Enter>** right after starting the command.
  Then you will get the opportunity to select a line that will be used to align the dimension to (*A on the illustration*).
- Then you have to give the two extension points, *points B and C on the illustration*.
- Finally you have to give the distance between extension points and dimension line, *point D on the illustration*.
- (Optional) Then you can still expand the chain dimension with additional extension points (*point E on the illustration*).

You can convert this chain dimension into an ordinate dimension using the command Dimension settings ( ).

With this command you can additional extension points to an existing chain or ordinate dimension.

With this command you can remove an extension line from a chain or ordinate dimension.

With this command you can change the origin points of an ordinate dimension.

With this command you can let Parabuild search for chain dimensions in the 2D sheet. If it finds dimensions that form a chain dimension but they are not merged, then it will merge these dimension to one chain dimension.
Merged chain dimensions can be converted at once into an ordinate dimension.

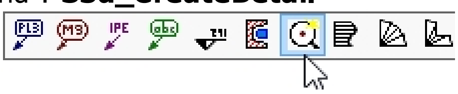
With this command you can manually merge dimensions into a chain dimension.

With this command you can manually disconnect a range of dimensions.

With this command you can convert a chain dimension into an ordinate dimension and vice versa. This command is also accessible by right-clicking a dimension.

## Creating a detail on a view

Command : **S3d_CreateDetail**

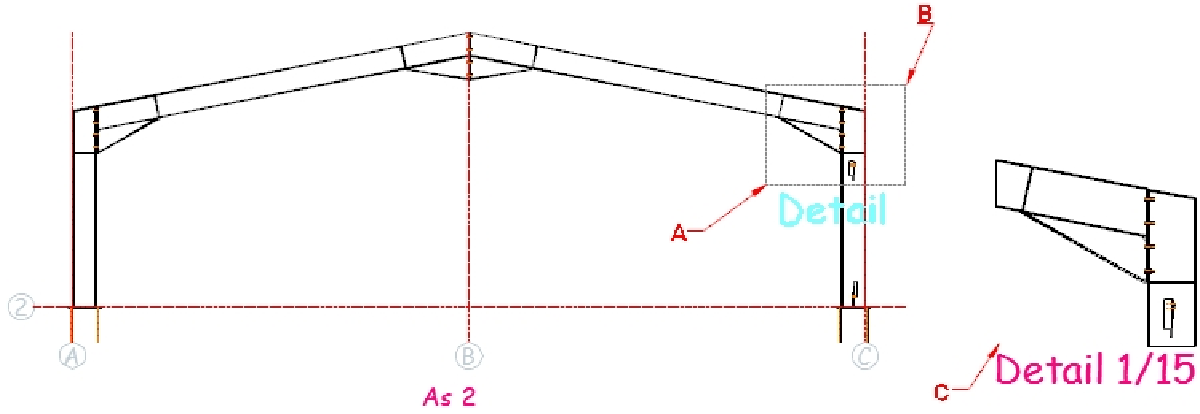


With this command you can create a detail view based on another existing view.

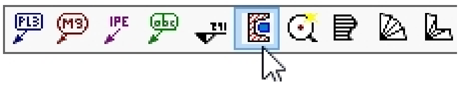
When you start this command you first need to indicate two points that form the rectangle of the detail.

You need to indicate this rectangle on an existing view (see A and B on the illustration).

Then you need to enter the scale and the placement of the new detail view (see C on the illustration).




## Creating a section of a part



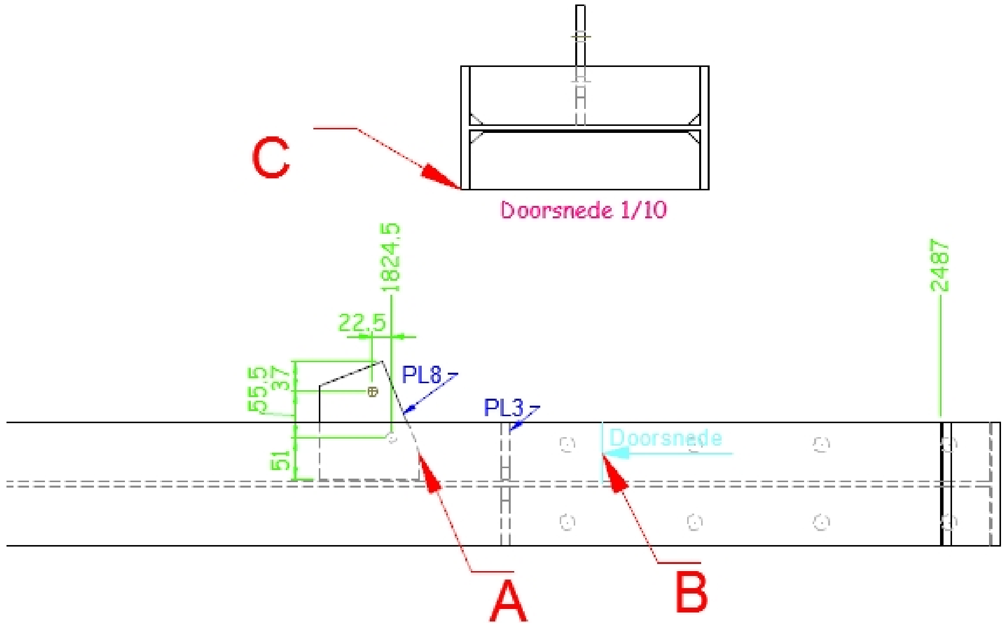
With this command you can add a section view to a position or assembly workshop drawing.

Before starting this command you need to activate a position or assembly workshop drawing.

For the first point you need to select the midpoint of the section on a side view (point A on the illustration).

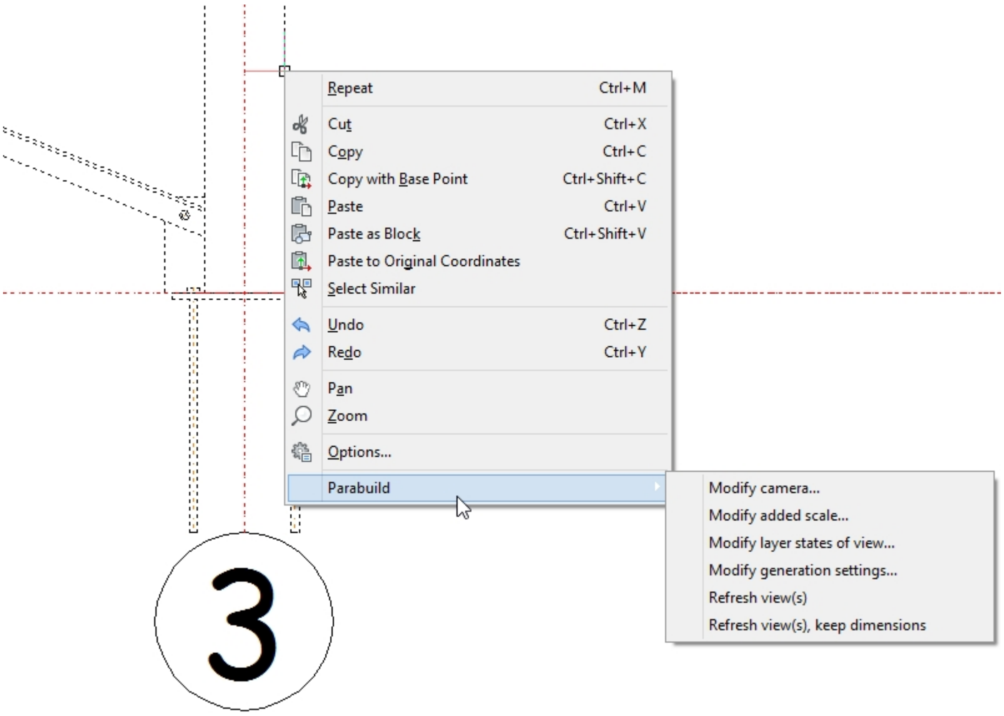
For the second point you need to select the viewpoint of the section. This point also determines the front view limitation of the new section view (point B on the illustration).

For the last point you choose the location of the new section view (point C on the illustration).

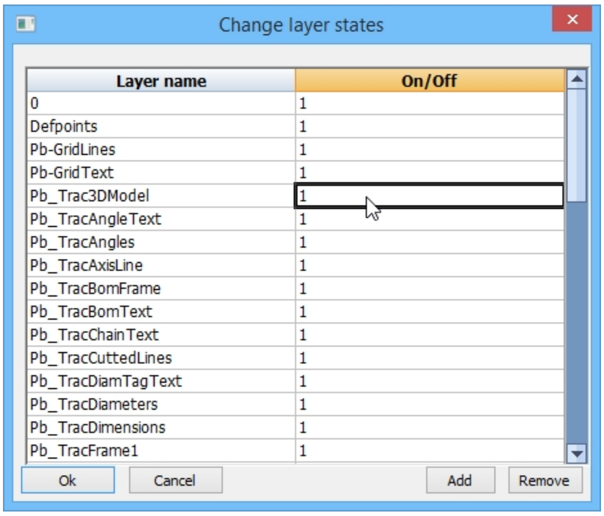



## Changing the visibility of the layers of a view

You can start this function by right-clicking on a view of a 2D sheet.



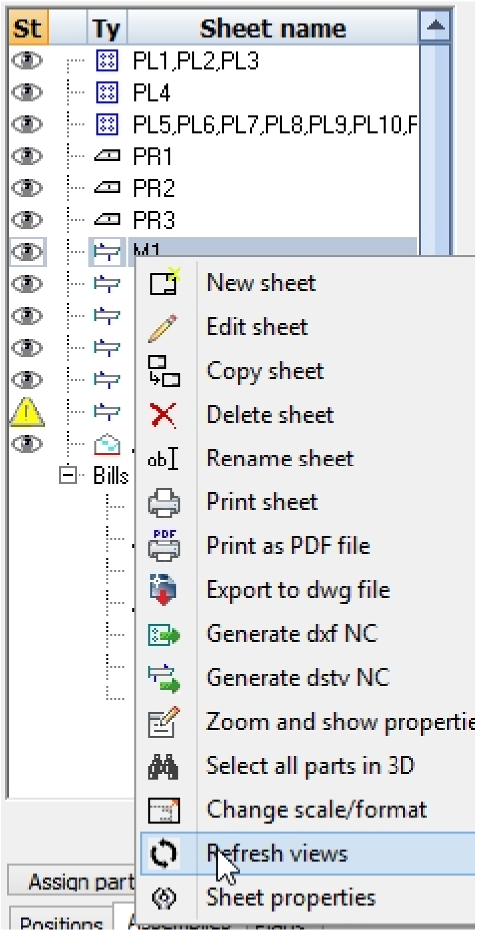

In the dialog box you can choose for each layer whether it should be visible in the view or not.
If you made a modification in this dialog box, then you need to [refresh the views](#) before the result will be visible on the view.



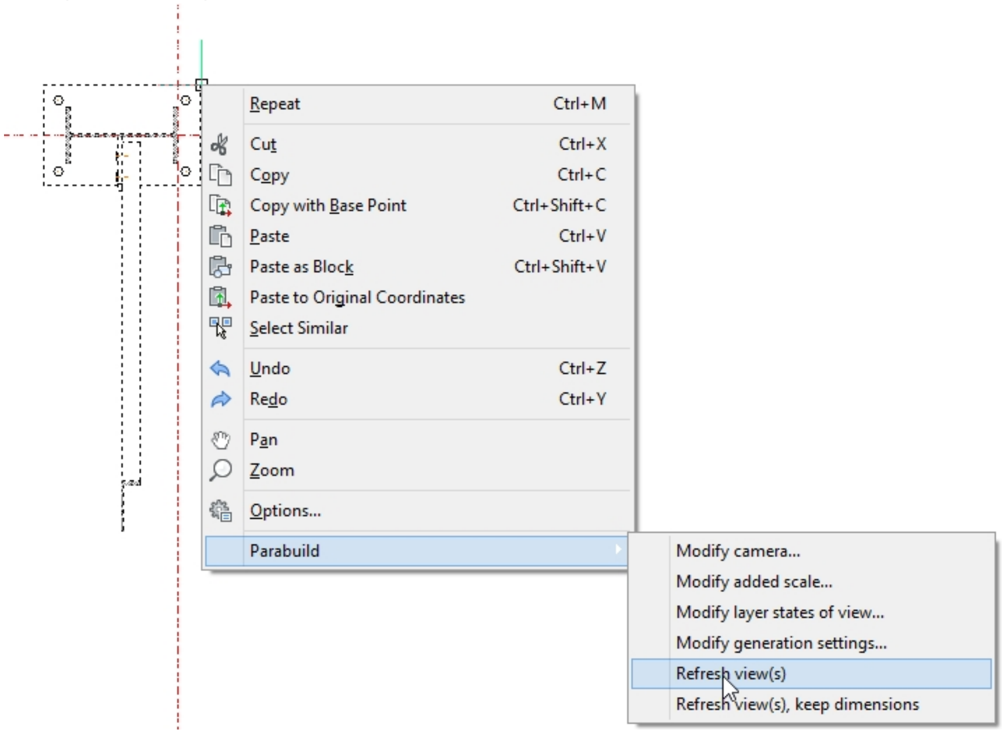

## Refreshing views

There are two methods ro refresh the view(s) of a 2D sheet.

- By right-clicking on a sheet in **sheets manager** .




- By right-clicking on a view.



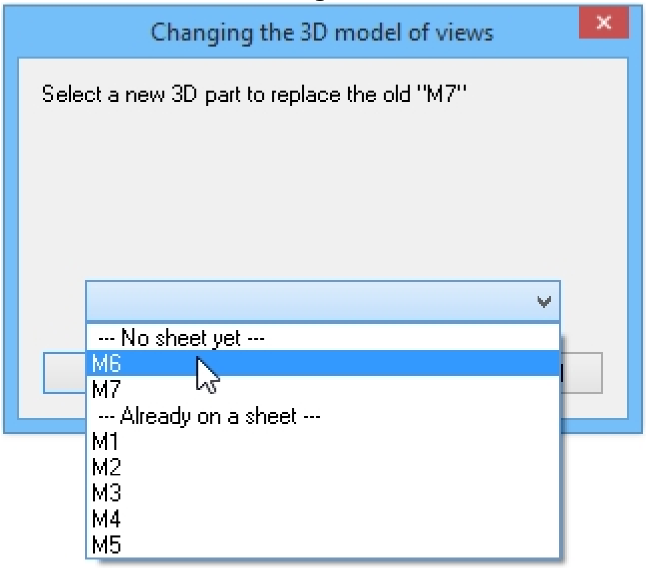

When you perform this function on a General Arrangement view, then the entire view will be refreshed. All the changes you made to the 3D model up to this point will be adopted by the view(s).
While updating the view, Parabuild will keep the dimensions and annotations that are drawn

on the view and if necessary they will be moved or stretched.
It is possible that the connection between 2D view and 3D model is lost. This happens when a 3D model was deleted. In this case the dimension remains untouched and the value of the dimension will contain '?' when it becomes unmeasurable.

When you perform this function on a position- or assembly workshop drawing, then the number of parts in the bill of the sheet will be recounted. The 2D views on the shop drawing will not be renewed because the views are connected to a position or assembly number.
When the subject of a sheet, the 3D position or assembly, doesn't exist any more, then the sheet will get a warning triangle in front of it's name. The views will be indicated as being deprecated so that no mistake can be made in case the sheet would be printed at this point.

## Reusing an expired workshop drawing

When a position or assembly drawing has expired (), then you can use the *Refresh views* tool on this drawing in order to reuse the drawing.
Once you do that, Parabuild searches for 3D positions and assemblies that are not yet inserted on a 2D drawing.

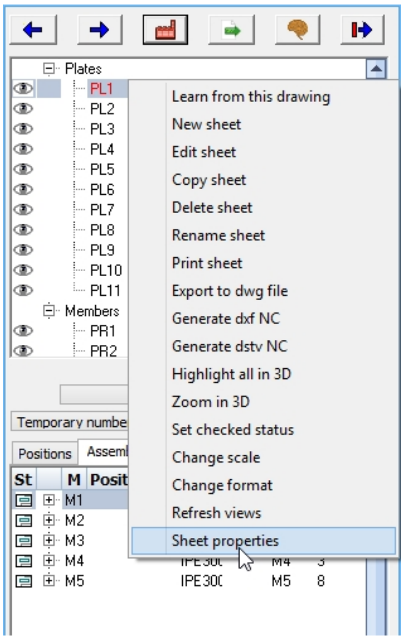



You need to choose from the list the number you want to apply to the expired drawing.
When you press Ok, Parabuild will only update the view of the position or assembly, and nothing further is changed or added.
You still need to modify the dimensions for the elements of the assembly that are added or modified.
This tool is especially useful for large assemblies in which you have invested time adapting them.

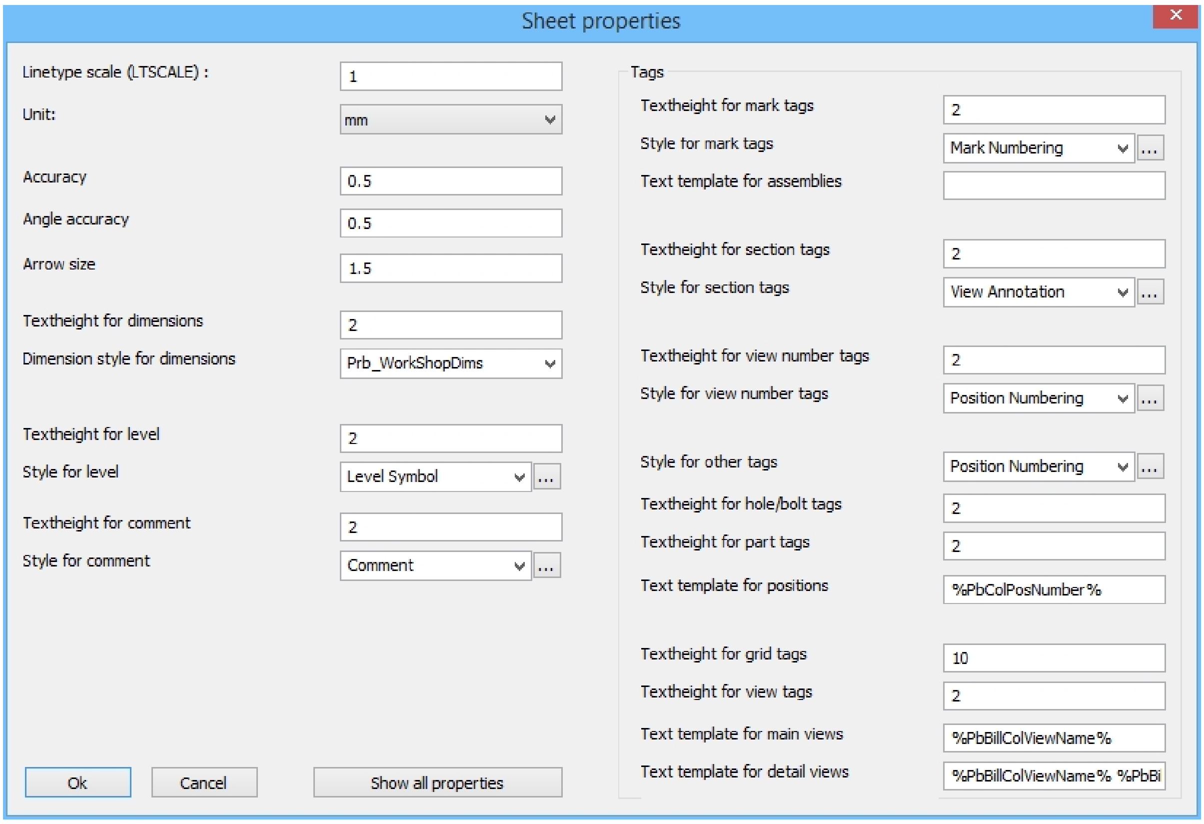
## Sheet properties

There are two methods for influencing the sheet properties.

- For an existing sheet you can start this function by right-clicking a sheet in **sheets manager** , and then clicking on **Sheet properties**.




- You can change the sheet properties of future sheets through the [settings of plates, members, assemblies and General Arrangement view](#).




In this dialog box you can change the following settings :

- **Linetype scale**. The scaling of dashed lines and center lines.
- **Unit**.
- **Accuracy**. The accuracy of measurement for linear dimensions.
- **Angle accuracy**. The measurement accuracy for angle dimensions.
- **Arrow size**. The size of the arrows of dimensions.
- **Textheight for dimensions**. The text height of all dimensions on the sheet.

- **Dimension style voor de bematingen**. The dimension style (AutoCAD) for all dimensions on the sheet.
- **Textheight/style for level**. The text height and style of all level annotations on the sheet.
- **Textheight/style for comment**. The text height and style of all comment annotations on the sheet.
- **Textheight/style for assembly annotations**. The text height and style of all assembly annotations on the sheet.
- **Text template for assembly annotations**. Enter the text that should be shown in the frame of the assembly annotation. Use variables that can be entered between % symbols. Look at all the available variable names by clicking the button Show all properties.
- **Textheight/style for section annotations**. The text height and style of the endplate views and section views on assembly workshop drawings.
- **Textheight/style for view number annotations**. The text height and style of view numbers 1,2,3,4 that are shown on workshop drawings.
- **Style for other annotations**. The style for all other annotations such ass members, holes, bolts, etc...
- **Textheight for hole/bolt/part annotations**.
- **Text template for positions**. Change this field if you want to show more information than just the position number in the annotation of position workshop drawing. Use variables that can be entered between % symbols. Look at all the available variable names by clicking the button Show all properties.
- **Textheight for grid/view annotations**. The text height of a grid annotation will generally be much larger than other annotations.
- **Text template for main views**. This setting applies to the annotation beneath the General Arrangement views. Enter the text that should be shown in the frame of the view annotation. Use variables that can be entered between % symbols. Look at all the available variable names by clicking the button Show all properties.
- **Text template for detail view**. This setting applies to the annotation below a detail views. It is interesting to add the scale of the detail to this annotation, because it will differ from the scale of the sheet. Enter the text that should be shown in the frame of the view annotation. Use variables that can be entered between % symbols. Look at all the available variable names by clicking the button Show all properties.

With the button **Show all properties** you can review all variables that can be used in the text template of annotations.

The text height and text template settings of this dialog box can be empty.
If for example the text height of an annotations is empty, then the text height in the annotation style will be used instead.
Also if the text template of an annotations is empty then the text template in the annotation style will be used.
The reason why these settings still exist here is so that we can still choose different text heights for different sheets.

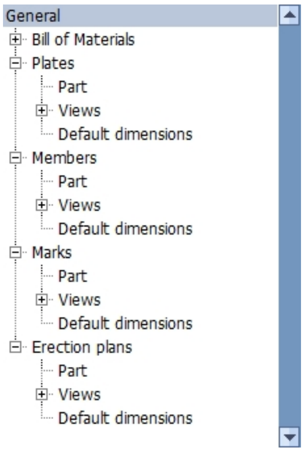
## Settings for generation of 2D sheets

This dialog box can be opened by clicking on 

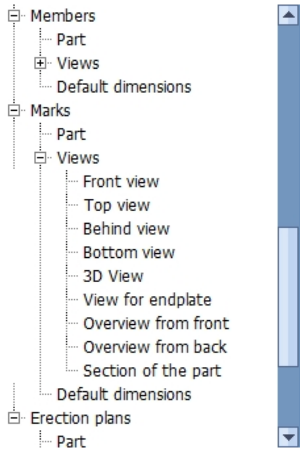
in **sheets manager** .

If you select an item in the list on the left then you can see to the right the settings that belong to that item.
We look at a part if the items in the list. The items are repeated for plates members, assemblies an General Arrangement views. That is why we look at only the items of

assemblies.




- **General**. Here you can change the learn groups, Dstv, Dxf and the accuracy of bills of materials.
- **Bill of materials**. Activate and change the bills of materials here.

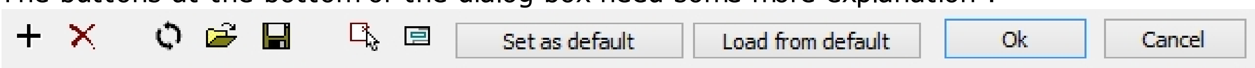



- **Assemblies**. Change here the sheet properties, the layers, title, bill and the priorty of the formats.
- **Part**. Change here the shortening of the sideviews and activate the extra section views and endplate views.
- **View**. Here you can change the sorting of the sideviews. Click the button 

 of 

 to sort the sideviews automatically according to EU or US projection.
    - **Views Front, Top, Behind and bottom**. Activate/ deactivate and change here all the settings of the sideviews.
    - **3D View**. Activate/deactivate and change here all the settings of the optional 3D view.
    - **View for endplate**. When one or more endplate views are drawn, the settings under this view item will be used for the endplate view.
    - **Overview from front**. Change here all the settings of the front overview.
    - **Overview from back**. Change here all the settings of the back overview.
    - **Section of the part**. When one or more section of the part are drawn, the settings under this view item will be used for the endplate view.
- **Default dimensions**. Change here the settings for default dimensions that are drawn without the learning library.

The buttons at the bottom of the dialog box need some more explanation :




 Will add a new group with it's own settings, useful for for example stair, bannister, truss.

 Removes a group that you added manually.

 Restore all settings to the 'Out of the box' defaults of Parabuild.

 Allows you to read all the settings from file, except for the bills of materials.

Allows you to save all the settings to file, except for the bills of materials.

Allows you to draw an example sheet of a 3D object that you select in 3D. If a plate item is active then the sheet of a plate will be generated, if a member item is active then a member sheet will be generated, ...

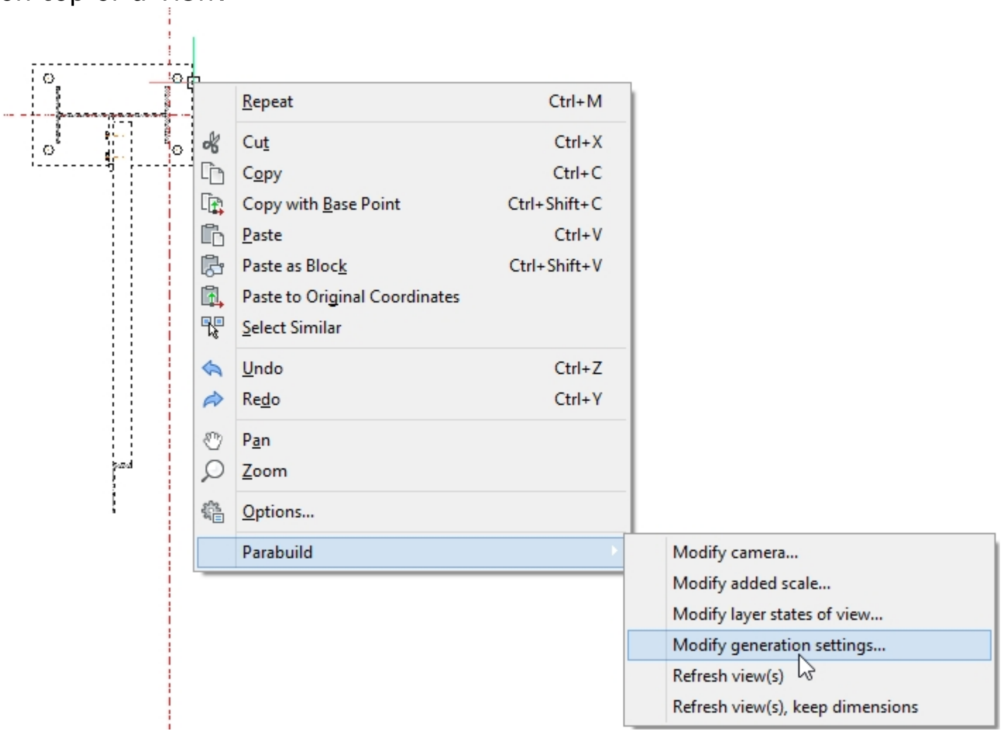
Shows an example sheet of the last selected 3D object.

Set as default Click here to save the current settings so that these settings will be loaded each time you open Parabuild.

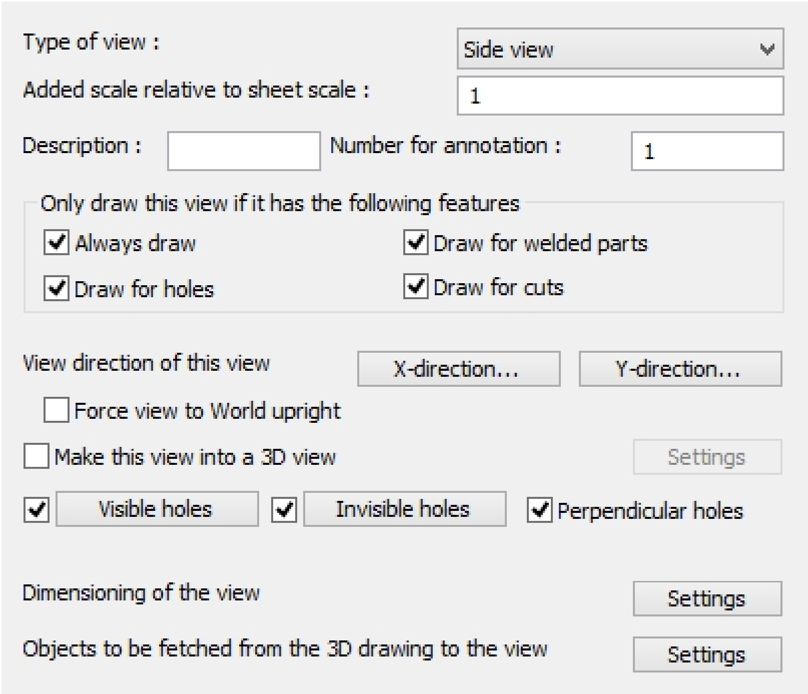
Load from default Click here to load the settings that Parabuild loads at startup.

## Settings of a view

- To change the settings of the future views, you can click on the button Settings in sheets manager , after which you select a view in the list on the left.
- To change the settings of an existing view you can right-click while the cursor is located on top of a view.



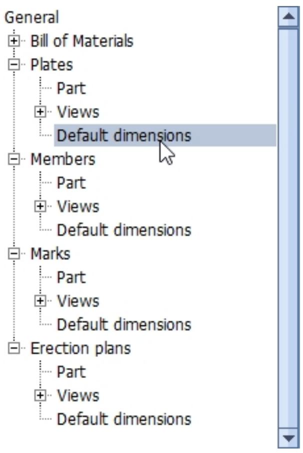

We explain the most important settings of this dialog box :



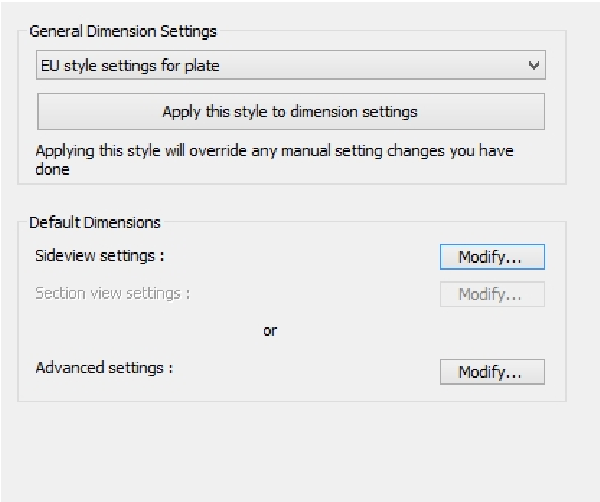

- **Use faster but less accurate 3D models**. If you activate this option then the Parabuild 3D models will be used for the generation of the 2D views. Arcs will be drawn tessellated with short straight lines. Use this option only if you are experiencing difficulties with the option deactivated.
- **Search for source objects of 2D lines**. This option must be activated for measuring of levels, for measuring shortened view and for the adjusting of dimensions and annotations when the view is refreshed. Thanks to this option the dimensions are measured in 3D, not in 2D.
- **Always draw**. This option allows you to deactivate the view (useful for sideviews of members and assemblies).
- **View direction for this view**. Determines the view direction of the view on the part. Only useful for members and assemblies. These settings have been set correct by default.
- **Force view to World upright**. If you activate this settings then a beam will be drawn horizontal, a column vertical and a stringer oblique.
- **Visible/Invisible/Perpendicular holes**. Allows you to change the appearance of holes, or completely disable them. Perpendicular holes are holes seen from the side.
- **Dimensioning of the view**. Allows you to activate or deactivate the dimensions and the different annotation types for the view.
- **Objects to be fetched from the 3D drawing to the view**. Choose whether the axis lines, grid lines, 3D Solids, ... should be copied over to the view.
- **Make this view into a 3D view**.

## Default dimensions

The default dimensions can be changed using the button Settings of the sheets manager .




Default dimensions are dimensions that Parabuild draws automatically for you on the workshop drawings of plates, members and assemblies.
To draw these dimensions Parabuild does not need any learned data.
These are programmed dimensions based on the settings in this dialog box.

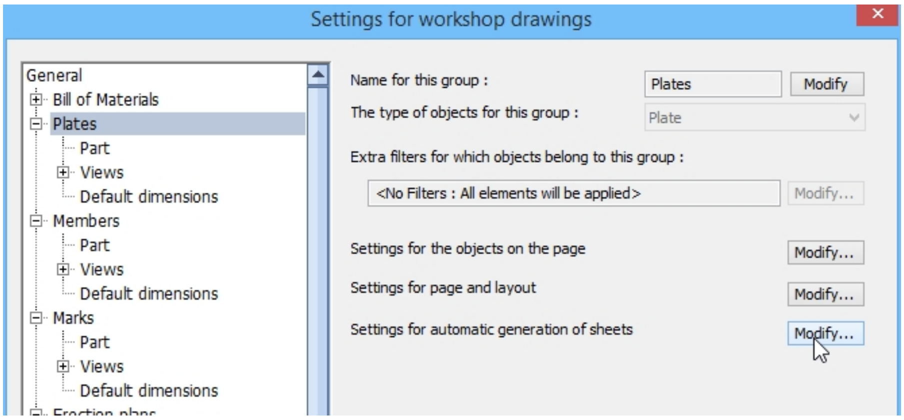



- **General dimension settings**. With this option you restore all the dimension settings to a certain dimensioning-style such as EU or US. Do not use option after you've changed other settings, otherwise your other settings will be lost.
- **Sideview settings**. The settings for the dimensions on all sideviews. This dialog box contains instant help for each settings.
- **Section view settings**. The settings for sections and endplate views. Only for assemblies.
- **Advanced settings**. With difficult but advanced settings you can change the dimensions even more to your needs. The settings you changed here may get lost when you change something in one of the settings above.

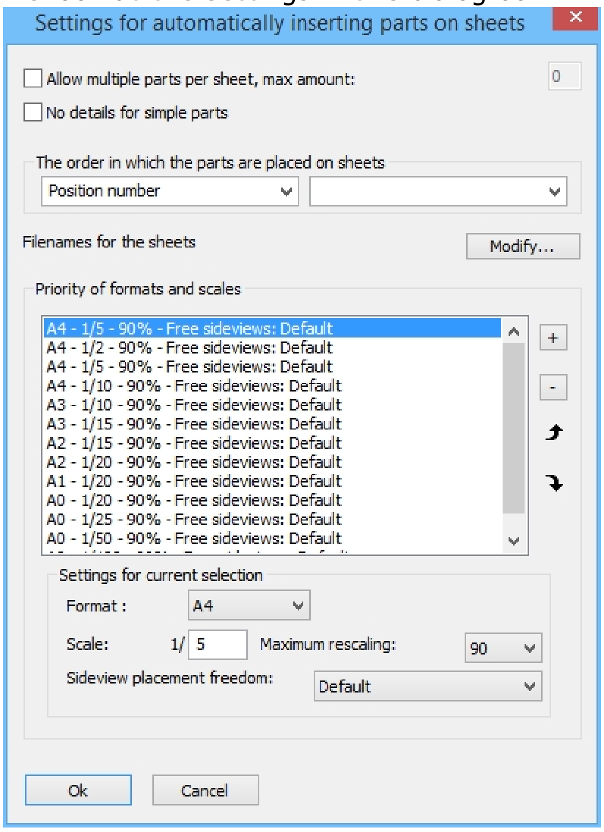
## Page settings for automatic generation

These automatic choices can be changed in the 

of **sheets manager** .
Then you activate *Plates*, *Members* or *Assemblies,* and then you click on **Settings for automatic generation of sheets**.




We look at the settings in this dialog box.




- **No details for simple parts**. Rectangular plates that don't have any holes will not get a workshop drawing. Members without holes or cuts will not get a workshop drawing. Assemblies without welded parts will not get a workshop drawing.
- **The order in which the parts are placed on the sheets**. Enter here 1 or 2 properties that should be used for the order in which the sheets should be generated. This way you can for example sort according to material so that concrete parts will be separated from the steel parts.
- **Filenames for the sheets**. Only necessary when you used a sorting method that differs from the position or assembly number.
- **Priority of formats and scales**. In this priority list you should put the preferred format and scale at the top of the list. Parabuild will try to fit the part on the first format+scale combination in the list. If it doesn't fit, then it will try the next format+scale item in the list until the part fits. You can choose multiple scales for the same format before switching to a different format. Make sure that the list doesn't become large (less than 30) because this can slow down the generation process.
    - **Format/Scale**. Enter the format and the scale that Parabuild should try.
    - **Maximum rescaling**. With which percentage can Parabuild rescale the views

even further in order to let the part fit on the sheet? This rescaling will simply scale the annotations and dimensions, so they will become a less readable. But this method can be a big space saver.

- **Sideview placement freedom**. Enter the freedom that Parabuild has in rearrangement of the sideviews.
    - **Always**. The sideviews can be drawn freely on the sheet.
    - **Default**. The sideviews will be outlined most of the time, but in some cases where the part is big and not linear Parabuild will not align the sideviews in order to achieve more efficient space usage on the sheet (for example for a bannister).
    - **Never**. The sideviews always need to be aligned with each other.

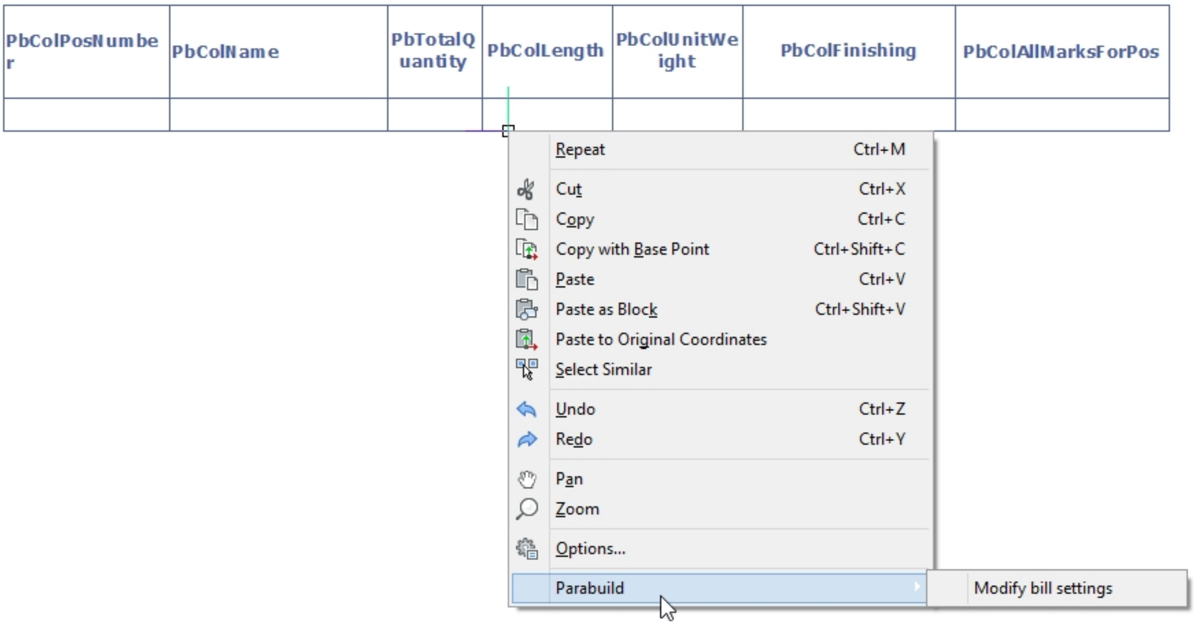
## Settings for the bill on 2D sheets

The bill on the workshop drawing is adjustable through the Settings of the **sheets manager**.

The bill that is shown in the template drawing can be adjusted to your own preferences.

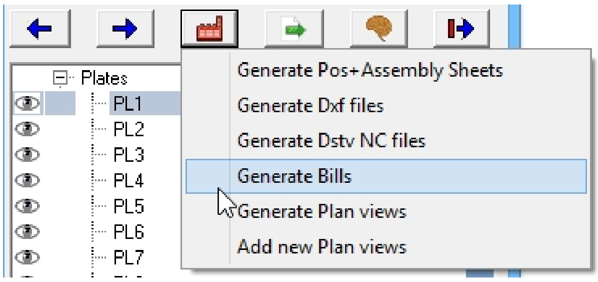

The first row is always the titles row.
The second row is used for filling the bill.
You can change the color, text style, text height and column widths of each field.
The appearance of the titles are adjustable separately from the filled in bill rows.
You can add or remove columns. The text in the title determines the property of the parts that need to be filled.

The sorting of the rows and the totals row are adjustable by right-clicking on the view :




## Generating all bills

You can start this function by clicking in **sheets manager**.

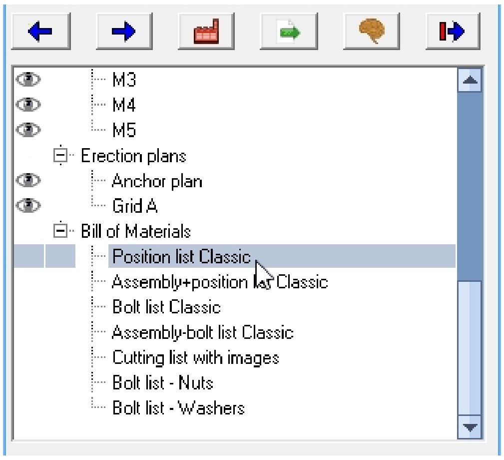



All bills of materials that are listed in the sheets manager will be generated.
The files will be stored in the same folder as the folder of the 3D dwg files.

If you want more or other bills in this list, then go to [Settings for bills of materials](#).

## Generating one bill

You can request a bill of materials by double-clicking or right-clicking on a bill in the list of the **sheets manager** .



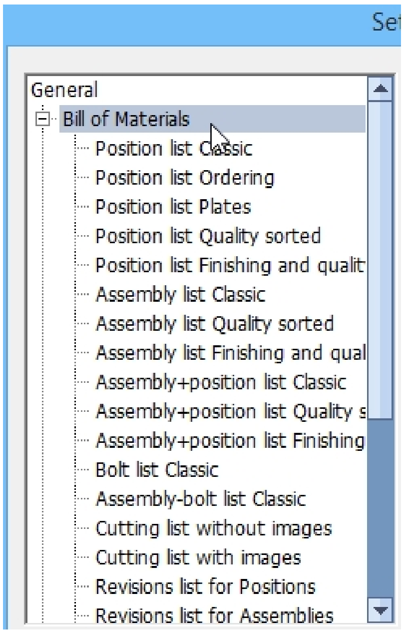

The bill will be generated and then immediately opened in Excel, Open Office or the Parabuild spreadsheet. The reader that is used depends on the output setting in the bill's settings.
The file will be stored in the same folder as the folder of the 3D dwg files.

If you want more or other bills in this list, then go to [Settings for bills of materials](#).

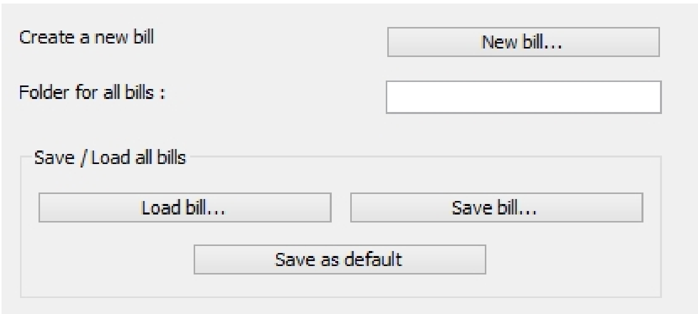
## Settings for bills of materials

You can start this function using the button 

in **sheets manager** .
In the general **Settings for workshop drawings** dialog box you need to open the *Bills of Materials* item in the tree structure at the top left.

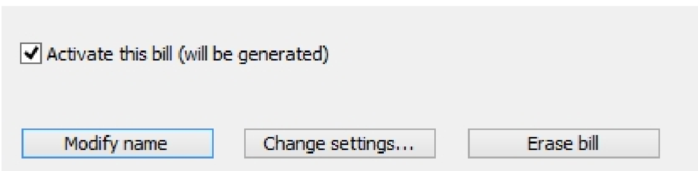



When *Bills of Materials* is selected in this list, then you can perform the following actions :




- **New bill...**. Create a new bill of materials from scratch.
- **Load bills...** Load a file that contains all your favourite bills.
- **Save bills...** The current bills will be store to a file.
- **Save as default**. The current bills will be stored as default, which means that these bills will be loaded each time you start Parabuild.

When a bill selected in the list to the left, then you can start the following functions :




**Activate this bill**. The bill will become visible in the list of the sheets manager , so that it can be generated.

- **Modify name**. Use this to change the name of the bill in this dialog box as well as in the sheets manager.
- **Change settings...** Will change the settings of the bill.
- **Erase bill**. Permanently erases this bill.

# Part list settings

A versatile part list that will satisfy the highest possible number of users is a part list that can be edited in a multitude of ways. This means a part list with a multitude of settings. This is the reason for the extent of this dialog window.

If the user is entirely satisfied with the standard part lists already available, this dialog window may be ignored.

When setting up a new part list it is advisable to run through the settings from left to right and from top to bottom.

## 1) Global settings

- **Bill name**: The name for both the eventual file name of the part list and the name in the list of part lists.
- **Bill type**: Select one of the 9 types of part lists. Every part list type has its own column options and is generated in a variety of ways.
- **Output type**: Regular text is an ASCII file that can be read or printed using notepad.
- **Separation line** between all columns: places a vertical division line between each column of the part list.
- **Repeat column-titles for every new page**: Repeats the column titles every time a new page is started.
- **Number of lines per page**: Determines the number of lines contained within a page, or simply how often the column titles are repeated.
- **Add totals to the end of the bill**: Adds totals at the end of the complete part list, this can be further developed as required using the sub totals table (more information see below).

## 2) Column sequence

This is a table in which every row represents a column of the part list. Every row has four settings:

- **Checkbox**: If the box at the left-hand side of the row is checked, the column will be included in the part list.
- **Column name** (description): Next to the tick box is the column type (e.g. Position number).
- **Column width**: The width of the column, for normal text this is the number of letters.
- **User column name**: This is the actual column name that will appear in the part list. As the description is often too long, an abbreviation or similar may be used.

Column name and column width are text fields that can be edited by double-clicking on them.

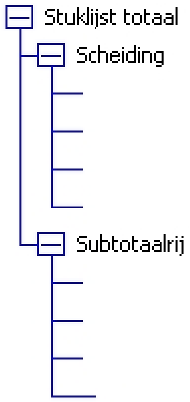
This table not only determines what columns are on, but also in what sequence. The top row in the list is the column that will eventually appear at the far left on the part list. To move a row up or down, first select the row and then click on one of the two arrows on the left-hand side of the table.

## 3) Column sorting

The columns that eventually appear on the part list are set using this table. How all of the elements (plates/profiles/bolts) are sorted within the part list can be set totally separate. This can be carried out using the following table. The system is almost the same as the previous table: the first checkbox indicates if the row has been sorted or not. The last checkbox determines if a sub total has to be made for this column (see following table).

## 4) Sub totals

This table is slightly more complicated, because every row no longer works independently. This table has a tree structure.




This illustration shows one tree.

One complete tree is one sub total line. A tree is added for every sub total that has been clicked on in the column sorting.

Every tree has a variety of settings:

- **Separation**: This describes the final appearance of the division between the normal elements of the part list and the sub total. All possible combinations can be selected by checking or unchecking the boxes. The sequence can also be edited using the arrows (e.g.: first a line and then a new page).
- **Subtotal row**: The columns where a subtotal is to be made. If an extra column is placed in the subtotals, but is not on, the description will be used.

## 5) Element filters

This can be used to determine from which elements the part list will be created. This can be useful to generate a separate order list for steel and concrete for example.

The filter system works in accordance with a list of rules. The rules are in an AND relation, this means that all rules have to apply to an element before it will be included in the part list. If no rules have been created then all elements will be applied to the part list.

To create or delete a new rule, click on the right-hand button. The field 'Editing rule' allows the rule to be edited. Click on the rule in the list and this will allow the rule to be edited. When the rule is edited, the change is displayed immediately.

By way of example the rule for a part list containing only steel elements:

*If the property Position Quality starts with SS,*

In another part list, but then for concrete the following rule would be used:.

*If the property Position Quality starts with concrete,*

## Exercise: Making a revision list

To practice working with part lists, use this exercise to create a part list.

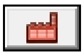
In this exercise, we will add 2 columns to the revision part list. The columns should be added manually, as they are project dependent. Countless revisions can be created all of which with their own name.

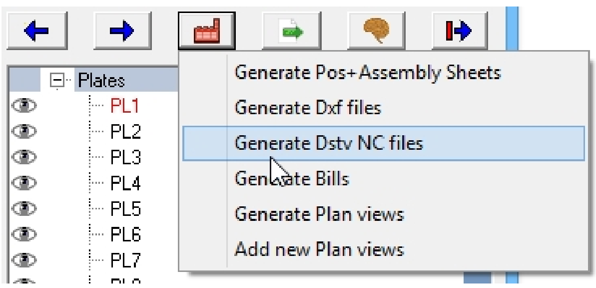
1) Ensure that a drawing is opened which contains at least two locked revisions (more information on locked revisions in the Revisions) chapter). For the purpose of this exercise, we will assume that the locked revisions are 0 and 1.
2) Open the dialog window of the **workshop drawings** and go to the **part lists** section.
3) Activate (check) and select the part list **Positions revision list** and then click on **Edit**.
4) Ensure that at the top left of the settings **Skip unchanged elements** is on.
5) At the top right-hand side, tick the two locked revisions **0** and **1** of the project.
6) Click on **Ok** and then once again on **Ok** to generate the part list.

If everything has been carried out correctly the end result should look similar to this:

```
-----------------
Pos     | 0 | 1
-----------------
PR2     | 1 | 0
PR3     | 0 | 1
```

## Generating all DSTV files

You can start this function by clicking the button  in **sheets manager** .




Dstv files can be read by most cutting- and drilling CNC machines.
Also some plate machines (laser, plasma) can read Dstv files.

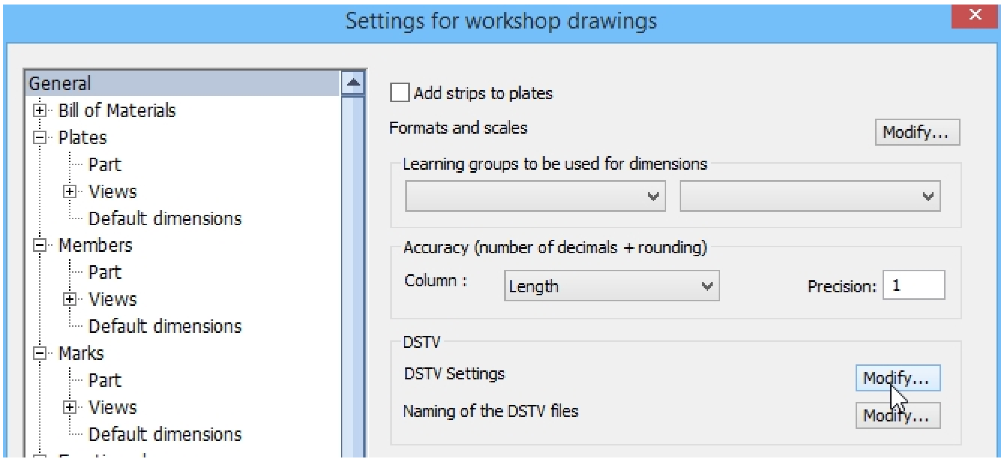
The Dstv files will be stored in the same folder as the location of the 3D dwg file.

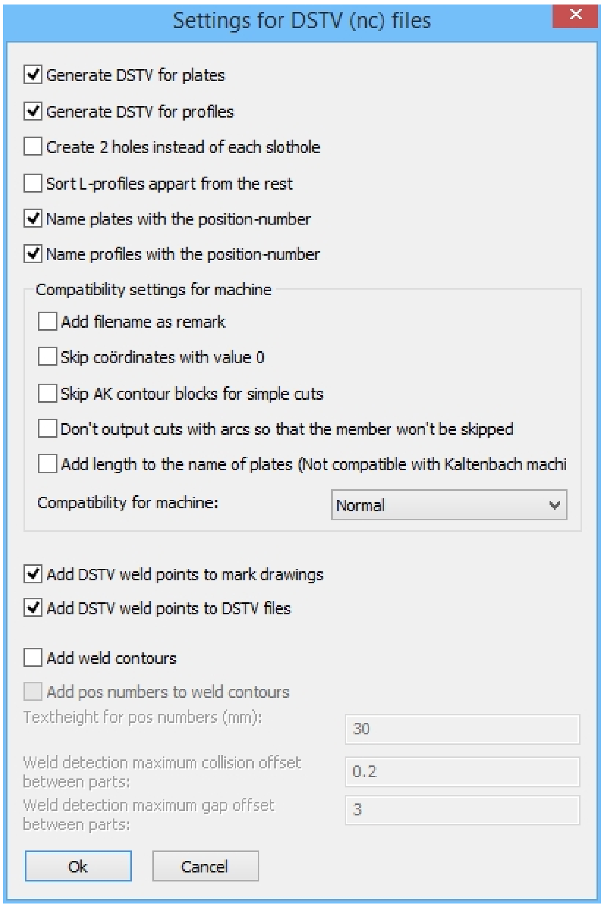
To change the settings of the Dstv files, you need to click the button 

in **sheets manager** .

In the list you need to click on General, so that in the middle of the dialog box you can click on the button 

next to *DSTV Settings*.




This dialog box has the following options :

- **Create 2 holes instead of each slothole**. If activated then Parabuild will insert two holes in the Dstv file instead of a single slothole. This is useful if the machine doesn't have a tool for creating slotholes.
- **Sort L-profiles apart from the rest**. If activated then the dstv files of L-profiles will be stored into a separate folder. This is useful when the L-profiles need to be done with a different machine.
- **Annotate plates with the position number**. The machine that supports this will engrave/write the position number on the plate.
- **Annotate profiles with the position number**. The machine that supports this will engrave/write the position number on the profile.
- **Compatibility settings for machine**. Some machines do not follow the Dstv standard, or are based on an old version of the Dstv standard. That is why some compatibility settings are necessary in order to support as many machines as possible. Only change

these settings if you are experiencing problems with reading the dstv files in your CNC machine.
  - **Add filename as remark**. Some machines expect this, and it does no harm.
  - **Skip coordinates with value 0**. Some machines do not expect any values with value 0 while this shouldn't make a difference.
  - **Skip AK contour blocks for simple cuts**. Some machines always need the AK block, others don't support the AK block at all.
  - **Don't output the cuts with arcs so that the member won't be skipped**. Parabuild skips the profiles that contain curved cuts. If you activate this setting then these profiles will not be skipped any more but the curved cuts inside the profiles will be skipped.
  - **Add length to the name of plates**. Normally the plates receive a name such as PL150x10. But some machines also expect the length in this name, so it will become PL150x10*200.
  - **Compatibility for machine**. These are a collection of changes to the standard that some old machines need.
- **Add DSTV weld points to assembly drawings**. For each weld point in Dstv Parabuild well draw a cross in the assembly drawing.
- **Add DSTV weld points to DSTV file**. The drill of the machine can be used to create a dot to facilitate the welder. You can find out more about weld points here.
- **Add weld contours**. The circumference of the touching face of the main part and the welded part will be engraved/written by the machine.
  - **Add pos numbers to weld contours**. The position number of the welded part will also be engraved close to the contour.
  - **Text height for pos numbers (mm)**.
  - **Weld detection maximum collision offset between parts**. If the welded parts collide with each other too much then the weld contour won't be added to the dstv file.
  - **Weld detection maximum gap offset between parts**. If the gap between the welded parts is too much then the weld contour won't be added to the dstv file.

### DSTV weldpoints

# DSTV weldpoints

DSTV weld points are points that are placed in the automatically produced DSTV nc-files. A point indicates on which spot an element must be welded on the profile. Because the cnc-machine can drill these points automatically, it can result in winning a lot of time because one has to measure less when welding.

The points will normally be made by the machine with the tip of the (currently present) drill.

Because of the flexibility of the numbering of profiles and plates the program must place the points automatically. With some general options you can influence the placement of the point for each element.

# Creating a points list

First of all we have to make some DSTV points that we will later use for each element.

In the **Parabuild Settings** dialog box (icon SET) there is a button “DSTV weld points”.

When clicking that button you will see a small dialog box with a list of weldpoint groups.

Here you have to set one group as current in order for the DSTV weld points to work. The points in the current group will be used for this drawing (the current group can be established separately for each drawing).

The purpose of several groups is that one can create a different set of weld points for a certain project, and thereby can keep the normal weld points intact.

# Creating a point

When you click on **Edit...**, a new dialog box will appear.

At the top of the dialog you see all points in the group. You can create new points or remove points with the buttons next to the list.

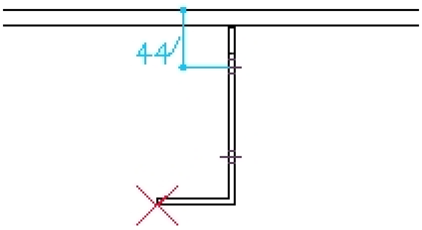
If you select a point from the list then the options for that point will be visible beneath it and you can modify these.

# Properties of a point

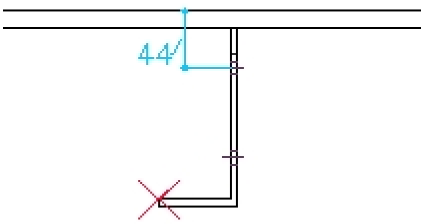
**Name**: The name for the point. We will later use this name if we want to use this point for a welded element.

**Choose corner**: You choose the corner where the weld point should be positioned on the welded element.

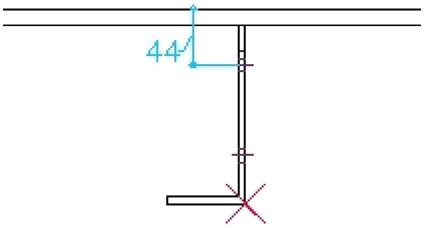
We illustrate all possible choices with an example:



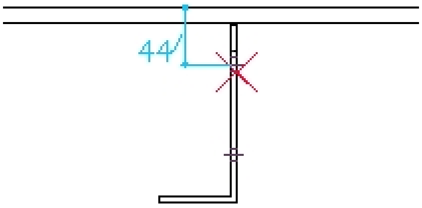

Corner left under.




Corner left above. The point will only be placed where there is material, in this case below but the upper part of the angle bracket.



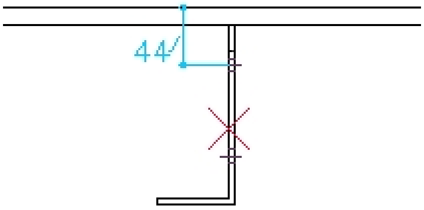

Corner right under.



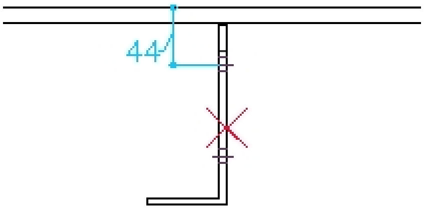

Corner right above. The point was not placed entirely at the top because the drill head cannot come this close to the flange. This clearance space is adjustable, see further in the manual.

**Choose intersection**: To understand this option we must imagine that a line is drawn through the welded element (on the weld plane). The weld point is placed on the first point where the line intersects the welded element. With this option you choose where this intersection line should be placed.

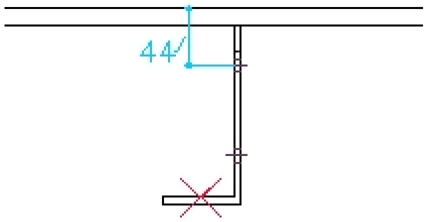
We illustrate all possible choices with an example:



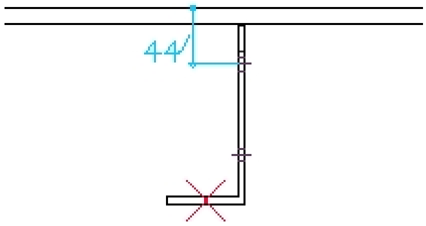

Intersection left.




Intersection Right.




Intersection Above




Intersection under + middle. With this example not only the option **Intersection under** was enabled but also the option **In the middle of the welded element's thickness**. The result is that the point will be placed in the middle of the bracket's thickness.

**In the middle of the welded element's thickness**: If this is activated then the point will be placed in the middle of the thickness at the local intersection.

**Choose view**: If this is set to **Automatic** then the software will always use the view that directly looks at the welding plane. However you can choose to manually set another view, in those cases where the automatic view is not desired (for example for brackets that are welded against the flanges of an I profile).

**Minimum distance to edge**: The machine can't move its drill too close to obstacles. If there is a point on the web of an I profile, then the weld point should keep clear a distance from the flange. Otherwise the drill would collide against the flange and cnc machine refuses to create the point.

The value you enter here is the clearance Parabuild will use automatically to avoid these occurrences.

**Drill complete hole**: If you activate this option, a complete hole will then be drilled instead of a point. This hole will never be visible in the 3D drawing. It will still be treated as if it is a weld point. Only during the communication with the cnc-machine a complete hole will be passed on instead of a point.

This hole can be useful, for example for stiffeners: we only have to drill one hole for both stiffeners and the hole is also more visible while welding.

**Hole diameter dependent on plate thickness**: If you activate this, you do not have to enter a diameter for the hole, but how much larger than the thickness the hole has to be (offset).

The points we just looked at have to be set up just once.

**Coupling weld points to elements (you have to do this for each project)**

Now you have to give each welded element that needs it a weld point.

This is possible using the AutoCAD Properties (see group 'Output').

In the field **DSTV weld point** of the properties you enter the name of the weld point.

This property is of course only useful for welded elements.

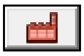
This property exists both for plates and profiles.

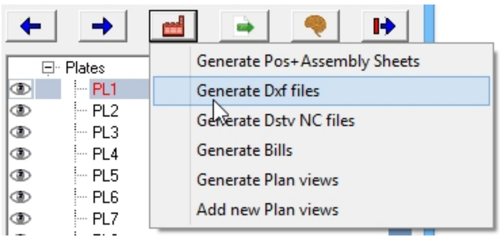
You can give one element multiple weldpoints.

When the above options were completed successfully, then you will see crosses on the mark (assembly) drawings. These are the weld points that will be passed on to the machine. The crosses were added to the assembly for the convenience of the welder. This way one can see more easily on which spot the welded element and the weld point must match.

Assembly drawings and DSTV files will also get different names. Normally you have one DSTV file for each position number, but that is no longer possible when that profile has weld points. In that case the file name of the assembly drawing and the DSTV file will contain both the mark number and the position number. The result is more files and drawings, but it is an inevitable disadvantage.

## Generating all DXF files

You can start this function by clicking on the button  in the **sheets manager** .



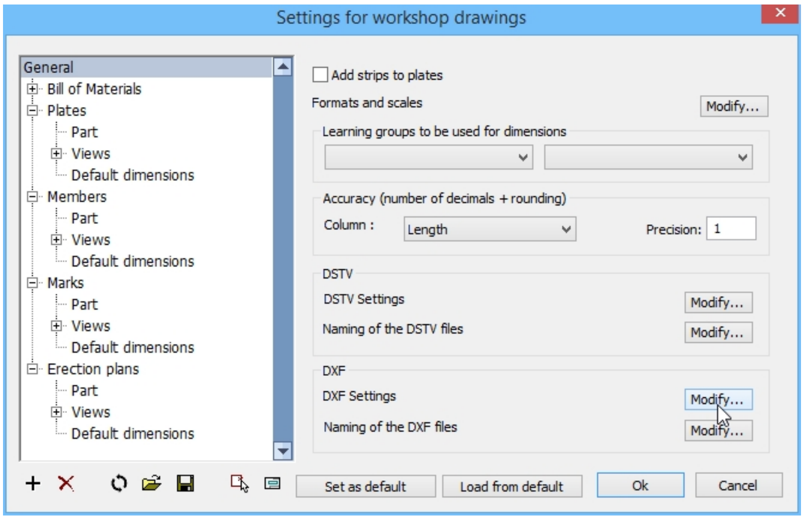

Parabuild will only create Dxf files for plates. These Dxf files are only meant for plate machines (laser, plasma) that only have 2D capacity.
The Dxf files will be stored in the same folder as the location of the 3D dwg file.

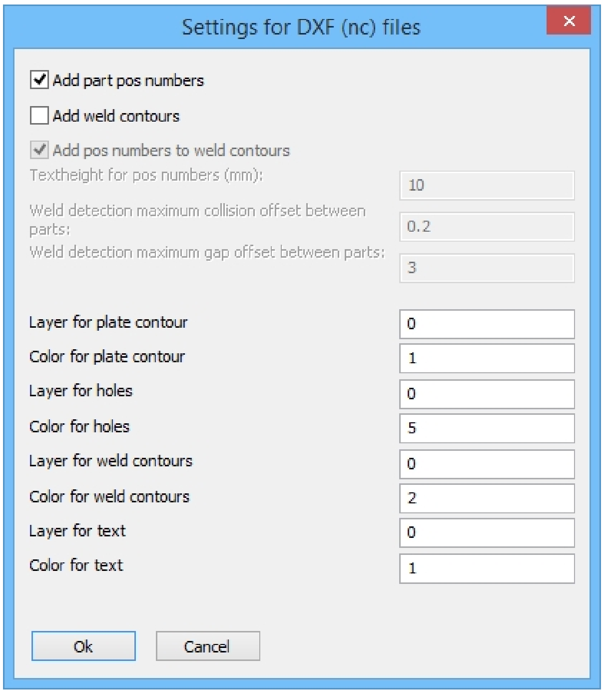
To change the settings of the Dstv files, you need to click the button 

in the **sheets manager** .

In the list you need to click on General, so that in the bottom of the dialog box you can click on the button 

next to *DXF Settings*.



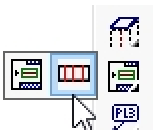




In this dialog box you can change the following settings :

- **Add part pos number.** The position number of the plate will be added as text to the Dxf file so that the machine can engrave it on the plate.
- **Add weld contours**. The circumference of the touching face of the plate with another part will be engraved by the machine.
- **Layer for xxx**. Enter the layer name that the machine expects, if necessary.
- **Color for xxx**. Enter the color number that the machine expects. Some machines expect another color number for the circumference than for holes.

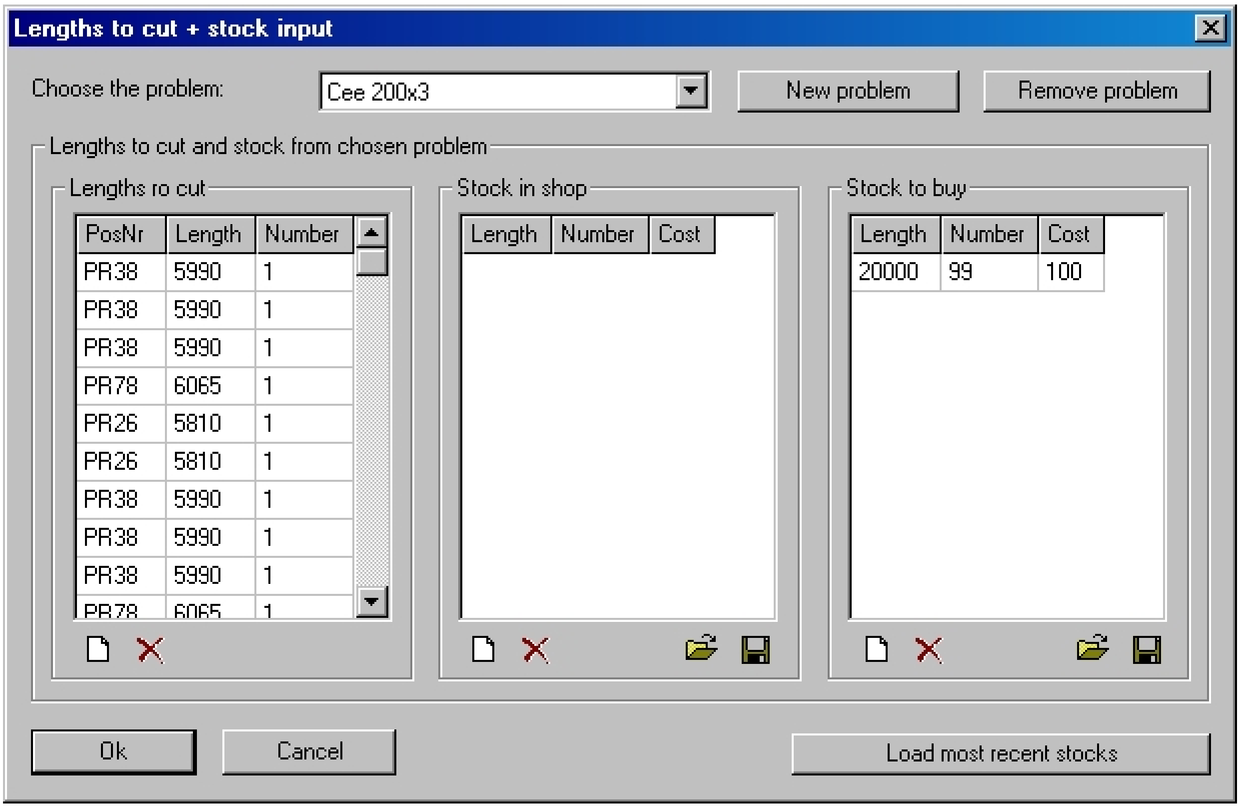
## Profile length optimisation

Command : **S3d_Opt**

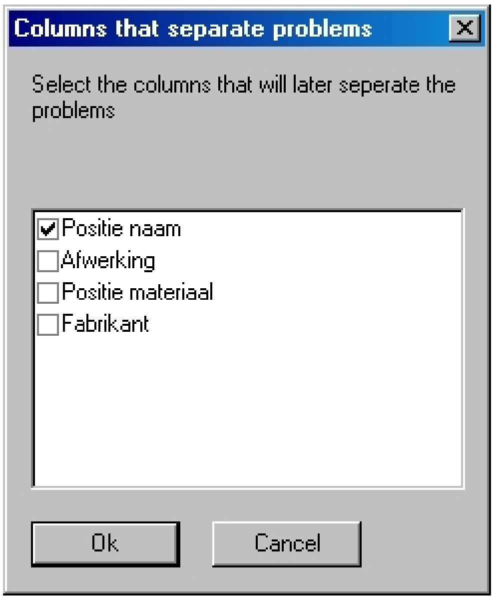


This command can search an efficient method for you to cut the required profile lengths from the available stock in order to waste as minimum material as possible.

First, it is required to input the required profile lengths and the available stock. To do this you have to fill the 2 first dialog boxes.




In the second dialog box you have to enter the lengths to cut and the stock for each problem. What is a problem? Each type of profile is a problem, for example HEA200. The problems are automatically retrieved from the drawing and entered into this dialog box using the properties you enabled in the first dialog box.




Also the lengths to cut are automatically retrieved from the drawing and entered into this dialog.

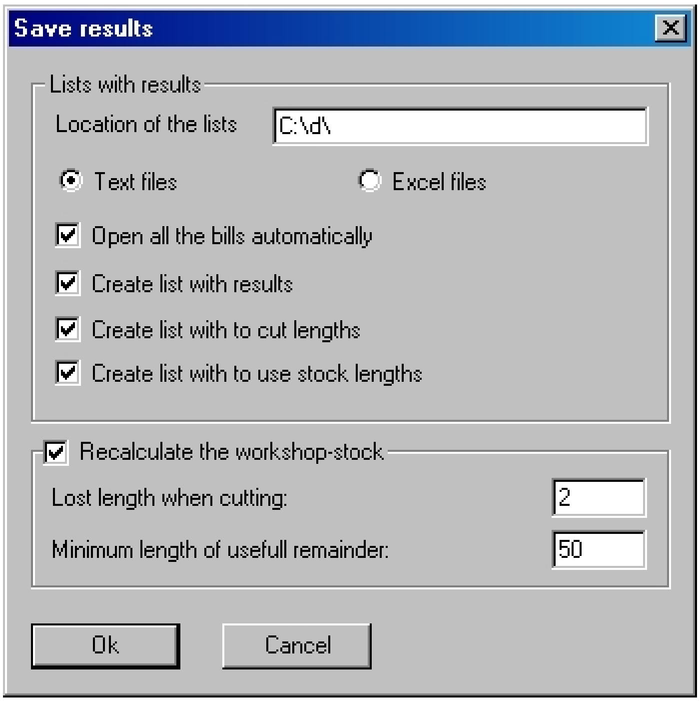
Of course you can always add, remove or adjust problems, lengths to cut and stock.

There are two types of stocks: 'Workshop stock' and 'To buy stock'. They are split apart so we can later easily renew the workshop stock: Removing used lengths and adding new remainder lengths. You can automatically load the most recently saved stocks with the button bottom right! The command will then search and load the newest workshop stock (.shp) and the newest to buy stock (.buy) for each problem. The location being searched is as follows:

"c:\Parabuildv1\S3d_Lib\Cutting Stock\HEA200\... "

c:\Parabuildv1\S3d_Lib\ is the location of the library; this location can differ depending on your installation. The last directory HEA200 is the name of the problem.

The field 'Cost' for stocks is a percentage of the length. Later while calculating efficient combinations these percentages will be taken into account. Lengths that have a lower cost will be used more quickly than the expensive. This means the to buy lengths should always have a cost of 100%, and the workshop ‘remainders' should have a lower percentage. The exact percentage depends on how fast you want to get rid of them.




The third dialog box is meant for saving the end results. You have to enter the location where the lists with results have to be saved, and also which type of lists you want.

Also in this dialog box you can choose whether the workshop stock should be renewed. If you enable this feature, you will get another dialog box after calculation that shows you the updated workshop stock for each problem. The used lengths will be removed. The new remainder lengths will be added. That last dialog box can also save all the generated workshop stocks in the following location:

c:\Parabuildv1\S3d_Lib\Cutting Stock\HEA200\Stock1.shp

The directory HEA200 is the name of the problem. The file Stock1.shp will become Stock2.shp should the first already exist, or Stock3.shp should it already exist too, or...