Parabuild manual

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Introduction

Parabuild runs on both AutoCAD and BricsCAD platforms. Structural steel objects can be modified through the standard AutoCAD and BricsCAD commands or using the Object Property Manager.

All Parabuild objects are integrated as if they are native AutoCAD/BricsCAD objects.

Fast and intuitive modeling

For easy detailing of structural steel - small parts or frame elements such as beams, columns, girders, struts, joists, purlins - use the Context Modeler for maximum productivity. Move the cursor around and solutions are offered in real time based on cursor location, nearby members and other context. The new element is automatically linked to the nearby members or to the grid.

Connections

Parabuild has a library of hundreds of standard steel connections. Detailing connections is easy; select the members and only valid types are proposed for that situation. The connection is immediately drawn and any changes made are shown instantly on the 3D model. Easily turn unconnected members into fully connected structural steel.

User-defined connections

Every single standard connection was created using tools that are available to any detailer, without programming a single line of code. This was done by using simple geometric rules on members and parts inside template drawings, which means you can create your own connections! They are integrated with the existing connection system for easy re-use in all your projects.

Geometric constraints

Parabuild is the only structural steel detailing software that allows fully constrained structures. Connections and macros can be constrained in template drawings and re-used in any project. That includes whole structures, trusses, wind bracing, cladding, stairs, … This means it is possible to customize Parabuild without hiring a programmer or writing any code.

Clash detection

Parabuild will automatically detect any collision between parts and show it on the screen. Bolt clearance and hole to edge/hole to hole distances are checked as well.

Numbering of identical parts

Parabuild automatically recognizes identical parts in the model, with the purpose of making the Bill of Materials shorter and for needing less shop drawings.

CNC file generation

Parabuild can write DXF, Dstv, Sat and Step files automatically for the purpose of using them for CNC machines.

Shop drawing generation

Parabuild will generate clear and readable shop drawings for the fabrication of all the parts in the drawing.

It supports 'Position' drawings, which are 'per part' drawings containing dimensions for holes and cuts.

It also support assembly drawings, which contain the dimensions for welding multiple single parts.

General Arrangement Drawings

Parabuild has many tools that allow you to quickly create GA drawings. It will also help with updating the existing GA drawings after changes have been made to the 3D model.

CAD Interface

AutoCAD / BricsCAD

AutoCAD is the most widely used CAD software in the world. It is easy to use, there is a large community behind it, a lot of documentation, and a lot of training material. Most drafters are familiar with AutoCAD and new detailers can be trained to master it with ease.

BricsCAD is very similar to AutoCAD in many ways, it uses the same file format, and is only a fraction of the price. Because each has its own advantages we feel the choice should be up to the detailer, so we are committed to supporting both platforms. Scroll down to read more about these two excellent programs.

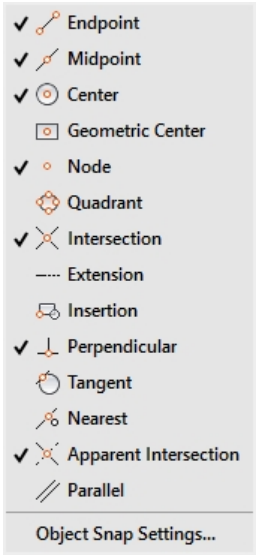
The combination of AutoCAD and BricsCAD, with Parabuild's unique features results in an easy to use yet powerful solution for drawing 3D structural models. For steel detailing Parabuild benefits even more because of the extensive 2D and dimensioning tools in both applications

Complete integration

Parabuild was completely integrated with both platforms. Here are some examples of this:

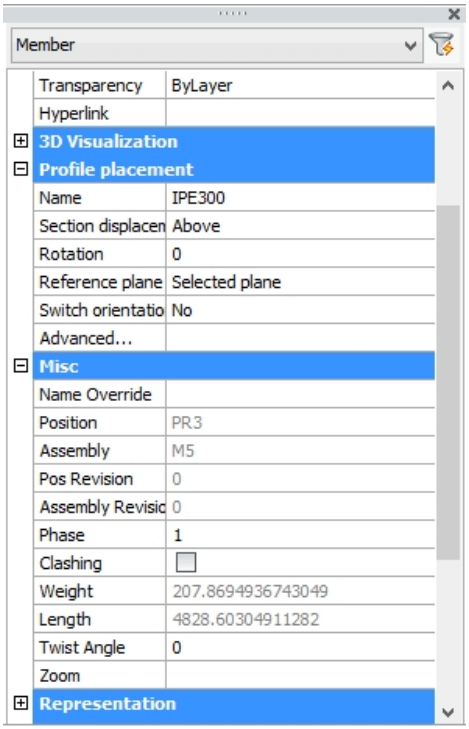
Osnap interface

You can use the powerful object snapping tools on any of the Parabuild entities and sub-entities.



Object Property Manager (OPM)

Parabuild objects can be modified, searched and filtered through the powerful OPM interface. This helps you make changes to many objects at the same time with minimal effort.



Standard commands

The standard commands can be used on Parabuild objects. That includes stretch, move, copy, array, mirror, …

DWG file format

Parabuild drawing information is entirely contained in the .dwg file, the most pervasive CAD file standard. The same drawing file is used in both AutoCAD and BricsCAD, so there is no need to

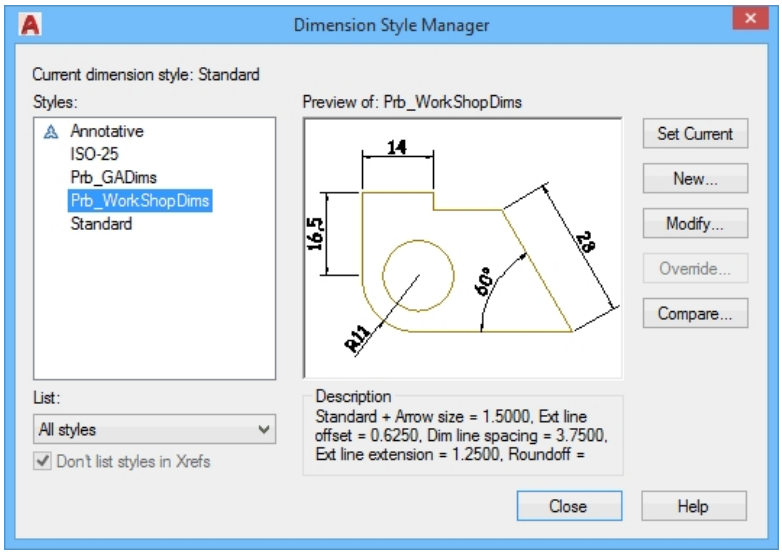
convert drawings and you can freely switch between both platforms at any time.

Easy to learn

Both platforms are easy to learn and if you are familiar with one of them, using the other will feel quite natural. Combined with the intuitive and efficient features of Parabuild this results in an environment that gets you to productivity quickly.

Extensive dimensioning tools

AutoCAD 2D sketching and dimensioning tools are popular and refined through decades. BricsCAD works similarly and has tools that are just as powerful and easy to use. Parabuild takes full advantage by drawing all automatic dimensions as native AutoCAD/BricsCAD dimensions making it easy for the detailer to make changes when needed.

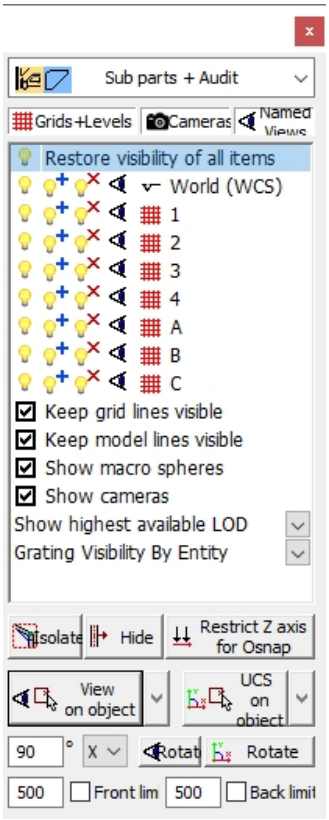


3rd party applications

Both platforms are open development platforms. That means 3rd party applications are available for a wide variety of CAD sectors, and these apps can be used simultaneously in the same drawing. The fact that all 3rd party apps work inside the same CAD environment and on the same drawing, is a unique environment that gives the draftsperson the freedom to combine many different disciplines within the same CAD environment. AutoCAD and BricsCAD are very popular development platforms, which has resulted in the availability of thousands of 3rd party applications on these platforms.

Additional visibility tools with Parabuild

AutoCAD and BricsCAD have many powerful functions that allow you to visualize your 3D model : visual styles, slices, view limitations, object hiding, transparency, etc… Parabuild adds extra visibility tools that are specific for structural steel detailing and they allow you to save time while modeling, or provide you with a better insight in the model.



AutoCAD or BricsCAD: two good choices

The choice between AutoCAD and BricsCAD is mostly one of preference. Both are good platforms and will help you get the job done efficiently. Parabuild is equally powerful on both platforms and we aim to keep it that way.

AutoCAD is the industry standard DWG platform. In development since 1980 and with millions of users, AutoCAD is a CAD environment that has been refined for decades. It is the current market leader for general CAD software. There are thousands of 3rd party applications based on the platform, whatever you need it will for sure exist as a 3rd party app on AutoCAD. We have been a Autodesk Developer Network (ADN) member for over 20 years, and we consider it an excellent CAD application.

BricsCAD is fully compatible with the DWG format. BricsCAD offers the same features as AutoCAD, and in some cases more. BricsCAD costs only a fraction of the price of AutoCAD. While there are currently less 3rd party apps on BricsCAD, this is rapidly changing. Some unique features in the premium versions of BricsCAD are direct modeling, an impressive sheet metal module and improved AEC tools. The team working on BricsCAD is very open and dynamic and is supportive and responsive to us as developers. This makes it a joy to have them as a partner and a very good choice as a CAD platform.

BIM - Building Information Modeling

About BIM

The key idea behind BIM (Building Information Modeling) is to have all information related to a

facility organized in one application-independent model. Ideally, this model is started at concept stage and expanded during each phase of construction and operation of the facility.

While BIM is centered around a 3D model, it is not limited to geometry. Other possibilities are costs, maintenance information, manufacturers, materials, names, dates and times, position and assembly numbers, phases, revisions, and so on.

While any application storing these properties with the 3D model often claims to be BIM software, for it to have practical benefit the software should support the free exchange of data with other applications. While there are several formats available, in the last few years the IFC file format has emerged as the best way to exchange BIM models. IFC stands for Industry Foundation Classes; it is an open-source file format developed by the independent organisation buildingSMART, and it is supported by all the major CAD vendors.

Why is BIM so important?

Information management is key to completing a project on time and within budget. Yet it is information that poses a recurring problem in the construction industry. Throughout all stages of a project a wide variety of different document types are generated by different tools and different teams, and at each stage the data is being manually reworked to fit the task at hand. Challenges include incompatible file formats, 2D/3D conversions, communication problems, limited transparency and more. This results in more errors and inefficiencies, ultimately increasing costs and causing delays.

Instead of moving and mutating data throughout the process, BIM methodology proposes that all teams work off the same data from start to finish, regardless of which BIM software is used. This not only facilitates communication and collaboration, but allows information to flow between the various tools used by the construction industry.

What are the main BIM benefits?

While the initial effort to set up a BIM model can be a bit bigger, there are benefits at each stage of a construction project. Even at the concept and design stage, the 3D aspect can help with visualization and communication with the client. Different variations can be more efficiently explored, and even at this early stage the client will have an increased understanding of what the end result will be.
This improved insight will also support better decision making early on, which helps the project progress smoothly.

The resulting model is an excellent communication tool for the next phases – either in-house or for external contractors. In addition to re-use of data and visual checking, the BIM model can be used for early-stage collision detection when data from different applications is combined.
The rich data model is also an excellent base for improved cost estimation for pricing, bidding or planning.

Model-based communication means improved collaboration and results in less friction and mistakes. Because some types of errors are avoided and some are detected sooner, fewer changes and revisions will be needed. The same 3D BIM model can be accessed on the field – by tablet or laptop – eliminating all doubts should any issues arise.

All of these result in better time and cost estimations, faster project completion, faster delivery and better business relationships.

Avoid vendor lock-in with IFC

A major benefit not often talked about is avoiding of CAD vendor lock-in. Most vendors use proprietary file formats, limiting data exchange between your preferred applications, and ensuring

you would need to keep licenses of their software just so you could still access your own data.

If you design your BIM strategy around the widely used and open-source file format IFC then you escape this kind of vendor lock-in. Not only can you choose which application is best for each aspect of your project, you are free to change your mind in the future, switch to a different application and still maintain access to the BIM models of your older projects.

This freedom of choice is especially important when outsourcing parts of a project. No longer is a specific application a job requirement – BIM is the requirement and that greatly expands the number of parties you can work with.

What BIM means for steel detailers

These days BIM is increasingly becoming a requirement for many jobs. Not only are companies realizing the benefits but some governments have mandated the use of BIM in public projects in order to speed up adoption. It is advisable to plan accordingly in order to remain competitive in the future. Getting on board sooner will help you get more jobs and finish them more efficiently.

Detailers that are still working in 2D or using a generalized 3D modeling application (without BIM information) will have a steeper learning curve because switching to a 3D steel detailing solution is a requirement, but they also have the most to gain. A specialized steel detailing application can save a lot of time and money over manual methods.

You certainly should not fear 3D. While some 3D applications are certainly very complex to use, Parabuild was designed to be intuitive and it may be a lot easier to pick up then you think. A lot of your repetitive work will become automated, allowing you more time to focus on the important things.

For detailers that are already using specialized 3D steel detailing software, changing to a BIM mindset will not be difficult. Most modern 3D steel detailing software already have all the required BIM data in a proprietary file format, and should support IFC import/export so the detailer's work can integrate with the rest of the data. There are also free tools available to help you combine, visualize, check and manage BIM models.

You will be able to use BIM data from Revit Structure and most structural analysis software, and once detailed the model will serve for collision detection with HVAC, piping, mechanical objects among other things.

On a side note; a format other then IFC used to be more popular in the steel industry – CIS/2- but that format was deemed too specific to structural steel and did not support enough of the other aspects of construction. Currently it looks likely that IFC will remain the dominant BIM file format for a very long time.

The LOD of steel detailing

When you're getting involved with BIM you will hear about LOD: Level Of Development (or a while ago Level Of Detail – which was deemed not adequately generalized). LOD describes the amount of information in a BIM model. It goes from LOD100 (conceptual data) to LOD500.

Steel detailers work at nearly the final level of development; LOD 400. That means the structural steel model is fully specified down to the nuts and bolts, and it is ready for fabrication and construction. This is just a matter of terminology – detailers have been documenting at this level for a long time. Now this work is defined and stored as LOD400 within the BIM model. The next level LOD500 is for use during and post-construction, it specifies data on the state of the project as it was built.

BIM in Parabuild

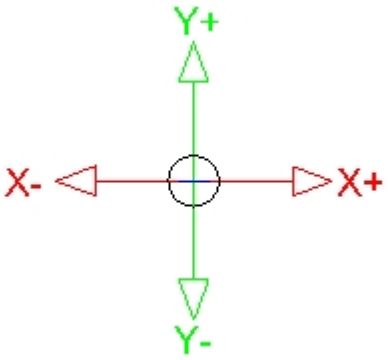
Parabuild has been an information-rich application right from the start, long before BIM/IFC was around. By working with Parabuild, you are automatically constructing your BIM model! We are proponents of keeping your data freely accessible and easily shared, so we support the best BIM file format IFC for both import and export. Try the free Parabuild trial to experience it first-hand!

The Coordinate System

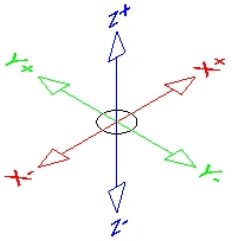
Every point in an CAD drawing file can be identified by its **X**,**Y**,**Z** coordinates. When viewed in plan, the **X** axis will pass from left to right, with movement to the right given as a positive (+) value, and to the left, a negative (-) value.

The **Y** axis will pass from top to bottom, with movement up given as a positive (+) value, and down as a negative (-) value.

The **Z** coordinate value will be 0.0 This system of coordinates is referred to as the **World Coordinate System**, or **WCS.**



2D Coordinate system



3D Coordinate system

Coordinate input

The most direct way to enter points precisely is to type numbers with the keyboard. CAD uses these keyboard coordinate entry formats:

- **Absolute Cartesian:** e.g. 20,40,60

  Values:

  X = 20

  Y = 40

  Z = 60

- **Relative X,Y,Z coordinates** e.g. @37,-25,50 (The **@** character defines coordinates relative to the last point (x = 20, Y = 40, Z = 60))

  Values:

  X = 37

  Y = -25

Z = 50

- **Relative polar coordinates** e.g. @25<45 = (@distance<angle)

  Values:

  @25 = distance from the last point

  <45 = Angle

User Coordinate Systems

You can define an additional coordinate system to more easily create drawings. This Non-World coordinate system, known as a **User Coordinate System** or **UCS**.

Why would you want to diverge from the standard WCS? Well, the most common reason is that its much easier to calculate and enter coordinates if they're based on the plane you want to work on in 3D.

Suppose that you're modeling an old-fashioned, wedge-shaped rubber doorstop and you want to add the manufacturer's logo to the sloping surface of the wedge. It isn't easy if you stay in the WCS, but AutoCAD lets you set a new UCS based on that sloping surface. After the UCS is made current, you draw in it just as you draw in the WCS.

Although originally intended for 3D work, a UCS can be useful on either two dimensions or three. The WCS assumes that the north direction is straight up, but you may be working on a building layout where one wing is at a 37.8 degree angle to the other.

No problem: Simply create a UCS thats aligned appropriately. You can look up this process in the on-line help system, but heres a quick hint: Click the UCS icon in the lower-left corner of the screen to make “grips” appear at the origin and the ends of the axis indicators. Then drag the icon by its grips to set a new UCS.

DRAW BY NUMBERS

You're CAD system will locate *absolute* X,Y coordinates with respect to the 0,0 point of the drawing — usually, its lower-left corner. AutoCAD or BricsCad locates *relative* X,Y coordinates and *relative* polar coordinates with respect to the previous point you picked or typed.

You will see how CAD uses all three coordinate formats to draw a pair of line segments that start at the absolute coordinates 2,1, and then move to the right 2 units and up 1 unit (@2,1) relative to the first point, and then (relative to that point) move 2 units at an angle of 60 degrees (@2<60).

Note in particular how the first two coordinate pairs use the same numbers (2,1) but the second pair defines a different point because of the leading @ symbol.

You can find out the X,Y location of the cross-hairs by moving them around in the drawing area and reading the coordinate values at the left end of the status bar. The X,Y coordinates should change as you move the cross-hairs. If the coordinates don't change, click the drawing coordinates area until you see Coordinates on in the command line.

If you're using the full version of AutoCAD, you may have noticed that *three* numbers are at the left end of the status bar. AutoCAD is showing you the X,Y coordinates of the cross-hairs and the current elevation. However, in 2D drafting, the Z value is 0, so you can continue calling them X,Y coordinates.

Although it isn't apparent at first, AutoCAD has, in fact, four coordinate display modes. Clicking the coordinates readout cycles through these modes:

- **Off -** (<Coords off>): The status bar coordinate readout is dimmed, and the coordinate values don't update until you pick a point.

- **On - showing X,Y coordinates** (<Coords on>): The coordinate readout appears black, and the absolute X,Y coordinates update continuously as you move the cross-hairs. If no command is active, clicking the coordinates readout alternates between this mode and <Coords off>.

- **On - showing polar coordinates (**<Coords on>**):** This mode, which displays distance and angle relative to the last point picked rather than absolute X,Y values, appears if a command is active and AutoCAD is waiting for you to pick a point.

- **On** - **showing geographic coordinates (**<Coords on>**):** This mode displays coordinates as latitude and longitude values, but it can be used only after you set the drawing's geographic location with the GeographicLocation command.

If you start a command such as Line, pick a point, and then click the Coordinates area a few times, the display changes from coordinates Off to live absolute coordinates to live polar coordinates. Displaying live polar coordinates is the most informative mode most of the time.

If you're working in AutoCAD's architectural or engineering units, the default unit of entry is *inches,* not feet. Here are some guidelines for entering numeric values when you work with feet and inches:

- **To specify feet,** you must enter the apostrophe (') symbol for feet after the number:

  6' is 6 feet.

- **To separate feet from inches,** enter a dash:

  6'-6” is 6 feet, 6 inches.

- **When you enter coordinates and distances,** both the dash and the inch mark are optional:

  6'6” and **6'6** are the same as **6'-6”**.

- **To type a coordinate or distance that contains fractional inches,** you *must* enter a dash — not a space — between the whole number of inches and the fraction:

  6'6-1/2 (or **6'-6-1/2**) represents 6 feet, 6-1/2 inches.

- **To enter partial inches**, use decimals instead:

  6'6.5 is the same as **6'6-1/2”** to AutoCAD, whether you're working in architectural or engineering units.

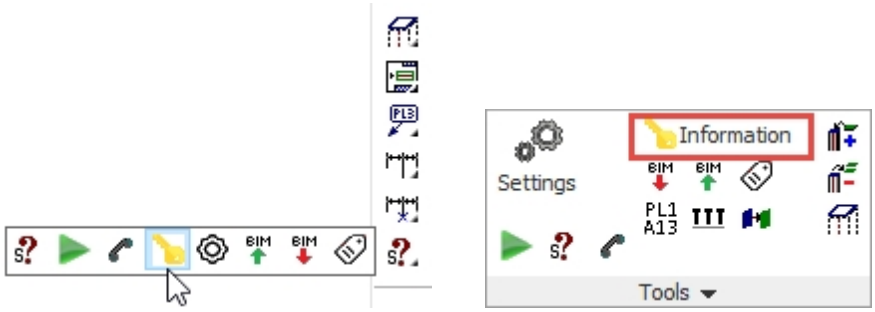
Reference Guide

This section covers all the commands listed in the 3 main toolbars and the ribbon, together with the Parabuild Settings and the Properties panel.

- Parabuild Settings
- The Properties Panel
- Main - Library
- Main - 3D Modeling
- Main - Output

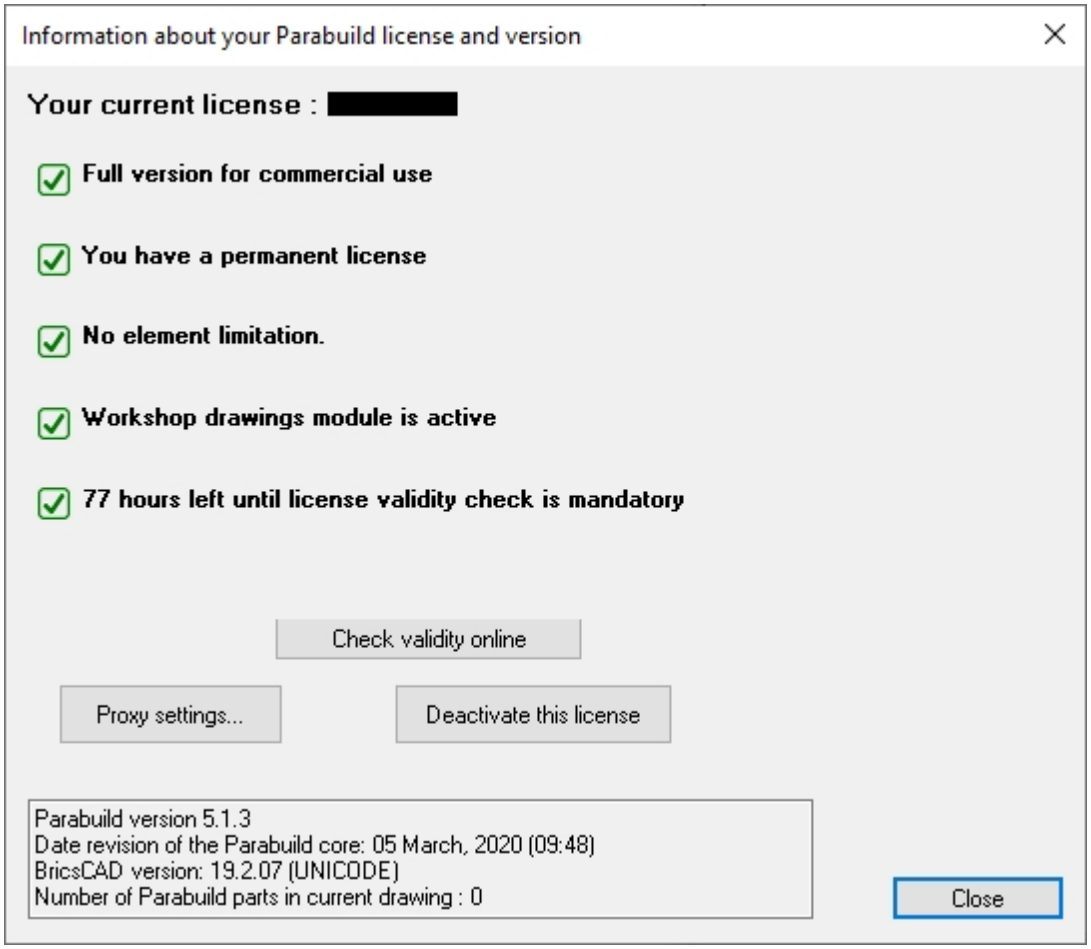
License management

Command - **PrB_Info**



This dialog box allows you to see information about the version and license of Parabuild.

You can also activate and deactivate the license here.



The items in the dialog are explored below :

Activate a license

When we activate a license from this dialog box, Parabuild will request a license activation from the internet server.

The server will only grant an activation if the license is not already activated on another computer.

Deactivate this license

By deactivating the license, you will free up the license on the server again.

This makes the license available again for activation on another computer.

Deactivating frequently

For managing licenses we use a third party service. Deactivation and Activation of the license is taxing on the servers. For this reason, we are charged per activation by the third party.

To avoid excessive costs, you are allowed to activate Parabuild for a maximum of 50 times per year if you have a maintenance contract for the license, or 10 activations per year if you do not have active maintenance for the license.

We will monitor the activation usage periodically to find the licenses that have surpassed this limitation.

If this happens we will then contact the customer to find a solution, which may include billing for excess activations.

Activation fails and you can't use Parabuild anymore

This can happen when a system crash has occurred on a system that had a Parabuild activation.

To fix this, we can reset the activations of your license on the server.

You can ask your reseller to help you with this problem.

For a quicker response, you can create a support ticket by sending an email to : support@parabuild.com

You can also contact customer relations to help you with this problem. You can email them at : contact@parabuild.com

Do not forget to mention your license ID and company name.

If you need **very urgent assistance** with this problem, you can call Parabuild directly at :

+32 3 216 49 66

This phone number will be answered during business hours in time zone GMT+1/CET.

Proxy settings

This is only needed in very rare occasions.

If your network requires you to use a proxy server, then enter the proxy information here.

Parabuild will then go through this proxy server to contact the Parabuild server.

Check validity online

Parabuild will periodically connect to a license server to verify the status of your license.

This check happens in the background, unless the server could not be reached for several days.

If this happens, the user will be notified and still has 3 days to connect to the internet.

If you expect to be working offline for a few days, you can prepare by pressing the **Check validity online** button.

If it is not yet visible then it means the license was recently verified and there is no need for a new check.

Element count

At the bottom of the dialog, we can see this line :

Number of Parabuild parts in current drawing

This shows the total amount of 3D parts as it would be counted for the licensing.

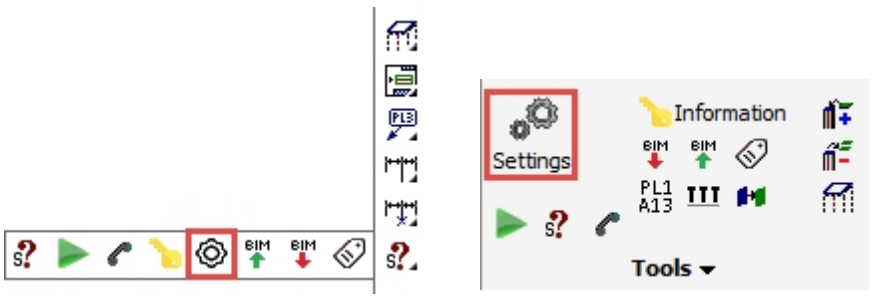
This number can be used to see how close the current drawing is to hitting the maximum amount of parts in the current drawing.
Only the LT versions of Parabuild have a limit on the number of parts in the drawing.

Only plates, profiles, structures and volumes are counted as 3D parts. Bolts are counted as well but the total number of bolts is divided by 4.

Once the maximum has been reached, the commands for drawing new 3D parts will stop working. Parabuild will start to function as a viewer from this point on, or until enough 3D parts have been removed from the drawing.

Parabuild Settings

Command: **PrB_Settings**



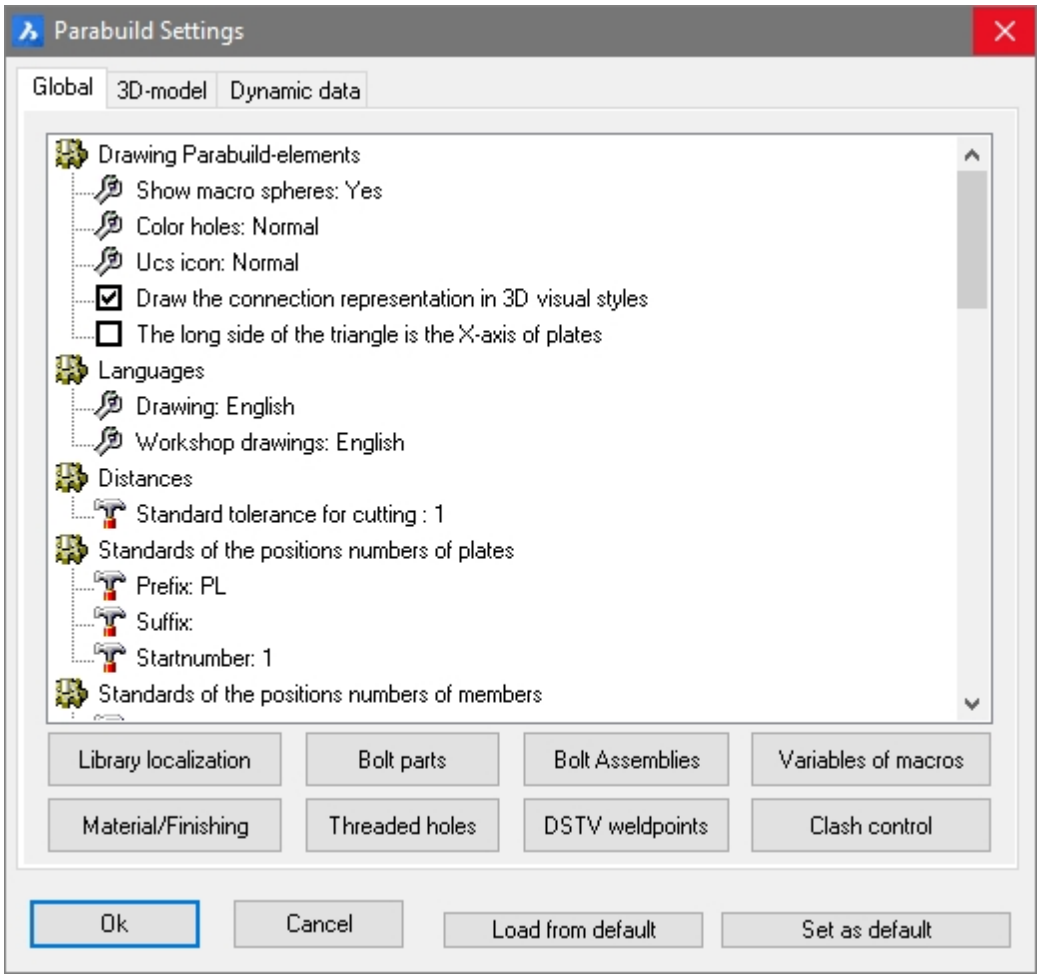
The Parabuild settings window has 3 main tabs:

Global - Where you can preset global standards and settings - these settings will override all AutoCAD and BricsCAD settings

The bottom buttons allow you to edit the various libraries and filters, and clash control settings

3D Model - These settings will determine how the model is presented and viewed

Dynamic data - This allows additional properties to be created to supplement the properties already offered by Parabuild

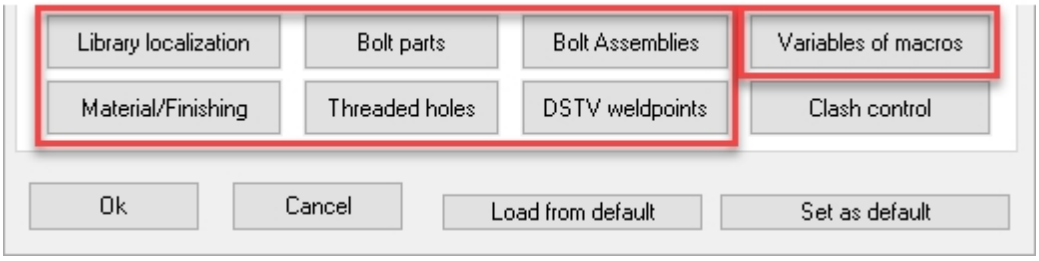


Where are the values of these settings stored?

Most of the setting values on the tabs **Global** and **3D-model** are stored in the 3D drawing (.dwg file).
So if you make another 3D drawing active, then you might see different values in the tabs **Global** and **3D-model**.

Where are the library settings stored?

All the settings behind the below 7 buttons are stored in the Parabuild library (fe c: \Parabuild\Pb_Lib)



The library folder can be switched to a different location with the help of the Parabuild **Startup settings** tool.

The purpose of this would be to let several Parabuild users share the same library within the same company.

If one of the users adds a new profile or a new material, then all the users will automatically get access to the new profile and material.

The procedure to do this is to first copy the **Pb_Lib** folder of one of the users to the shared location and then use the **Startup settings** tool on all of the Parabuild installations to switch the library to the shared location.

A condition for doing this set-up is that all the Parabuild installations should work with the same version of Parabuild.

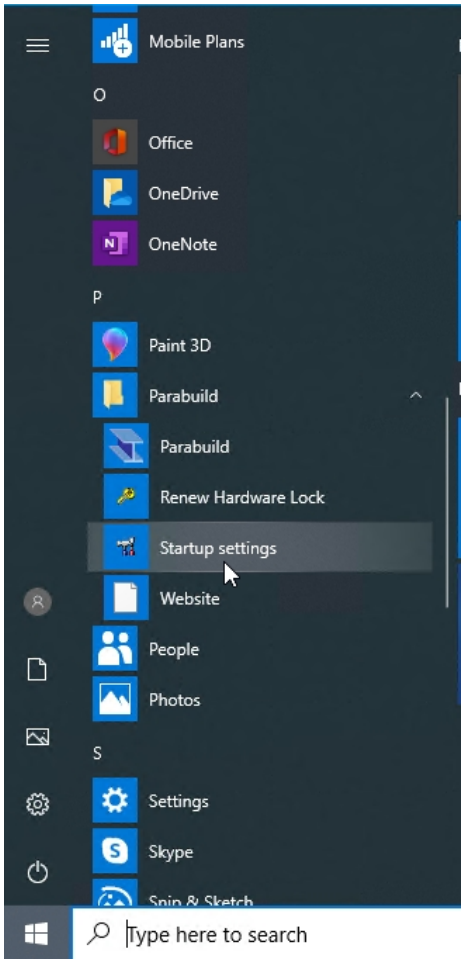
We recommend to user one of the cloud drive services as the location for the shared Parabuild library folder (DropBox, Microsoft OneDrive or Google Drive).

These cloud drives will store a local copy of the library, which ensures maximum speed while using Parabuild commands.

But still if the user is disconnect from the internet such as with a laptop, he/she will still have access to the local Parabuild library to continue working with Parabuild.

The cloud drive's synchronization tool will update the local library with changes from the cloud and will upload your changes to the cloud drive in the background when there is an internet connection.

The **Startup settings** tool can be found in the Windows start widget :



How to change these values for existing parts?

This dialog box will not modify the settings of existing parts in the drawing.

The settings in this dialog box only applies to new parts that will be drawn.
If you want to change the settings of existing parts, then use the Properties panel to change the settings of a single part or several parts at the same time.

How to store these settings?

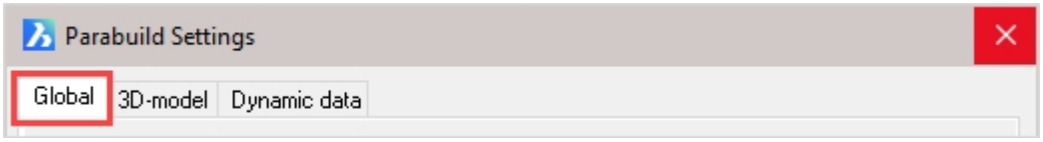
You can use **the Load/Set from default** buttons at the bottom of the dialog box to store or retrieve the setting values in the dialog box.

This refers only to the settings on the **Global** and **3D-model** tabs.

- **Set as default** will store all of the settings in a file in your Parabuild installation folder (\Parabuild\defset.dat).
  All new 3D drawings that you create starting from a template drawing will assume these defaults.

- **Load from default** will read the defaults that have been stored in \Parabuild\defset.dat and will apply all the values to the current 3D drawing.

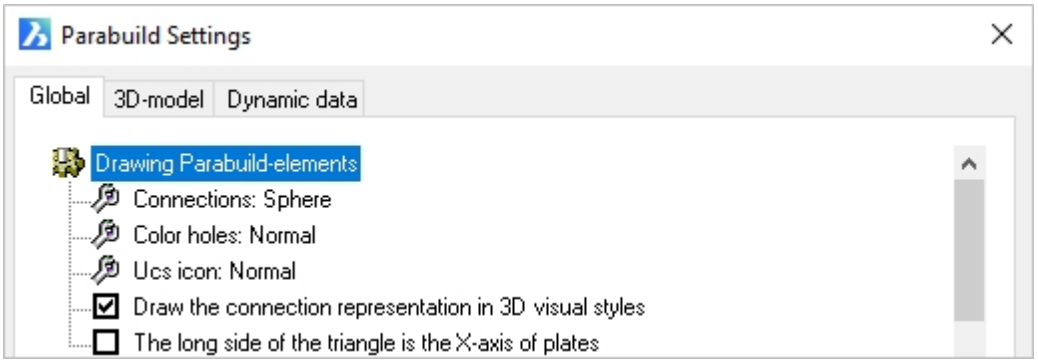
Global



The global settings are divided into the following categories:

- Drawing parabuild elements
- Languages
- Standard tolerances for cutting
- Distances
- Standards of the position numbers of plates
- Standards of the position numbers of members
- Standards of the position numbers of structures
- Standards of assembly numbers
- Global settings
- Standard values for newly drawn members and plates
- General plate settings

Drawing Parabuild Elements



Connections: Refers to the macro indicator - Options = Not / Sphere / Triangle

Color Holes: Hole color - Options = Normal / Red

UCS icon: This refers to the coordinates system of plates and profiles. Options = Normal, which draws a triangle at the origin / Detail, which draws a UCS icon at the origin.

Draw the connection representation in 3D visual styles: A switch, which when checked will show the macro spheres when one of the 3D visual styles are active. When unchecked all spheres will be hidden.

The long side of the triangle is the X axis of the plates: The origin triangle direction may be switched from the Y axis (checkbox UnChecked) to the X axis (Checked). This option will only come into effect when the UCS icon is set to Normal

Languages

The language option for **Drawing** refers to the language of the user interface in Parabuild. That means all the dialog boxes, command line prompts and toolbars.

The language option for **Workshop drawings** refers to the language of the automatically generated workshop drawings only. More specifically the title block and Bill of Material on these drawings.

The available languages include:

- English
- Dutch
- French
- German
- Korean
- Traditional Chinese

Distances

Standard tolerance for cutting: Is the dimensional tolerance for cutting lengths. Commands for adding cuts to parts will follow this default value.
Default value = 1 mm.

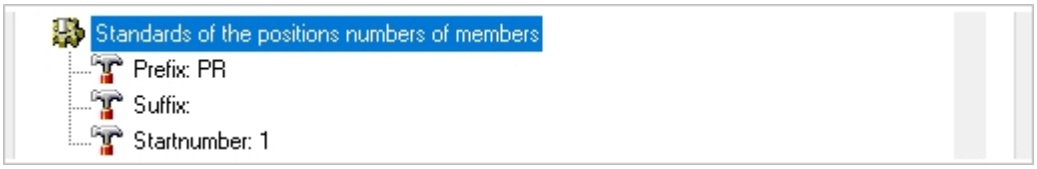
Standards for the position numbers of plates

Prefix: Is the prefix given to plate numbers i.e. **PL** 1 - this may be edited to suit

Suffix: Is the suffix given to plate numbers i.e. PL 1 **01** **-** this is optional

Start number: Is the first number given to the plate i.e. PL**500**. This number will be automatically incremented if the given start number is already occupied.

Standards for the position numbers of members

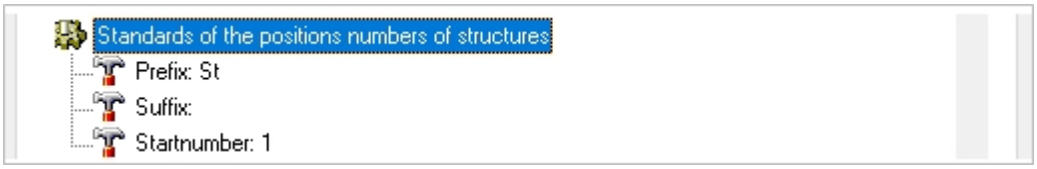


Prefix: Is the prefix given to member numbers i.e. **PR** 1 - this may be edited to suit

Suffix: Is the suffix given to member numbers i.e. PR 1 **01** **-** this is optional

Start number: Is the first number given to the member i.e. PR**500**. This number will be automatically incremented if the given start number is already occupied.

Standards for the position numbers of structures



Prefix: Is the prefix given to structure numbers i.e. **ST** 1 - this may be edited to suit

Suffix: Is the suffix given to structure numbers i.e. ST 1 **01** **-** this is optional

Start number: Is the first number given to the structure i.e. ST**500**. This number will be automatically incremented if the given start number is already occupied.

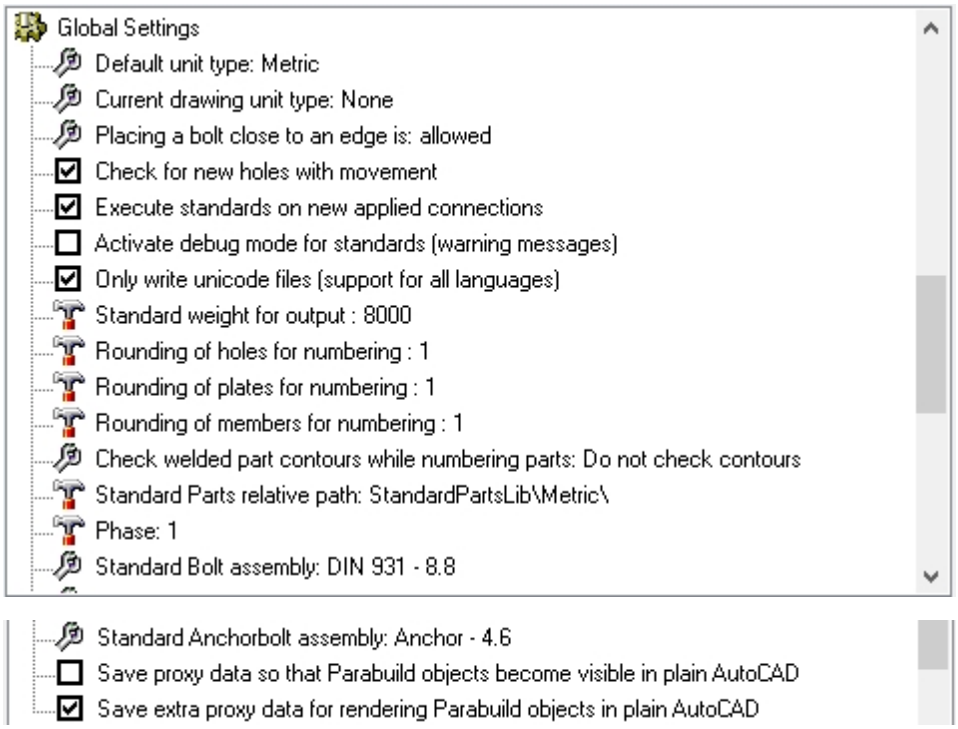
Standards of the assembly numbers

Prefix: Is the prefix given to assembly numbers i.e. **M** 1 - this may be edited to suit

Suffix: Is the suffix given to assembly numbers i.e. M 1 **01** **-** this is optional

Start number: Is the first number given to the assembly i.e. M**500**. This number will be automatically incremented if the given start number is already occupied.

Global Settings



Default unit type: Use **None** if you want to work in both Metric and Imperial projects. Choosing **Metric** will cause Parabuild to assume Metric in drawings with conflicting units variables. The same applies to the **Inches** option. Options = None / Metric / Inches

Current drawing unit type: Change this option only when Parabuild has assumed the wrong units for the current 3D drawing. Options = None / Metric / Inches

Placing bolt close to an edge: Options = Not allowed / Allowed
(The minimum distance between a bolt and the edge of a beam or plate can be set in the Clash Control dialog)

Check for new holes with movement: Options = Off / On
(When a bolt is moved, the corresponding holes are moved with it. You can switch this off)

Execute standards on new applied connections: Options = On / Off (Refer to Standards for Connections)

Activate debug mode for standards (Warning messages): Options - On / Off

Only write Unicode files (Support for all languages): Options - On / Off

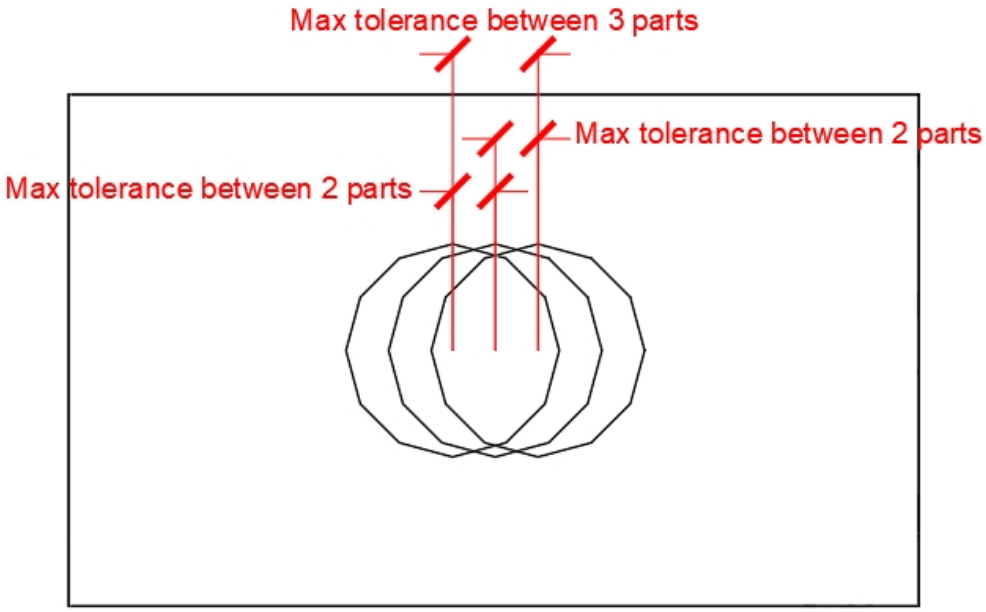
Standard weight for output: This value is the default weight that Parabuild should assume for objects that have no material assigned. The default value is set at 8000 kg/m³ - which is the approximate value for 1m3 of steel (7860 kg/m³).

Rounding of holes for numbering: This is the tolerance that Parabuild should use when searching for identical parts and assigning numbers to them.

In practice, Parabuild will assume half of this tolerance when comparing 2 parts. This means that the total maximum difference between 3 parts will be the full tolerance.

The below illustration shows the difference between the tolerance of 2 parts and the total tolerance of 3 parts or more.

It shows 3 plates on top of each other, but with slightly offset hole positions.



Rounding of plates for numbering: Same as above, but this refers to the tolerance of plate edges for number assignment.

Rounding of members for numbering: Same as above, but this refers to the tolerance of member edges and cuts for number assignment.

Standard parts relative path: This value should point to a folder inside the Parabuild library (the full path is by default c:\Parabuild\Pb_Lib\StandardPartsLib\Metric\).

This tool is meant to create library drawings that contain pre-made “off the shelf” parts.

You will assign recognizable part numbers to the "off the shelf" parts so that the persons in the shop know which part to take. Typically there is no shop drawing needed for these parts.

This is how it works :

During the numbering stage, Parabuild will open all the 3D drawing files in this folder, and it stores all of the plates, profiles and structures in those files into memory.

Then when it encounters a part in your current drawing with the same geometry as in the standard parts, then the part number will be inherited from the part that was stored in the standard parts file.

Also a limited number of properties such as Skip shop drawing generation of will be inherited.

Phase: Sets the phase that all newly drawn objects should receive.

Standard bolt assembly: The default bolt assembly that connections and bolt tools should use by default. (See Bolt Assemblies)

Standard anchor bolt assembly: The default bolt assembly that anchor plate macros should use by default. (See Bolt Assemblies)

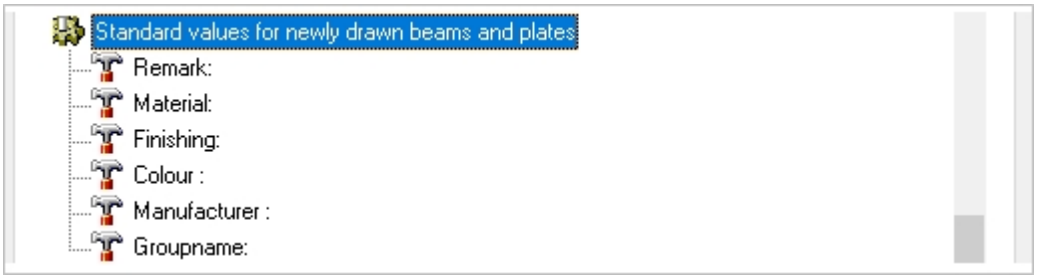
Save proxy data so that Parabuild objects become visible in plain AutoCAD/BricsCAD: This option allows the 'Proxy' details of a 3D-drawing to be saved. Parabuild creates its own objects (profiles, plates, bolts ...) which means that without Parabuild, AutoCAD will not recognize these objects and therefore 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 2 to 3 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 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***.

Save extra proxy data for rendering Parabuild objects in plain AutoCAD/BricsCAD: this is a sub-set of the previous option. This will add additional proxy data for rendering of faces with 3D objects. Note that this option will not work independently as the edges are still needed to determine the faces. Therefore, both check boxes must be active if you want the rendering data to be written to file.

Standard values for newly drawn beams and plates



Remark: Here you may enter a standard Remark for newly drawn parts

Material: Here you may enter a standard material for newly drawn parts

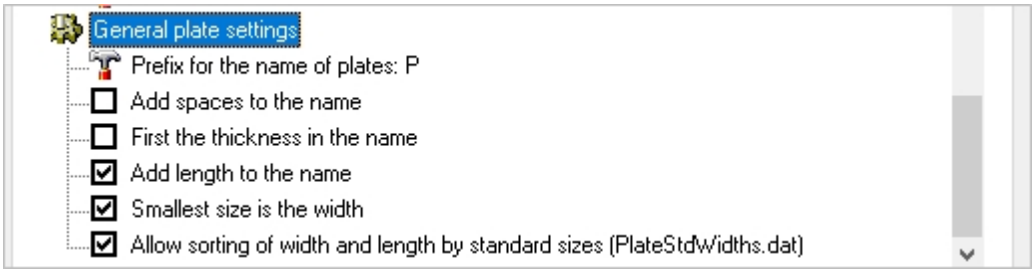
Finishing: Here you may enter a standard Finishing for newly drawn parts

Color: Here you may enter a standard Color for newly drawn parts

Manufacturer: Here you may enter a nominated Manufacturer for newly drawn parts

Group name: This property is not actively used by Parabuild, so it can be considered a free to be used "User property".

General plate settings



Prefix for the name of plates: The default value is PL - this may be changed to suit preference (i.e. PL8x100)

Add spaces to the name: Will add spaces between the characters (i.e. PL 8 x 100)

First the thickness in the name: Will place the plate thickness before the plate size (PL8x100 vs PL100x8)

Add length to the name: Will add the plate length to the plate name (i.e. PL8x100x190)

Smallest size is the width: Will assume the smallest size is the plate width (i.e. PL8x135x200 vs

PL8x200x135)

Allow sorting of width and length by standard sizes (PlateStdWidths.dat): If enabled, Parabuild will use this file to determine what the standard plate widths are : c:\Parabuild\Pb_Lib\PlateStdWidths.dat.

The buttons at the bottom of the dialog are referenced here:

Library Localization

Bolt Parts database

Bolt Assemblies

Variables of Macros

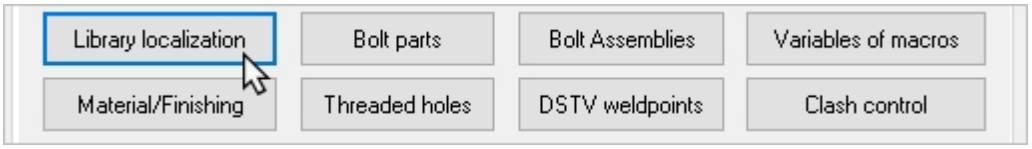
Material / Finishing

Threaded Holes

DSTV WeldPoints

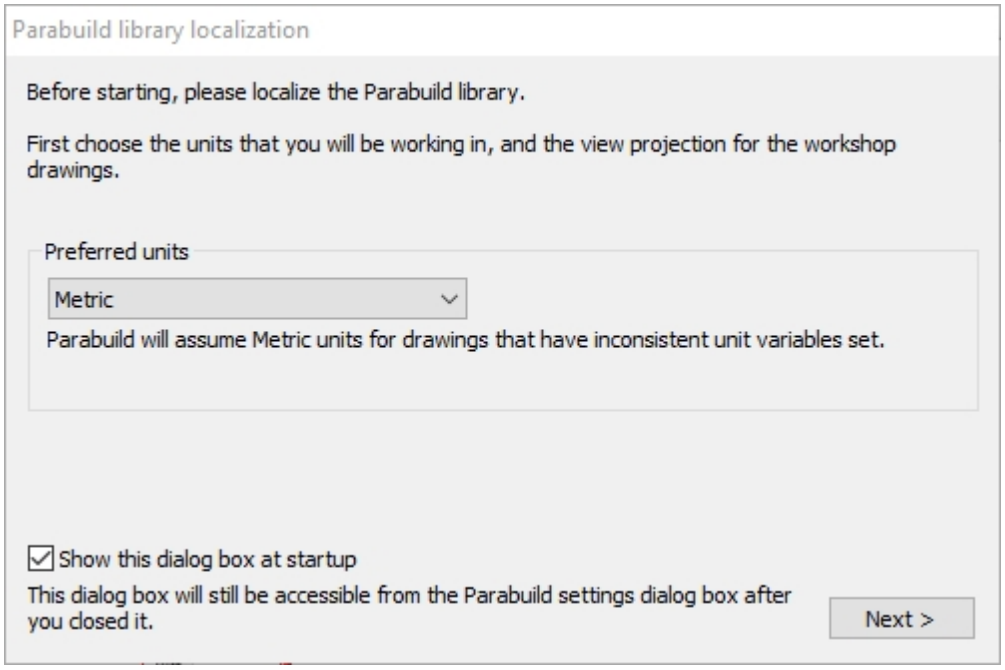
Clash Control

Library Localization



This is the same dialog box that is shown when starting Parabuild for the first time.

First you will be prompted to select the preferred drawing units:



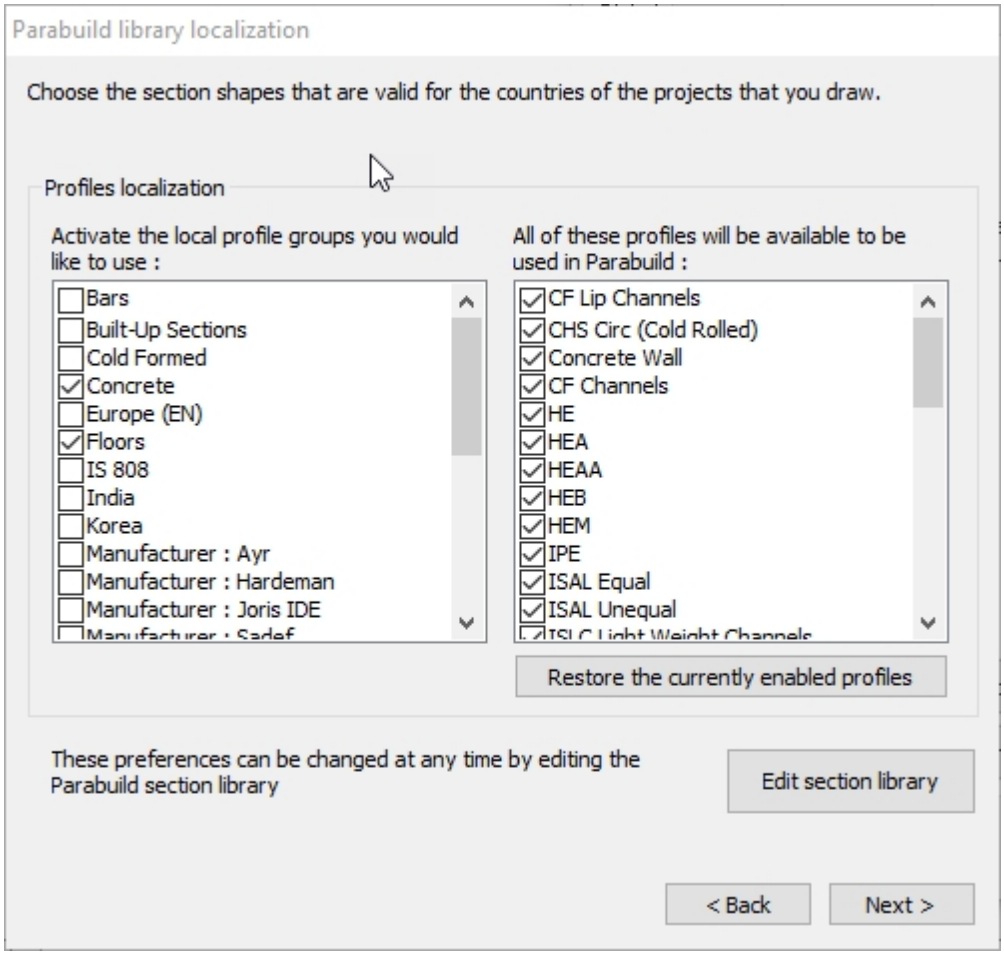
The options include: **None** / **Metric** / **Inches**

Choosing **None** would suggest that the user may sometimes work on metric drawings, and at other times on Imperial drawings.

Choosing for **Metric** or **Inches** will often solve issues that can arise when the user has mistakenly started drawing with a template of the wrong units.

Choosing **Metric** or **Inches** will also cause less options to be proposed in the next steps.

Click **Next** to open the Parabuild Profile library localization.

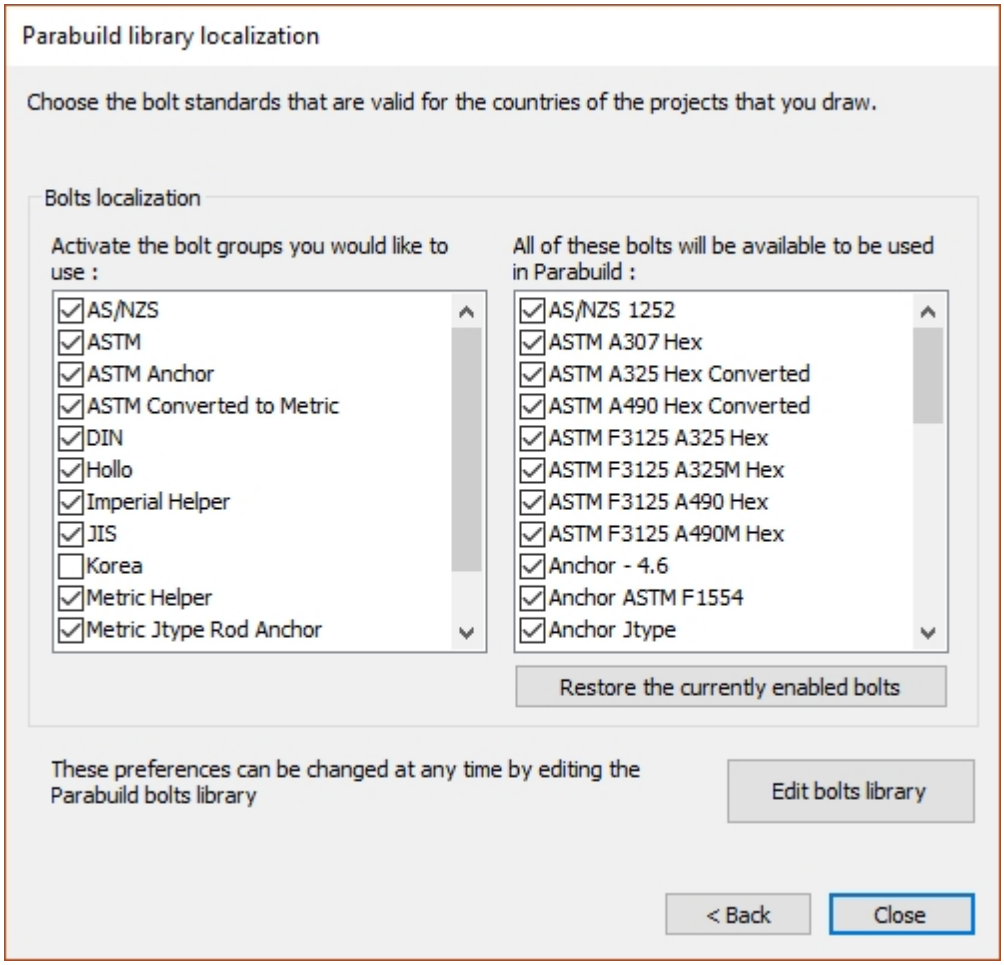


Step 2 of the process allows you to select the desired profiles. The window on the left is essentially a keyword filter allowing you to select profiles and shapes according to their generic group which includes - shapes, country of origin, profiles, and manufacturer specific profiles. Activating the checkbox will display the options contained within that generic group in the right hand window. These preferences may be changed at any time by editing the Parabuild profile library

Restore the currently enabled profiles - will restore the state of the profiles to how it was before opening this dialog box.

Edit the profile library - allows you to edit the profile library

Click **Next** to open the bolt library localization

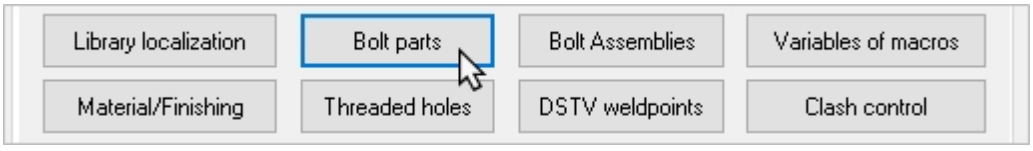


Step 3 of the process allows you to select the desired bolts. As with profile selection above, the window on the left is a keyword filter allowing you to select bolt assemblies according to their generic group which includes - types, standards, country of origin, and manufacturer specific bolts. Activating the checkbox will display the options contained within that generic group in the right hand window. These preferences may be changed at any time by editing the Bolt parts database

Restore the currently enabled bolts - will restore the state of the bolts to how it was before opening this dialog box

Edit the bolts library - allows you to edit the bolts library

Bolt Parts Database



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

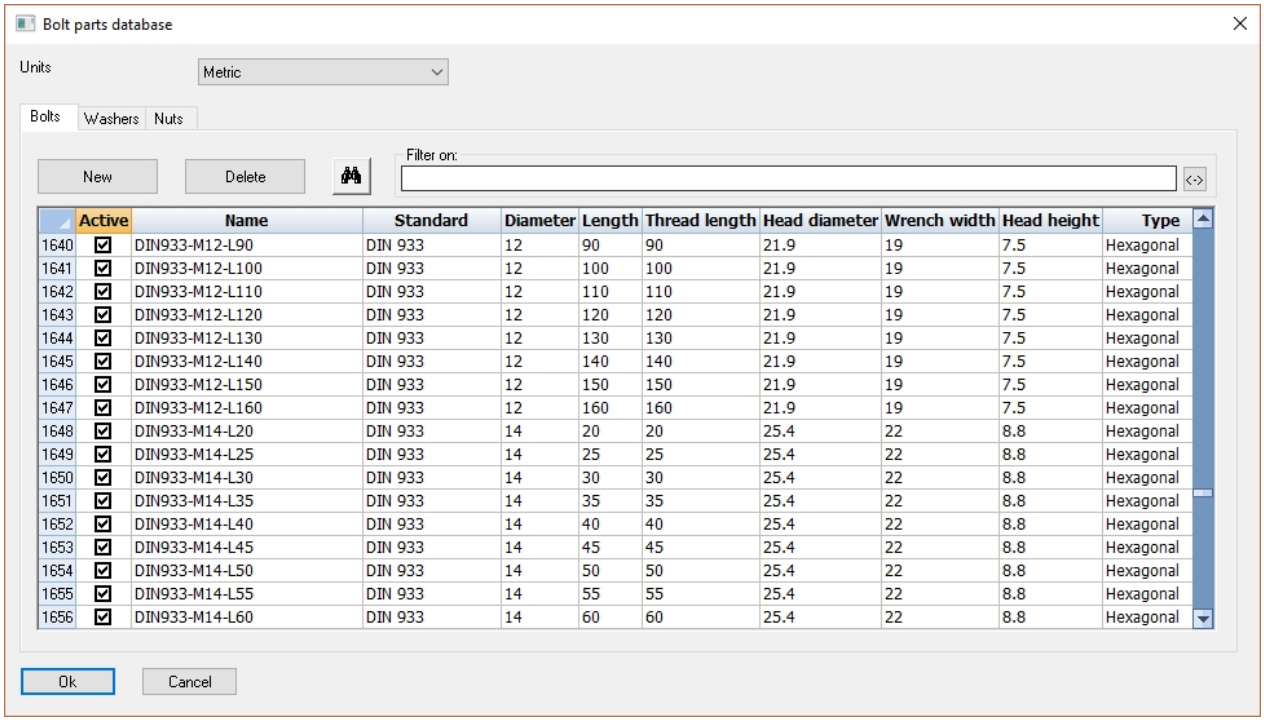
This database contains every bolt with its exact dimensions (diameter, length, thread length, ...).

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

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.

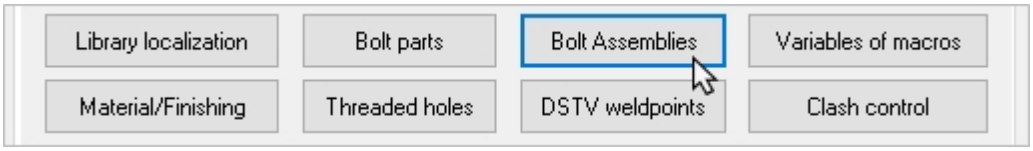
All parts in the available lengths and diameters for the respective standards have already been entered.

Because the number of parts can total more than 1000, a filter can be applied. Type in DIN933 in the box in the top 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.

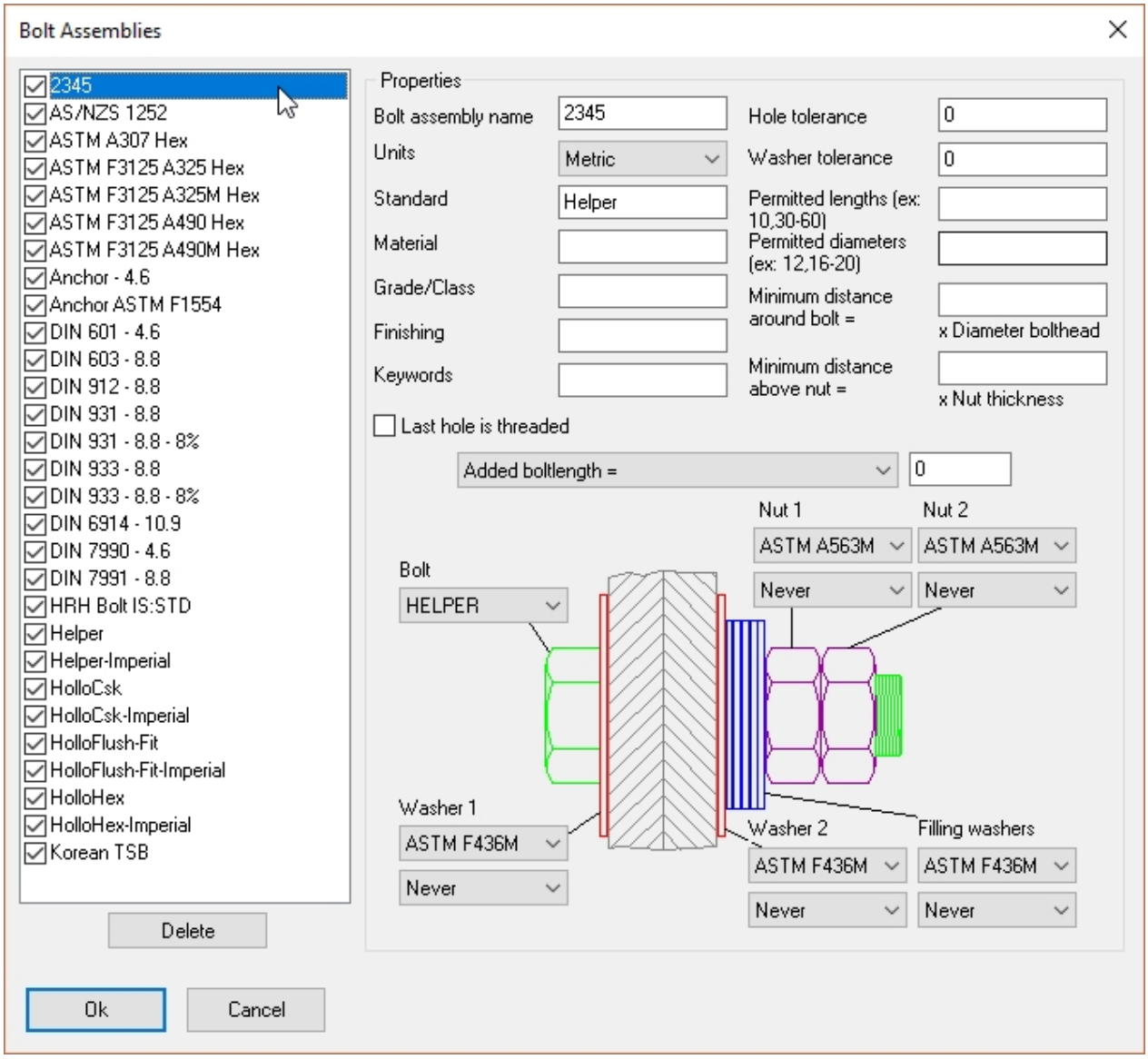


	Active	Name	Standard	Diameter	Length	Thread length	Head diameter	Wrench width	Head height	Type
1640	☑	DIN933-M12-L90	DIN 933	12	90	90	21.9	19	7.5	Hexagonal
1641	☑	DIN933-M12-L100	DIN 933	12	100	100	21.9	19	7.5	Hexagonal
1642	☑	DIN933-M12-L110	DIN 933	12	110	110	21.9	19	7.5	Hexagonal
1643	☑	DIN933-M12-L120	DIN 933	12	120	120	21.9	19	7.5	Hexagonal
1644	☑	DIN933-M12-L130	DIN 933	12	130	130	21.9	19	7.5	Hexagonal
1645	☑	DIN933-M12-L140	DIN 933	12	140	140	21.9	19	7.5	Hexagonal
1646	☑	DIN933-M12-L150	DIN 933	12	150	150	21.9	19	7.5	Hexagonal
1647	☑	DIN933-M12-L160	DIN 933	12	160	160	21.9	19	7.5	Hexagonal
1648	☑	DIN933-M14-L20	DIN 933	14	20	20	25.4	22	8.8	Hexagonal
1649	☑	DIN933-M14-L25	DIN 933	14	25	25	25.4	22	8.8	Hexagonal
1650	☑	DIN933-M14-L30	DIN 933	14	30	30	25.4	22	8.8	Hexagonal
1651	☑	DIN933-M14-L35	DIN 933	14	35	35	25.4	22	8.8	Hexagonal
1652	☑	DIN933-M14-L40	DIN 933	14	40	40	25.4	22	8.8	Hexagonal
1653	☑	DIN933-M14-L45	DIN 933	14	45	45	25.4	22	8.8	Hexagonal
1654	☑	DIN933-M14-L50	DIN 933	14	50	50	25.4	22	8.8	Hexagonal
1655	☑	DIN933-M14-L55	DIN 933	14	55	55	25.4	22	8.8	Hexagonal
1656	☑	DIN933-M14-L60	DIN 933	14	60	60	25.4	22	8.8	Hexagonal

Bolt Assemblies



The assembly of a bolt determines the standards of the bolt, washers and nuts. We can also determine the default tolerance of the holes in the assembly.



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 on the directly. Activating another assembly will save all the changes that you made to the current assembly. If another assembly name is assigned to the assembly then a new assembly with that name will be created.

The standards of the bolts, nuts and washers that are presented in the dropdown boxes are retrieved 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 effectively determine whether a new bolt drawn in 3D will have the nuts/washers enabled/disabled by default. When you choose always or never, you will not be able to disable/enable them after drawing the bolt.

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 after the bolt was drawn.

The settings **Permitted lengths** and **Permitted diameters** are useful quick filters. As the bolt standard contains many bolt diameters and lengths, but in practice most of these bolts are not

used. The diameters and lengths that you enter here should match the bolts that the erectors bring to the site. This way, the Parabuild bolts list will accurately match the bolts that the erectors will need on site.

This is the sequence that Parabuild uses when a new bolt is drawn :

1. Parabuild searches within the selected assembly for the standard for the new bolt, washers and nuts that should be used.
2. Using this standard, Parabuild refers to the bolt parts database and selects a bolt using the following details: Standard, diameter and length (length = penetration length + washers + nuts + added length).
3. Only the bolt parts that fall within the above **Permitted lengths** and **Permitted diameters** filters are used by Parabuild

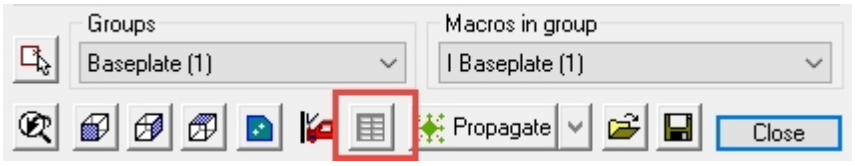
Variables of Macros

In this dialog box we can influence the variables that are used by the Macro standards.

The purpose of the macro standards is to change the macros default values to some sensible values that are dependent on the base profiles of the macro.

The Macro standards is a tool that is run automatically on each macro that is inserted from the library using any of these commands :

It is also possible to run the standards on a macro manually, with this button in any macro dialog box :



You can influence the standards by changing the value of a variable in the below dialog box. 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 the available 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.

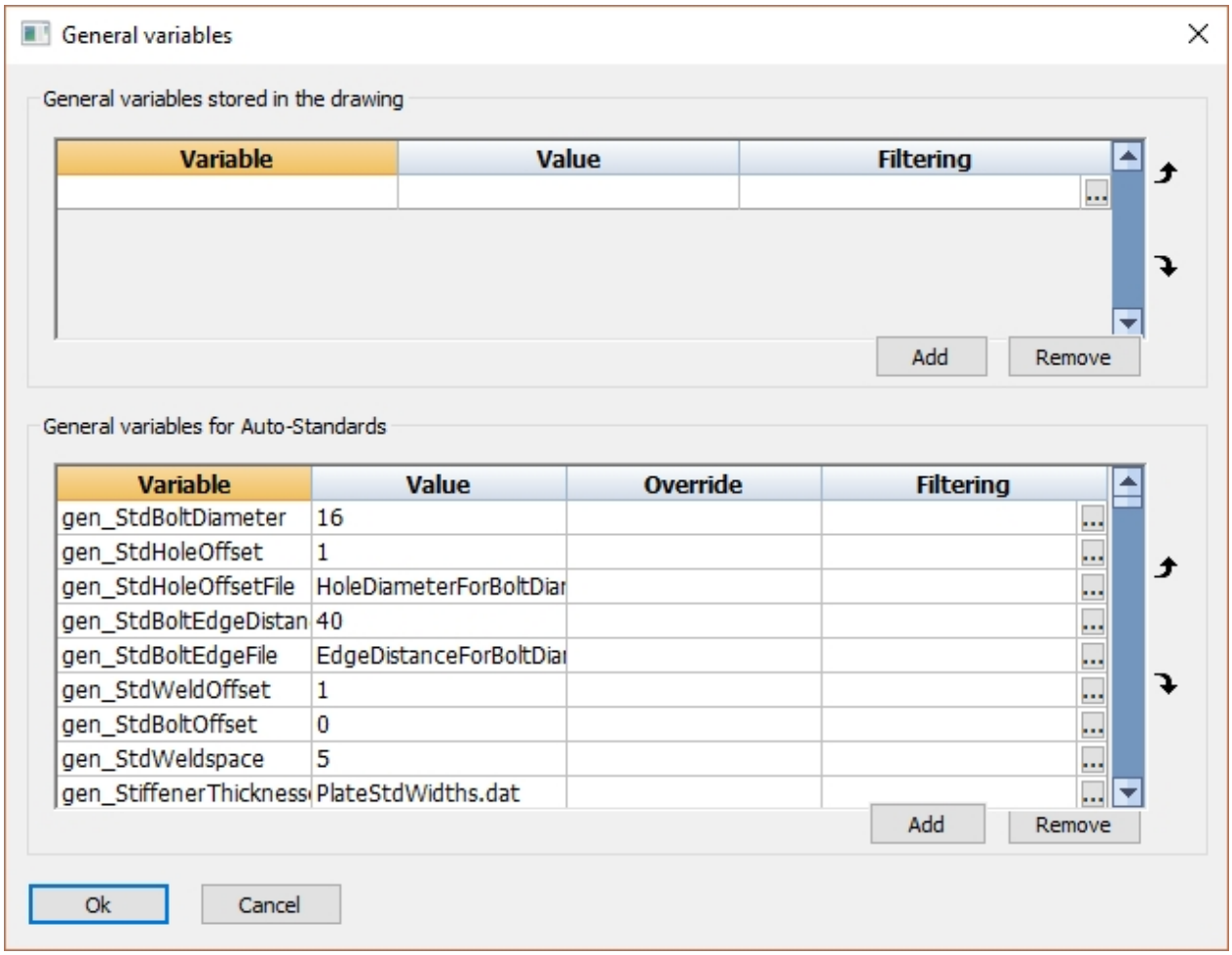
gen_StdHoleOffsetFile : This variable contains a filename as its value. In the file itself you can make changes to influence the hole tolerance. See the next chapter for more information.

gen_StdHoleOffset : The (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.

If you the file "\Pb_Lib\PlateStdWidths.dat", 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

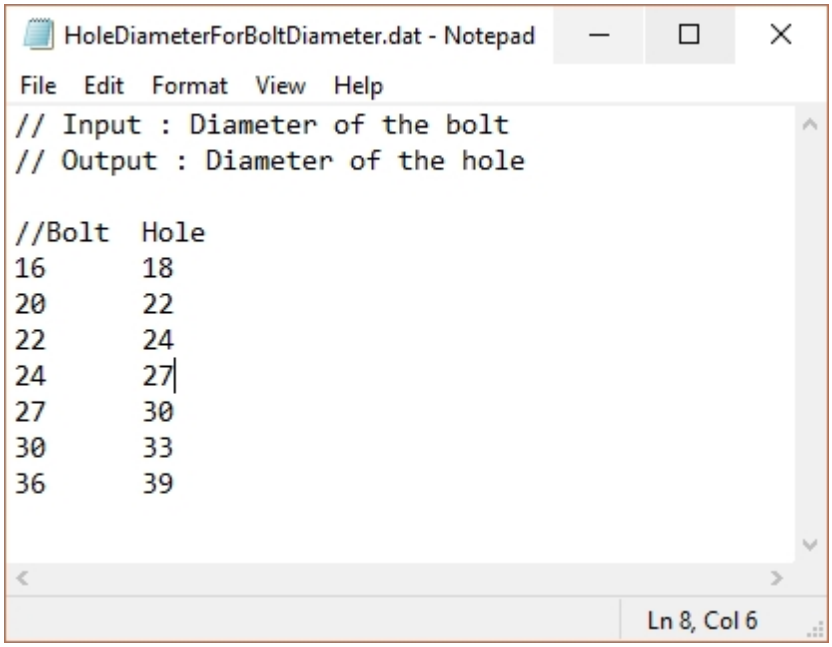
A part of the standards variables refer to a file. You will find these files in the following location on

your computer:

\Parabuilld\Pb_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:



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.

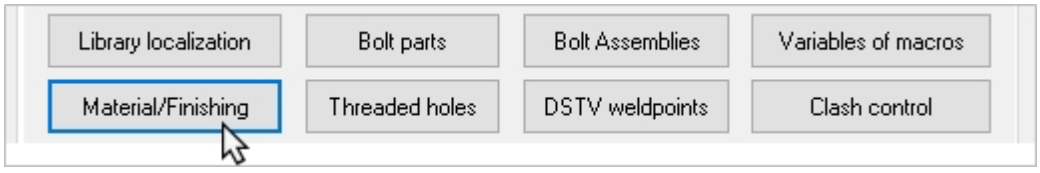
The purpose of *HoleDiameterForBoltDiameter.dat* is to choose a hole diameter for each bolt diameter.

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.

Material / Finishing



The following dialog appears - The configuration options include:

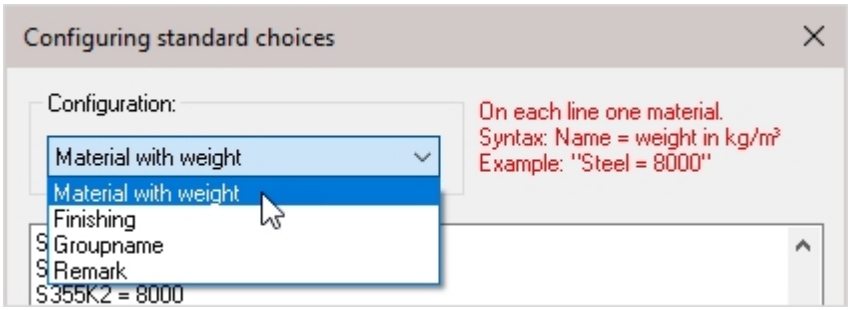
Material with weight - Here you may enter a new material followed by the '=' symbol, followed by the weight.

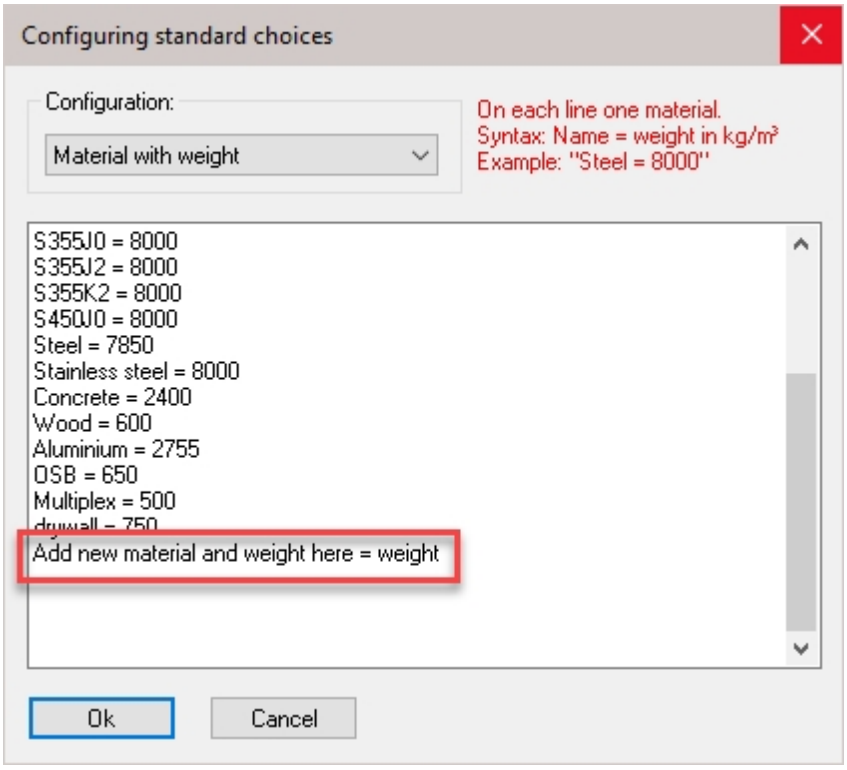
Finishing - Here you may enter a finish, which may be Paint / Galvanizing etc...

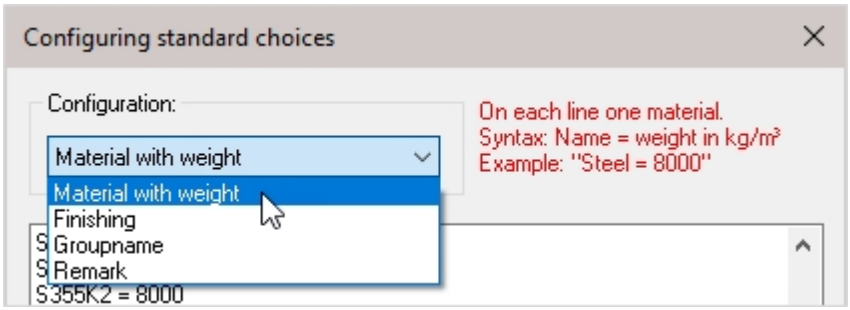
GroupName - Here you may enter the desired group name

Remark - Here you may enter a standard remark

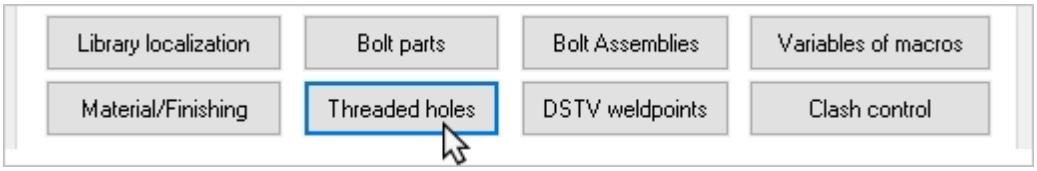
All values entered here will appear as options in the various dialog boxes - so that you don't have to type them in each time.







Threaded Holes



In this dialog box you can determine which drill diameter the shop drawings should use for threaded holes. For illustration purpose threaded holes have some extra lines in 3D similarly to the thread of a real hole.

If you convert a regular hole into a threaded hole, the diameter of that hole will be set to match the diameter of the bolt.

Here you can adjust for each threaded hole diameter the drill diameter that the annotation on the shop drawing should contain.

By default the annotation of a threaded hole will look like this : **Drill ø10.2 Tap M12**

The annotations in workshop drawings that contain the variable %PbColCommentForHoleDriller% will contain this string.

Parabuild uses the first 2 columns to compose this string.

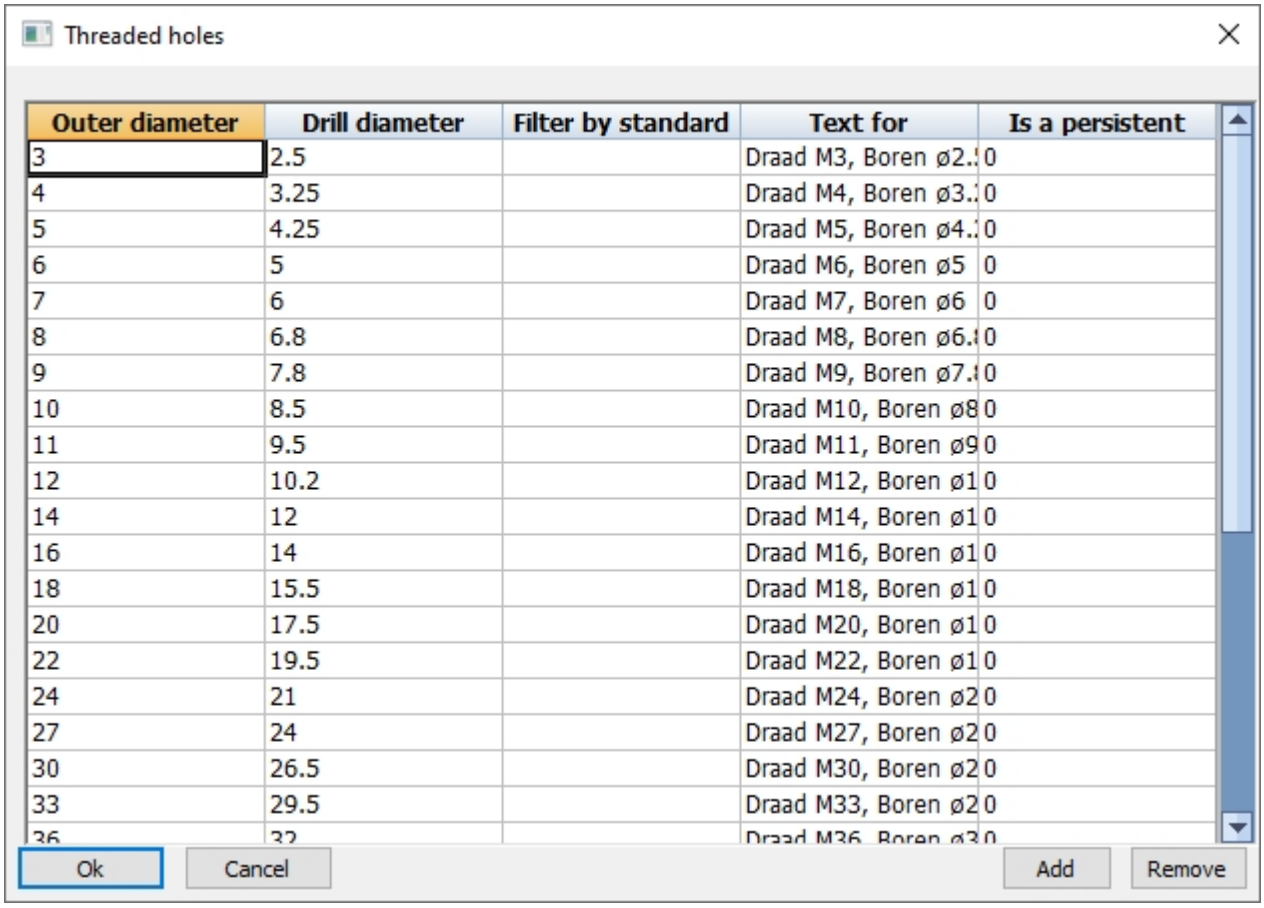
The column **Text for annotations** is a separate string that will be used instead of the above string, but only if the last column **Is a persistent override** is active for the row.

The column **Filter by standard** applies an extra filter for the entire row. Just enter the standard of the bolt here, and the row will only be applied to those bolt standards.

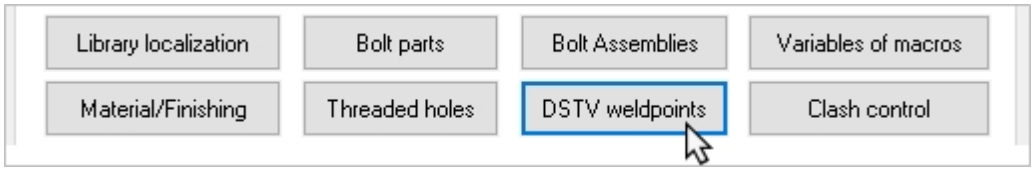
Thanks to this we filter can create multiple rows for the same diameter, but with different drill diameters. This is useful when different tapping tools are used in the same 3D drawing.

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

If there are 2 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.



DSTV WeldPoints



DSTV weld points are points that are placed in the automatically produced DSTV NC-files.

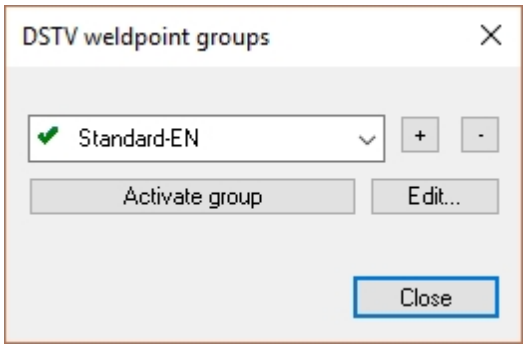
A point indicates on which spot an element must be welded on the profile. Because the CNC-machine can drill these points automatically, it can result in gaining time because one has to measure less when welding.

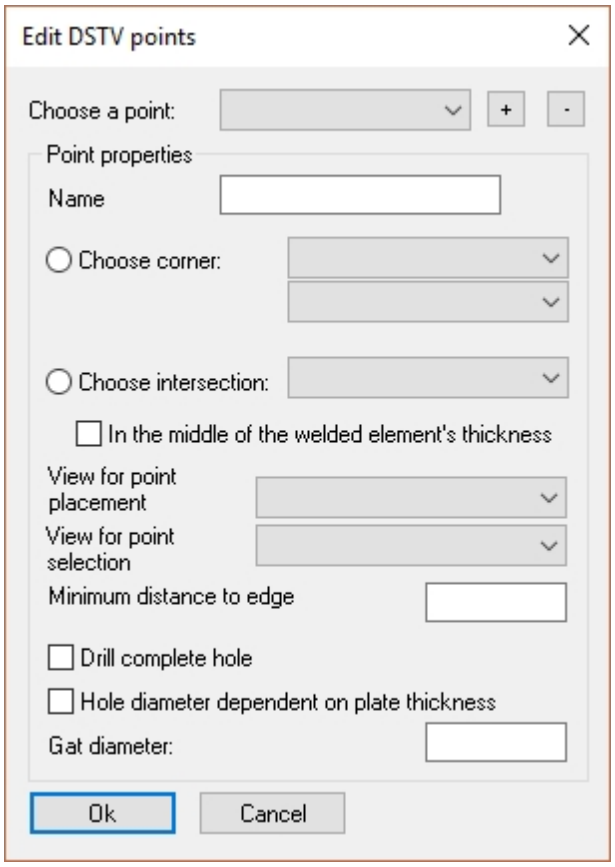
The points will normally be made by the machine with the tip of the (currently present) drill.

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.

Related Articles:

DSTV Weld Points





Creating a points list

First of all we have to make some DSTV points that we will later use for each element.

When you open the *Dstv Weldpoints* 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 **WeldpPoint** of the properties you enter the name of the weld point.

This property is of course only useful for welded elements.

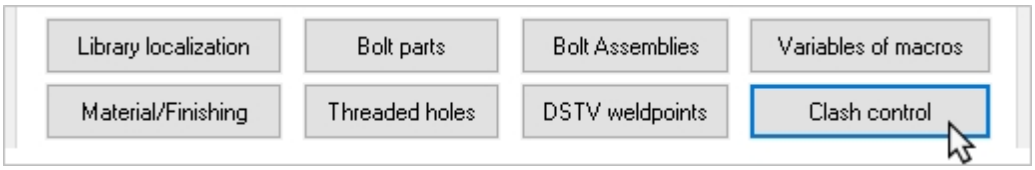
This property exists both for plates and profiles.

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.

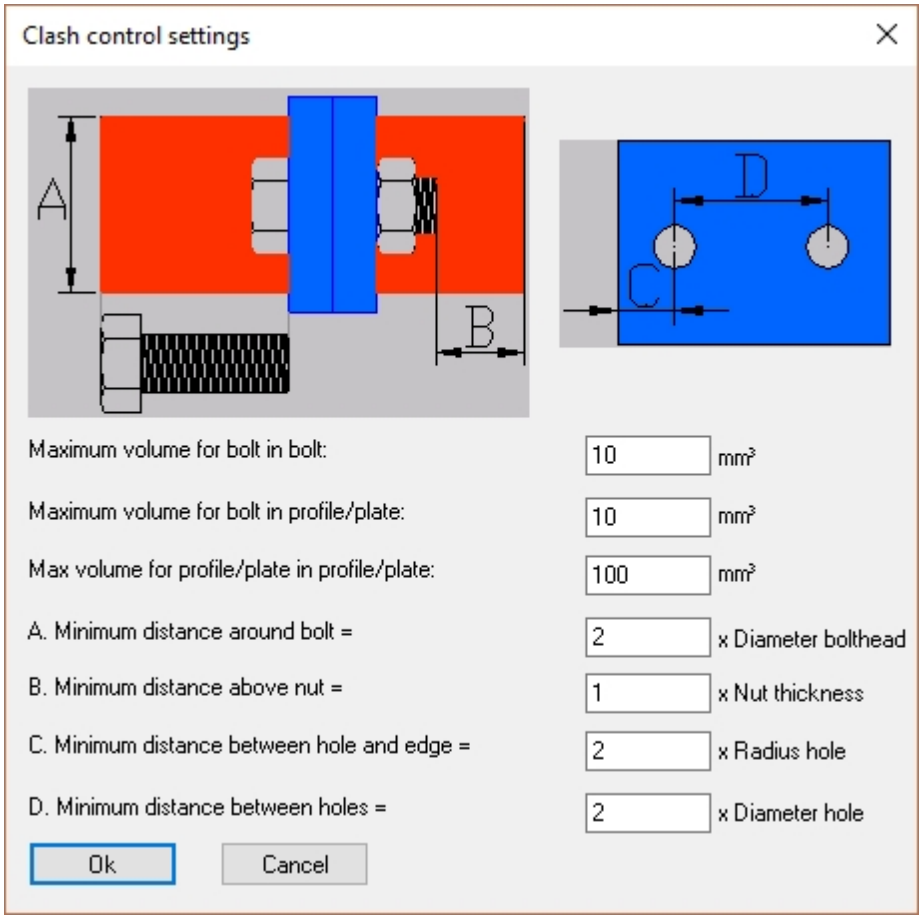
Assembly drawings and DSTV files will also get different names. Normally you have one DSTV file for each position number, but that is no longer possible when that profile has weld points. In that case the file name of the assembly drawing and the DSTV file will contain both the mark number and the position number. The result is more files and drawings, but it is an inevitable disadvantage.

Clash Control



The default settings may be revised to suit user preferences.

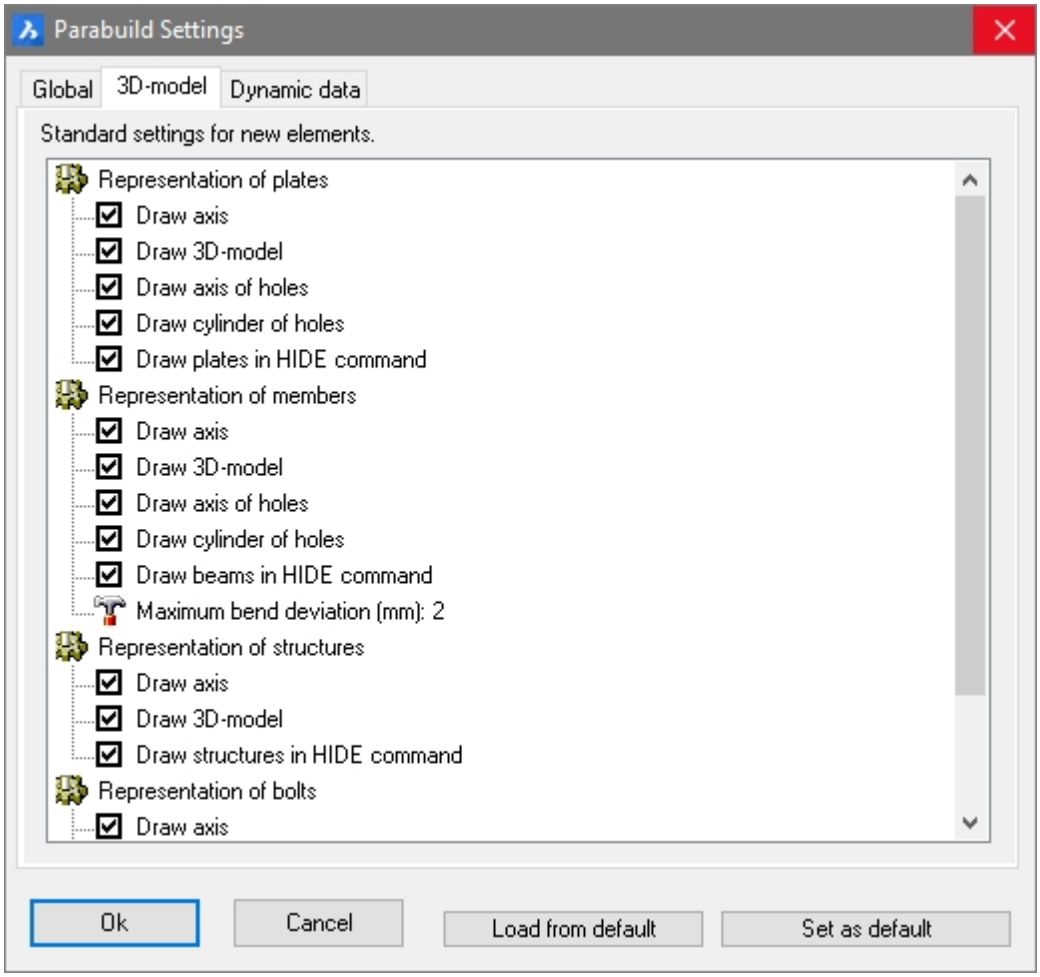
Values greater than the maximum, or less than the minimum will be marked on the drawing as 'Clashing'



3D Model

These settings determine how the 3D models are presented.

All of the settings in this list will apply to newly drawn objects - they do not affect existing objects.

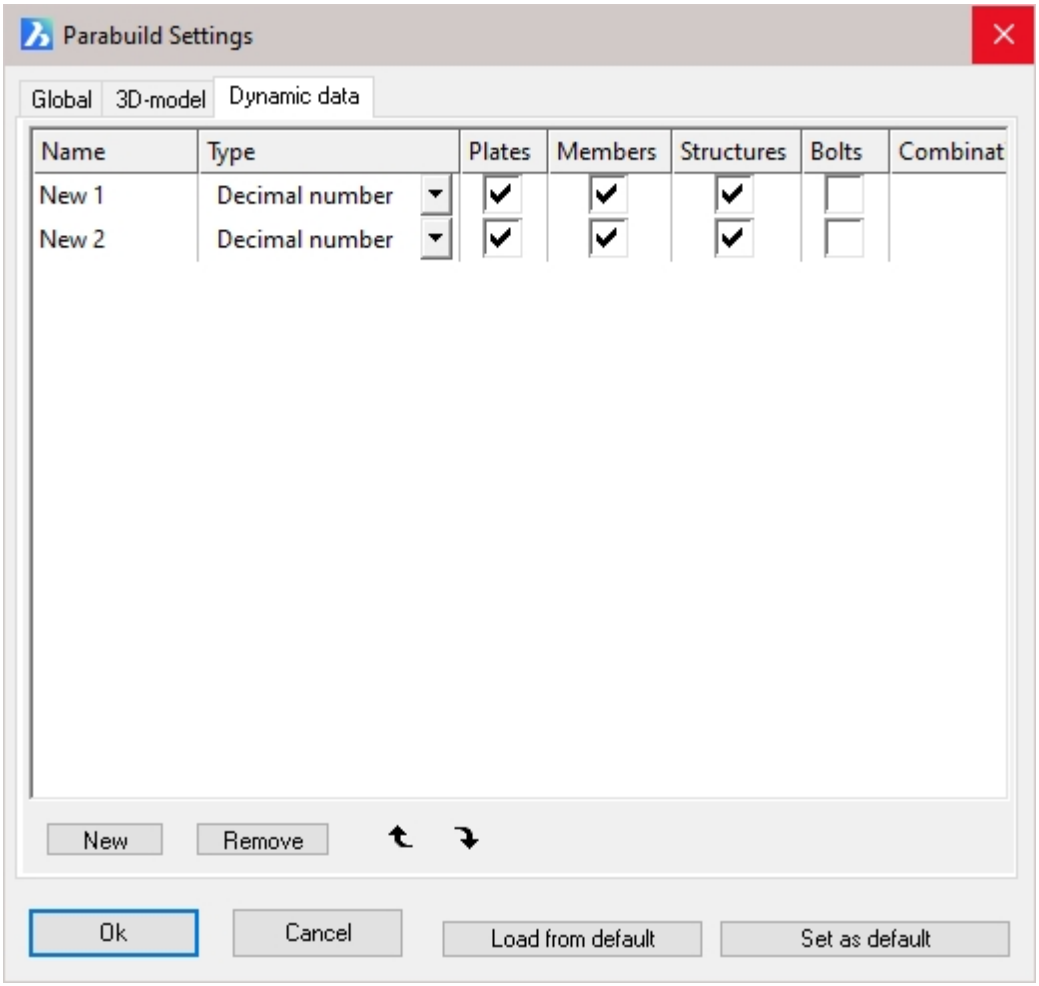


Dynamic Data

Dynamic properties are custom 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 Properties panel. Every dynamic property can also be activated as a column in the parts list (also on workshop drawings).



Name:- Click the button 'New' to add a new property. The name may be edited to suit.

Type:- Is the determined value of the property. From the drop-down menus select either:

Decimal number - e.g. 00

Comma delimited number - e.g. 00.0

Text - e.g. Stock No. 5

Combination-text - This can be used to contain a fixed number of predetermined texts. These texts should be entered in the last column

Plates:- Tick the checkbox to apply the property to plates

Members:- Tick the checkbox to apply the property to members

Structures:- Tick the checkbox to apply the property to structures

Bolts:- Tick the checkbox to apply the property to bolts

Combination text:- The texts should be separated with a comma. E.g. "steel,concrete,aluminum" will allow you to make a selection of these three materials.

The Properties Panel

The Properties of all Parabuild objects can be changed using the Properties window. This window can be started in one of 4 ways:

1. From the top (CAD) menu, select **Modify** > **Properties**
2. At the command line, enter **Properties**
3. Select an object using the left mouse button, click on the object with the right mouse button, and select **Properties** from the drop-down menu
4. Double click the object using the left mouse button (this will only work on some objects such as lines)

The properties window can be resized to suit, and docked at the top, bottom, left, or right sides of the viewing area. When the dialog window is open, all commands are still accessible without affecting the Properties window.

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

Note that editing an object from the Properties panel will only affect the selected object(s) and will have no impact on the default settings.

Just a note of caution, that when selecting multiple objects with different properties, for example 2 members of different sections, the 'Properties' value will, in most cases, be read as 'Varies'. Editing this value, will change both members. At any time, these changes may be reversed by clicking on the **Edit** > **Undo** button from the top (CAD) menu.

Note! The default values visible in the Properties panel are set by the AutoCAD or BricsCAD settings – these values may be overridden by the Parabuild Global Settings. The Parabuild settings will take precedence over all CAD settings.

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

Types of objects

Each object type has its own distinctive properties.

The following object types are explained in more detail :

- Arc
- Line
- Coordinate system
- Annotation
- Member
- Macro
- Plate
- Pattern
- Bolt
- Structure

- Array
- View
- Camera
- Annotation

Dynamic Properties

All Parabuild parts can have an unlimited number of user properties.

These are shown at the bottom of the properties list.

To create these user properties, see the topic See Dynamic data.

Arc

The 'Arc' object type has the following headings:

General

3D Visualization

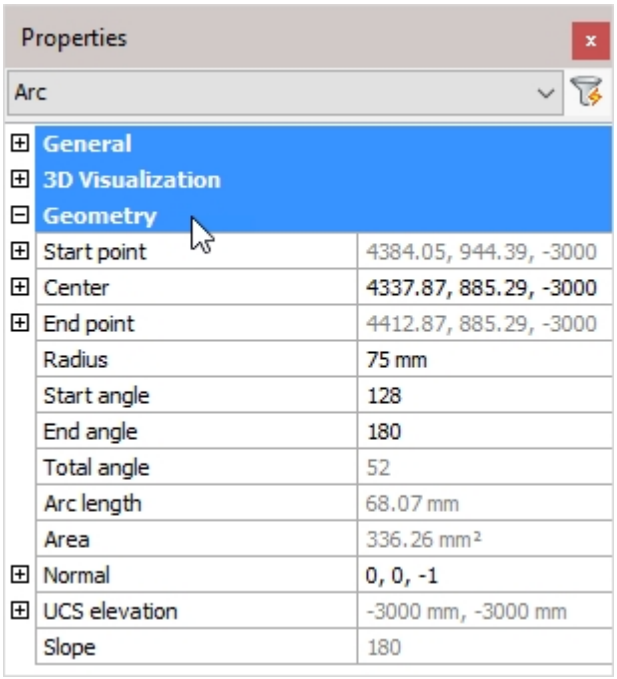
Geometry

General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Geometry:



Note! - * *indicates that the given value is a resultant of the sum of other values in the panel - and cannot be individually edited.*

Start point: * Start point of the arc – expressed as the X, Y, and Z coordinates

Center: The geometric center of the arc – expressed as the X, Y, and Z coordinates

End point: * End point of the arc – expressed as the X, Y, and Z coordinates

Radius: Arc radius

Start angle: Measured from the horizontal plane

End angle: Measured from the horizontal plane

Total angle: * = (End angle - start angle)

Arc Length: * Calculated arc length

Area: * calculated area

Normal: Determines the 3D unit normal vector of a selected circle, arc, or PolyLine arc segment. This normal vector is the Z coordinate of the object coordinate system (OCS) of the selected object.

UCS elevation: * UCS elevation (Reference)

Slope: * Slope angle relative to the horizontal plane (Reference)

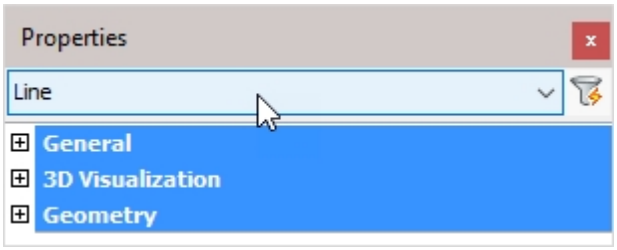
Line

The 'Line' object type has the following headings:

General

3D Visualization

Geometry

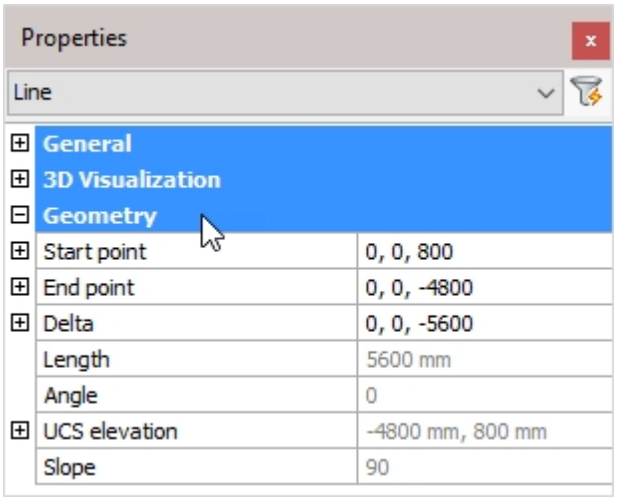


General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Geometry



Note! - * *indicates that the given value is a resultant of the sum of other values in the panel - and cannot be individually edited.*

Start point: Line start point (X, Y, Z coordinates)

End point: Line end point (X, Y, Z coordinates)

Delta: Delta (X, Y, Z coordinates)

Length: * Calculated line length

Angle: * calculated line angle

UCS elevation: * UCS elevation (Reference)

Slope: * Slope angle relative to the horizontal plane (Reference)

Coordinate System

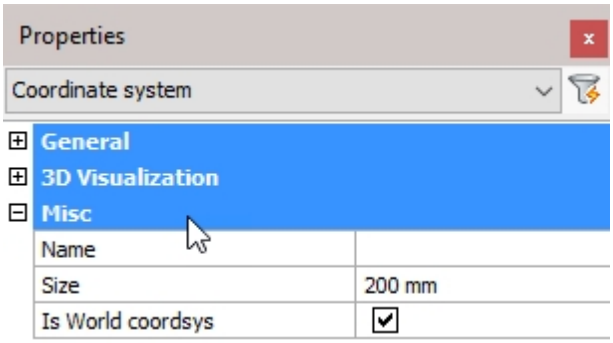
The 'Coordinate system' object type has the following headings:

General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Misc:



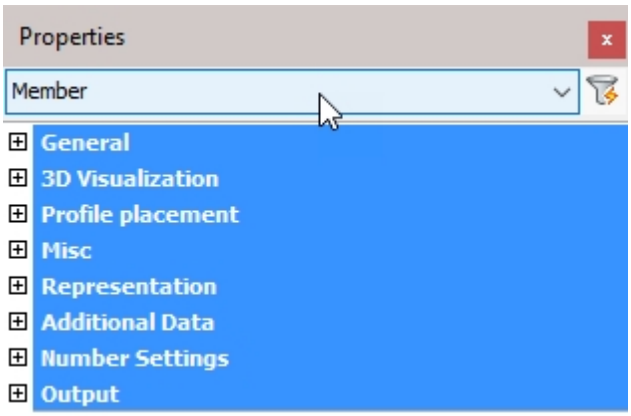
Name: User coordinate system name

Size: User coordinate system Icon size

Is World coordsys: This value can't be changed as there is always 1 World coordinate system in the drawing. Parabuild creates this coordinate system automatically.

Member

The 'Member' object type has the following headings:

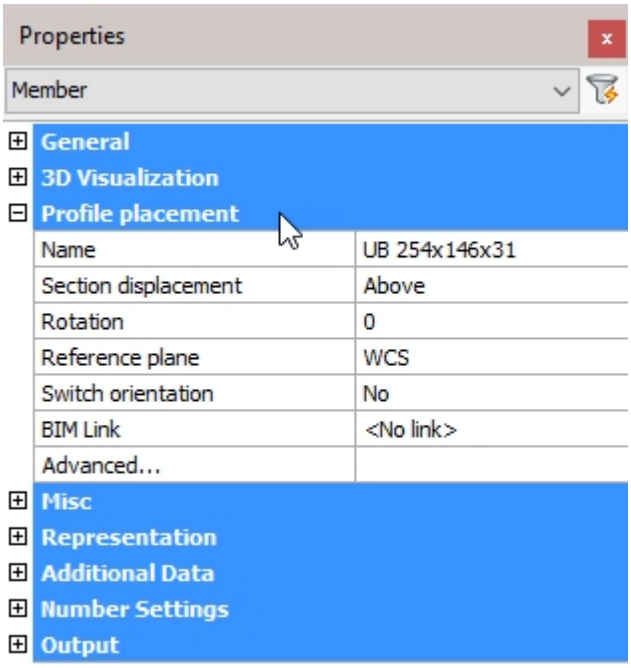


General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Profile placement



Name: Clicking the button will open the *Select Profile* dialog where you can select a different profile from any of the groups. Changing the profile will reflect immediately on the drawing.

Section displacement: Select a new section displacement point from the drop-down menu (This value will be *Unavailable* when the member is not determined by a macro)

Rotation: Enter a new value for the rotation of the profile (This value will be *Unavailable* when the member is not determined by a macro)

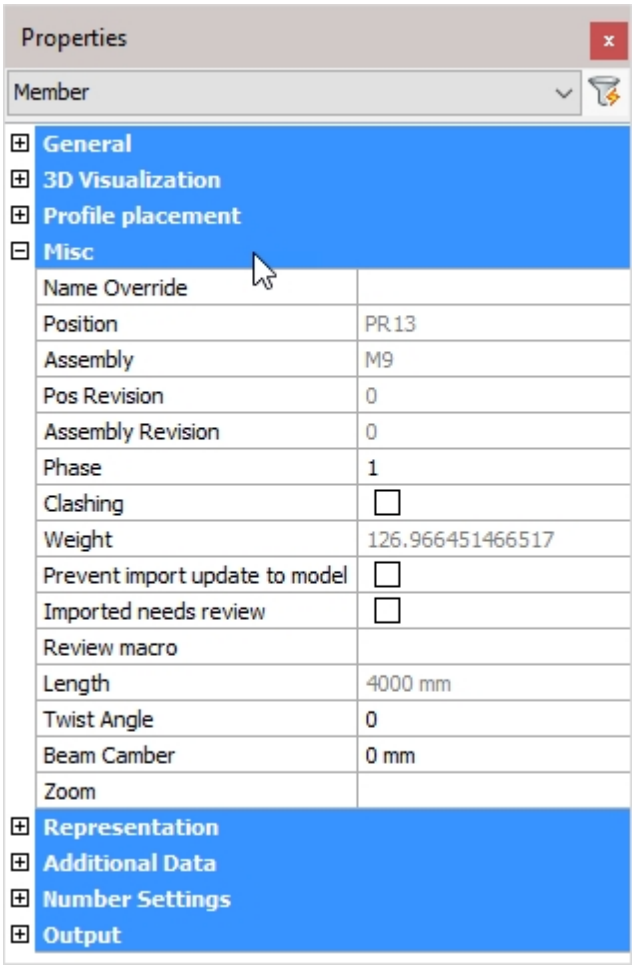
Reference plane: Select a Reference plane from the drop-down menu (This value will be *Unavailable* when the member is not determined by a macro)

Switch orientation: Enables you to switch the orientation of the profile. *For more information, refer to* Switch Profile Triangle Direction (This value will be *Unavailable* when the member is not determined by a macro)

BIM link: Select options from the drop-down menu (This value will be *Unavailable* when the member is not determined by a macro)

Advanced: Will open the Profile Placement dialog (This value will be *Unavailable* when the member is not determined by a macro)

Misc



Note! - * *indicates that the given value is a resultant of the sum of other values in the panel - and cannot be individually edited.*

Name override: The replacement of the normal profile name. This replacement name will be used in all part lists and workshop drawings.

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 suffixes.

Assembly: * The Assembly number automatically given to elements by Parabuild

Pos revision: * Position revision number

Assembly revision: * Assembly revision number

Phase: Change the project phase of the selected element

Clashing: 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 primarily included in properties for search objectives. For more information read the chapter Clash Check

Weight: * Calculated weight of the selected member

Prevent import update to model: Activating the checkbox will prevent any import update to the model. This refers to CIS/2 or Ifc incremental imports.

Imported needs review: This value will be set automatically for parts that were updated after an incremental CIS/2 or Ifc import.

Review macro: Will open the macro edit dialog, where the macro may be viewed and/or edited

Length: * Calculated member length

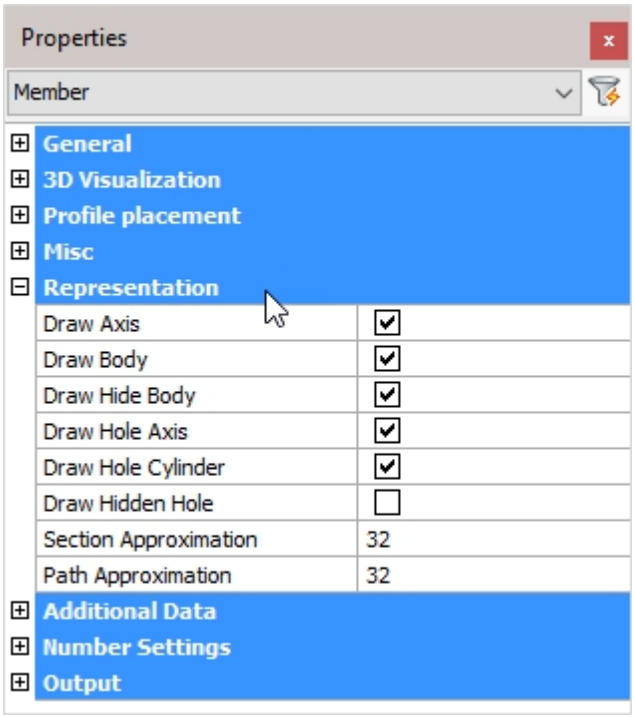
Twist angle: The angle that the profile has to turn over the length of the profile. This should only

be used for profiles on a spiral path or other special 3D-forms.

Beam camber: The camber (distance) to be used for the beam. This value influences the position number of the beam. This value is exported to KISS files.

Zoom: Zoom to the extents of the selected member

Representation



Draw Axis: Draws the Axis of a profile

Draw Body: Draws the complete 3D-model of the profile in 2D wireframe

Draw Hide Body: Draws the complete 3D-model of the profile in 3D visual styles

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

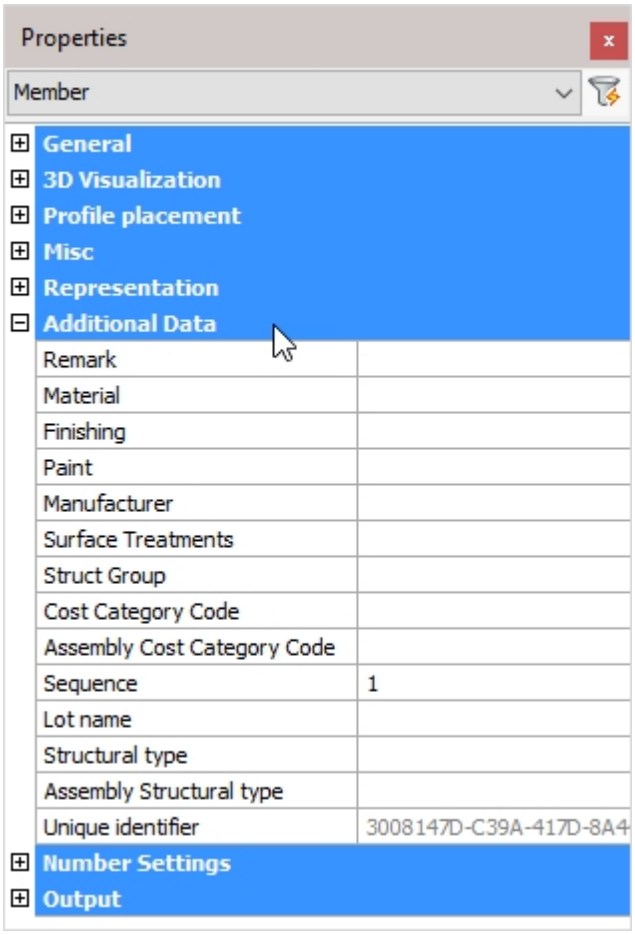
Draw Hole Cylinder: Draws the cylindrical path of all holes in the profile in 2D wireframe

Draw Hidden Hole: Draws the cylindrical path of all holes in the profile in 3D visual styles. One can activate the hole subtraction here for a better visual representation, but do note that when a bolt is drawn through this hole, you would not see the difference. On top of that, the subtraction is heavy on resources. This is the reason why this option is disabled by default. The edges of the hole are always drawn independently of this setting.

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 on curved profiles

Additional data



Remark: 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: 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/assembly number). Two elements that are identical, but have another material assigned to, will be given another position number. This enables a total categorization of different materials in part lists and workshop drawings.

The weight factor of every material can be changed in the Parabuild Properties - Global - Material/Finishing. Parabuild will use this weight factor to calculate the weight in the part lists.

Finishing: Same as Remark

Paint: Same as Remark

Manufacturer: Same as Remark

Surface Treatments: Will open the Surface Treatments dialog

Struct Group: This property can be freely used for the user's own purpose.

Cost category Code: These codes are usually determined while a quote for the project is made. This code can then later be assigned to the 3D model when the project was ordered. This way the parts can be traced from estimation to production and to completion inside ERP/MIS software. This value is exported to KISS files.

Assembly Cost category Code: See 'Cost category Code'

Sequence: The erection sequence of the part. This value is exported to KISS files

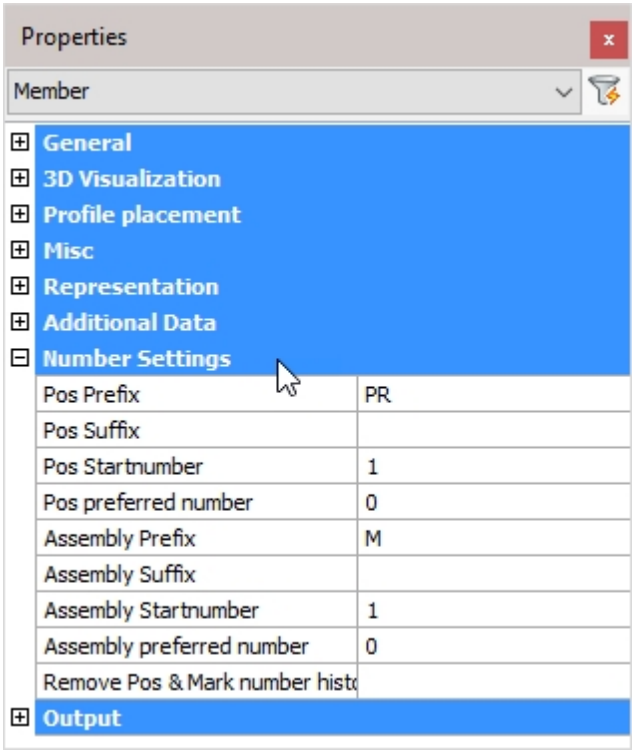
Lot Name: The lot that the part belongs to. This value is exported to KISS files

Structural Type: ** For example: Bracing/Rafter/Stringer/Handrail/Post/etc....**

Assembly Structural Type: **For example: Stair/Railing/etc....**

Unique Identifier: * Unique identifier allocated by Parabuild, or received from a CIS/2 or Ifc file import.

Number settings



Pos Prefix: Determines the prefix of the position number.

Pos Suffix: Determines the suffix of the position number.

Pos Start number: Determines the start number of the position number.

Pos preferred number: Forces Parabuild to use the preferred number as position number. Parabuild will comply if this number is not already in use by a different part.

Assembly Prefix: Determines the prefix of the Assembly number

Assembly Suffix: Determines the prefix of the Assembly number

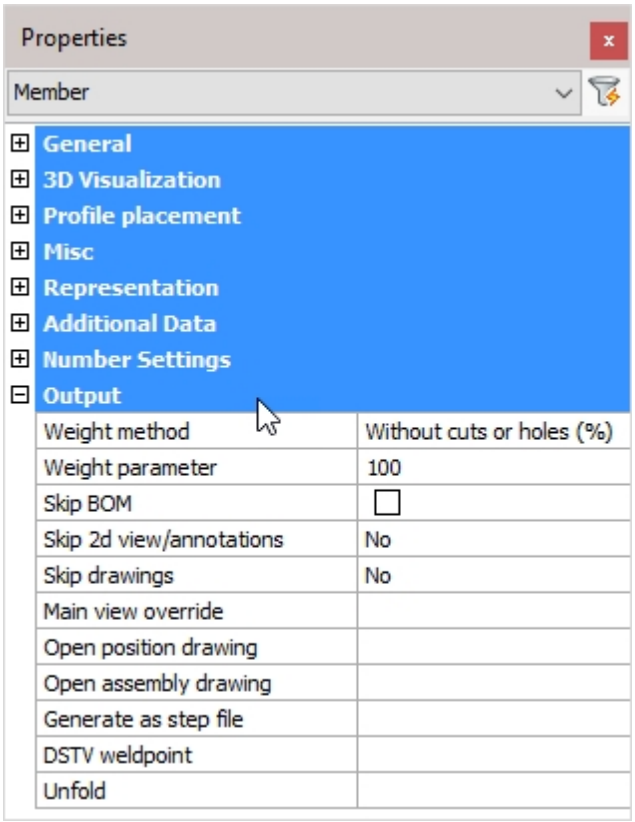
Assembly Start number: Determines the start number of the Assembly number

Assembly preferred number: Forces Parabuild to use the preferred number as assembly/mark number. Parabuild will comply if this number is not already in use by a different part.

Remove Pos & Mark number history: Lets Parabuild forget the preferred numbers and also the previous numbers that the part had.

Note! For more information on any of these items, refer to Numbering of elements

Output



Weight method:

- **Without cuts or holes (%)**: 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.
- **Cut**: The cuts in the profiles are subtracted to calculate the weight.
- **Cut and drilled**: The cuts 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 or lbs (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 this property to 'Yes', this element will not appear in the BOMs

Skip 2D view/annotations: Adjust this to skip this element for the 3D annotations or the 3D annotations + 2D view

Skip drawings: You can skip only the Position drawings, only the mark/assembly drawings, or both the Position and Mark drawings

Main view override: Select a plane that determines the main view's direction

Open position drawing: Will open the position drawing of the selected part

Open assembly drawing: Will open the assembly drawing of the selected assembly

Generate as a STEP file: STEP file is a CAD file format, usually used to share 3D models between users with different CAD systems. Note! This command will only work in BricsCAD and when the Communicator module is installed and licensed. *For more information, refer to* Settings for Workshop Drawings > Generate STEP Files

DSTV weld point: 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. *For more information, refer to* parabuild Settings > Global > DSTV Weld points

Unfold: Will unfold or develop a sheet metal object. Note! This command will only work in BricsCAD and when you have an active sheet metal license.

Macro

The 'Macro' object type has the following headings:

General

3D Visualization

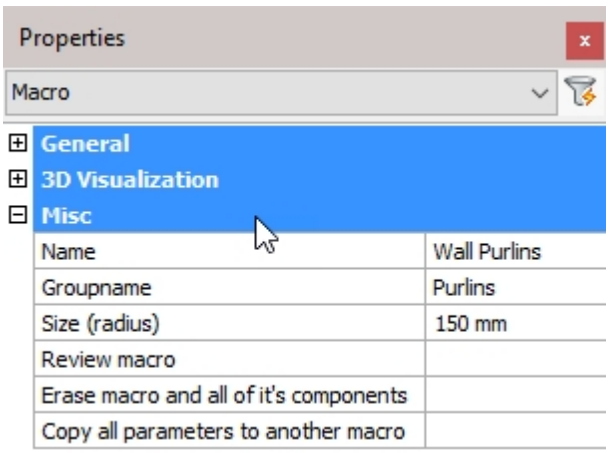
Misc

General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Misc



Name: The name of the macro – this may be edited and will reflect in the macro dialog box

Group name: The group name of the macro – this value is used for grouping together similar macros in the macro dialog box

Size (radius): Adjust the size of the macro sphere

Review macro: Activating this button will open the macro edit dialog box for the selected object

Erase macro and all of its components: Will Erase macro and all of its components

Copy all parameters to another macro: Will Copy all parameters to another macro

Plate

The 'Plate' object type has the following headings:

General

3D Visualization

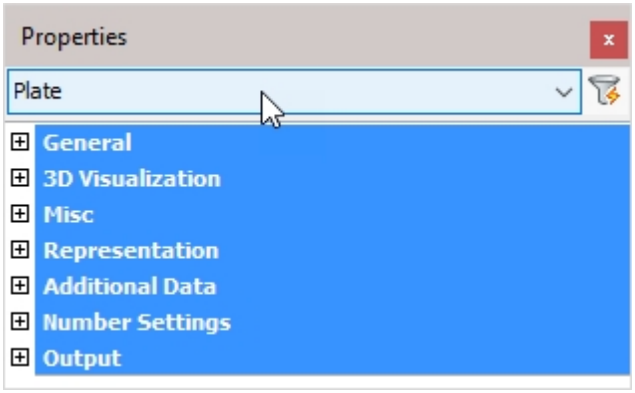
Misc

Representation

Additional Data

Number Settings

Output

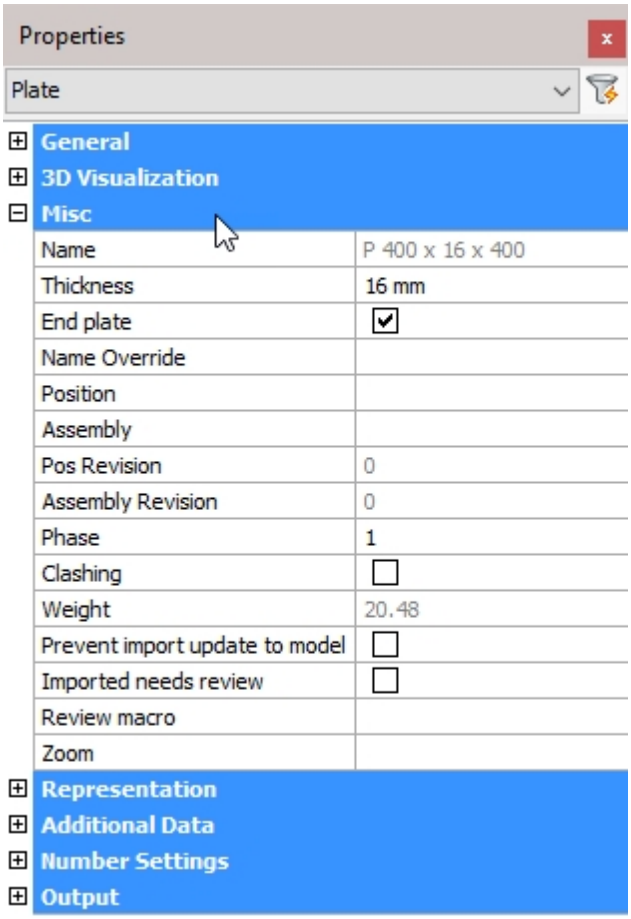


General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Misc



Note! - * *indicates that the given value is a resultant of the sum of other values in the panel - and cannot be individually edited.*

Name: * This is the plate size, also shown in the BOM. Refer to the Settings dialog box for influencing this name

Thickness: Plate thickness - editing this value will immediately reflect on the model

End Plate: A plate, designated as an 'End plate' will receive an additional view on the assembly drawing

Name Override: The replacement of the normal plate name. This replacement name will be used in all part lists and workshop drawings.

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 suffixes.

Assembly: * Assembly number

Pos Revision: * Position revision number

Assembly Revision: * Assembly revision number

Phase: Change the project phase of the selected element

Clashing: 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 primarily included in properties for search objectives. For more information read the chapter Clash Check

Weight: * Calculated weight of the selected member

Prevent import update to model: Activating the checkbox will prevent any import update to the model. This refers to CIS/2 or Ifc incremental imports.

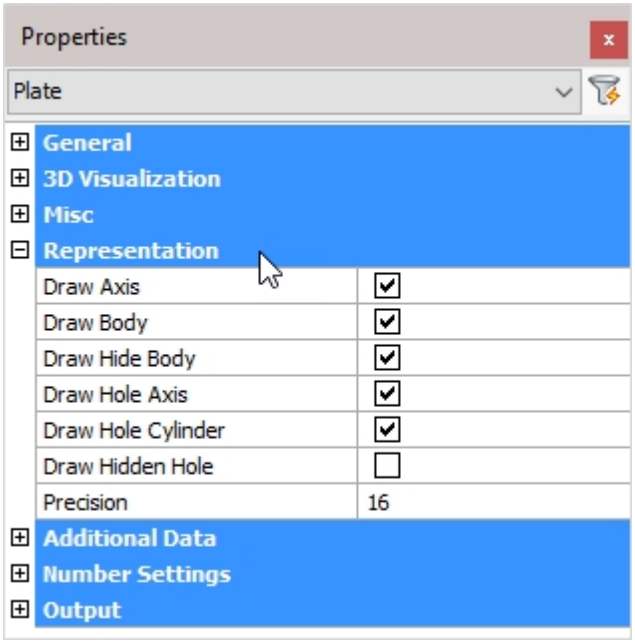
Imported needs review: This value will be set automatically for parts that were updated after an

incremental CIS/2 or Ifc import.

Review macro: Will open the macro edit dialog, where the macro may be viewed and/or edited

Zoom: Pressing the button will zoom to the extents of the selected plate

Representation



Draw Axis: Draws the Axis of a profile

Draw Body: Draws the complete 3D-model of the profile in 2D wireframe

Draw Hide Body: Draws the complete 3D-model of the profile in 3D visual styles

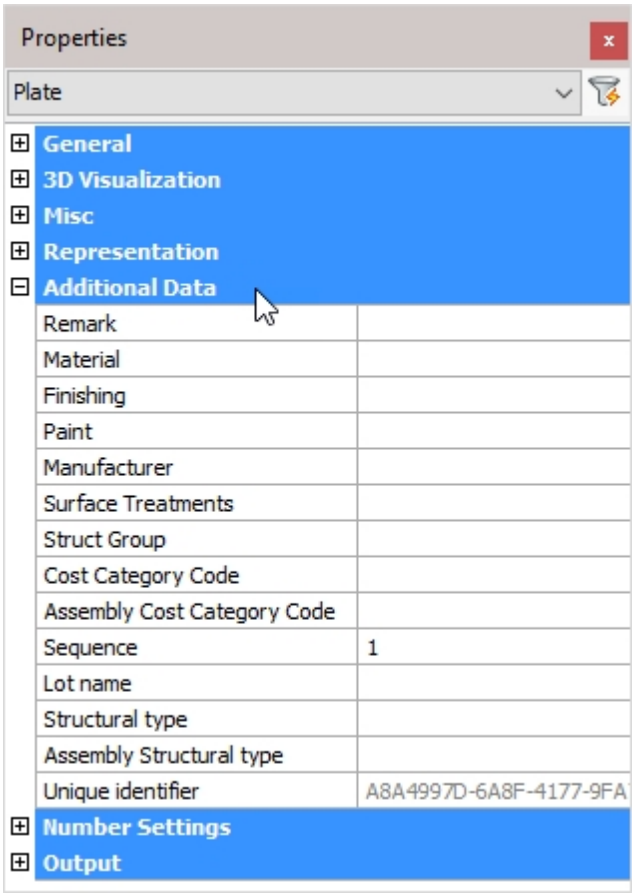
Draw Hole Axis: Draws the axis of every hole in the plate

Draw Hole Cylinder: Draws the cylindrical path of all holes in the plate in 2D wireframe

Draw Hidden Hole: Draws the cylindrical path of all holes in the plate in 3D visual styles.
One can activate the hole subtraction here for a better visual representation, but do note that when a bolt is drawn through this hole, you would not see the difference. On top of that, the subtraction is heavy on resources. This is the reason why this option is disabled by default.
The edges of the hole are always drawn independently of this setting.

Precision: This value applies to the precision of curves in the plate's internal or external contours.

Additional Data



Remark: 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: 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 categorization of different materials in part lists and workshop drawings.

The weight factor of every material can be changed in the Parabuild Properties > Global > Material/Finishing. Parabuild will use this weight factor to calculate the weight in the part lists.

Finishing: Same as Remark

Paint: Same as Remark

Manufacturer: Same as Remark

Surface Treatments: Will open the Surface Treatments dialog

Struct Group: This property can be freely used for the user's own purpose.

Cost category Code: These codes are usually determined while a quote for the project is made. This code can then later be assigned to the 3D model when the project was ordered. This way the parts can be traced from estimation to production and to completion inside ERP/MIS software. This value is exported to KISS files.

Assembly Cost category Code: See 'Cost category Code'

Sequence: The erection sequence of the part. This value is exported to KISS files

Lot Name: The lot that the part belongs to. This value is exported to KISS files

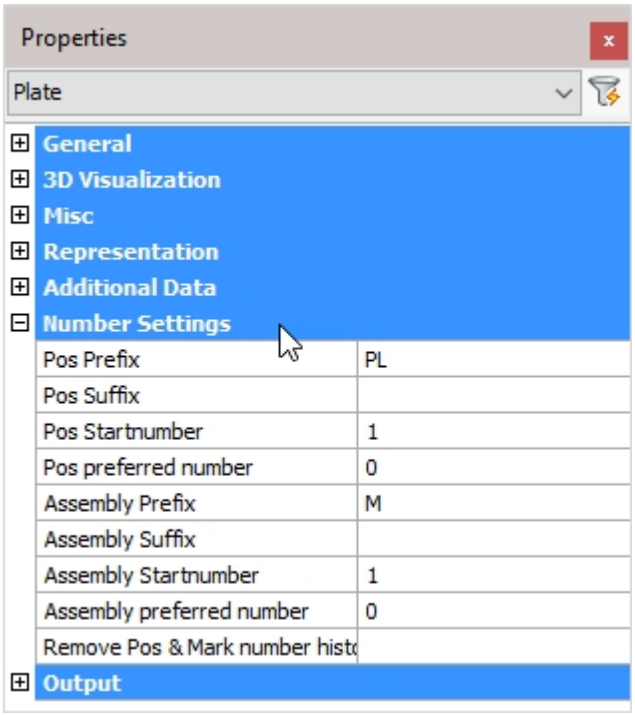
Structural Type: ** For example: Bracing/Rafter/Stringer/Handrail/Post/etc....**

Assembly Structural Type: **For example: Stair/Railing/etc....**

Unique Identifier: * Unique identifier allocated by Parabuild, or received from a CIS/2 or Ifc file

import.

Number Settings



Pos Prefix: Determines the prefix of the position number.

Pos Suffix: Determines the suffix of the position number.

Pos Start number: Determines the start number of the position number.

Pos preferred number: Forces Parabuild to use the preferred number as position number. Parabuild will comply if this number is not already in use by a different part.

Assembly Prefix: Determines the prefix of the Assembly/Mark number.

Assembly Suffix: Determines the prefix of the Assembly/Mark number.

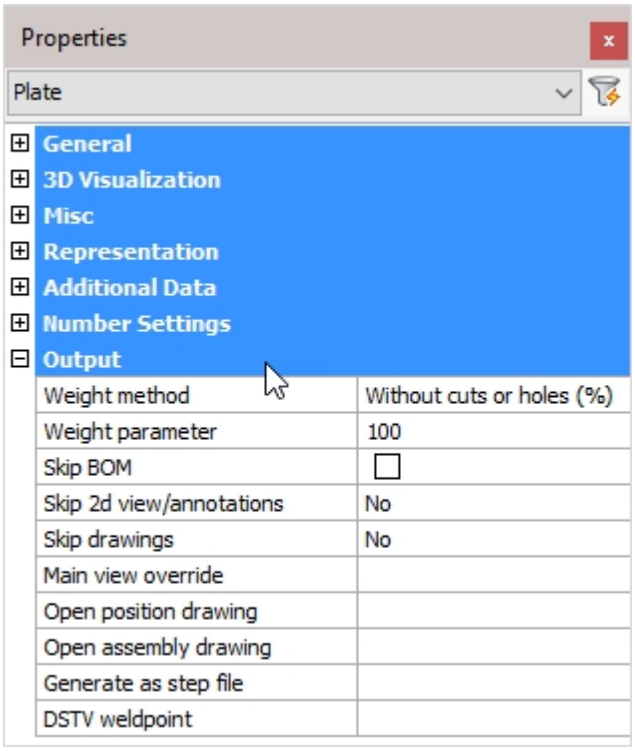
Assembly Start number: Determines the start number of the Assembly/Mark number.

Assembly preferred number: Forces Parabuild to use the preferred number as assembly/mark number. Parabuild will comply if this number is not already in use by a different part.

Remove Pos & Mark number history: Lets Parabuild forget the preferred numbers and also the previous numbers that the part had.

Note! For more information on any of these items, refer to Numbering of elements

Output



Weight method:

- **Without cuts or holes (%)**: 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.
- **Cut**: The cuts in the profiles are subtracted to calculate the weight.
- **Cut and drilled**: The cuts 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 or lbs (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 this property to 'Yes', this element will not appear in the BOMs

Skip 2D view/annotations: Adjust this to skip this element for the 3D annotations or the 3D annotations + 2D view

Skip drawings: You can skip only the Pos drawings, only the mark drawings, or both the Pos and Mark drawings

Main view override: Select a plane that determines the main view's direction

Open position drawing: Will open the position drawing of the selected part

Open assembly drawing: Will open the assembly drawing of the selected assembly

Generate as a STEP file: STEP file is a CAD file format, usually used to share 3D models between users with different CAD systems. Note! This command will only work in BricsCAD and when the Communicator module is installed and licensed. *For more information, refer to* Settings for Workshop Drawings > Generate STEP Files

DSTV weld point: 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. *For more information, refer to* parabuild Settings > Global > DSTV Weld points

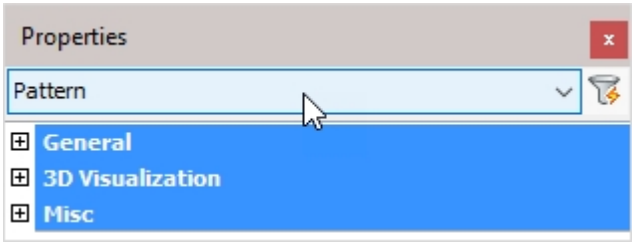
Pattern

The 'Pattern' object type is used by macros to draw and manage a range of bolts. It has the following headings:

General

3D Visualization

Misc



General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Misc

Name: Will show the pattern name

Bolt

The 'Bolt' object type has the following headings:

General

3D Visualization

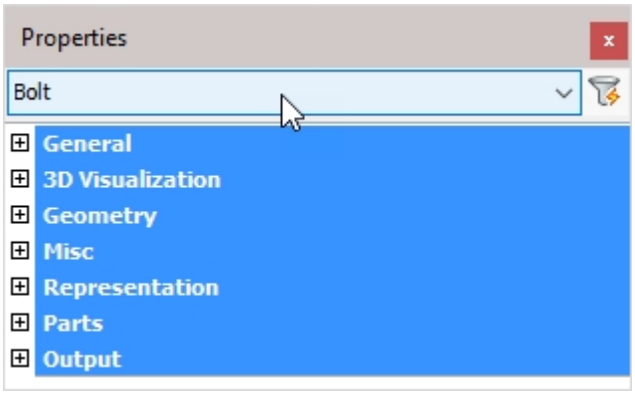
Geometry

Misc

Representation

Parts

Output

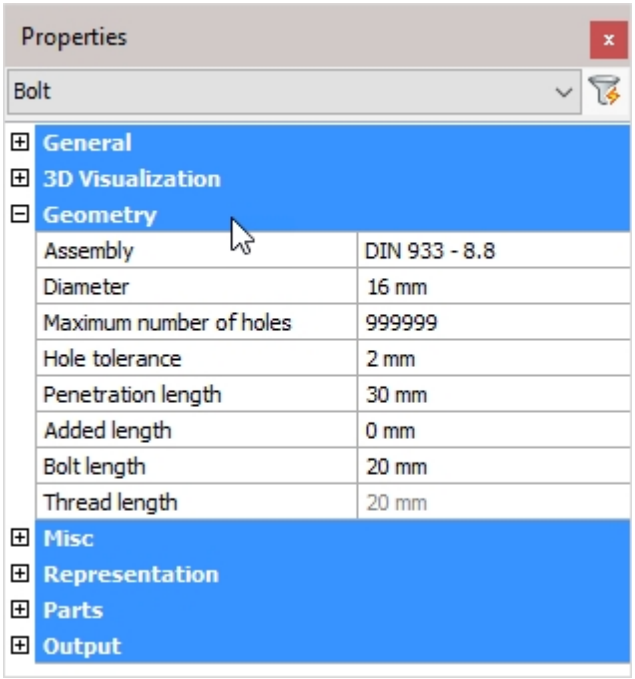


General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Geometry



Assembly: Select a Bolt assembly from the drop-down menu. For more information refer to Bolt Assemblies

Diameter: Edits the bolt diameter

Maximum number of holes: You can enter a limit on the number of holes that this bolt can have. Parts will get holes starting from the bolt head until the maximum number of holes limit has been reached.

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

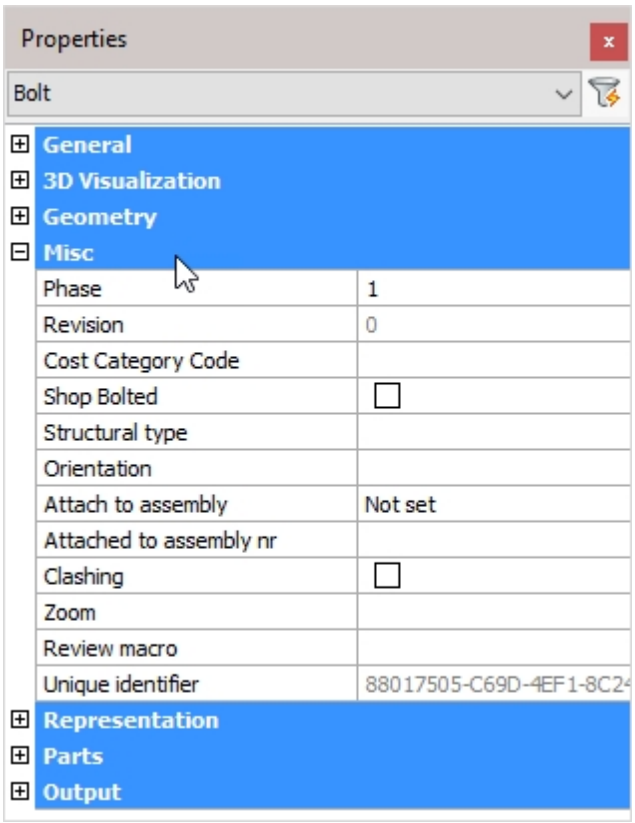
Penetration length: A combination of the thickness of all the drilled part(s)

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

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 use the command Check for new holes on the tube and the bolt. The bolt will be given the correct length and the second hole will be drawn.

Thread length: The thread length is taken from the Bolt parts database

Misc



Note! - * *indicates that the given value is a resultant of the sum of other values in the panel - and cannot be individually edited.*

Phase: Change the project phase of the selected element

Revision: * Position revision number

Cost category code: These codes are usually determined while a quote for the project is made. This code can then later be assigned to the 3D model when the project was ordered. This way the parts can be traced from estimation to production and to completion inside ERP/MIS software. This value is exported to KISS files.

Shop Bolted: Activating the checkbox will set the bolt to shop-bolted - See Bolts/Holes > Shop Bolts

Structural type: ** For example: Bracing/Rafter/Stringer/Handrail/Post/etc....**

Orientation: Clicking on the button will switch the orientation of the bolt

Attach to assembly: From the drop-down menu, select which element the bolt should be attached to. Options include 'Closest to bolt head' or 'Closest to bolt end'

Attached to assembly nr: This will indicate which assembly number the bolt is attached to

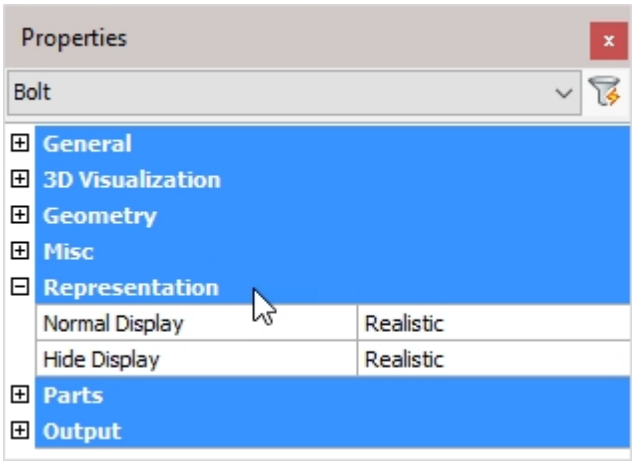
Clashing: This is set to yes or no and refers to whether the profile is currently clashing with other elements or not. This property is automatically set by the clash control, and is only included in properties for search objectives. For more information read the chapter Clash Check

Zoom: Zoom to the extents of the selected bolt(s)

Review macro: Will open the Macro Edit dialog, where the macro may be viewed and/or edited

Unique identifier: Unique identifier allocated by Parabuild, or received from a CIS/2 or Ifc file import.

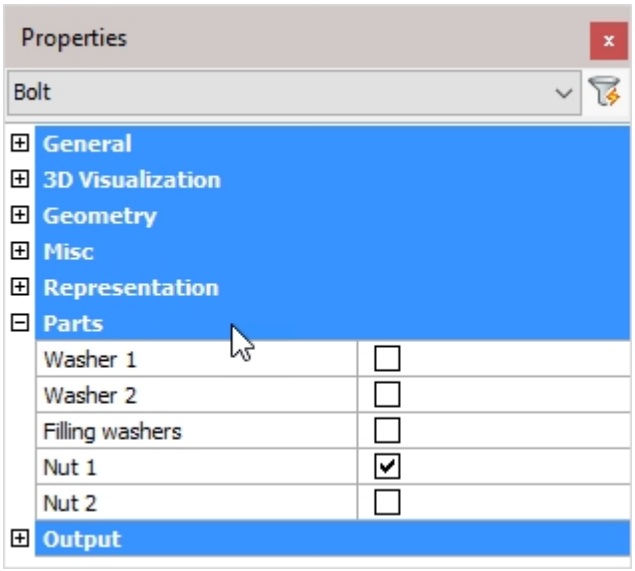
Representation



Normal Display: Selects the bolt display style in 2D wireframe

Hide Display: Selects the bolt display style in 3D Visual styles

Parts



Washer 1: Enables or disables washer 1. The standards used for these parts are recorded in Bolt Assemblies

Washer 2: Enables or disables washer 2. The standards used for these parts are recorded in Bolt Assemblies

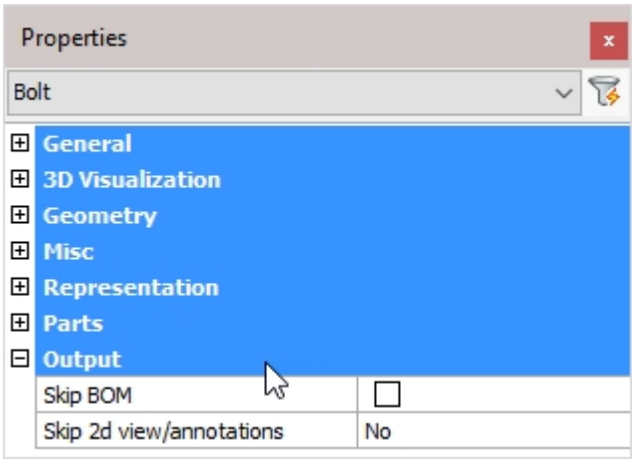
Filling washers: Enables or disables filling washers. The standards used for these parts are recorded in Bolt Assemblies

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

Nut 1: Enables or disables nut 1. The standards used for these parts are recorded in Bolt Assemblies

Nut 2: Enables or disables nut 2. The standards used for these parts are recorded in Bolt Assemblies

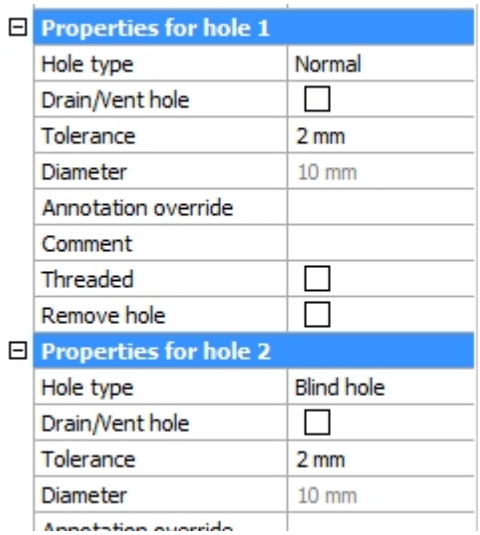
Output



Skip BOM: If this property to 'Yes', this element will not appear in the BOMs

Skip 2D view/annotations: Adjust this to skip this element for the 3D annotations or the 3D annotations + 2D view

Holes



Properties for hole 1	
Hole type	Normal
Drain/Vent hole	☐
Tolerance	2 mm
Diameter	10 mm
Annotation override	
Comment	
Threaded	☐
Remove hole	☐
Properties for hole 2	
Hole type	Blind hole
Drain/Vent hole	☐
Tolerance	2 mm
Diameter	10 mm

The last properties of the bolt are always the holes that the bolt may have.

All holes will be shown here separately, starting from the bolt's head the first hole will get number 1 and so on.

These are the properties for holes.

Hole type : Normal, Slotted, Counter sunk or Blind. For more information about the hole types, see their respective topics.

Drain/Vent hole : All the holes that are designated as 'drain / vent' holes will not receive any dimensions on the assembly shop drawings.

Tolerance : The tolerance for the hole in relation with the bolt.

Diameter : This value is only informative because it is derived from the bolt diameter and the tolerance.

Annotation override : When you enter a text in this field, this text will always be shown in the annotation of this hole on the position shop drawing. It will override the default text which would be the diameter, slotted, countersunk and thread information if they it is such a hole type.

Comment : Feel free to use this field to add comment to the hole. It will only have an effect if the variable %PbColHoleComment% is used in an annotation or in a parts list.

Threaded : See the threaded hole topic for more info about this.

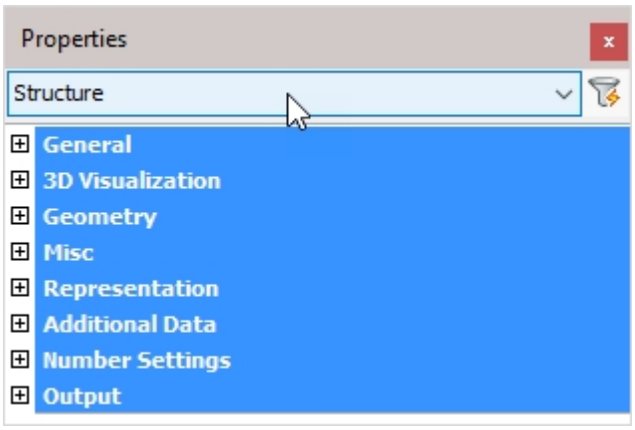
Remove hole : This is an action. Clicking on the button will erase this hole.

Structure

The 'Structure' object type has the following headings:

General

3D Visualization

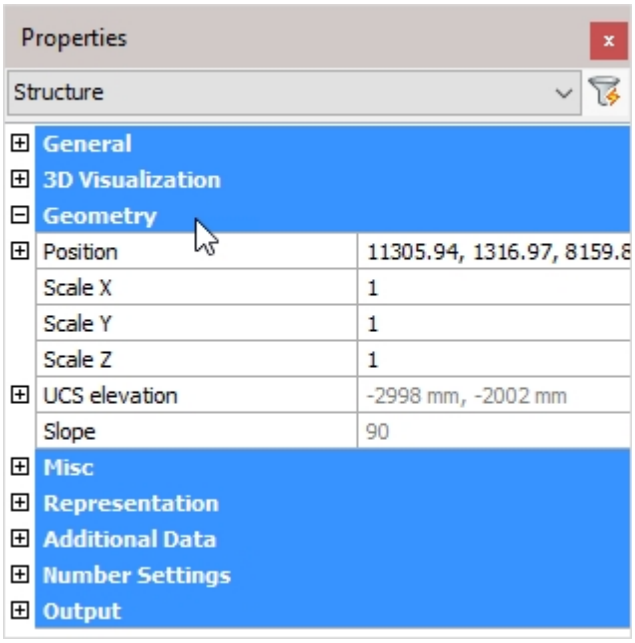


General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Geometry



Note! - * *indicates that the given value is a resultant of the sum of other values in the panel - and cannot be individually edited.*

Position: Position of the structure (X, Y, and Z) relative to the UCS coordinates

Scale X: Scale factor X

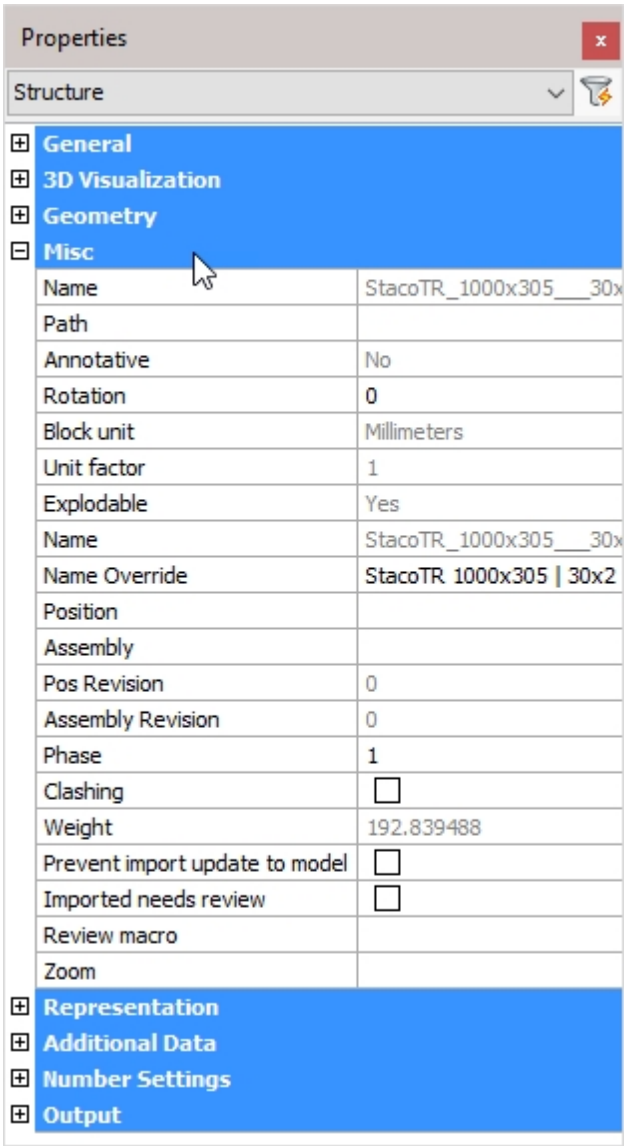
Scale Y: Scale factor Y

Scale Z: Scale factor Z

UCS elevation: * UCS elevation (Reference)

Slope: * Slope angle relative to the horizontal plane (Reference)

Misc



Note! - * *indicates that the given value is a resultant of the sum of other values in the panel - and cannot be individually edited.*

Name: * This is an AutoCAD and BricsCAD setting

Path: This is an AutoCAD and BricsCAD setting

Annotative: * This is an AutoCAD and BricsCAD setting

Rotation: This is an AutoCAD and BricsCAD setting

Block unit: * This is a AutoCAD / BricsCAD function. set block units to meters / mm / inches / etc.

Unit factor: * This is a AutoCAD / BricsCAD function. set block unit factor

Explodable: * Structure is explodable

Name: * Structure name used in the parts lists and workshop drawings

Name override: The replacement of the normal structure name. This replacement name will be used in all part lists and workshop drawings.

Position: * Position number

Assembly: * Assembly number

Pos revision: * Position revision number

Assembly revision: * Assembly revision number

Phase: The project phase of the selected element.

Clashing: 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 Check

Weight: * Calculated weight of the selected member

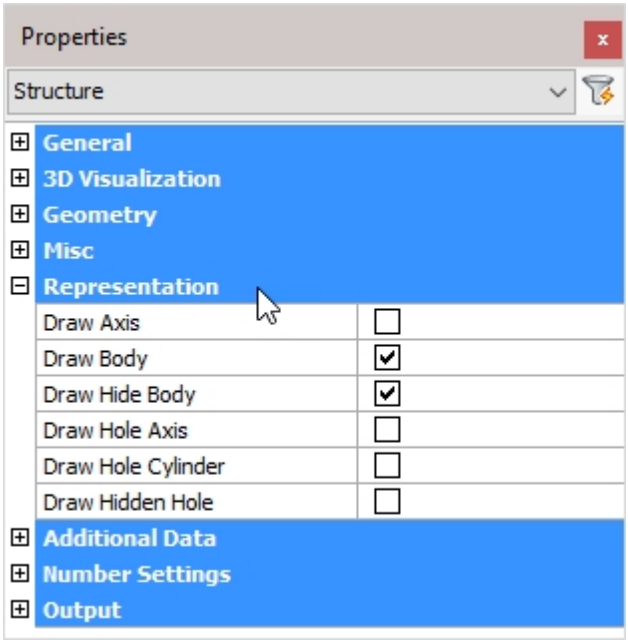
Prevent import update to model: Activating the checkbox will prevent any import update to the model. This refers to CIS/2 or Ifc incremental imports.

Imported needs review: This value will be set automatically for parts that were updated after an incremental CIS/2 or Ifc import.

Review macro: Will open the macro edit dialog, where the macro may be viewed and/or edited

Zoom: Zoom to the extents of the selected structure

Representation



Draw Axis: Draws the Axis of a profile

Draw Body: Draws the complete 3D-model of the profile in 2D wireframe

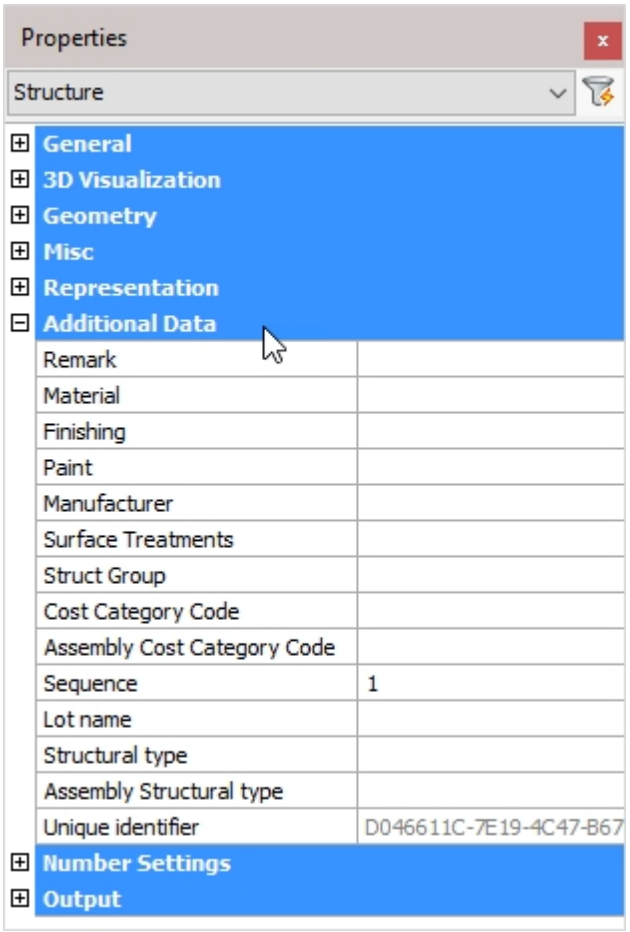
Draw Hide Body: Draws the complete 3D-model of the profile in 3D visual styles

Draw Hole Axis: Draws the axis of every hole in the structure

Draw Hole Cylinder: Draws the cylindrical path of all holes in the structure in 2D wireframe

Draw Hidden Hole: Draws the cylindrical path of all holes in the structure in 3D visual styles. One can activate the hole subtraction here for a better visual representation, but do note that when a bolt is drawn through this hole, you would not see the difference. On top of that, the subtraction is heavy on resources. This is the reason why this option is disabled by default. The edges of the hole are always drawn independently of this setting.

Additional Data



Remark: 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: 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 categorization of different materials in part lists and workshop drawings.

The weight factor of every material can be changed in Parabuild Properties - Global - Material/Finishing . Parabuild will use this weight factor to calculate the weight in the part lists.

Finishing: Same as Remark

Paint: Same as Remark

Manufacturer: Same as Remark

Surface Treatments: Will open the Surface Treatments dialog

Struct Group: This property can be freely used for the user's own purpose.

Cost category Code: These codes are usually determined while a quote for the project is made. This code can then later be assigned to the 3D model when the project was ordered. This way the parts can be traced from estimation to production and to completion inside ERP/MIS software. This value is exported to KISS files.

Assembly Cost category Code: See 'Cost category Code'

Sequence: The erection sequence of the part. This value is exported to KISS files

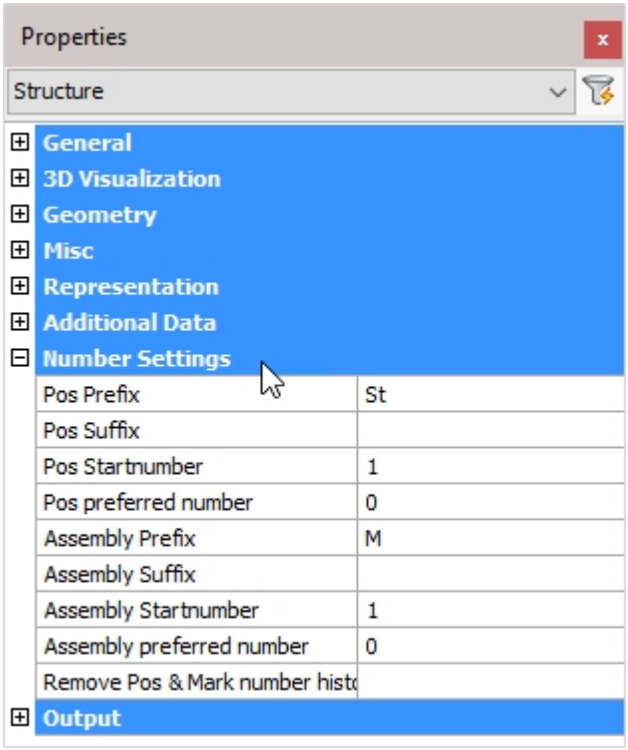
Lot Name: The lot that the part belongs to. This value is exported to KISS files

Structural Type: ** For example: Bracing/Rafter/Stringer/Handrail/Post/etc....**

Assembly Structural Type: **For example: Stair/Railing/etc....**

Unique Identifier: * Unique identifier allocated by Parabuild, or received from a CIS/2 or Ifc file import.

Number Settings



Pos Prefix: Determines the prefix of the position number.

Pos Suffix: Determines the suffix of the position number.

Pos Start number: Determines the start number of the position number.

Pos preferred number: Forces Parabuild to use the preferred number as position number. Parabuild will comply if this number is not already in use by a different part.

Assembly Prefix: Determines the prefix of the assembly/mark number.

Assembly Suffix: Determines the prefix of the assembly/mark number.

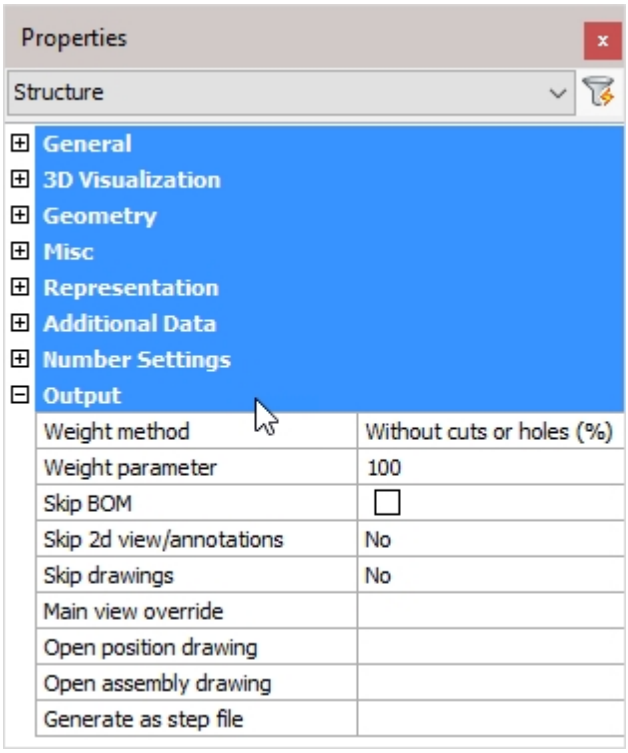
Assembly Start number: Determines the start number of the assembly/mark number.

Assembly preferred number: Forces Parabuild to use the preferred number as assembly/mark number. Parabuild will comply if this number is not already in use by a different part.

Remove Pos & Mark number history: Lets Parabuild forget the preferred numbers and also the previous numbers that the part had.

Note! For more information on any of these items, refer to Numbering of elements

Output



Weight method:

- **Without cuts or holes (%)**: 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.
- **Cut**: The cuts in the profiles are subtracted to calculate the weight.
- **Cut and drilled**: The cuts 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 or lbs (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 this property to 'Yes', this element will not appear in the BOMs

Skip 2D view/annotations: Adjust this to skip this element for the 3D annotations or the 3D annotations + 2D view

Skip drawings: You can skip only the Position drawings, only the mark drawings, or both the Position and Mark drawings

Main view override: Select a plane that determines the main view's direction

Open position drawing: Will open the position drawing of the selected part

Open assembly drawing: Will open the assembly/mark drawing of the selected assembly

Generate as a STEP file: STEP file is a CAD file format, usually used to share 3D models between users with different CAD systems. Note! This command will only work in BricsCAD and when the Communicator module is installed and licensed. *For more information, refer to* Settings for Workshop Drawings > Generate STEP Files

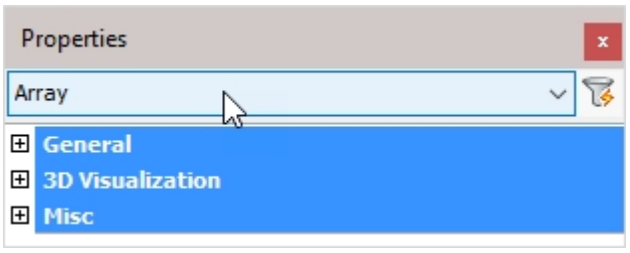
Array

The 'Array' object type is used for creating and updating arrays of macros. It has the following headings:

General

3D Visualization

Misc

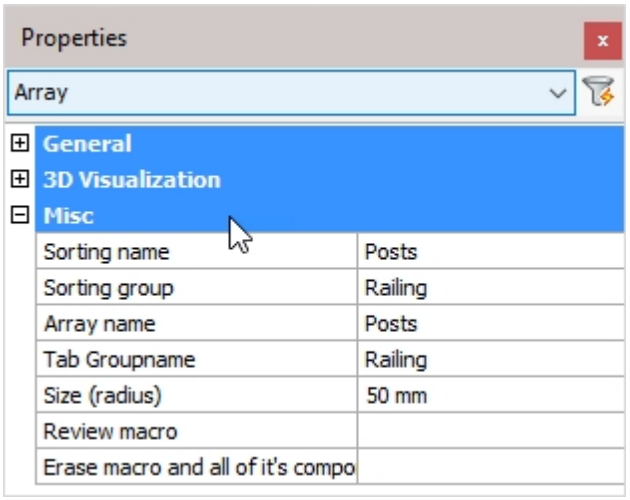


General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Misc



The name of the macro – this may be edited and will reflect in the macro dialog box

Group name:

Sorting name: The macro name to which the array should belong to in the macro edit dialog box

Sorting group: The group name to which the array should belong to in the macro edit dialog box

Array name: The name of this array

Tab Groupname: The name of the tab to which this array should be assigned to in the macro edit dialog box

Size (radius): The size of the triangle shown in the 3D model

Review macro: Will open the macro edit dialog, where the macro may be viewed and/or edited

Erase macro and all of its components: Will erase the array, all of its subject macros and all of the parts owned by these macros

View

The View object type is used in workshop drawings and on GA plans for showing a 2D representation of the 3D model or a part of it.
A view is a collection of 2D lines that are connected to their 3D model counterparts.
The view has the following headings :

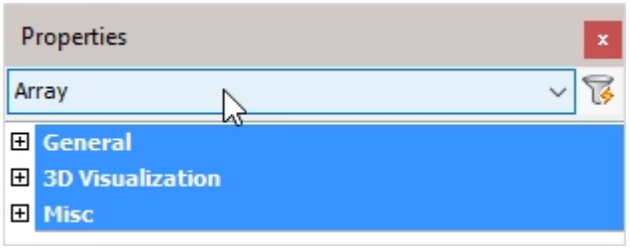
General

3D Visualization

Misc

Model appearance

Camera options



General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Misc

Misc	
Name	
Name for annotation	World (WCS)
View type	2D Plan
Assembly type	GA view
Added scale	1
Unfold	

Name - The name view for description purpose

Name for annotation - The short name of the view, as it will be shown in annotations

View type - The main purpose of this type is for Parabuild so that it know whether this is a side view, an endplate view or something else

Assembly type - This relates more to the type of sheet on which the view is located (plate, member, assembly or GA)

Added scale - This scaling property is used to enlarge or shrink the view relative to the sheet scale. The sheet scale is always 0. Setting this to smaller than 1 will shrink the view. Setting it to larger than 1 will enlarge the view.
Dimensions and annotations that are connected to the view will be adapted but not scaled.

Unfold - Clicking on this button will generate an unfold of the part that is displayed by the view. This button will only work on supported plates. It is supposed to be used on the views of workshop drawings.
This option currently only works when Parabuild is run inside BricsCAD Mechanical.

Model appearance

Model appearance	
Visible holes	☑
Invisible holes	☐
Perpendicular holes	☐
Invisible model lines	☐
Workshop bolts	☑
Site bolts	☑
Draw Axis for bolts	Only on views without
Grid lines	☑
Lines	☑
Polylines	☑
Minimum size for lines	399
Shortening lines	☑
3D Solids	☑
Profile rounding	☐
Always show profile axes	☐
Profile axes only for cente	☐
Profile axes only for roun	☑
Hatches	☑
Modify layer states of vie	
Modify generation setting	
Refresh views (F5)	...

Practically all of the properties in this section are already explained in the topic.

But these 3 properties do need some more explanation **:**

Modify layer states of view - This button will open up a dialog where you can disable or enable layers. The 3D objects that are on the enabled layers will be shown in the view. All the other 3D objects will be ignored.

Modify generation settings - This is a link to the Modify view Settings dialog of this view

Refresh views - This will refresh this view only. See note.

Note!

After applying a change to any of the properties in the Model appearance section, you will need to refresh the view in order to see the changes on the view.

You can refresh views with any of these 3 methods :

- The function key <F5> will refresh all the sheets that are currently visible on your screen
- The refresh views button of the view itself
- You can right-click on a sheet in the sheets manager to refresh all the views on the sheet

Camera options

These properties refer actually to the camera that is linked to this view.

All of these properties are already explained in the camera topic.

Camera

The camera is used as the driver for views.
It is always located in the 3D model, and determines :

- The viewing direction in 3D
- View clippings
- Additional filters on 3D model selection

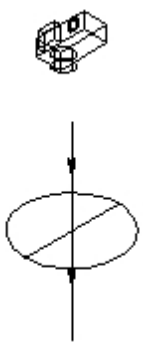
Misc	
Name	World (WCS)
Camera type	GA view
Clipping type	Front and Back view li
View limitation front:	500 mm
View limitation back:	500 mm
Box width	0 mm
Box length	1000 mm
Box height	0 mm
Included parts	0 Parts
Excluded parts	0 Parts
Switch orientation	
Take over from current vi	

The camera has the following properties :

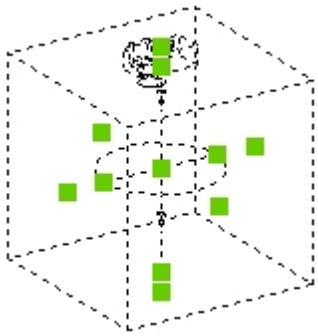
Name - The name of the camera, as it will be shown in the sheets manager

Camera type - This relates more to the type of sheet on which the view is located (plate, member, assembly or GA)

Clipping type - Clipping is used to filter out unwanted 3D models from the view, as they would obstruct the view on the subject models that we want to show on the view.



A camera with Front/Back limitation



A camera with Box limitation

View limitation front/back - This will only work the *Front/Back* clipping type is enabled.
Front limitation is measured from the circle of the camera (see illustration) to the camera symbol.
Back limitation is measured from the circle to the opposite side.

Box Width/Length/Height - This will only work the *Box* clipping type is enabled.
The grip points can be used to stretch the box. The grip points are shown when you select the camera in 3D (they are shown as green squares in the illustration).

Included parts - With this button you can force the view and camera to only show a certain selection set of 3D models on the view.

Excluded parts - With this button you can force the view and camera to never show a certain selection set of 3D models on the view.

Switch orientation - This button will switch the orientation of the camera in 3D.

Take over from current view - This button will rotate the camera as such that it matches the

current 3D view you are currently using.

Note!

After applying a change to any of the properties in the camera, you will need to refresh the view in order to see the changes on the view.

You can refresh views with any of these 3 methods :

- The function key <F5> will refresh the visible views on a sheet
- The view has a refresh property button
- You can right-click on a sheet in the sheets manager to refresh all the views on the sheet

Annotation

The 'Annotation' object type has the following headings:

General

3D Visualization

Misc

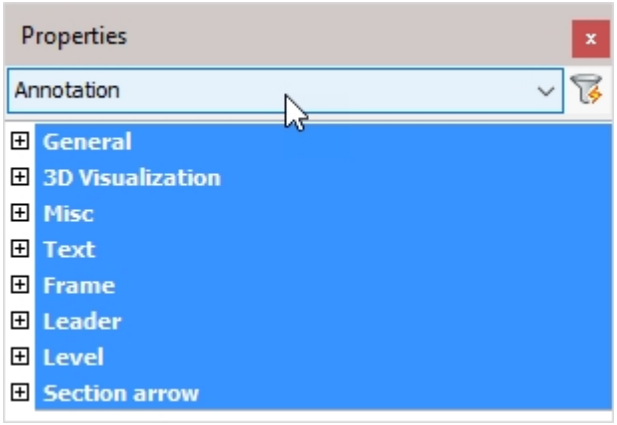
Text

Frame

Leader

Level

Section Arrow



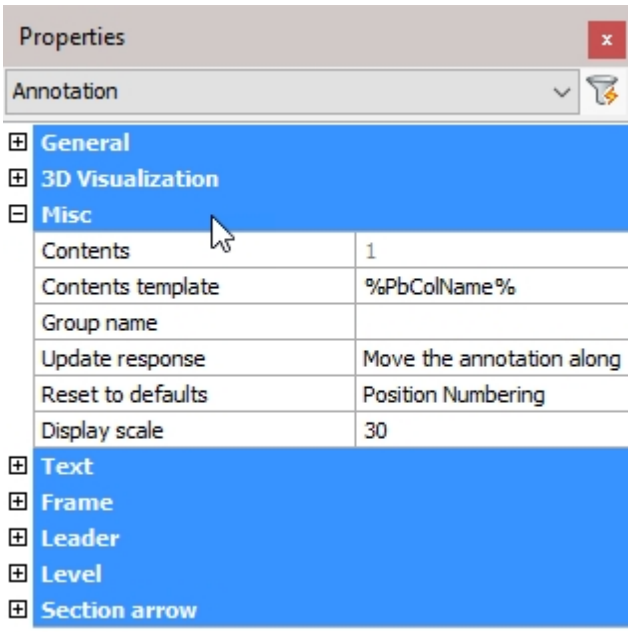
General: Is common to all object types, and covers general AutoCAD and BricsCAD settings, including: Color / Layer / Linetype / Linetype scale / Plot style / Lineweight / Transparency / Hyperlink / Handle /

3D Visualization: Is also common to all object types and includes: Material.

The material that is referred to here is only the visual representation material. For changing the

actual material of parts, see the *Additional Data* section of plates, profiles or structures.

Misc



Note! - * *indicates that the given value is a resultant of the sum of other values in the panel - and cannot be individually edited.*

Contents: * Displays the text string

Contents template: determines the contents template

Group name: Add a group name

Update response: Options include:

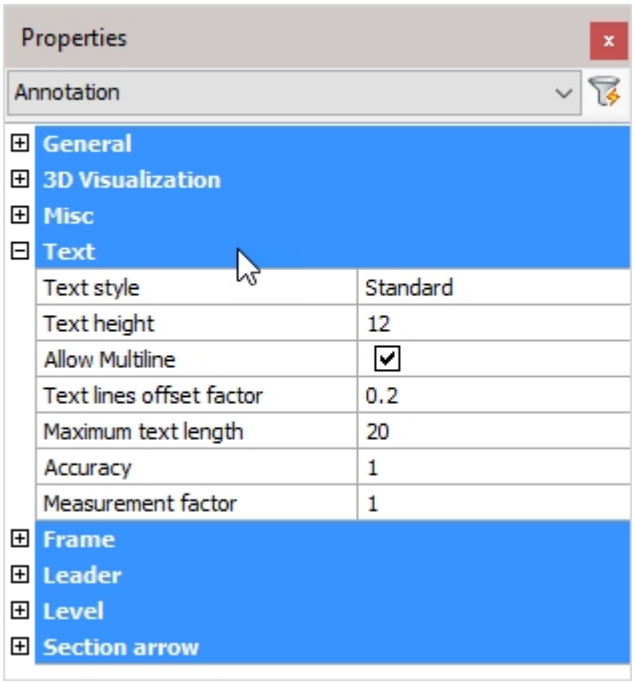
Move the annotation along

Stretch the arrow

Reset to defaults: Press the button to open the 'Reset settings to the default values' dialog, and choose the annotation style you want to reset from the drop-down menu

Display scale: Set the annotation scale

Text



Text style: Set the text style

Text height: Set the text height

Allow Multi line: Activate the checkbox to allow multi line text

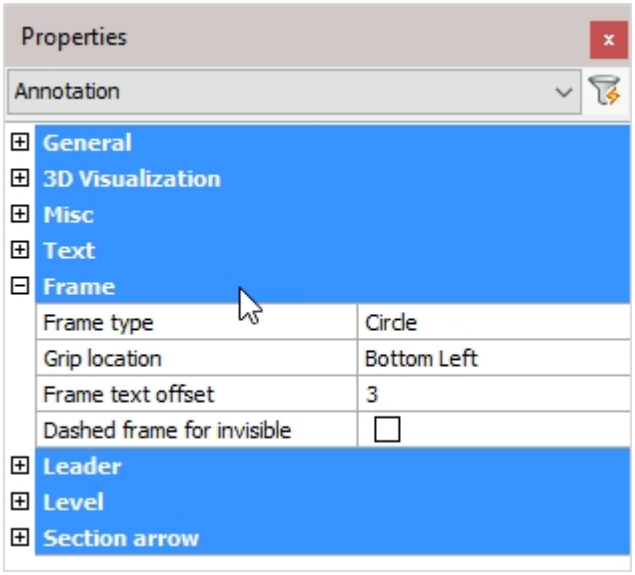
Text lines offset factor: Change the masking around the text – the value given is a percentage of the text height

Maximum text length: Set the maximum text length

Accuracy: Set the numerical text accuracy factor

Measurement factor: Set the numerical text measurement factor

Frame (If applicable)



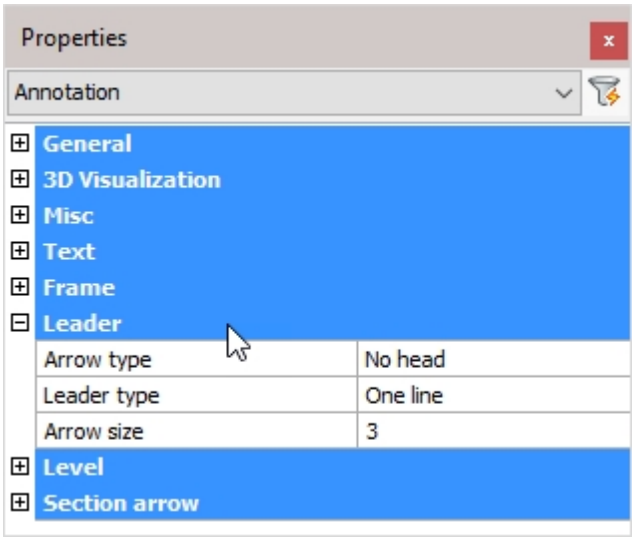
Frame type: Select the frame type from the drop-down menu

Grip location: Select the grip location from the drop-down menu

Frame text offset: Reset the offset (Padding) between the text and frame (mm)

Dashed frame for invisible: Activating the checkbox will make the frame invisible

Leader (If applicable)

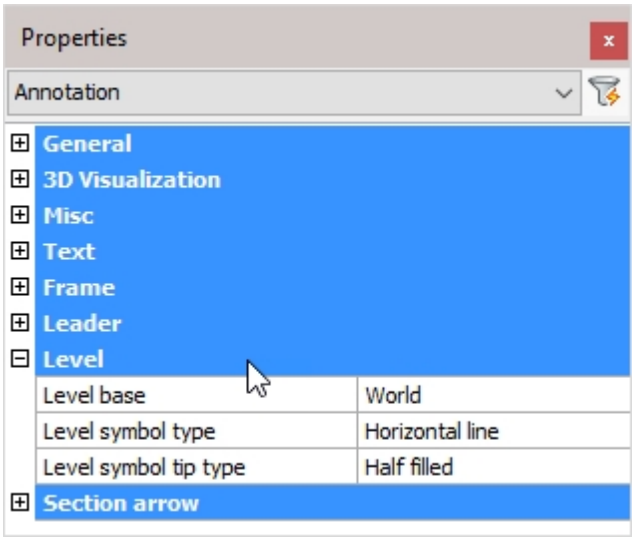


Arrow type: Select the Arrow type from the drop-down menu

Leader type: Select the Leader type from the drop-down menu

Arrow size: Set the arrow size (mm)

Level

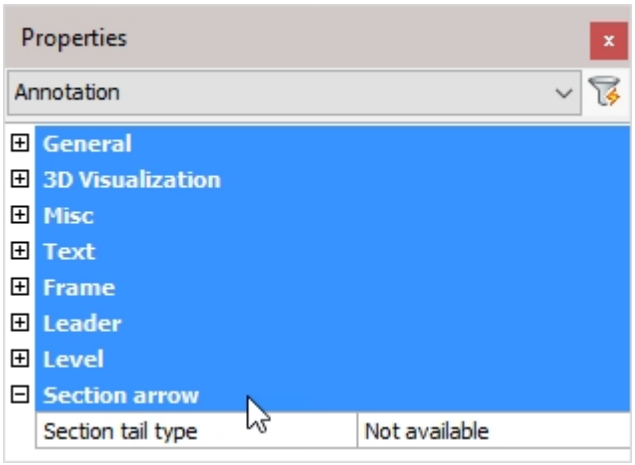


Level base: Enter the base level (datum) – all levels are measured from this value

Level symbol type: Select Level symbol type from the drop-down menu

Level symbol tip type: Select Level symbol tip type from the drop-down menu

Section arrow



Section tail type: Select Tail type from the drop-down menu

Main - Library

This chapter covers the commands in the Main Library toolbar, and includes:

- Profile Libraries
  - Editing the Profile Libraries
- Custom Shapes
- Elements Library

Profile Libraries

These are all the different section shapes that can be used to draw profiles :

I - Profiles

C - Profiles (Channels)

L - Profiles (Angles)

T - Profiles (Tees)

Built-Up Profiles (Plate Girders)

Hollow Sections (Circular, Square, Rectangular)

Flats / Bars / Rounds

Cold Formed Profiles(Purlins and Side-Rails)

Floors (Plate and Grating)

Others

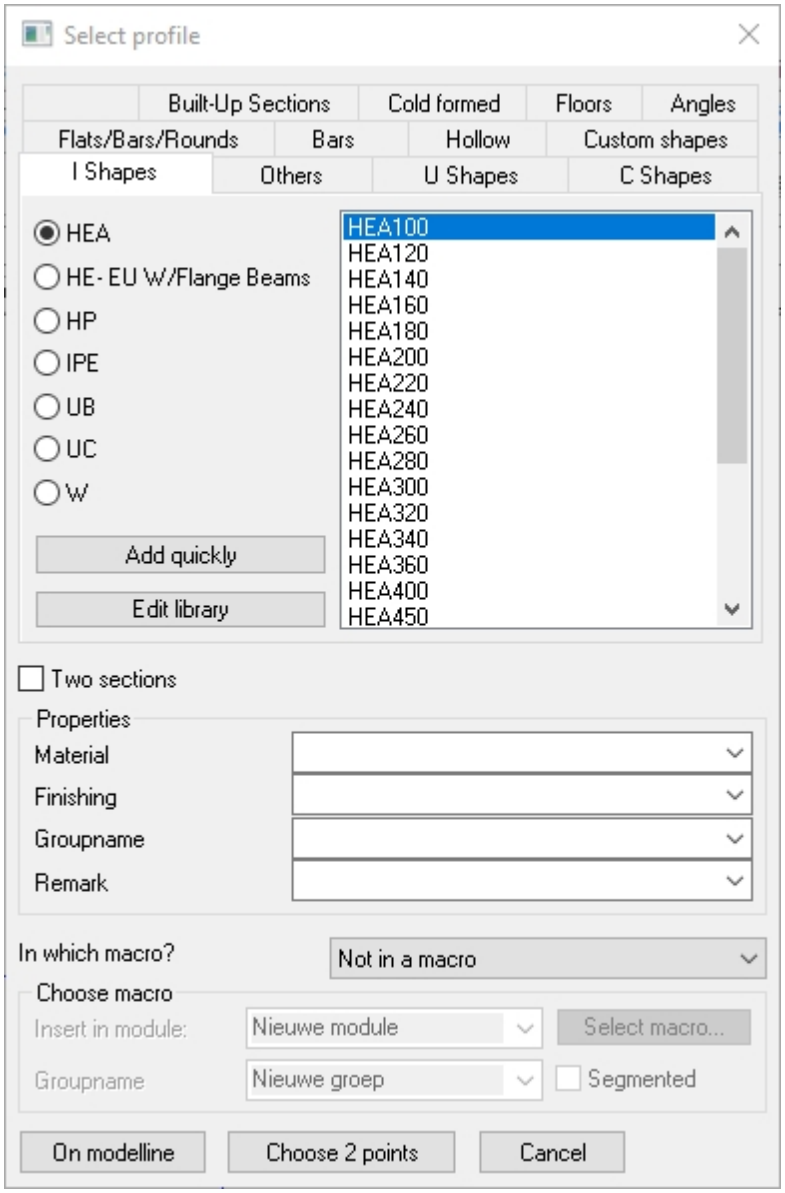
Custom Shapes - User Defined

Elements Library

Other Parts

Select Profile

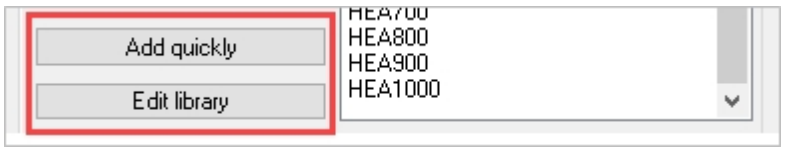
Selecting any of the above will bring up the following **Select profile** dialog.



The top of this dialog lists the profile **Groups**, which correspond with the icon bar.

The left side column lists the **Types** contained within the specified group - the radio button, when activated, will display the **Value tables** of the various profiles of the specified type.

There are options to **add** to, or **edit** the library. See Editing the Profile Library

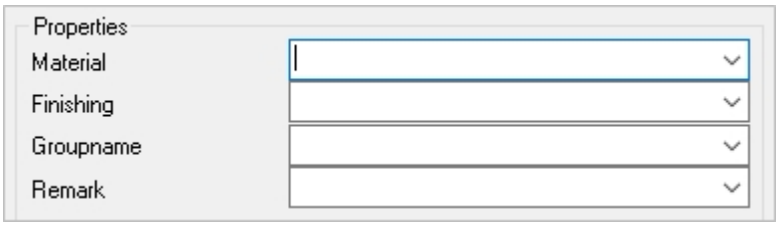


Morph 2 Sections

With the Two sections checkbox checked, you may Morph 2 Profiles

Profile Properties

Below the profile section you may enter 4 properties that the new profiles will receive. To avoid typing these values each time, they may be entered as standards by using the Parabuild settings > Global tab

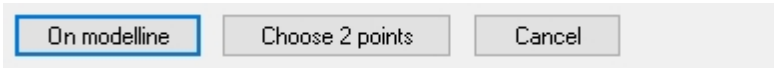


Add new profiles to a macro

These settings are relevant if you wish to add the new profiles to a macro. This is useful if you wish that the profile makes itself dependent 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.

Draw Profile

Profiles may be drawn on a model line or freehand - See Drawing Profiles

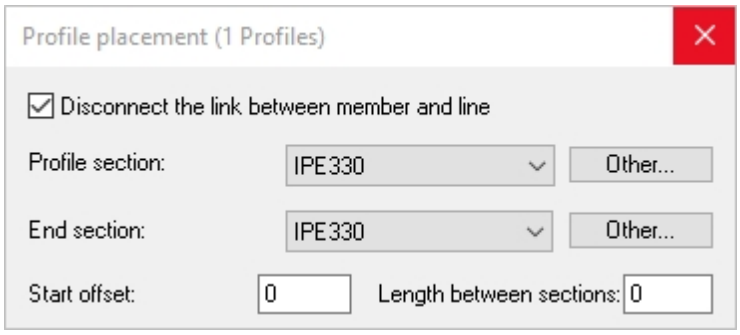


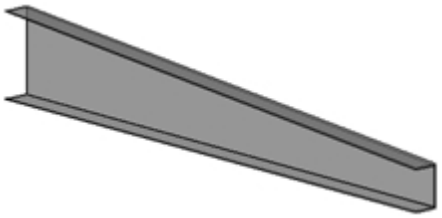
Morph 2 Profiles

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

You begin by drawing the profile the normal way - once the selected profile has been drawn, the Profile placement dialog will re-appear enabling you to enter the second profile - the morphed profile will be drawn.

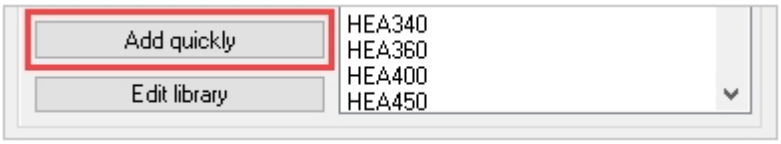
A prerequisite for this function is that the two sections have the same number of segments. Otherwise, a logical sequence cannot be created; The transition is being made from segment to segment (a segment is a section of straight line or curve section of a PolyLine)



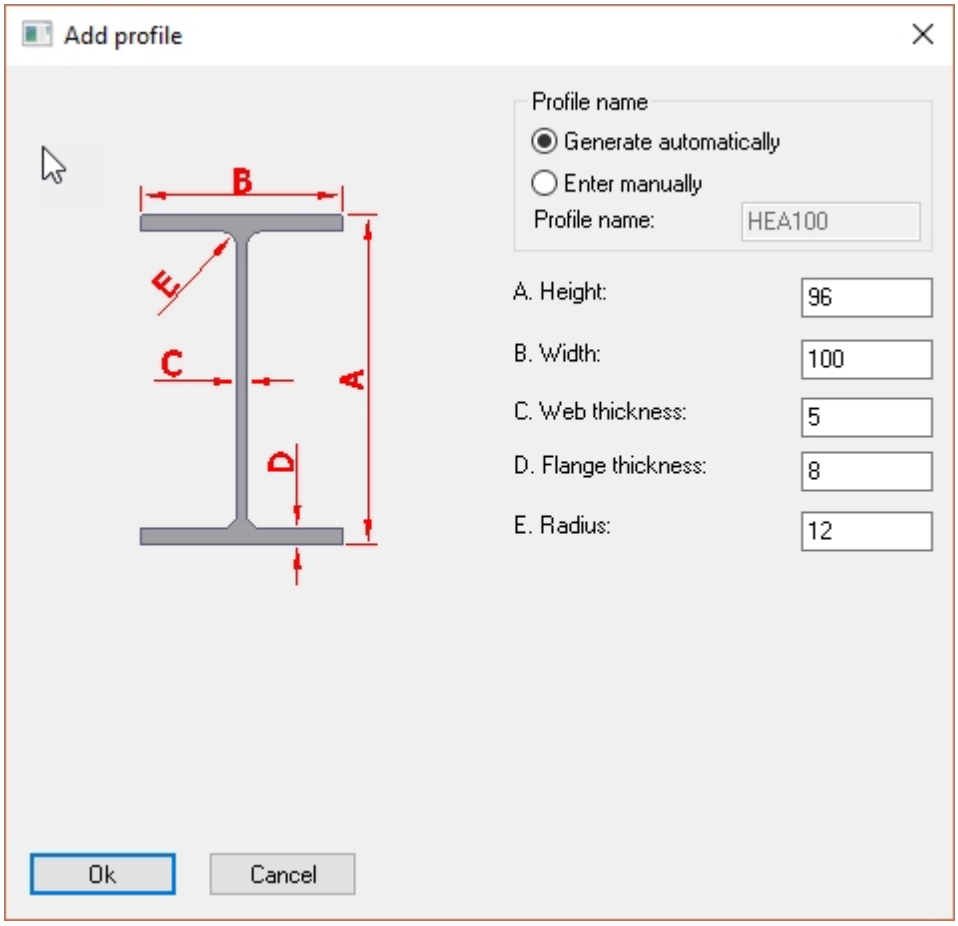


Editing the Profile Library

Add a Profile to the Profile Library



The **Add quickly** button will add a new profile to the active Value Table.



Profile name

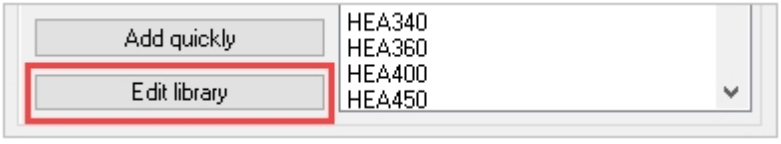
The options are to **Generate automatically** or **Enter manually**.

Selecting 'Generate automatically' will automatically add the Type to the profile i.e. IPE. In this mode the Profile name window will be grayed out.

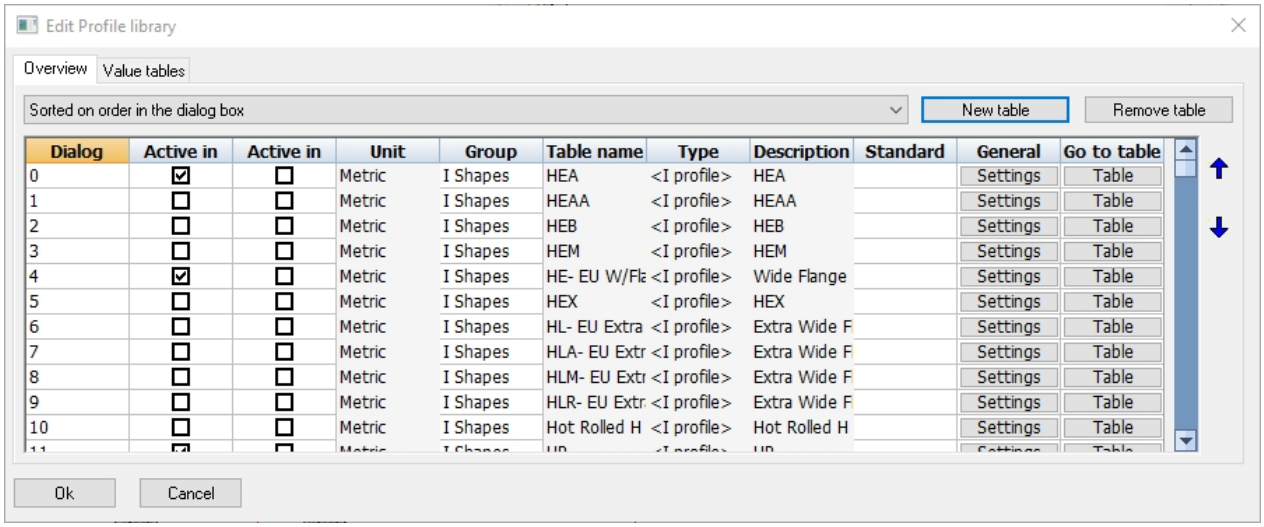
Selecting 'Enter manually' will allow the user to change the profile name manually.

By adding the relevant dimensions the new profile will be added to the active Value table.

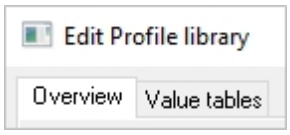
Edit the Profile Library



Selecting this tool will bring up the *Edit profile library* dialog. At the top left of the dialog there are 2 tabs: *Overview* and *Value tables*



Overview tab

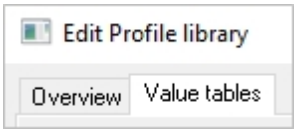


Selecting the *Overview* tab will show a table, wherein each row represents a profile table.

The purpose of each columns are explained below:

1. **Dialog priority** - which establishes the order of the profile table in the Select profile dialog box. The lower the is value, the higher it will be listed in the *Select profile* dialog box.
2. **Active in Metric** - If you are working on a Metric 3D drawing and this checkbox is activated, then this profile table which will become visible in the *Select Profile* dialog.
3. **Active in Imperial** - If you are working on a Imperial 3D drawing and this checkbox is activated, then this profile table which will become visible in the *Select Profile* dialog.
4. **Units** - Indicates whether the contents of the profiles table is in millimeters or in inches
5. **Group name** - Indicates which Group the row belongs to. Each group name represents a tab in the *Select profile* dialog box
6. **Table name** - The table filename this profile group is linked to. You can find this filename in the folder : \Parabuild\Pb_Lib\Prof\
7. **Type** - Refers to the section type that the profile table is using
8. **Description** - Provides a description of the profile. This description is the name of the table as it is shown in the *Select profile* dialog box.
9. **Standard** - Describes the international standards for the selected profile table
10. **General settings** - Will bring up the *Section Table Options* dialog, whereby the user may modify general settings of the profile table.
11. **Go to Table** - will take the user to the *Value tables* tab dialog, where all the contents and options of the table can be modified. See below.

Value tables tab



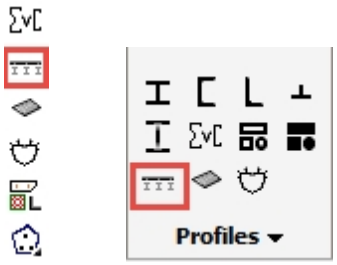
The Value tables will vary according to the profile Type, however, the first 4 columns are consistent for all Types:

1. **File name** - is the file name for the table which has the suffix .pr which is stored in the folder : c:\parabuild\Pb_Lib\Prof\
2. **Sysname** - The system name of the profile that Parabuild needs internally for the unique recognition of the profile. The value must not be one 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 usually 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.
3. **I/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.
4. **Name** - 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.

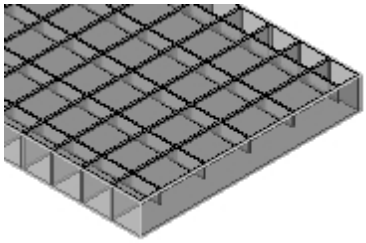
Floors

Command - **Prb_CreateProfFloors**

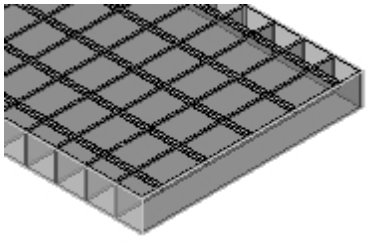


Activating this command will open the Select profile dialog - where you will be prompted to select a flooring profile.

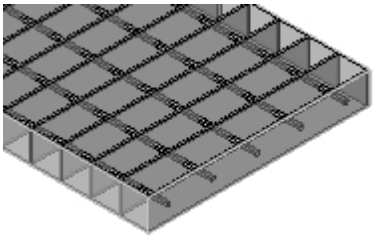
Floor types are divided into 4 groups:



Press Lock Bar Grating

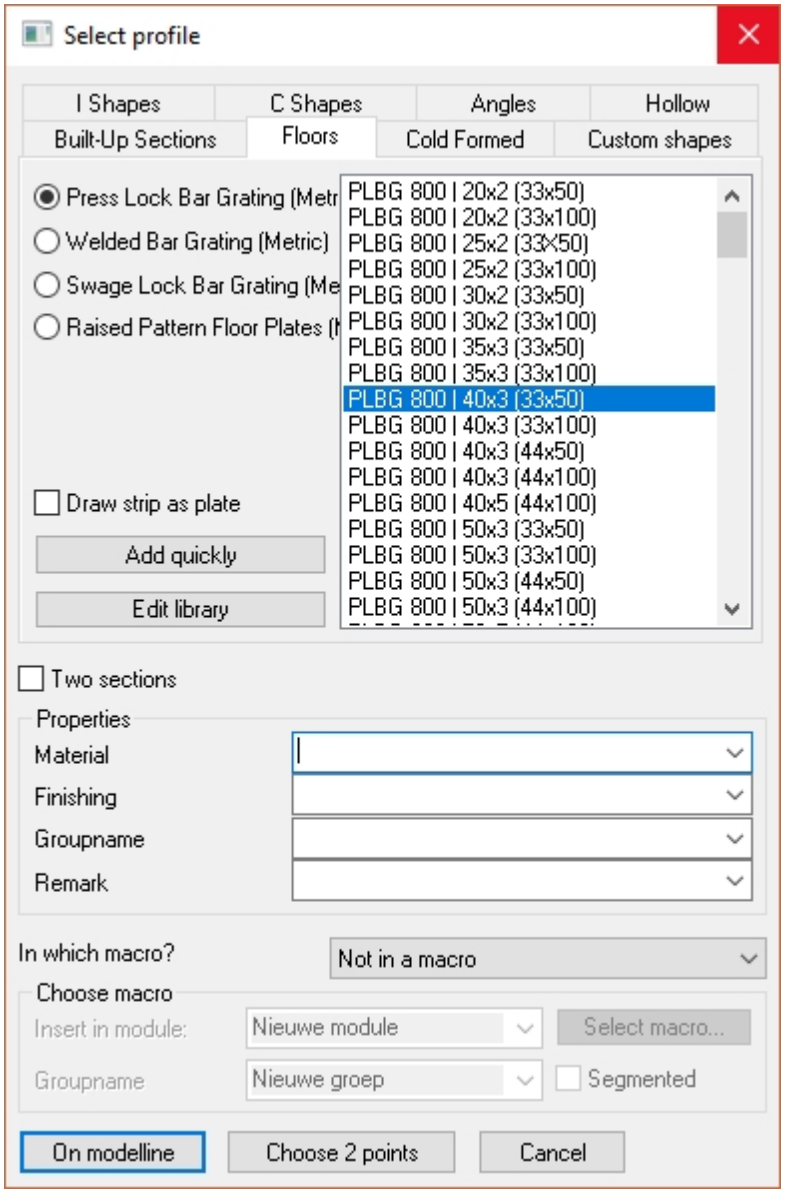


Welded Bar Grating



Swage Lock bar Grating

Raised Pattern Flo Plates



From this dialog you may select the required flooring type by selecting the appropriate radio button - the column to the right will display the value tables applicable to the selected floor type.

The value table will display the floor panel characteristics which may be interpreted thus:

PLBG 800 | 40x3 (33x50)

PLBG = Floor panel type - in this instance, Press Lock Bar Grating

800 = Standard panel width

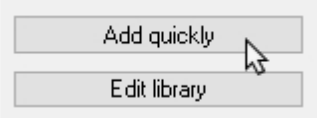
40x3 = The bearing bar size

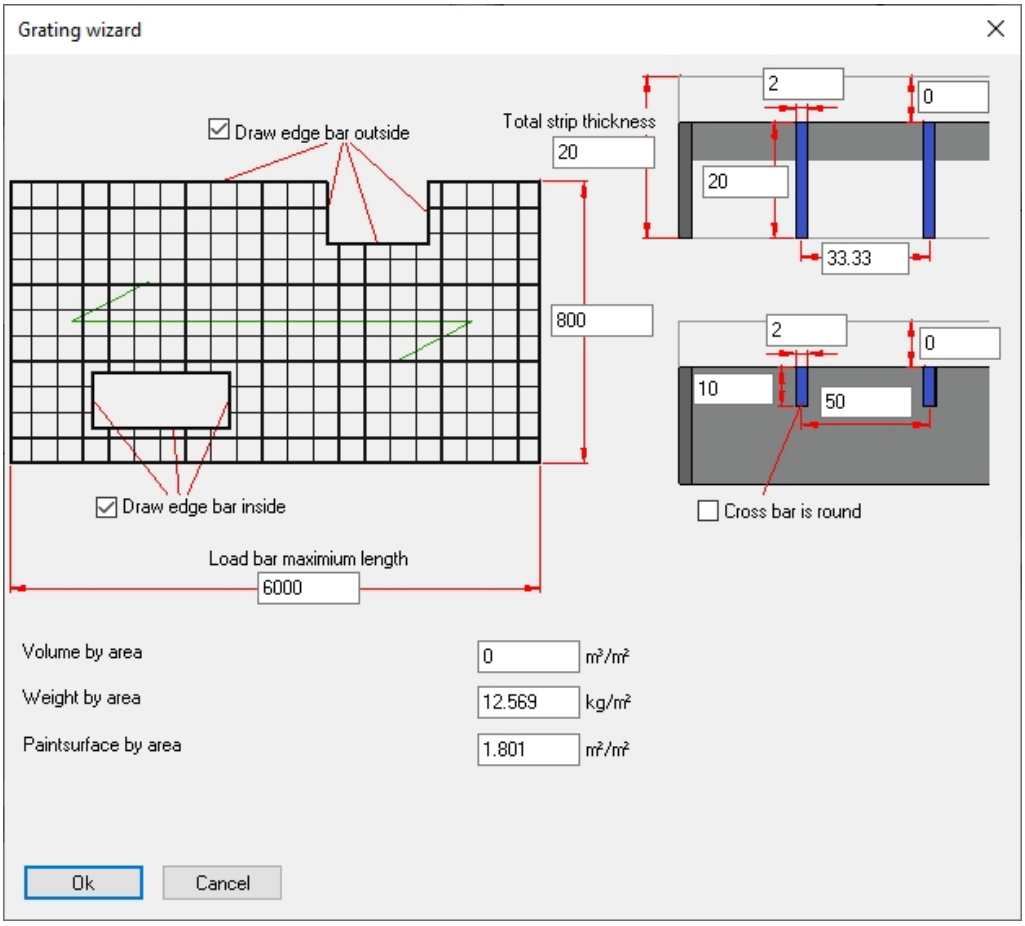
(33x50) = the grating opening size. 33 = the bearing bar centers, while the 50 = the transversal bar spacing

Draw Strip as plate - activating this checkbox will draw the floor panel as a plate (see Convert strip / plate)

Adding and Editing floor panels in the library

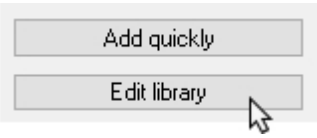
Custom Floor panels may be added to the library by clicking the **Add quickly** button in the **Select Profile** dialog which will open the **Add profile** dialog





This function will add a new profile to the active Value Table.

Edit Library



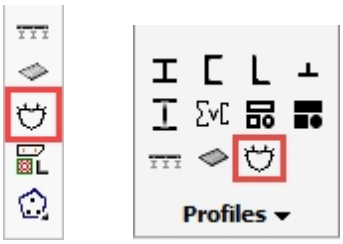
This function will open the Edit profile library dialog where the properties of specific floor panels may be edited.

For more information on editing the library - in general terms, go to Editing the profile library

The remaining settings are explained under Profile Libraries

Custom Shapes

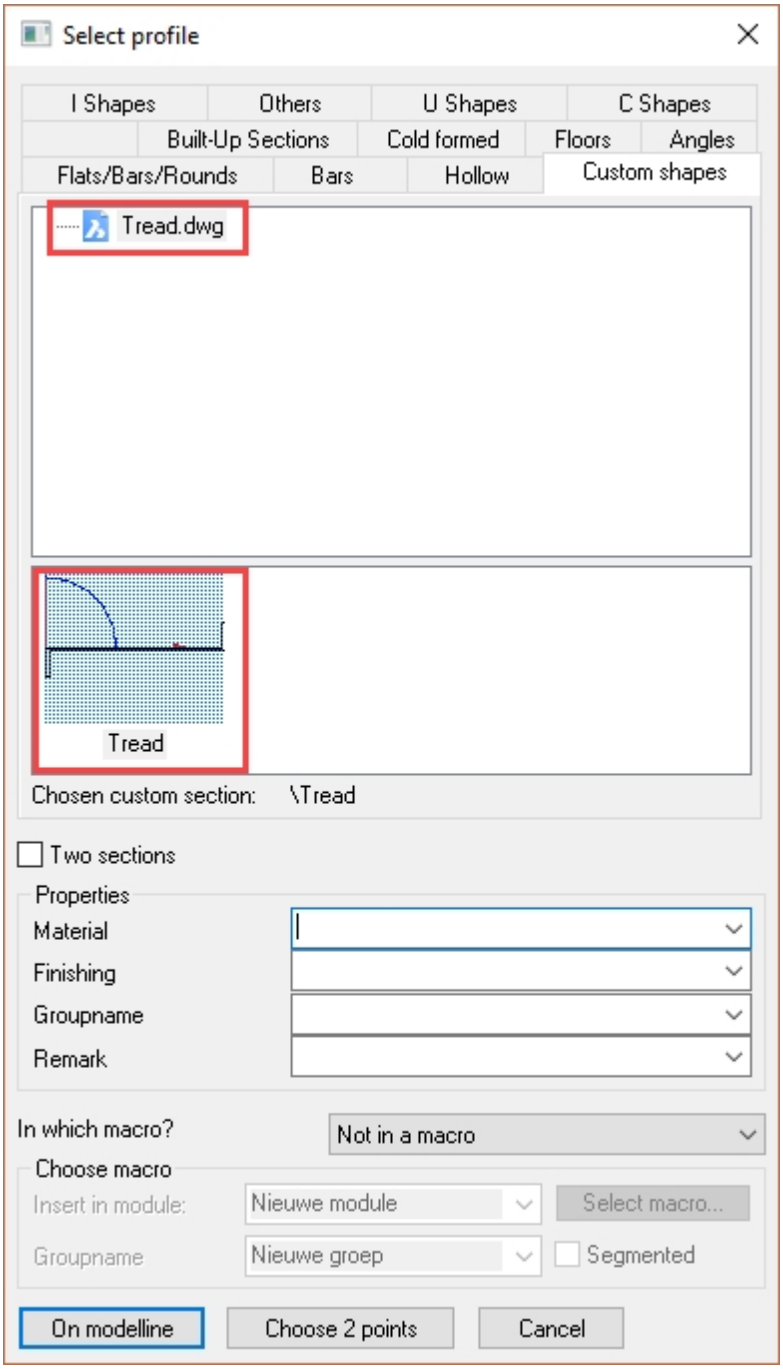
Command - **Prb_CustomProf**

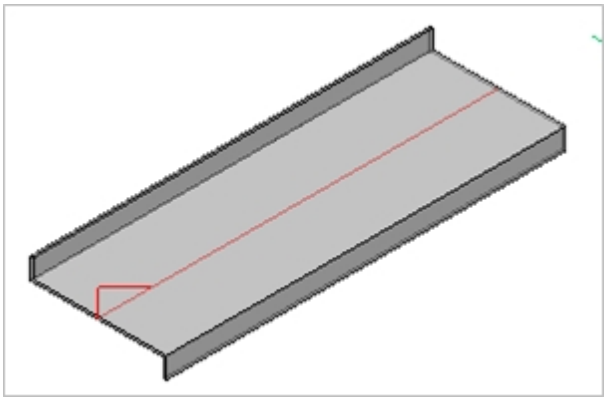


Custom shapes are user-defined shapes which are not contained within the standard profiles library.

Activating this command will bring up the **Select profile** dialog with the Custom shapes tab open.

Select the profile by selecting the title or the icon - the profile is drawn in the same manner as all profiles.





New custom shapes can be added using the [Create new custom shape](#) command.

Elements Library

This system is similar to Custom shapes

There is a command to add new 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 : **PrB_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 new library element and click on **Ok**.

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 the current drawing

Command : **PrB_ElementLib**

To use this tool, simply 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 topic.

Other Parts

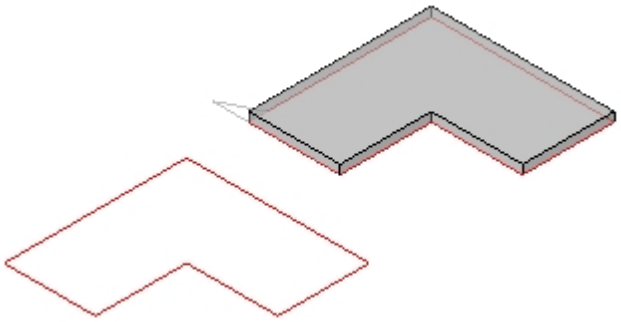
This chapter covers the commands for creating and editing custom parts, and includes the following Sub-Chapters:

- Plate with PolyLine
- Lowering (Splitting) a member
- Sandwich sections
- Miscellaneous sections
- Add to elements library
- Add to custom shapes library

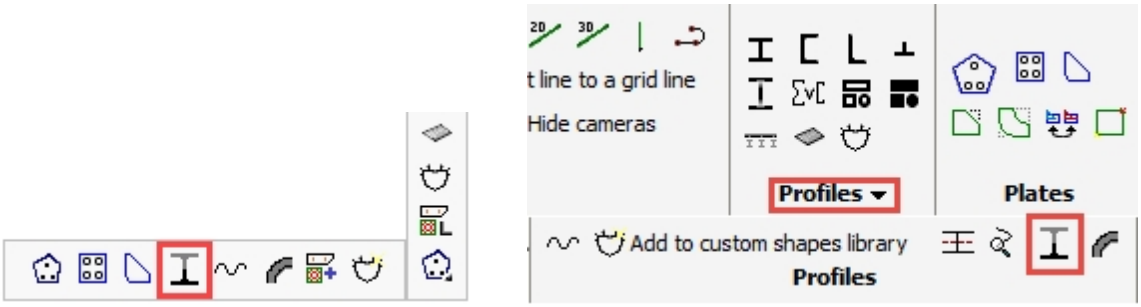
Plate with PolyLine

The PolyLine must at least have 3 vertices, i.e. a Square, Rectangle, Triangle. An exception to this is a circle, which is also allowed.

1. At the command prompt, indicate the PolyLine shape and press <**Enter**>
2. At the command line prompt enter the plate thickness, press <**Enter**>

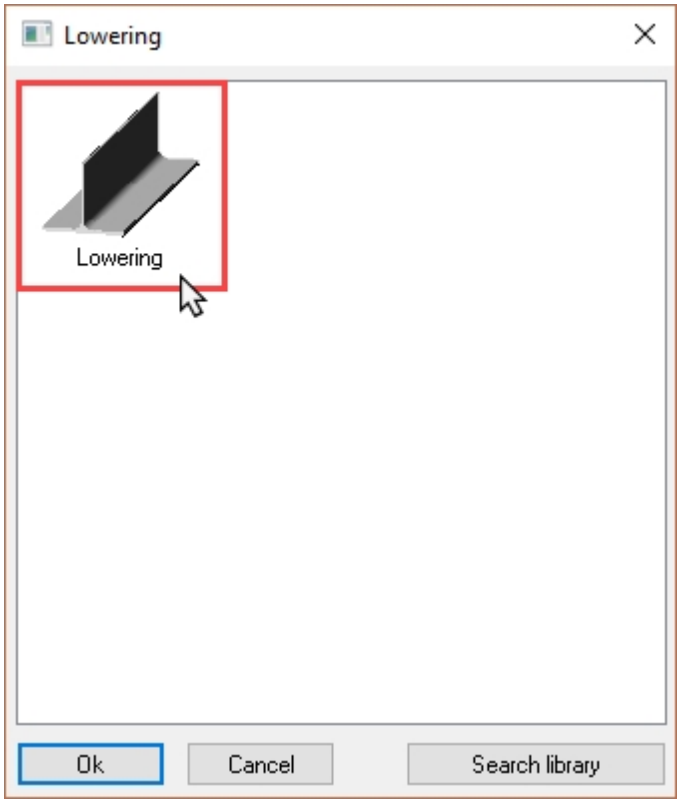


Lowering a Member



This command will create a Tee section from a standard profile.

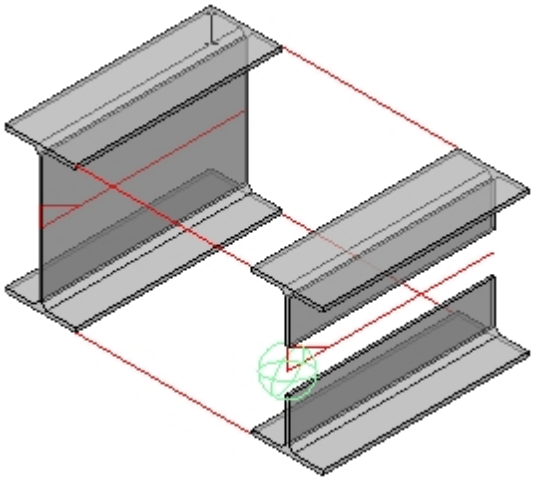
Starting this command will bring up the Lowering dialog where the appropriate icon should be selected

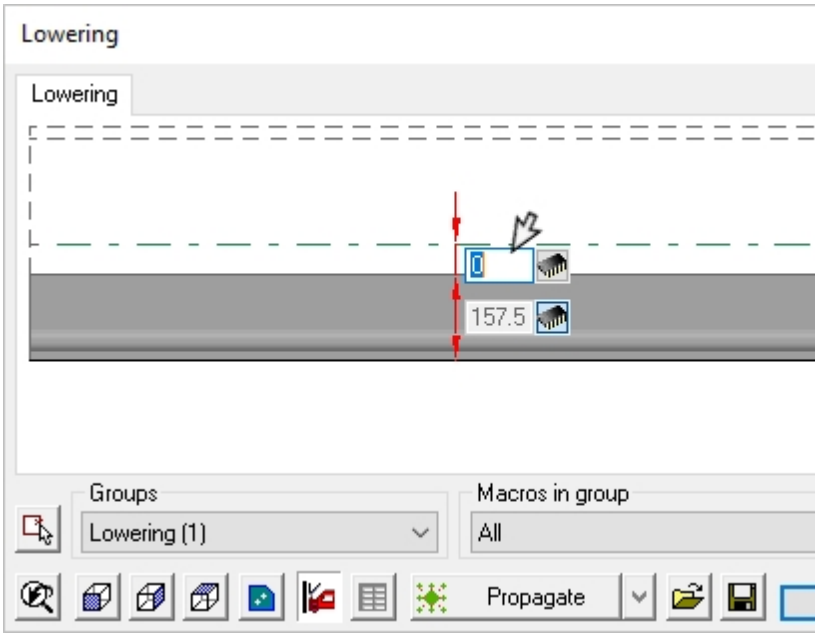


Selecting the profile top-flange will trim the lower half of the profile, while selecting the bottom flange will trim the upper half.

By default, the profile will be trimmed at the vertical center on the profile. By editing the zero value on the customization dialog the profile web will be modified accordingly.

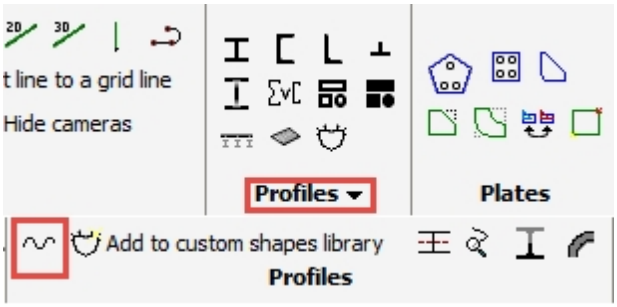
A plus value will shorten the web, while a negative value will lengthen the web





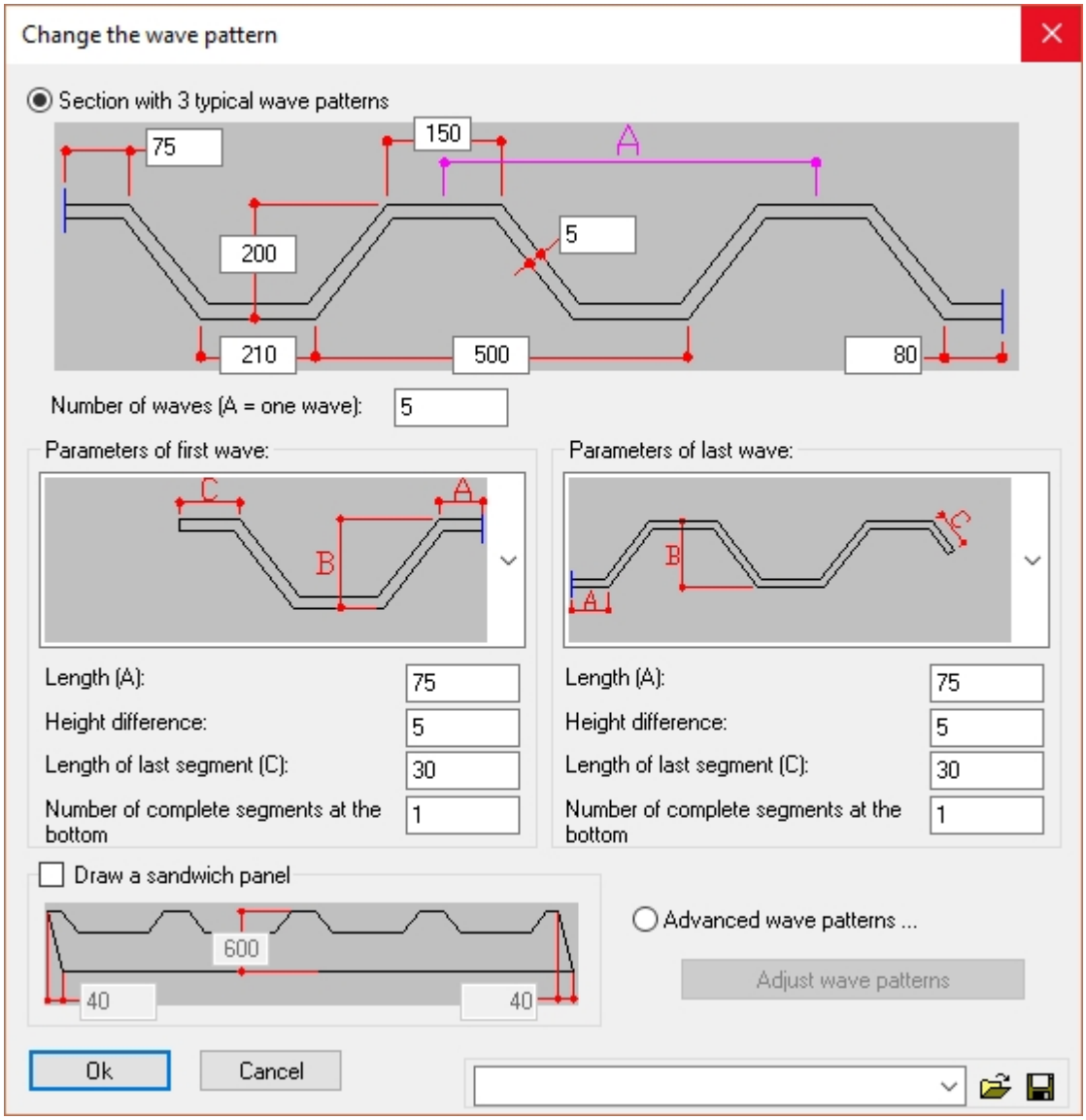
Sandwich Sections

Command - **Prb_Wave**



This command will draw the section of Sandwich 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**.

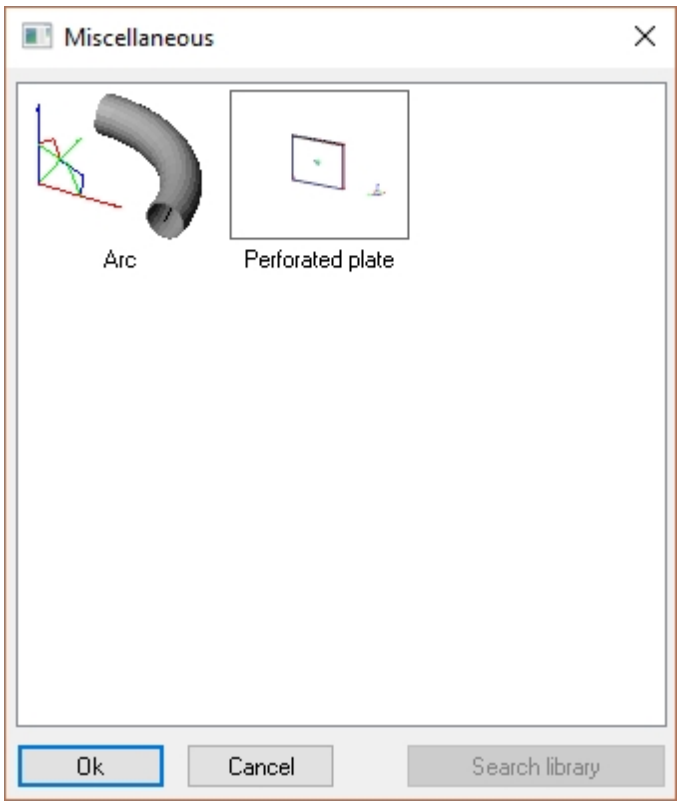


Miscellaneous

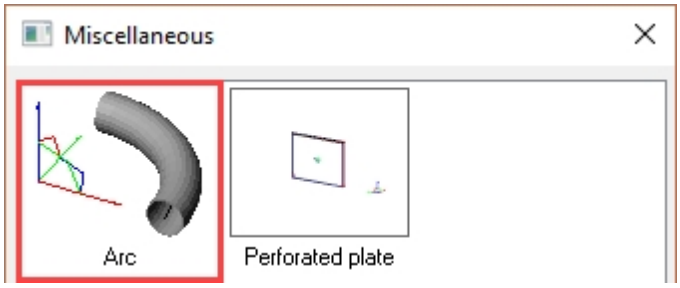
Command - **(Prb_MacroGroup "Misc")**

This command offers 2 options:

- Arc
- Perforated Plate

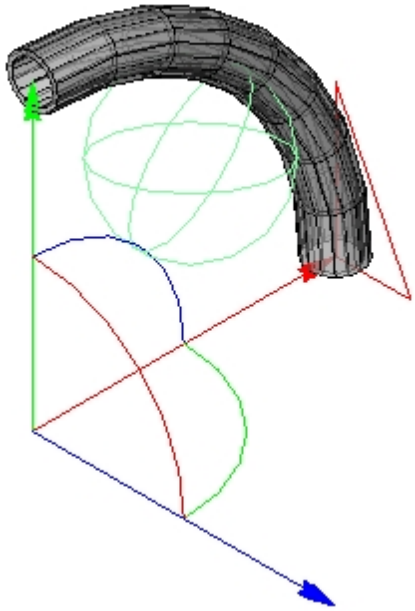


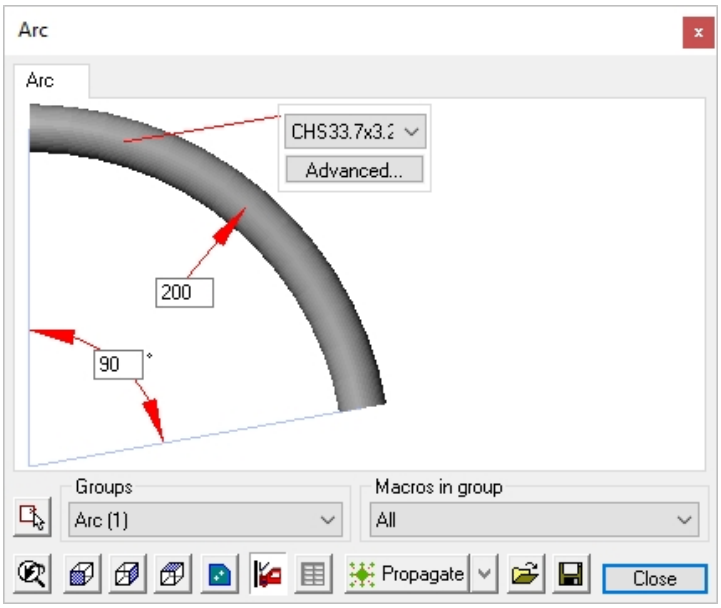
Arc



This command will 'bend' any profile or section through a prescribed radius and angle

On selecting the 'Arc' icon, you will be prompted to 'Select the point that will determine the placement'. this point will be the center-line intersection of the radius. After selection, the default profile will be drawn with the following customization dialog:



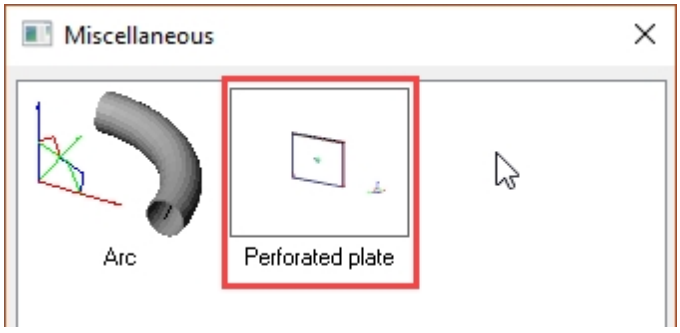


Here you may modify the profile size - or a completely new profile may be selected using the 'Advanced' tab

You may also modify the bend radius and segment angle.

The changes made in the customization dialog will be immediately be shown on screen.

Perforated Plate

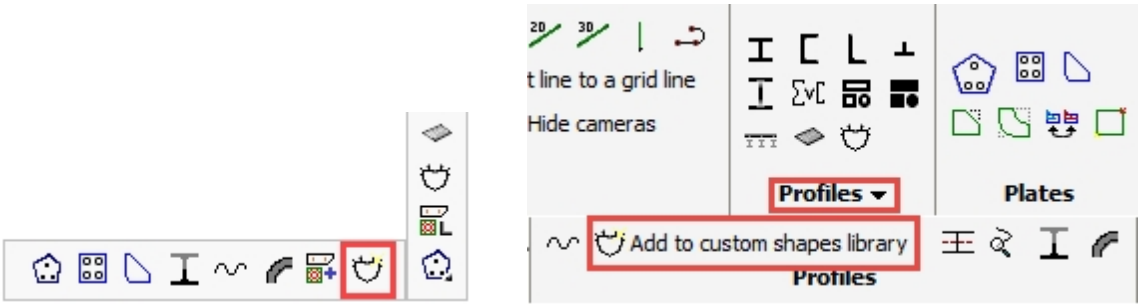


Add to Elements Library

For more information, see the Elements Library topic.

Add to Custom Shapes Library

Command - **Prb_NewCustomProf**



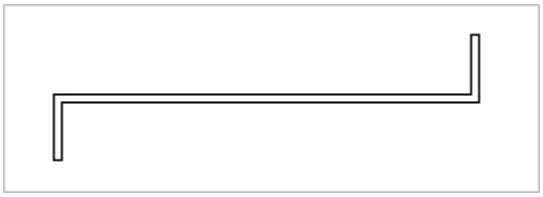
Customized sections are sections the dimensions of which can be determined by the user.

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 shape

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

For this example we have drawn a simple solid stair tread-plate



- 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 customized section
- On the left-hand side, select the name for the new customized profile - press <**Enter**>

  This will store the profile in the folder **c:\Parabuild\Pb_Lib\User_Sections**

  If folders are to be created or deleted, use Windows Explorer and go to the directory **c:\Parabuild\Pb_Lib\User Sections\...**

  (if c:\Parabuild 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.

The name of the custom shape that you enter here will be used in the library and as the profile name for all Parabuild functions (part lists, workshop drawings...) when the drawing of the profile is completed.

Main - 3D Modeling

This chapter covers the commands for 3D Modeling, and includes the following Sub-Chapters:

Draw Profiles

There are 3 ways to draw profiles:

1. Using Model-lines
2. Selecting 2 points
3. Using the Context Modeler

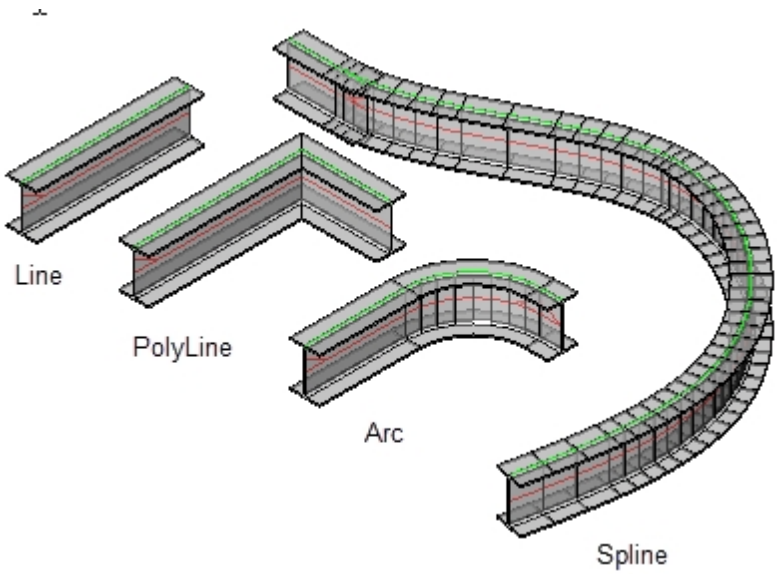
Using Model Lines

Use one of these icons to open the *Select profile* dialog.

And in the *Select Profile* dialog, select **On model line**

You can place profiles on the following line types:

- Line
- Arc
- 2D PolyLine
- 3D PolyLine
- Spline
- Parabuild helix - Command : PrB_Helix
- Parabuild 3D polyline, which has a bit more flexibility with regards to arcs in 3D than the AutoCAD/BricsCAD 3D polyline. Command : PrB_3dPoly



At the command line prompt, indicate the model line that the profile should follow.

Note that PolyLines, Arcs, and Splines should be 'open-ended'. This command will not work with circles or rectangles. Also the model lines should never be self-intersecting.

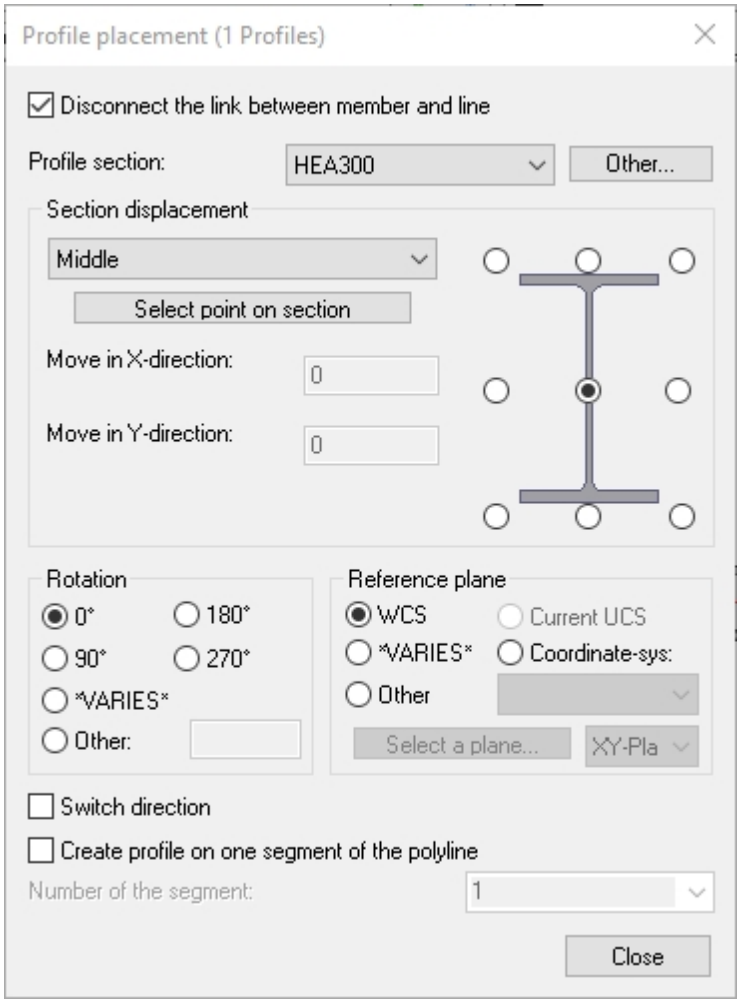
Drawing Profiles - Using 2 Points

The user is prompted for the first and second points on the baseline, after which the above dialog and options are displayed.

The points may be placed anywhere on the drawing.

After the profile is drawn, the following dialog will appear, offering the following options:

Profile placement



- **Disconnect the link between the member and line -** With the checkbox Unchecked, the profile will be constrained to the line. That is to say, the member cannot be moved independently of the line. See Constrained Profiles

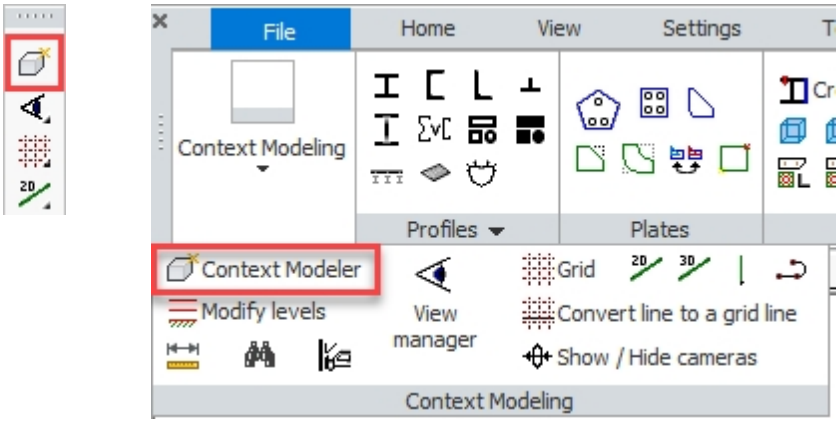
With the checkbox activated, the member may be moved independently of the line.

- With the **Profile section** DropDown menu, the user is able to change the profile section - while selecting **Other**, the user is able to select a profile from another group.
- The **Section displacement** enables the user to select the appropriate displacement point on the member, this may be done by activating the appropriate radio button, or selecting an option from the drop-down menu.
- It is possible to match the rotation of the member to another part by manually selecting a reference plane
- With the checkbox **'Switch direction'** activated, the user is able to switch the direction of the member on its axis.
- With the checkbox '**Create profile on one segment of the PolyLine**, the user is able to select

the applicable segment.

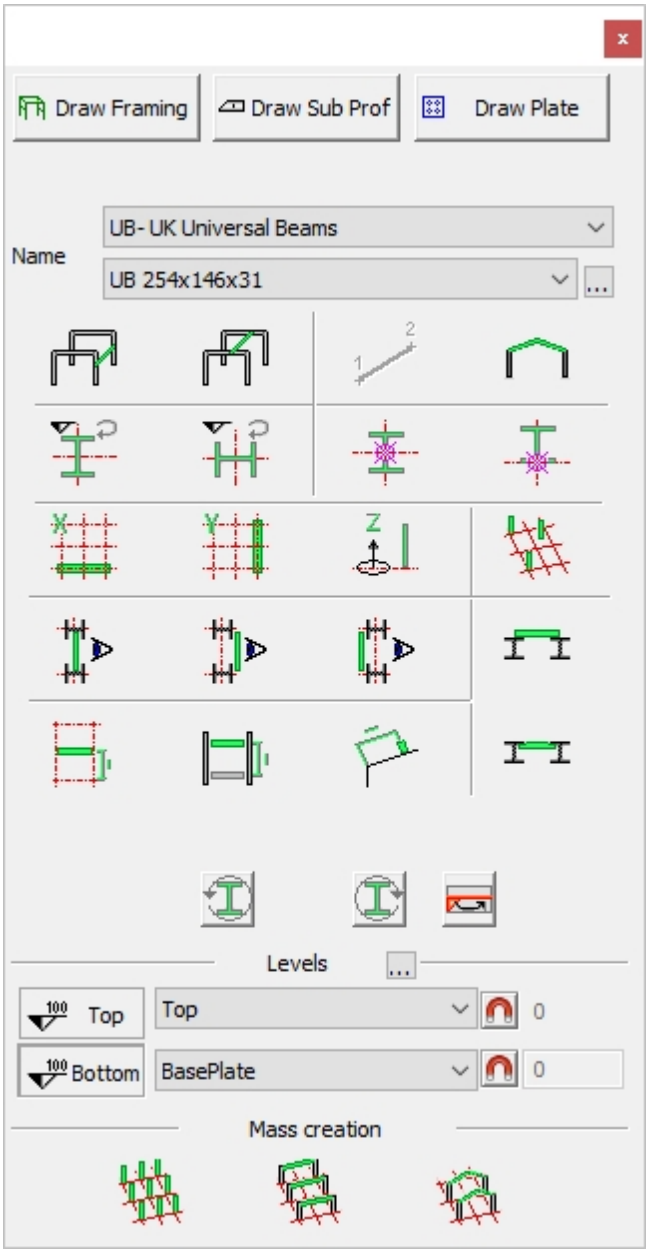
Context Modeling

Command - **PrB_ContextModeler**



The Context Modeler is unique to Parabuild - It allows you to create a complete accurate skeletal structure, simply and intuitively from the commands contained within the modeler.

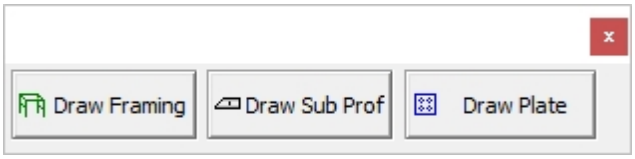
Activating this command will open the **Context modeler** dialog, which will remain open, or may be docked to the side of the working area for the duration of the work session. It may be closed at any time by clicking the  button.



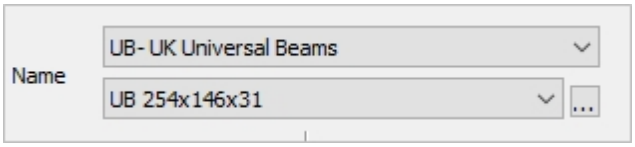
This dialog covers 5 main modeling operations:

1. At the top are the three action buttons and include:

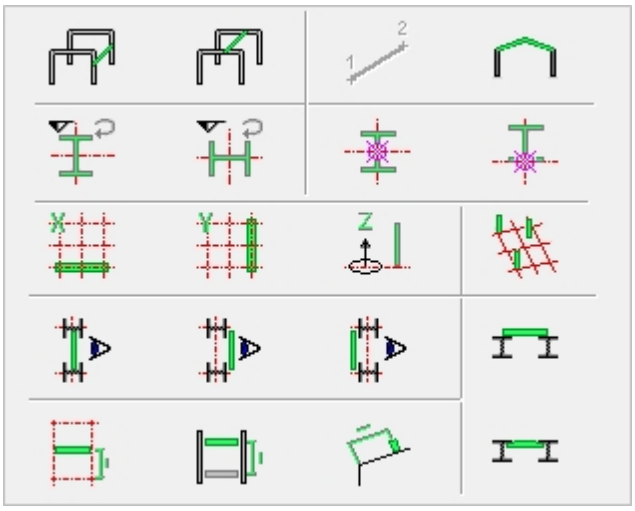
- Draw Framing
- Draw Sub Profiles
- Draw Plate



2. Below are the Selection Options - where you may select profiles from the Profile Libraries

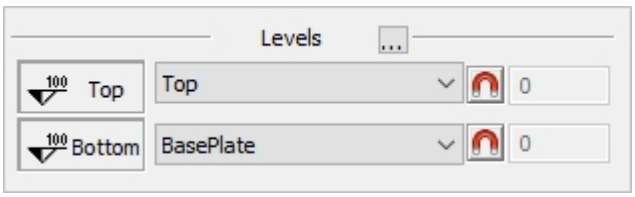


3. Next are the Context Buttons which offer a set of limitations and impositions so that you can more easily draw the type of profile that you need

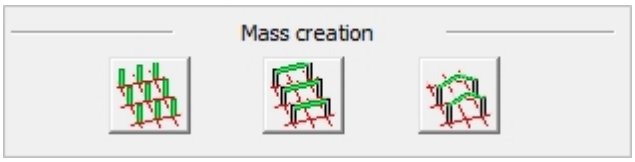


4. The Rotate Profile buttons can rotate the section of a profile through 90 °. As an alternative for these buttons in the dialog box, you can also use the keyboard **Arrow left** and **Arrow right** buttons to change the rotation.
You can also switch the start and end of the profile.

5. Using the Context Modeler, it's possible to set and establish Levels



6. The last section on the dialog is for Mass Creation where you are able to draw a range of profiles of the same type

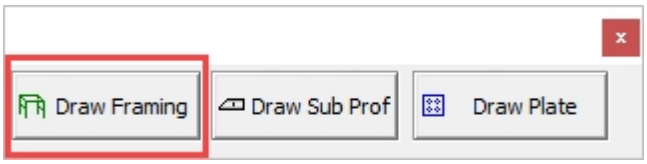


Some other keys that are available :
<**Space**>: When pressed, searching for solutions is temporarily suspended. The last element will remain on the screen. You can now freely move the cursor without changing the element, and you can also adjust the element using the arrow keys. Then press <**Enter**>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

Left Mouse Button: Draw the element that is currently visible

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

Draw Framing



With the *Draw Framing* action button you can draw columns, beams and rafters. Additionally, you can also draw side rails, purlins, joists and all longer profiles of your structure.

The profiles that you draw are based on :

1. Grid lines
2. Levels
3. Other profiles that already exist in the drawing
4. The World coordinate system (in case there are no grid lines)

Drawing profiles will become easier when using the functions offered by the Context Buttons

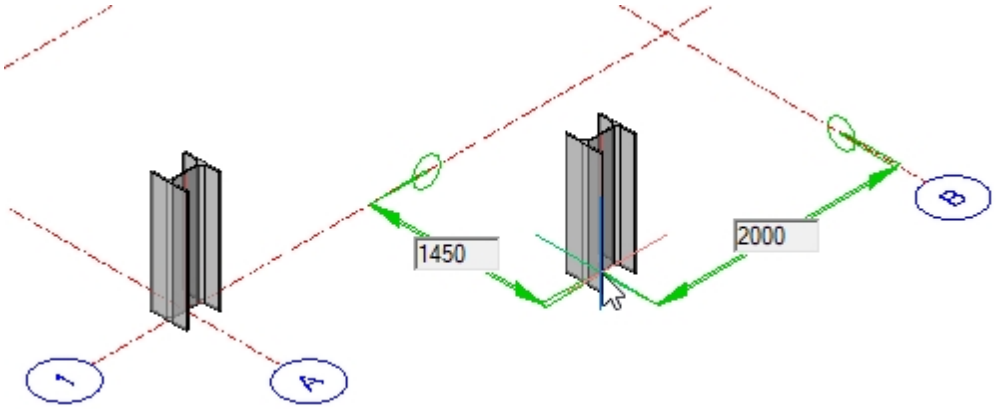
Before drawing the skeletal frame, it is recommended but not obliged to first:

1. Establish a Grid
2. Establish Levels

This is so that you may take advantage of the View manager functions and importantly, when planning and preparing 2D views in the General Arrangement drawings.

An additional advantage will be the relations between the members, the grids and the levels. The context modeler will create the relations between them automatically and later when you modify the grid or a level, the profiles will follow suit.

Drawing Columns



Ensuring that the levels are set as required, select a profile from the Selection options and press **Draw framing** - the column will be attached to the mouse pointer which may be placed anywhere on the model relative to the grid lines. When placing the column on a grid line, its orientation may be changed by moving the mouse

pointer along either the **X** or **Y** axis - when satisfied with the orientation, press the left mouse button to accept. You may continue adding columns - but the selected orientation will remain. Terminate the operation by either clicking the right mouse button, or pressing <**Enter**>

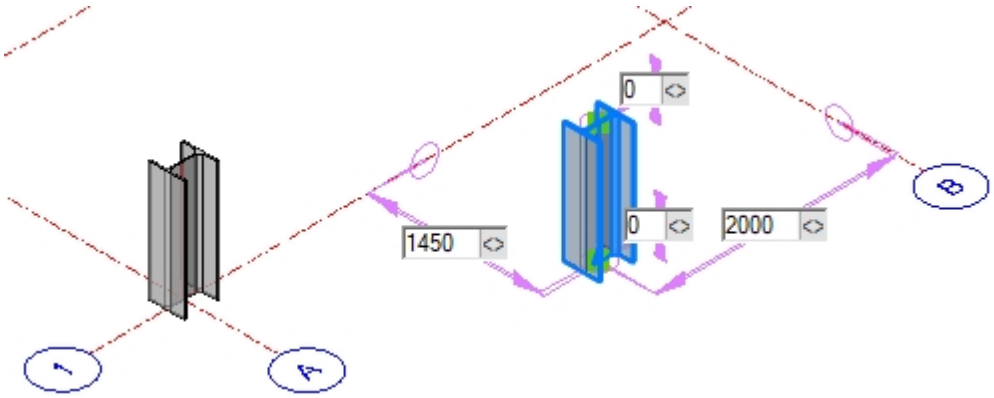
When placing columns off grid lines, Parabuild will automatically recognize the closest grid intersection, and any offsets will be indicated by editable dimension boxes.

If there are no grid lines, the column will be placed relative to the WCS

Once the column has been placed it is effectively constrained to the placement point and cannot be moved with the AutoCAD/BricsCAD move command. To move or modify that point, it's necessary to edit the constraints.

Alternatively, one can use the **Advanced** property of the profile to disconnect the profile from it's macro, so that manually moving the profile becomes possible.

Modifying Constraints



Clicking on the column will reveal the editable constraints - the relative points include dimensions on the **X**, **Y**, and **Z** axes. Here you may enter different values which will immediately be reflected on the model.

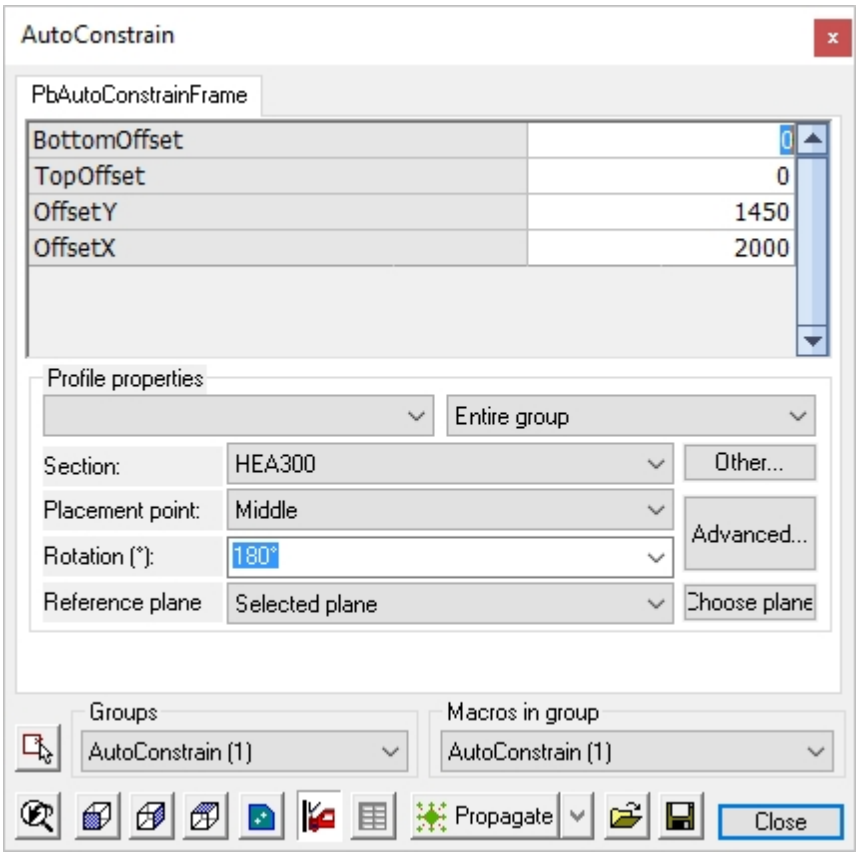
Alternatively, you may edit the constraints by using the drag constraint function 

Activating this function you will be prompted to: **Move the mouse to modify the constraint, then left-click to accept**:

Once complete, click the right mouse button or press <**Enter**>

Double-clicking the column will open the **AutoConstrain** dialog. Here you are presented with more options for editing the constraints, which include the offsets listed above, but in tabular form, and additionally:

- **Section** - Where you may choose another profile from the group by selecting from the drop-down menu
- **Other** - Will open the *Select profile* dialog where you may select a different group and profile
- **Placement point** - Where you may select a new placement point from the drop-down menu - alternatively you may select *Advanced*.
- **Advanced** - Which will open the *Profile Placement* dialog
- **Rotation** - Where you may change the profile rotation
- **Reference plane** - Where you may select the reference plane
- **Choose plane** - Where you are prompted to select a reference plane to match the section rotation to another part

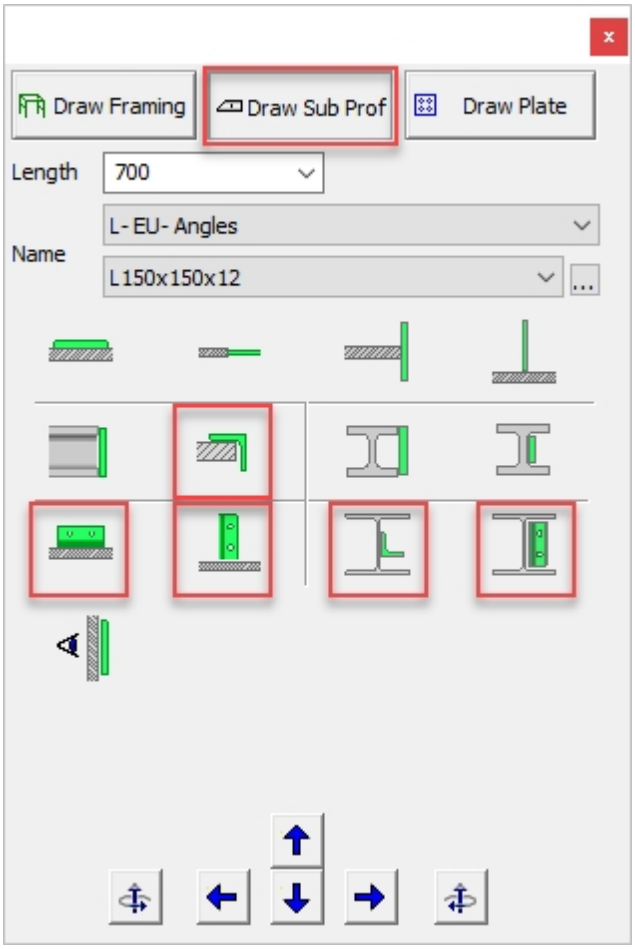


Draw Sub-Profiles

With the *Draw Sub Prof* button you can draw smaller profiles. These profiles will be attached to existing profiles or plates on the drawing.

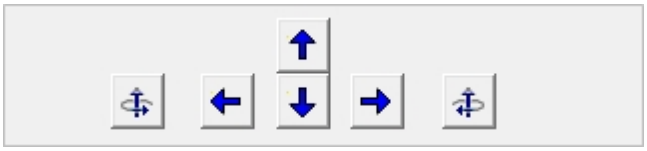
This button shares the dialog with **Draw Plate** - the relevant buttons for this command are highlighted below

1. Select the section profile and length
2. Move the mouse cursor to a part on the drawing to which the profile is to be attached



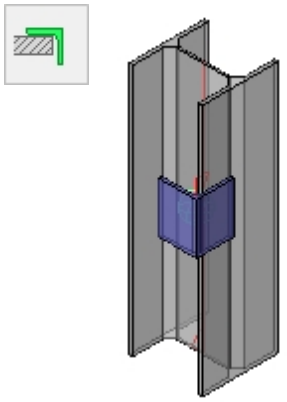
While drawing smaller profiles:

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

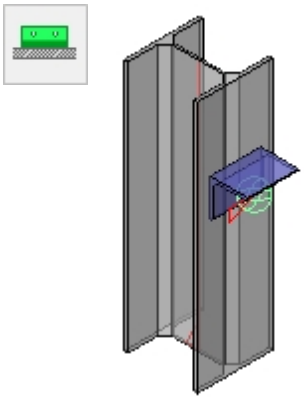


The rotate buttons will rotate the element on it's base surface (the base surface is the plane to which the element was oriented)

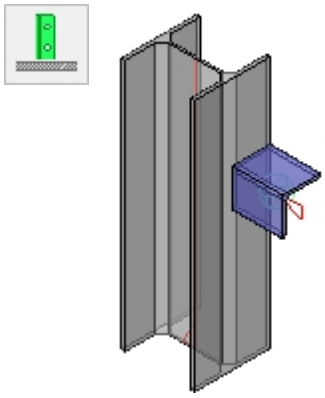
TIP : Use the arrows and *<Page Up/Down>* keys on the keyboard to quickly access these functions without having to move the cursor to the window!



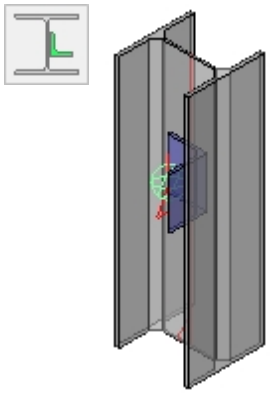
The Inside of the new section is attached



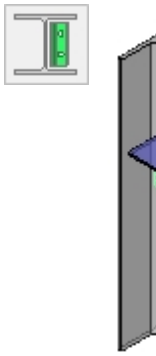
The new section is attached at the side



The new section is attached at the end

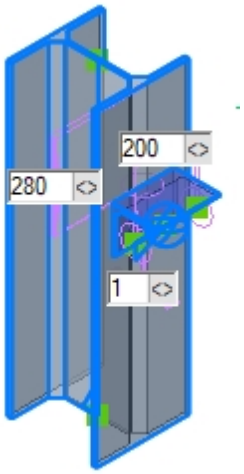


The new part axis is parallel to the base part axis

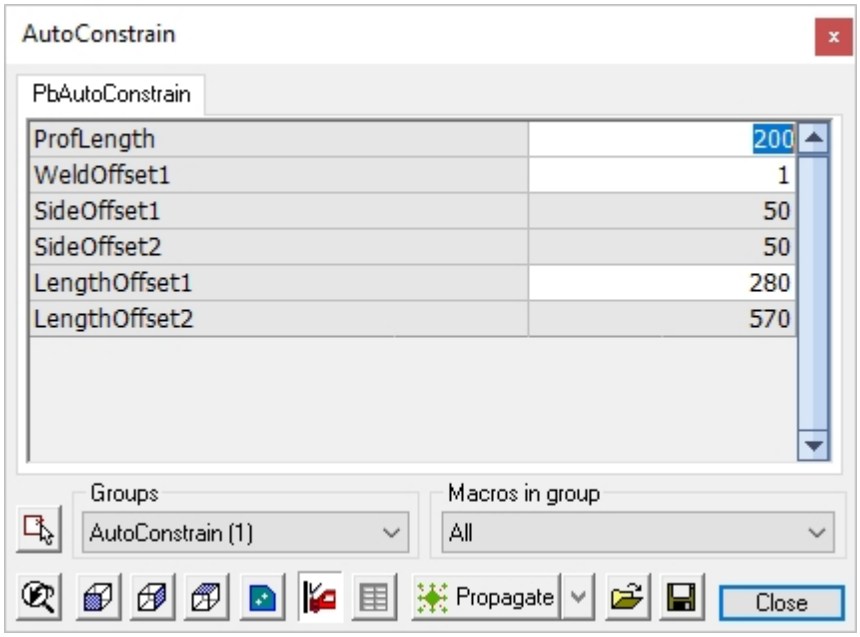


The new pa perpendicu base pa

At any time after the sub-profile is drawn, it's constraints may be edited by either:



- Selecting the part which will reveal the constraint dimensions which may be edited to suit requirements. When the context modeler dialog is closed, these constraints will not be shown.
- Double clicking the part to open the **Auto Constrain** dialog

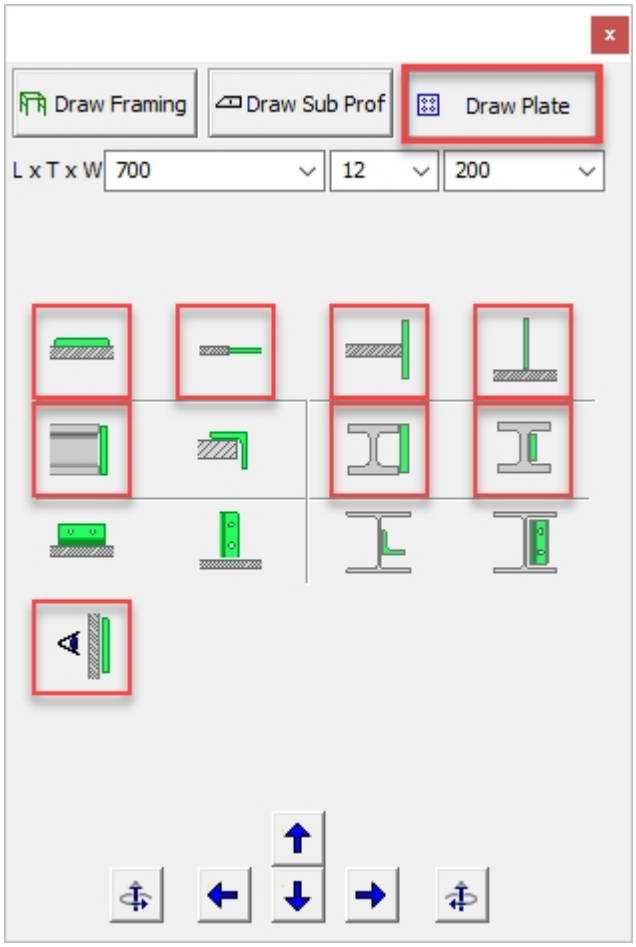


Draw Plates

With the *Draw Plate* button you can draw plates which will be attached to existing sections or plates on the drawing.

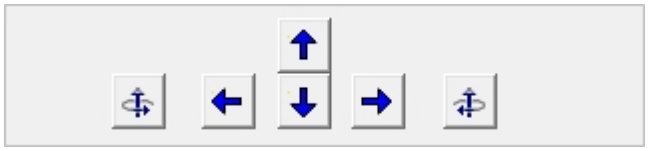
This button shares the dialog with Draw Sub-Profiles - the relevant buttons for this command are highlighted below

1. Select the plate size and length
2. Move the mouse cursor to a part on the drawing to which the profile is to be attached



While drawing smaller plates:

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



The rotate buttons will 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!

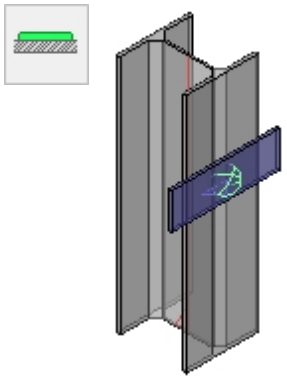


Plate faces touching

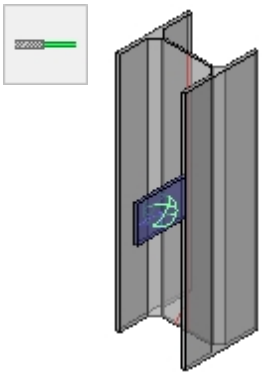
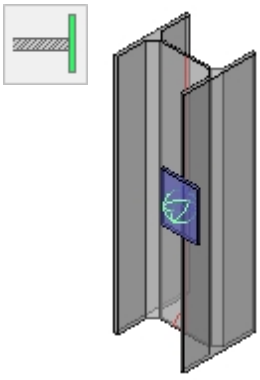
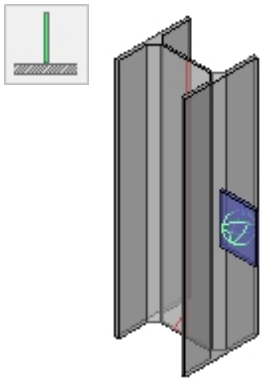


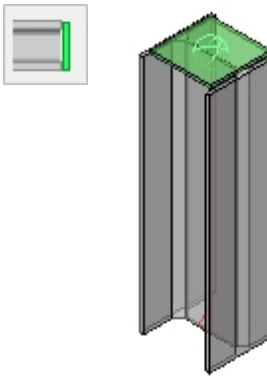
Plate sides touching



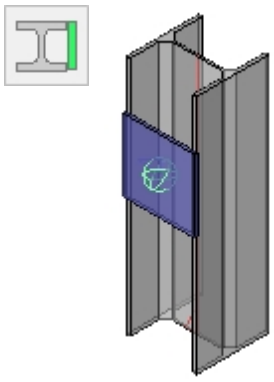
New plate face touches base plate side



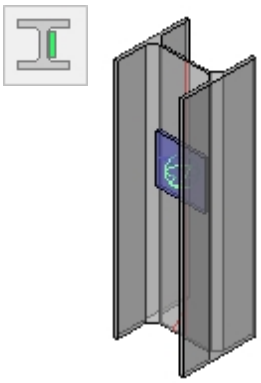
New plate side touches base plate face



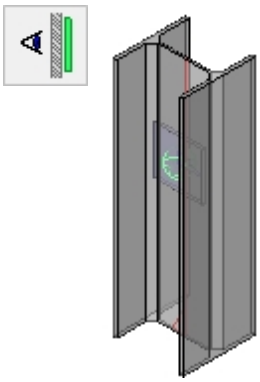
Attach plate to the en a member



The new part attached to the flanges of a member

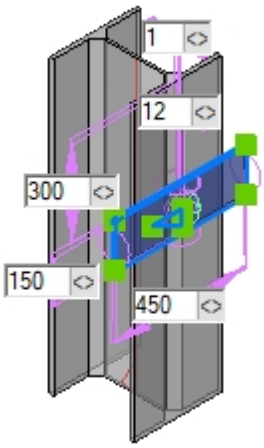


The new part's face is attached to the web of a member

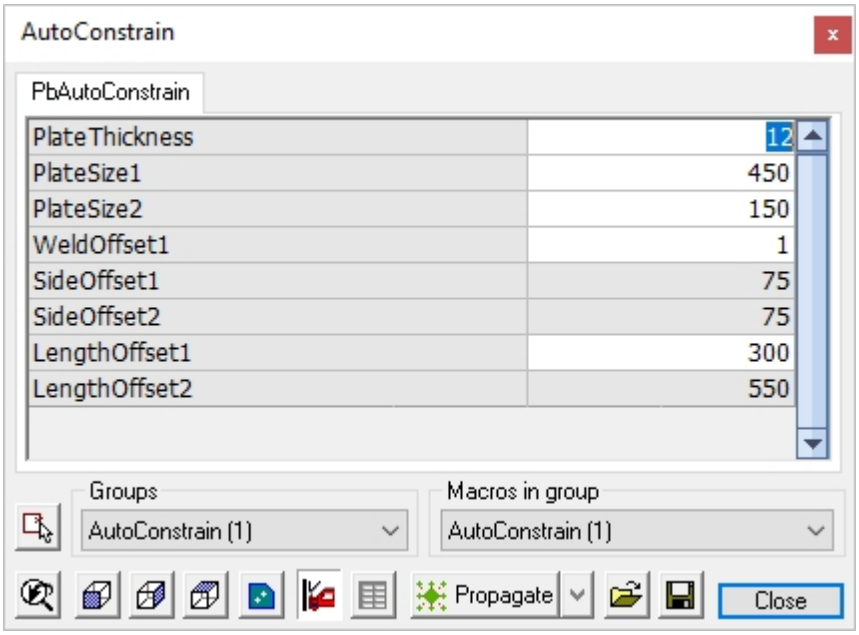


The new part is located on the back side of the base part

At any time after the plate is drawn, its constraints may be edited by either:



- Selecting the part which will reveal the constraint dimensions which may be edited to suit requirements. When the context modeler dialog is closed, these constraints will not be shown.
- Double clicking the part to open the **Auto Constrain** dialog



Selection Options

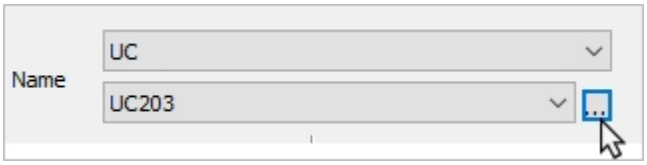
The selection options apply to Framing members or Sub-Profiles

Sections may be selected from the drop-down menus:

The Top drop-down will show the section types which have been activated in the Profile libraries

The second drop-down will provide the options available within the selected section type.

The button to the right, when clicked, will open the Select Profile dialog - from where you may select any profiles from all the groups.



Context Buttons

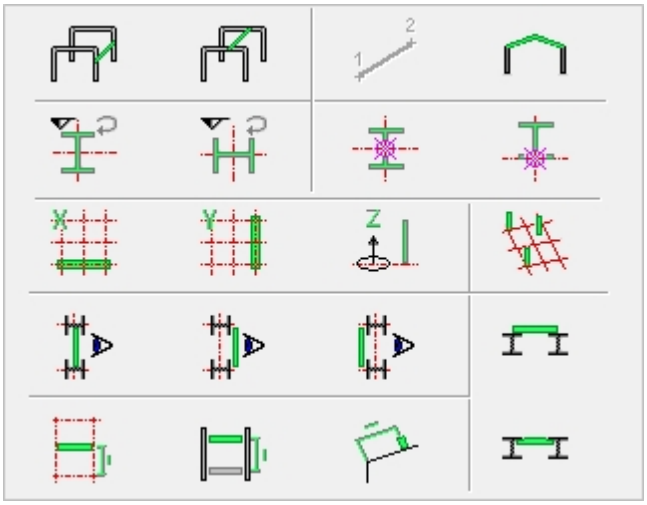
More about the context buttons (restrictions)

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.

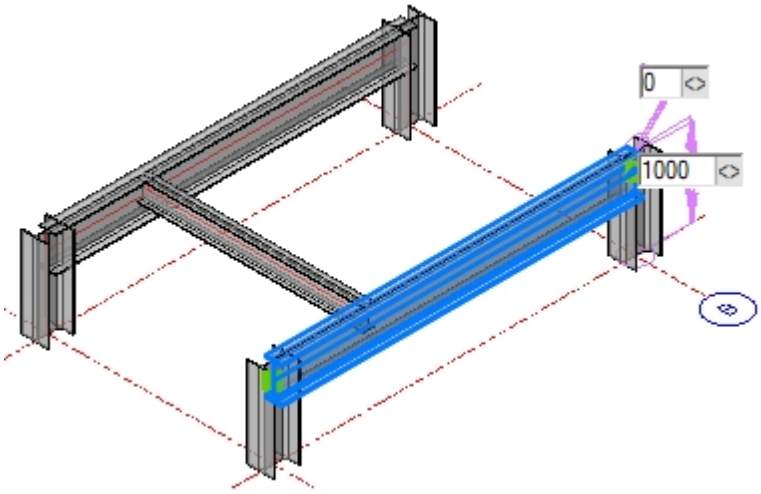
This is sometimes very useful, because in some situations, there are many different possible solutions for Parabuild to consider. Parabuild can not always show all the possible solutions at once. In that case the context buttons offer a solution. For example, if you want to draw a beam, you can press one or both beam buttons. Then Parabuild will not propose to draw a column or rafters.

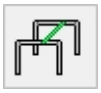
Note, that when selecting one of the context buttons, ensure all other buttons are deactivated unless you explicitly want the extra filter to be active.



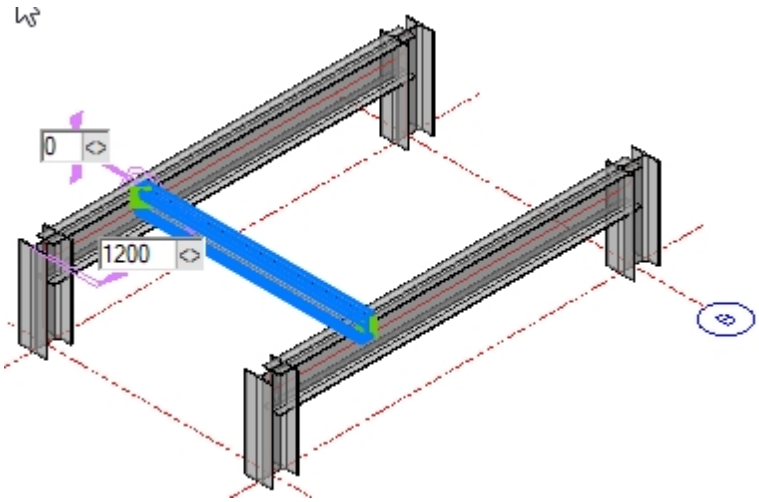
 **Draw beams between columns**

Move the mouse pointer between the columns to which the beam is to be connected - the selected beam will be automatically drawn showing the reference elevation.



 **Draw beams between beams**

Move the mouse pointer between the beams to which the beam is to be connected - the selected beam will be automatically drawn showing the reference dimension from the closest grid line. When satisfied with the position, click the right mouse button or press <**Enter**>



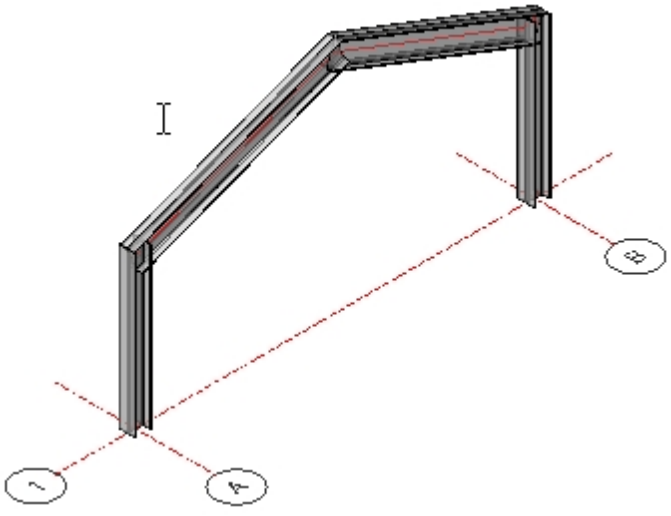
 **Draw a Pitched Rafter**

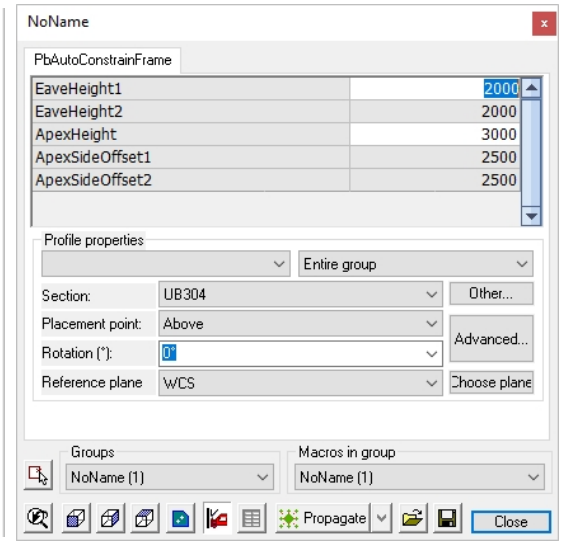
Activate the **Pitched rafter** button (Ensuring all other Context buttons are deactivated) and select the rafter profile - press the Draw framing button and move the mouse cursor between the two columns.

The eaves height of the rafter is usually established at the top level of the column - the apex height is set according to the chosen levels.

Left click the mouse to draw the rafter in the current position.

The dimensions may be edited at the time of setting the rafter in position by editing the reference dimension - Alternatively you can double click one of the rafter legs to open the Edit dialog.



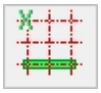


 **Parallel to World Y** - A column's orientation may be adjusted by the user by moving the cursor over either the **X** or **Y** axis of the grid line. This command will maintain the orientation to the **Y** axis - this is particularly useful when placing multiple columns.

 **Parallel to World X** - As above, but maintaining the **X** axis.

 **Grip on member axis** - This will maintain the grip on the geometric center of the member.

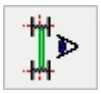
 **Grip on member side** - This will maintain the grip on the outside face of the flange.

 **Parallel WCS X Axis** - Will draw a beam between members while maintaining the **X** Axis

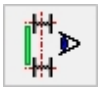
 **Parallel WCS Y Axis** - Will draw a beam between members while maintaining the **Y** Axis

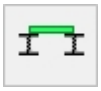
 **Parallel WCS Z Axis** - This button effectively forces Parabuild to only draw columns

 **Draw member based on Grid** - Will force Parabuild to draw columns based on the grids only.

 **Centered on Section** - Will draw a member in line with the centers of 2 other members

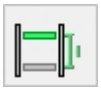
 **Near-side of Section** - Will draw a member against the near-side flanges of 2 other members

 **Far-side of Section** - Will draw a beam against the far-side flanges of 2 other members

 **New beam above beams** - Will draw a beam sitting on the top flanges of existing beams

 **New beam aligned with top of beams beams** - Will draw a beam in line with the top flanges of existing beams

 **Length offset from grid -** Will force Parabuild to draw horizontal members based on the grids only. This effectively disables the *Offset from parallel* obligation : 

 **Length offset from parallel** - Will force Parabuild to draw horizontal members based on other members only. This effectively disables the *Offset from grid* obligation : 

 **Length offset by model line ends** - Will force Parabuild to use model lines as reference point wherever it can (instead of the grid or other members).

Rotate Profile

The **Rotate Profile** tool will only work while the beam or column is still attached to the mouse pointer. To rotate the profile after its been placed you will need to double click the profile to open the Auto Constrain dialog.

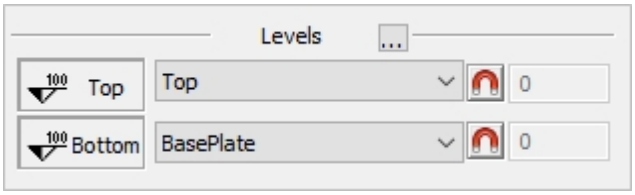
As an alternative for these buttons in the dialog box, you can also use the keyboard **Arrow left** and **Arrow right** buttons to change the rotation.

Additionally, you may Switch the Profile StartPoint (Triangle) by activating the  button

Levels

From within the Context Modeler, it's possible to set and establish Levels

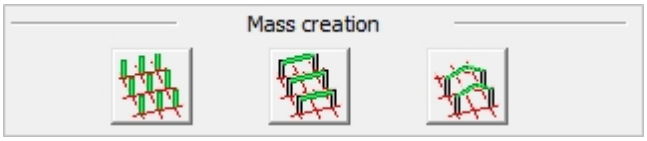
Pressing the  will open the Modify Levels dialog. Use this function to set and establish levels.



When the Anchor button  is activated, it will lock the level forcing Parabuild to draw members on that level only. The edit button to the right enables you to fix an offset relative to the level. Here you may enter positive or negative values.

Mass Creation

These 3 tools allow you to quickly draw a range columns, beams or rafters.



 This tool will draw a column on each intersection between the grid lines that you selected.

 This tool will draw a beam between all columns that it can find on the grid lines that you selected.

 This tool will draw 2 rafters between all columns that it can find on the grid lines that you selected.

View - Visibility

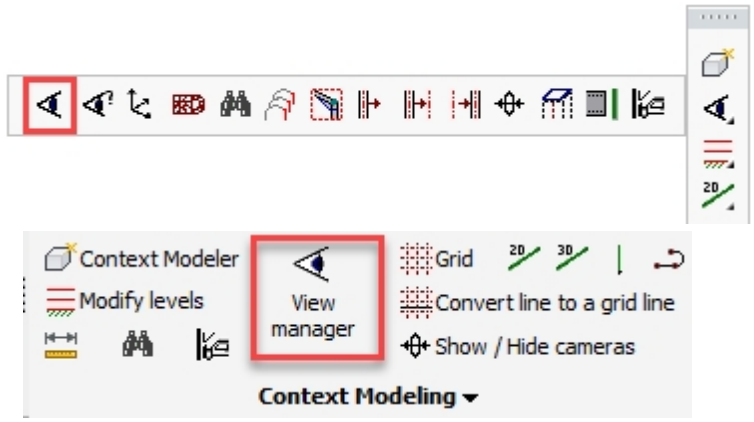
This chapter covers the visibility options for the 3D model, enabling you to hide and isolate objects, sub-chapters include:

- Visibility Manager

- Saved Views
- View Angle
- Search for Objects
- Isolate Objects
- Isolate a Selection
- Hide Objects Entirely
- Volumes -> Axes
- Axes -> Volumes
- Show / Hide cameras
- Assembly / Single Object Selection
- Weld Check
- Clash Check

Visibility Manager

Command : **PrB_VisibilityMgr**

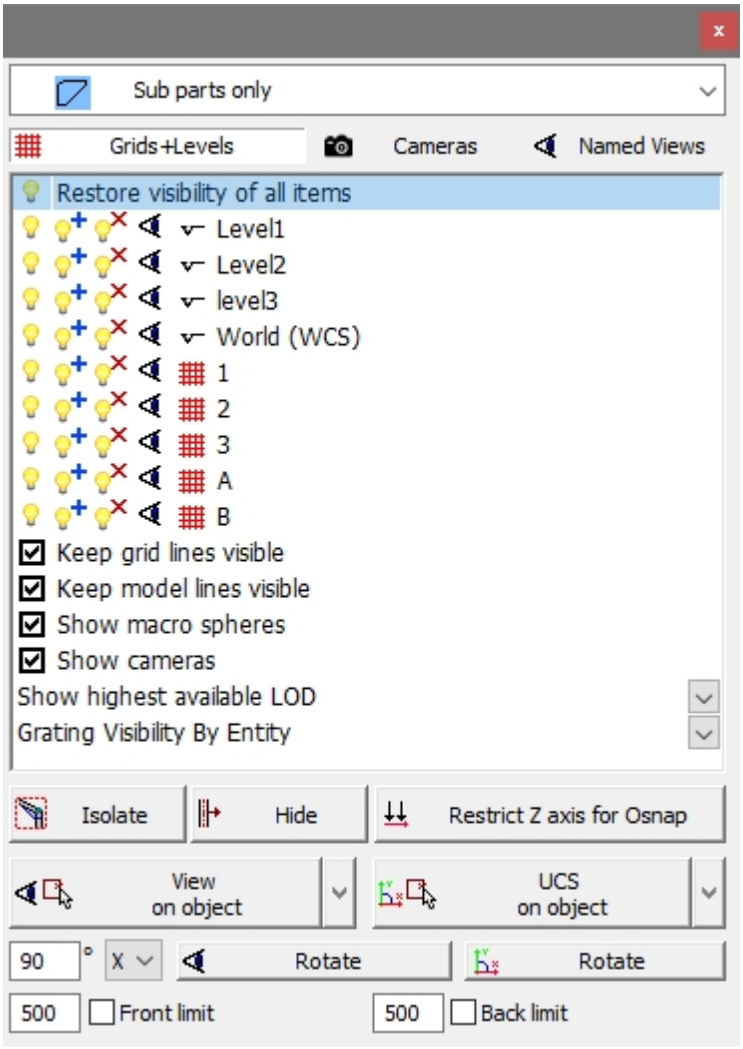


Selecting this command will open the **Visibility manager** window.

This dialog box may be docked by dragging it by the title bar to one of the edges of the application window.

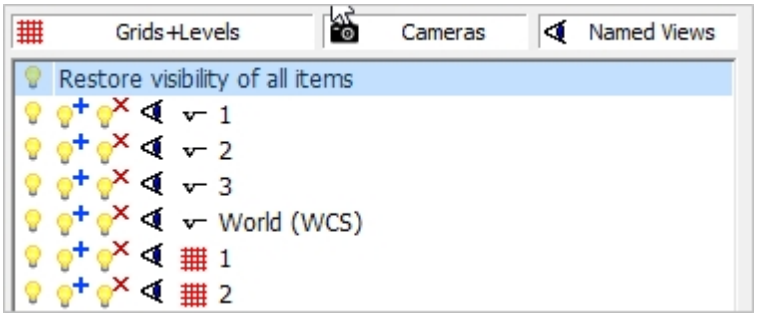
This window contains a set of tools that will enable you to:

1. Change the color of all elements
2. Hide elements
3. Change the view
4. Change the UCS and other tools related to the UCS



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 or blue). Or if you want the clashing elements in a striking color (Yellow)

When one of these buttons are activated, the corresponding element types will be shown in the list below:



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

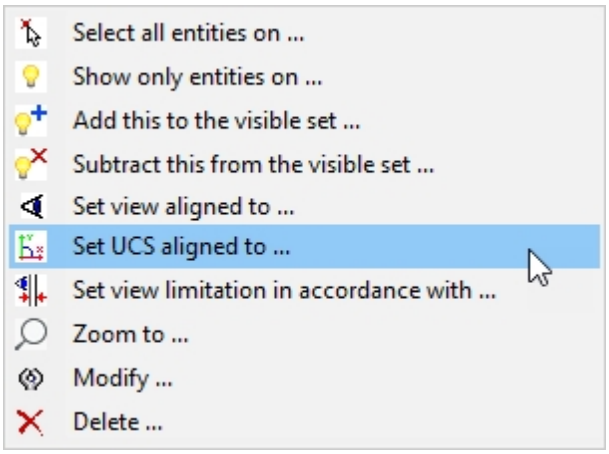
 Clicking on the lamp next to *grid 1*, all elements in the drawing will be hidden except the elements that are close to *grid 1*

 This lamp can be used to restore visibility of all elements close to *grid 1* in case they were rendered invisible by a previous lamp button operation

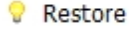
 With this lamp, you can hide all elements close to *grid 1*

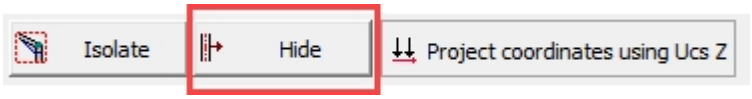
 Click on the eye next to an item to align the view with the level / grid / camera

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.



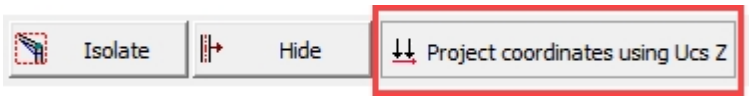
By selecting the Isolate button you will be prompted to select the objects to be isolated - press <Enter> and all other objects will be hidden.

To restore the visibility of all elements, select  at the top of the list in this dialog.



By selecting the Hide button you will be prompted to select the objects to be hidden.

To restore the model, select Restore visibility of all items at the top of the list in this dialog

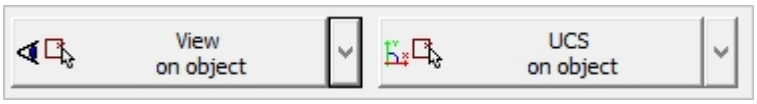


By activating this button, all Z coordinates will be projected to the XY plane. The Z coordinates will thus always be 0. Note: This button affects only the Object Snap tool of AutoCAD / BricsCAD.

(This tool works by modifying the AutoCAD variables OSNAPZ and ELEVATION)

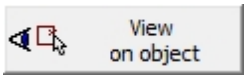
This is a useful tool if you want to temporarily work in the current UCS XY plane. One can then easily draw something in a roof if the UCS was set to align with a rafter.

Do not forget to disable this button again after you're done. When this button is enabled and you're not aware of it, then your object snaps will not work as you would expect it.



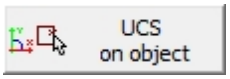
These buttons provide access to a range of view options - press the arrow button to see the full range of available options.

We will explain the most important tools.

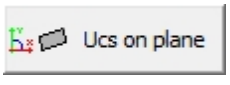


This is a good tool because you just need to select 1 object, and the view will be oriented according to the geometry of that object. If the object has an axis then this will be the new X coordinate of the view. Otherwise it will choose the longest geometry as X axis (this tool works on any type of object).

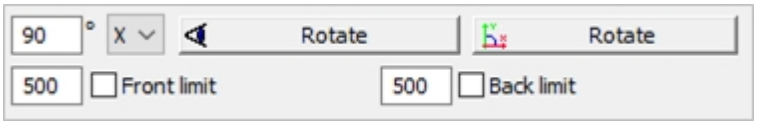
This tool is very powerful because we can determine with just 2 clicks which surface the view should be aligned to. As opposed to *View on object* this tool does not assume an X axis automatically. Instead it relies on the surface that you selected as basis for the view's new orientation (This tool only works on Parabuild objects).



Same as *View on object* but in this case the UCS will be aligned to the object.



Same as *View on plane* but in this case the UCS will be aligned to the surface.



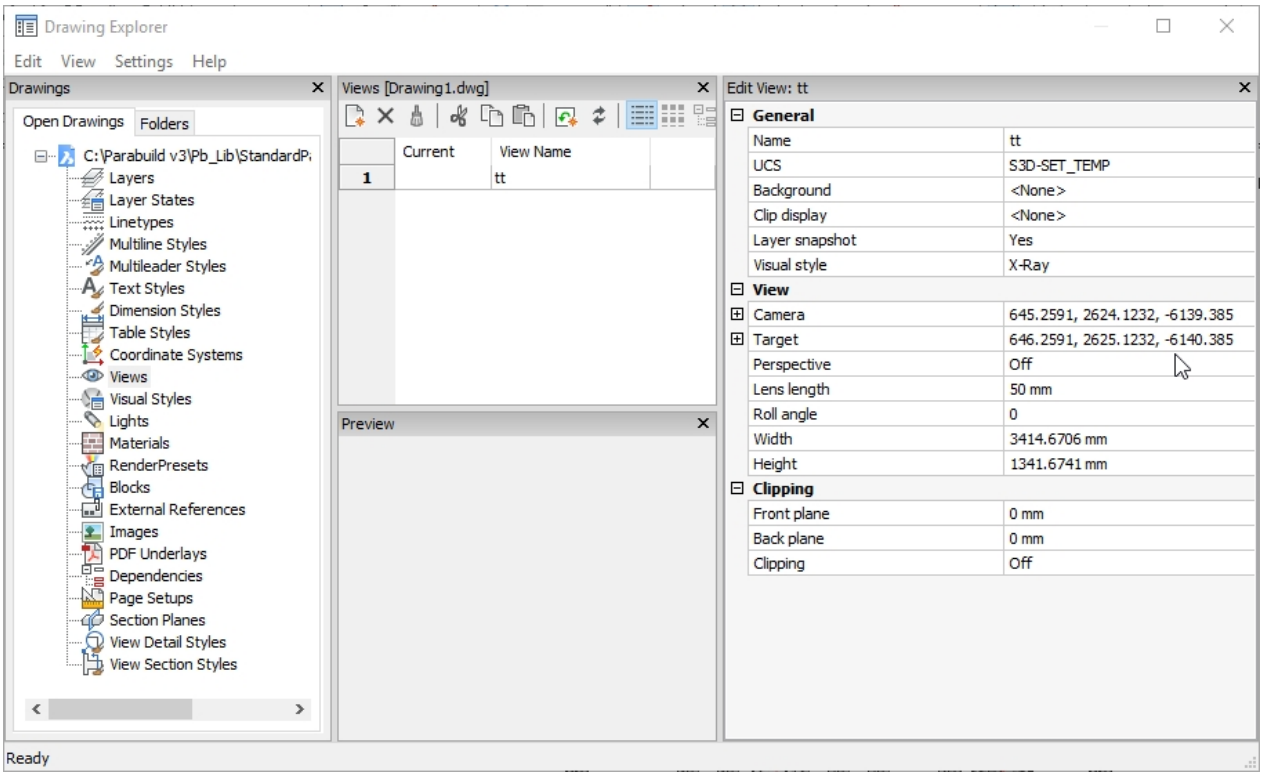
With these buttons it is possible to rotate the view or UCS.

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

Saved Views

Command : **ddview**

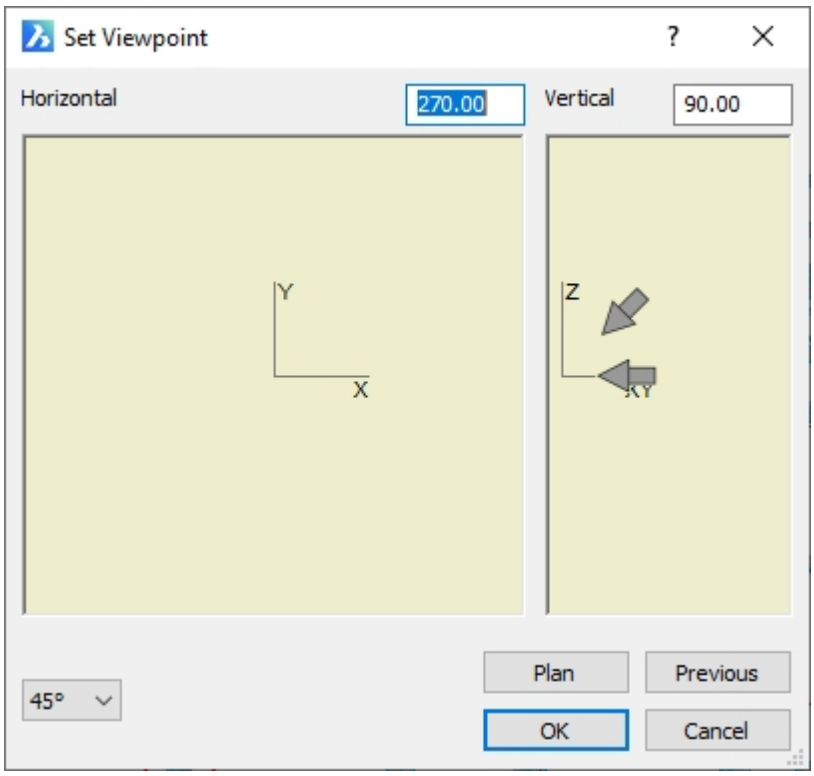
Selecting this command will open the Drawing Explorer window, where you can view and edit saved views



View Angle

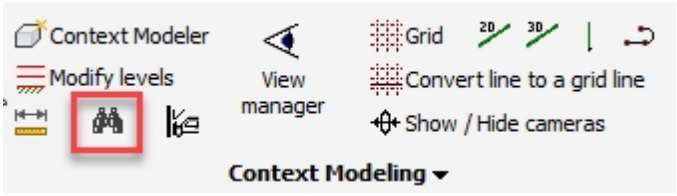
Command : **ddvpoint**

Selecting this command will allow you to set the view angle



Search for Objects

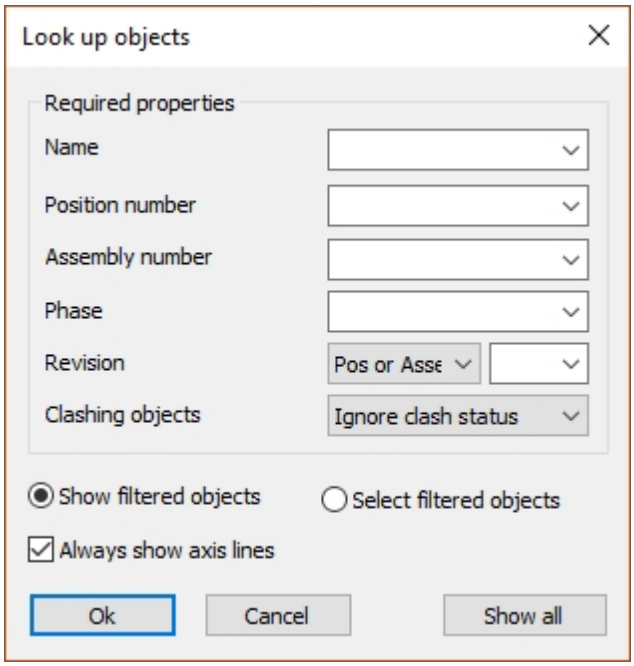
Command : **Prb_LookUp**



Selecting this command will allow you to search for objects.

Selecting an object value from any of the DropDown lists will hide all items that do not comply to the filter.

To restore the model - select: View manager > Restore visibility of all items.



The radio button at the bottom allows you to select the filtered objects instead of highlighting them.

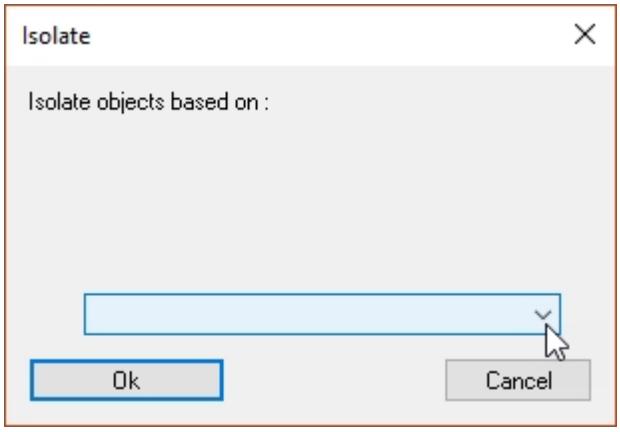
Isolate Objects

Command : **Prb_IsolateObjects**

Selecting this command will allow you to isolate selected objects.

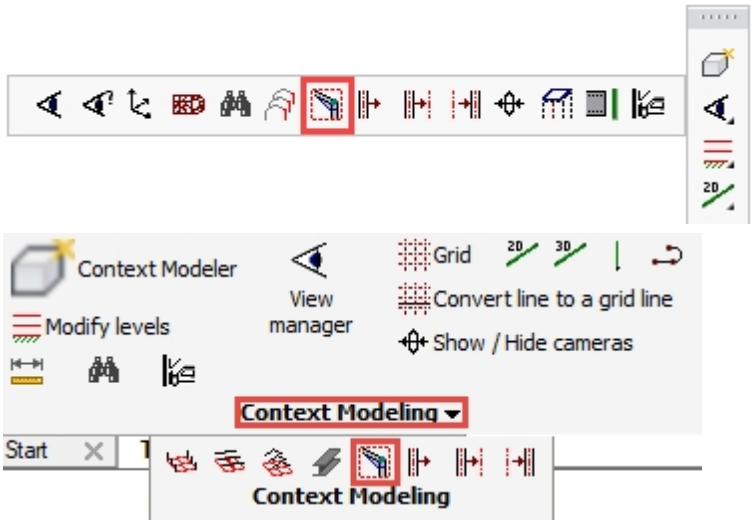
Clicking the DropDown menu will list the current levels and grid-lines - selecting one will hide all objects not close to that level or grid line.

To restore the model - select View manager > Restore visibility of all items



Isolate a Selection

Command : **PrB_IsolateSelection**



Selecting this command will isolate the selected objects.

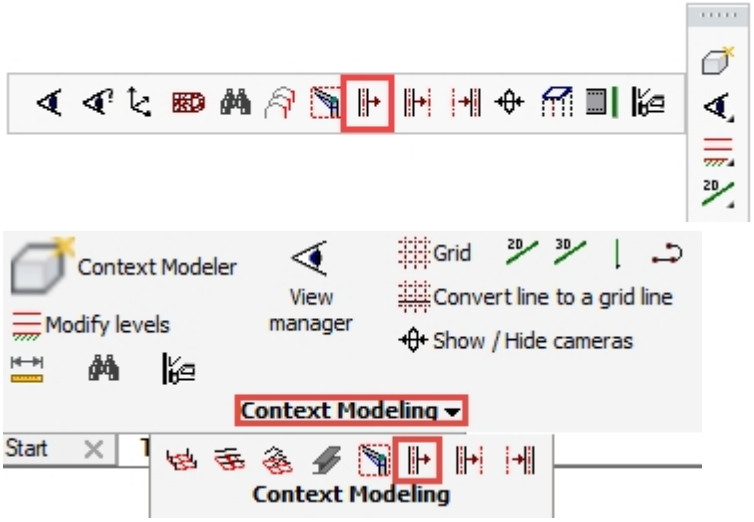
At the command prompt, select the entities you wish to isolate, then press <**Enter**>.

All objects except those selected will be hidden.

To restore the model - select View manager > Restore visibility of all items.

Hide Objects Entirely

Command : **Prb_HideObjectsNoAxis**



Selecting this command will hide all selected objects.

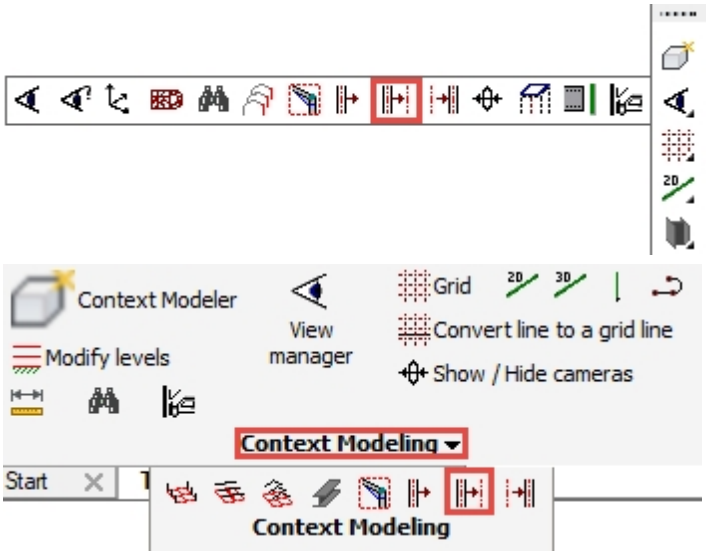
At the command prompt, select the entities you wish to hide, then press <**Enter**>.

The selected objects will be hidden.

To restore the model - select View manager > Restore visibility of all items

Volumes -> Axes

Command : **PrB_HideObjects**



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

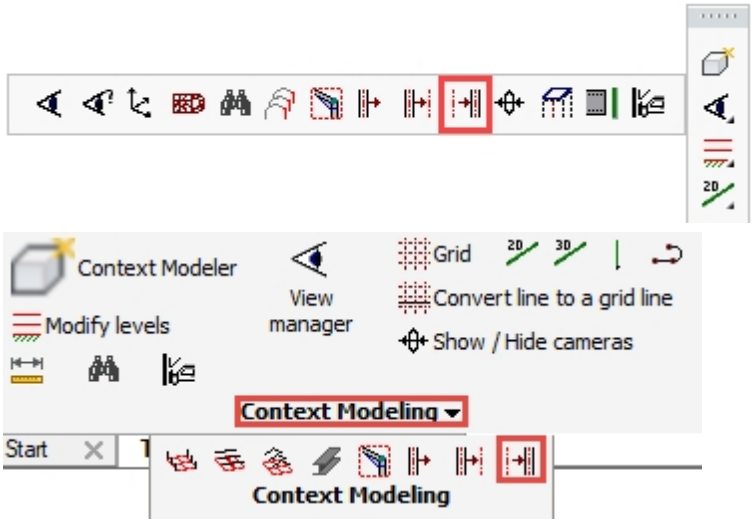
At the command prompt, select the entities you wish to hide, then press <**Enter**>.

All objects except those selected will be hidden, however, the axes of profiles and plates remain visible

To restore the model - select View manager > Restore visibility of all items

Axes -> Volumes

Command : **PrB_ShowObjects**



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

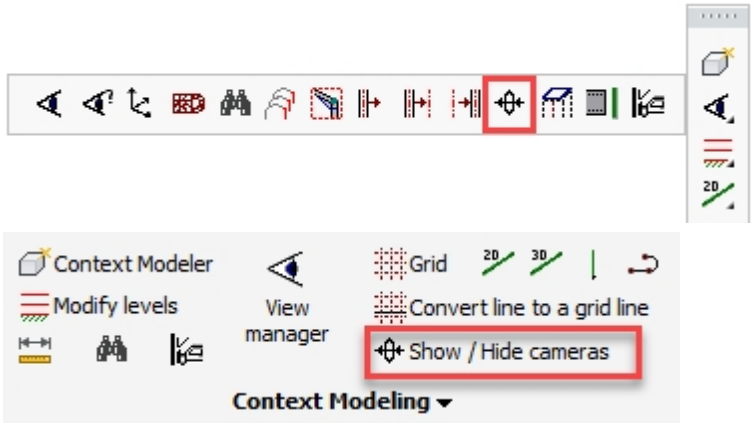
The volumes of which only the axes were visible can be selected and their full visibility restored.

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

Alternatively - select View manager > Restore visibility of all items

Show / Hide Cameras

Command : **Prb_SwitchViewDefVisibility**

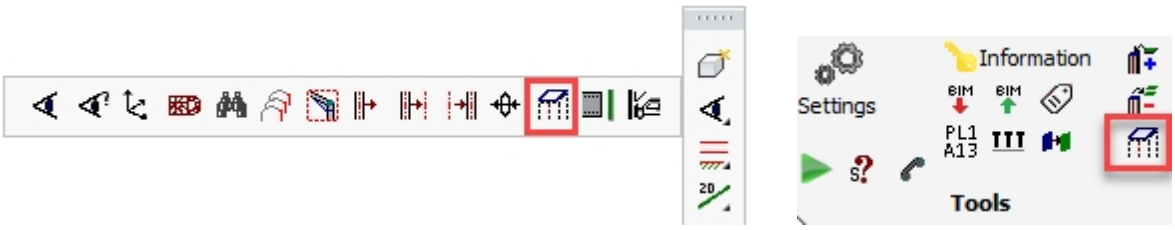


This command works as a switch.

It will hide all the cameras in the 3D drawing if they are currently visible or show them if they are currently invisible.

Assembly / Single Object Selection

Command : **Prb_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.

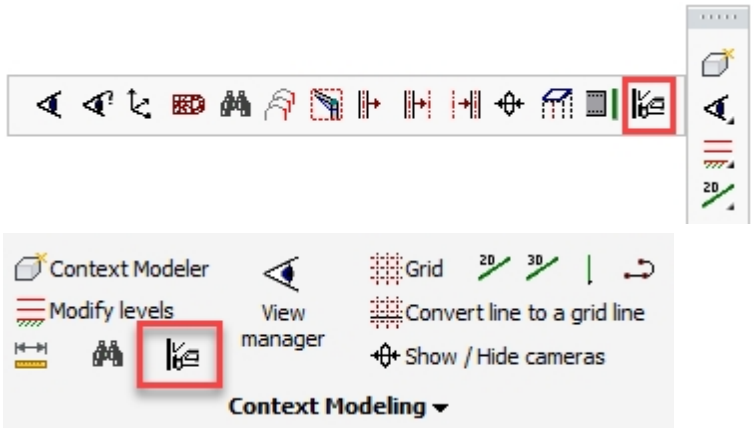
Weld Check

Command : **PrB_WeldCheck**

Currently not fully implemented yet

Clash Check

Command : **PrB_FullClash**



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

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

Bolts are colored yellow when they are :

- Too close to a plate or beam, thus warning for a possible lack of space surrounding the bolt for tightening it
- Too close to the edge, thus warning for a possible *material weakness* issue

So a yellow bolt could be ignored by the draftsperson if they know that in that situation it will be possible to tighten the bolt, or that there is no material weakness issue in that situation.

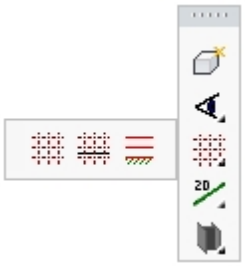
These maximum distances between the bolt and the edges of other parts can be set in the Global settings dialog box.

Bolts will be colored red if they are drawn in the air or if they are colliding with a part because there is no hole drawn.
Bolts usually get all of their holes automatically.
But in these (impossible) situations the holes are skipped by Parabuild :

- The hole would extend beyond the material. Parabuild does not draw these partial holes because most CNC machines would not drill this hole anyway. If you need to draw this hole anyway, then use this command to draw a special cut for it : . You can just use the bolt as the cylinder that defines the special cut.
- The bolt head or the bolt nut touches too much the radius between the flange and web of a member.
  Mounting the bolt there would create an unforeseen tension on the bolt head or the nut that touches the radius.

So therefore a red bolt indicates a serious problem and should never be ignored by the draftsperson.

General - Grids

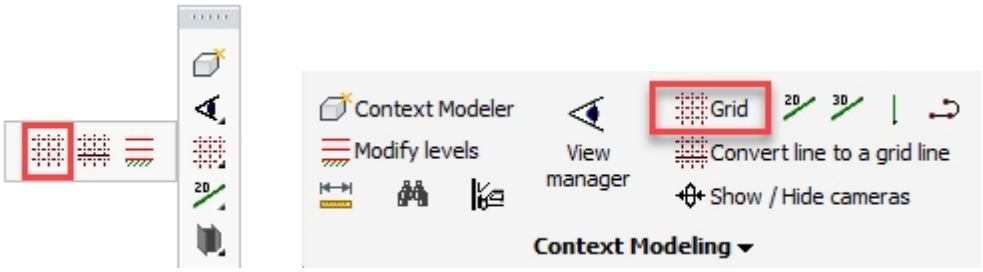


This chapter covers the use of Grid lines and Levels. The sub-chapters include:

- Draw Grid
- Convert Line to Grid-Line
- Modify Levels

Draw Grid

Command - **Prb_DrawGrid**



This tool will open the *Draw Grid* dialog.

To draw grids that are unequally spaced then you should enter in the **Distance between grid lines** field the distances separated by spaces or comma's. If you are working in inches units, then you can only use the comma to separate the distances because the space can be used to denote feet and inches.

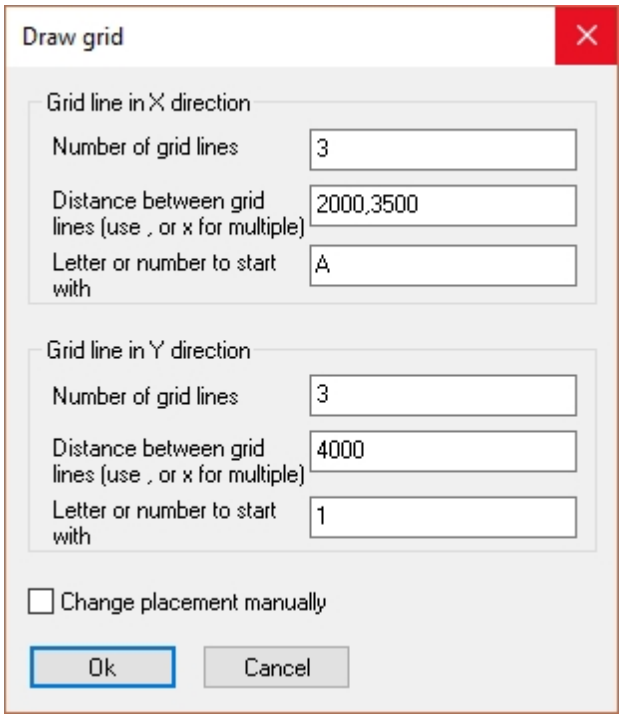
i.e. 1000,2000

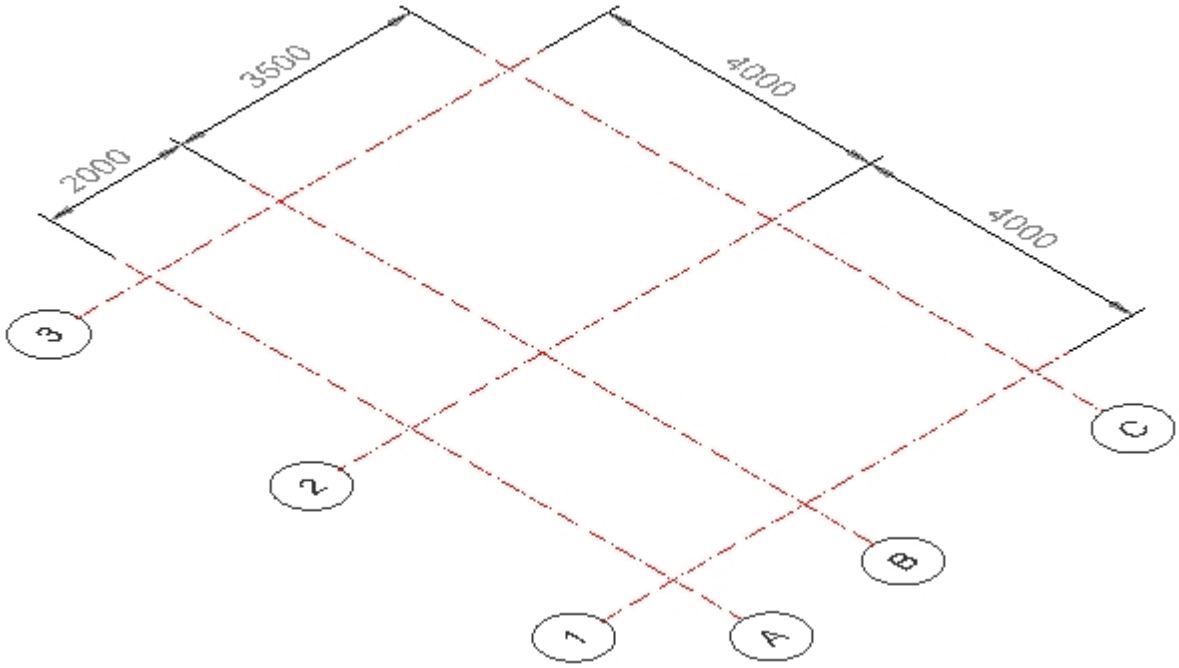
When you click **OK**, the grid will be drawn at the origin.

By checking **Change placement manually** you will be prompted to select the insertion point and indicate the **X** and **Y** axes.

You can Move / Rotate / and Stretch the resulting grid lines. They are regular lines which have their grid name attached.

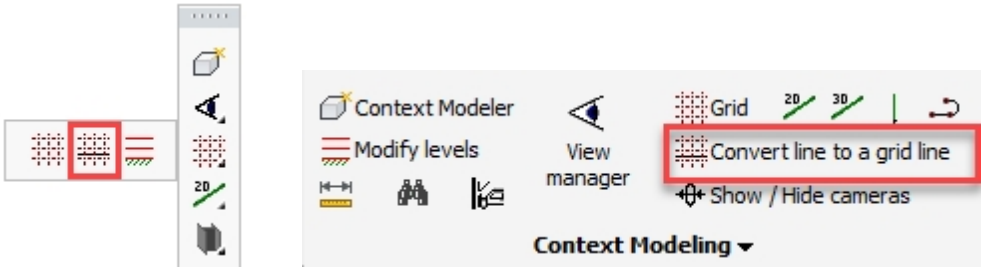
By convention, the grid line numbering system will begin at the bottom-left hand corner, incrementing along the X and Y axes.





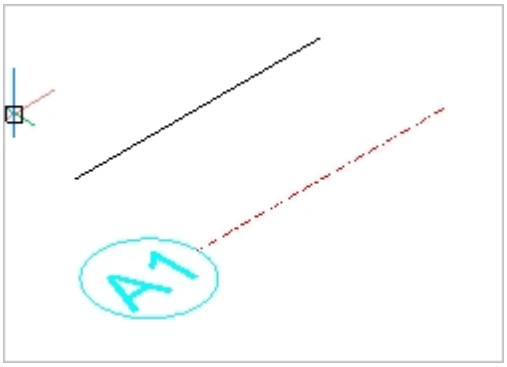
Convert Line to Grid-Line

Command - **Prb_SetAsGridLine**



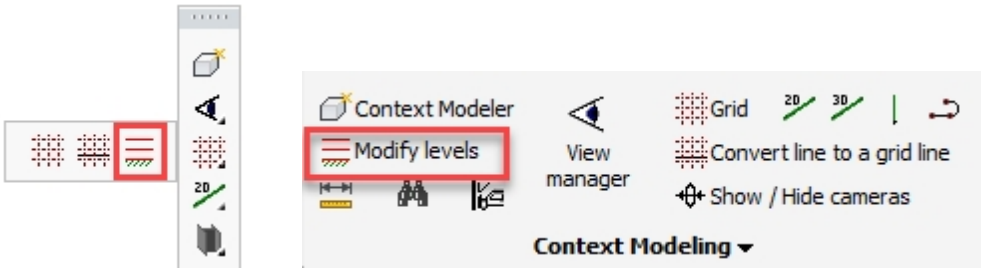
This command allows us to convert a regular line into a grid line.

It also allows us to change the grid names of existing grid lines.



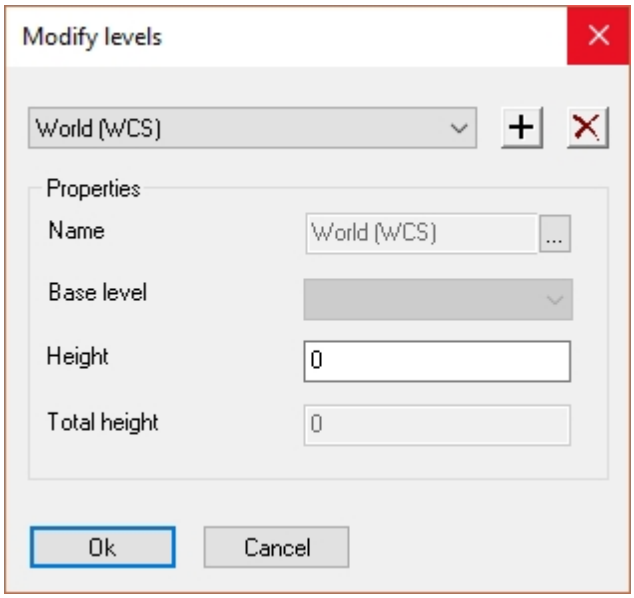
Modify Levels

Command - **Prb_ModifyLevels**



This command may be activated from the Context modeler, or from the **General** / **Grids** toolbar FlyOut**.**

Activating this command will open the **Modify levels** dialog. Here you can add, subtract, or modify levels and elevations.



To add a level - press the  button to open the **Name** dialog where you should add the name for the level. Press **Ok** to close the dialog. Enter the height of the level in the **Modify levels** dialog and press **Ok** again.

Repeat this process for all other levels.

At any time you may review and/or modify the levels by activating the drop down menu.

Any modifications to the levels will be immediately reflected in the model when you close this dialog.

To delete a level, highlight it from the drop down and press 

The base level may be the WCS (World Coordinate System) or another level.

Model Lines

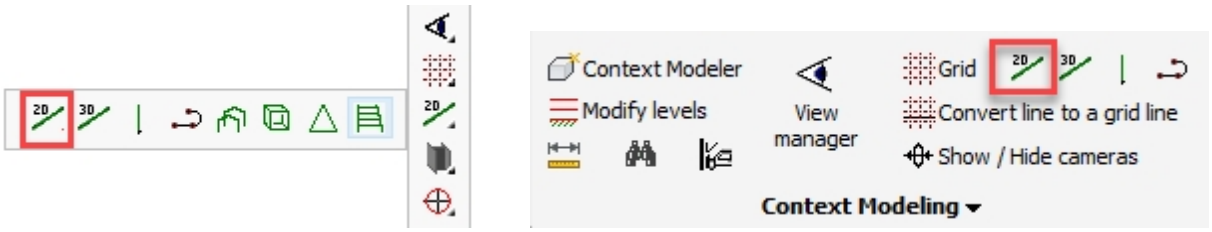
This chapter covers the use of Model Lines and the principles of WireFrame modeling. The sub-chapters include:

- 2D Line
- 3D Line
- Vertical Line

- 3D PolyLine
- Portal Frame
- Tower
- Pyramid
- Ladder

2D Line

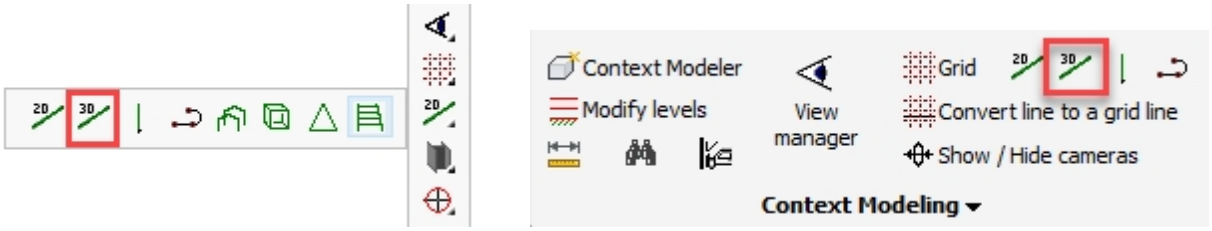
Command - **PrB_modlijn2D**



Draws a 2D Line that is restricted to use X-Y plane of the current UCS.

3D Line

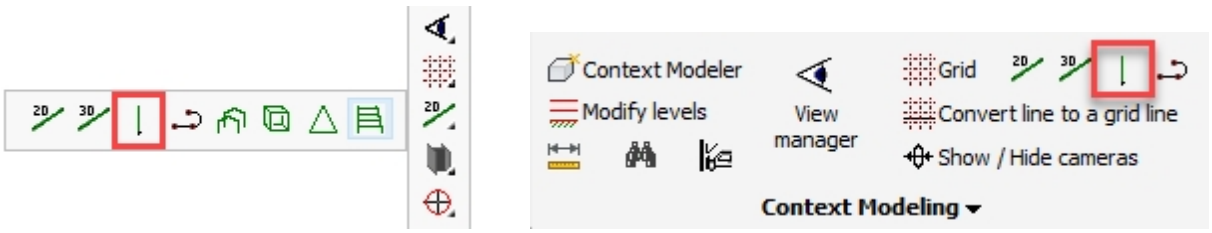
Command - **PrB_modlijn2D**



Draws a 3D Line, which does the same as the AutoCAD/BricsCAD line command.

Vertical Line

Command - **S3d_NokLijn**

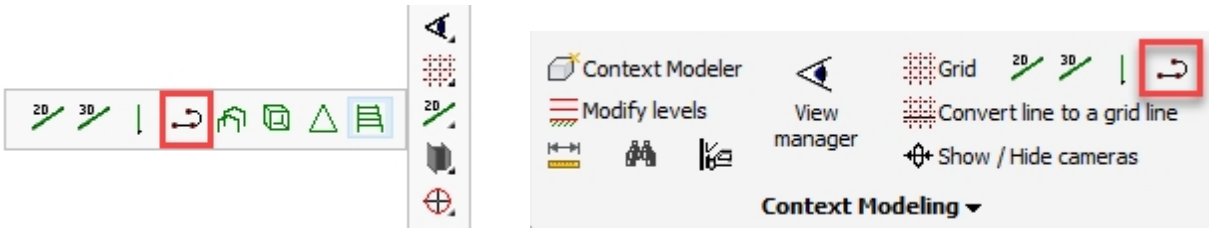


Draws a vertical line. The tool asks for the insertion point and the height of the line.

The new line will be parallel to the current Ucs Z axis.

3D PolyLine

Command - **PrB_3dPoly**



Draws a 3D PolyLine that can handle arcs in more 3D situations than the AutoCAD/BricsCAD 3D polyline.

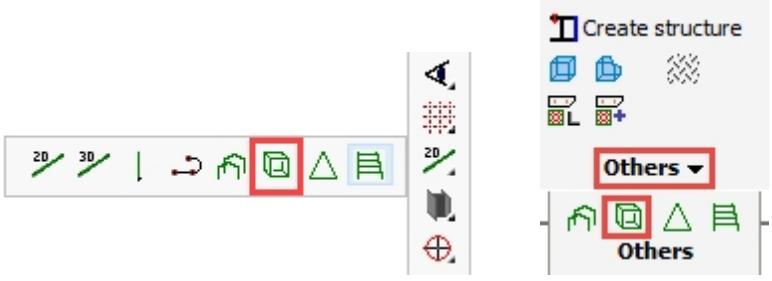
Portal Frame

Command -

This command will be renewed soon

Tower

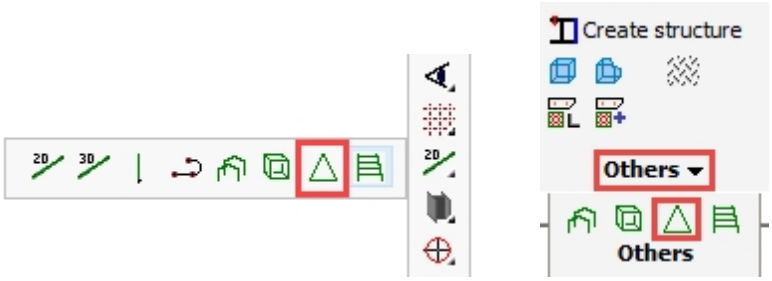
Command -



Command will be renewed soon

Pyramid

Command -



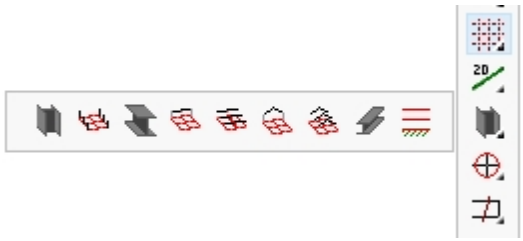
Command will be renewed soon

Ladder

Command -

Command will be renewed soon

Constrained Profiles



All of the following commands will draw members that are constrained to the grids, the levels and other members.

- Column with Point
- Columns with Grid-Lines
- Beam with Points

- Beam between Columns
- Beams with Grid-Lines
- Apex between Columns
- Apex with Grid-Lines
- Free Member
- Modify levels

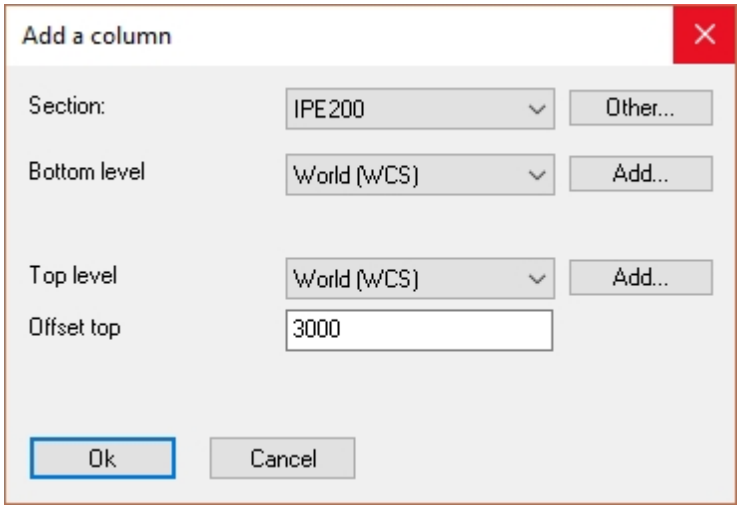
For more information about geometric constraints, see the chapter Macros and modules

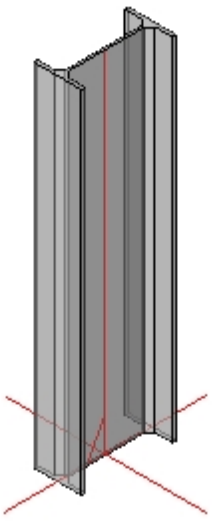
Column with Point

Command - **Prb_PointColumn**

Draws a column on a point.

On activating this command, the **Add a column** dialog will appear prompting you to select the profile to be drawn, together with the top and bottom levels. If levels have already been established, they will be reflected here, additionally, you have the opportunity to create new levels or choose an extra height offset of the column by entering a value for **Offset top**.





After selection, you will be prompted to choose a point. You may then select multiple points if you so desire, and then press <**Enter**> to end the command.

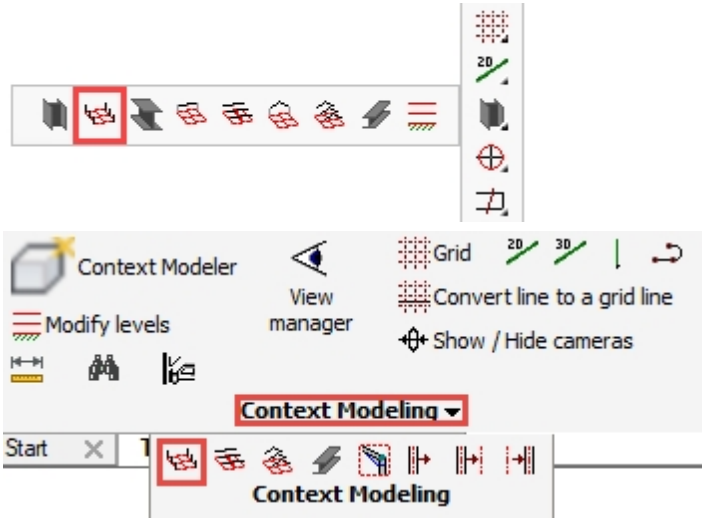
The column will be constrained to the grid lines that are closest to the point. If you selected a point at an offset from a grid line, then this offset will become modifiable in the macro dialog. Just double-click the column to open this dialog box.

The column cannot be moved independently of the grid lines, unless if the profile is disconnected from the macro.

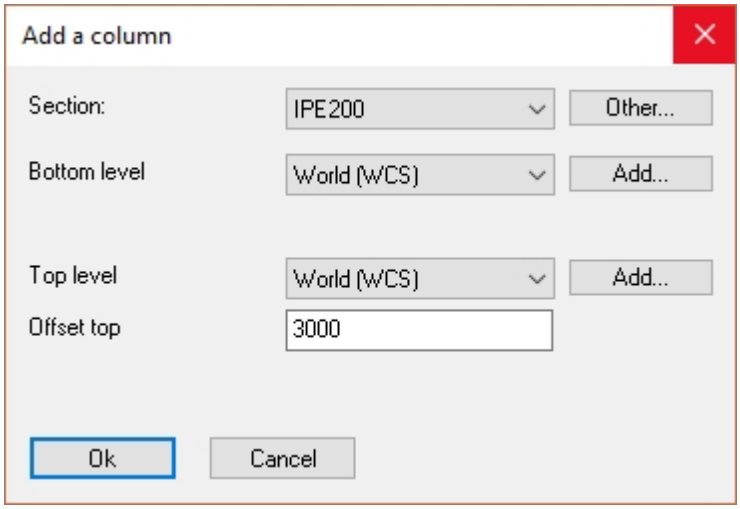
Columns with Grid Lines

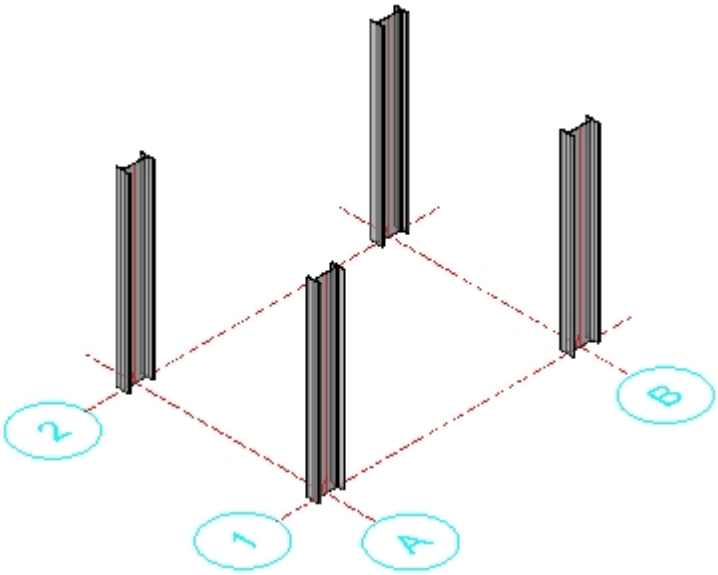
Command - **PrB_Columns**

Draws columns on the intersections of grid lines.



On activating this command, the **Add a column** dialog will appear prompting you to select the profile to be drawn, together with the top and bottom levels. If levels have already been established, they will be reflected here, additionally, you have the opportunity to create new levels or or choose an extra height offset of the column by entering a value for **Offset top**.





The command will draw a column on each intersection it finds between the grid lines that you select.

The columns will be drawn constrained to the grid lines. Moving or adjusting the grid line centers will automatically move the columns.

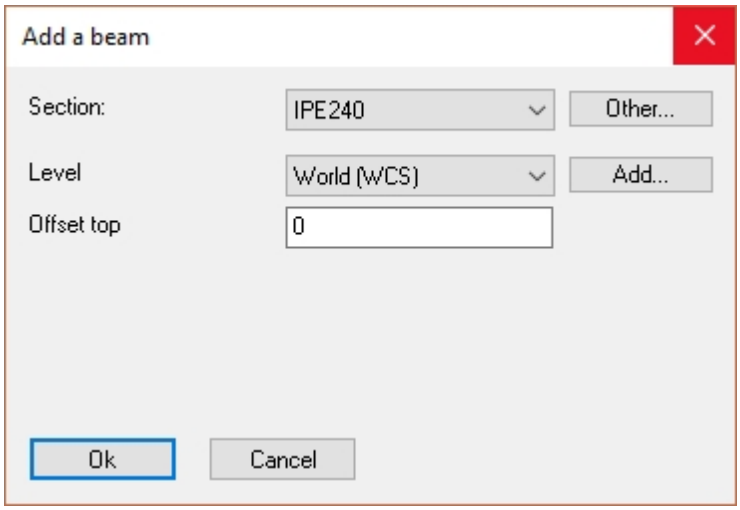
The columns cannot move independently of the grid lines, unless if the profile is disconnected from the macro.

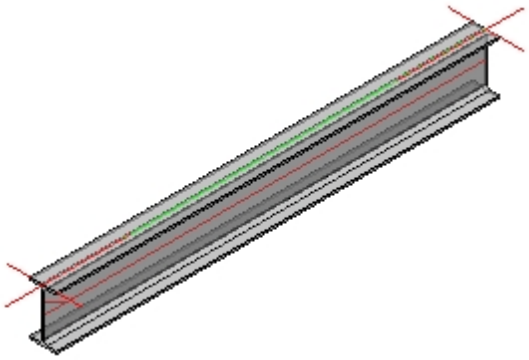
Beam with Points

Command - **PrB_PointsBeam**

Draws a beam constrained between 2 points

On activating this command, the **Add a beam** dialog will appear prompting you to select the profile to be drawn, together with the top level for the beam(s). If levels have already been established, they will be reflected here, additionally, you have the opportunity to create new levels.





You will then be prompted to **Select the first point**, followed by the **next point** - on completion, press <**Enter**>.

The horizontal beams(s) will be drawn at the selected points, but at the height that was chosen in the first step. Moving or adjusting the points will automatically move the beams.

The beams will be constrained to the grid lines that are closest to the selected point. If you selected a point at an offset from a grid line, then this offset will become modifiable in the macro dialog. Just double-click the beam to open this dialog box.

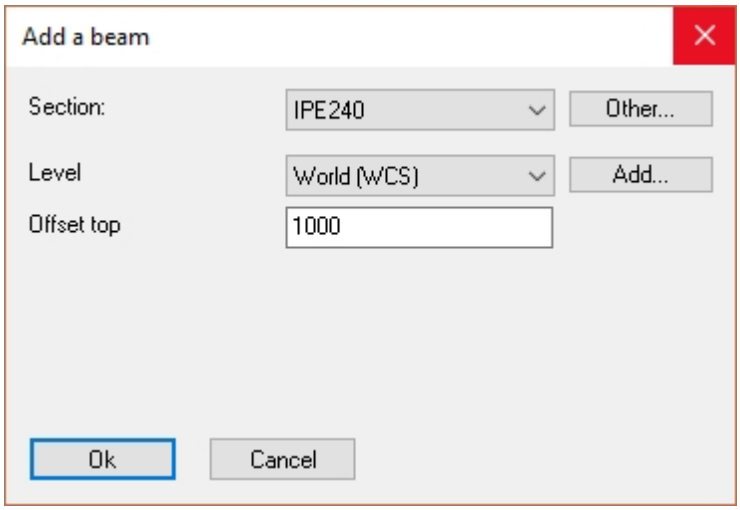
The beams cannot move independently of the grid lines, unless if the profile is disconnected from the macro.

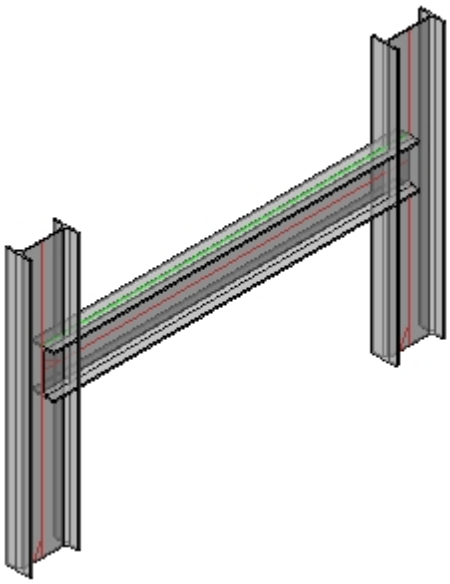
Beam between Columns

Command - **PrB_BeamBetweenCol**

Draws a beam between 2 columns.

On activating this command, the **Add a beam** dialog will appear prompting you to select the profile to be drawn, together with the level for the new beam. If levels have already been established, they will be reflected here, additionally, you have the opportunity to create new levels.





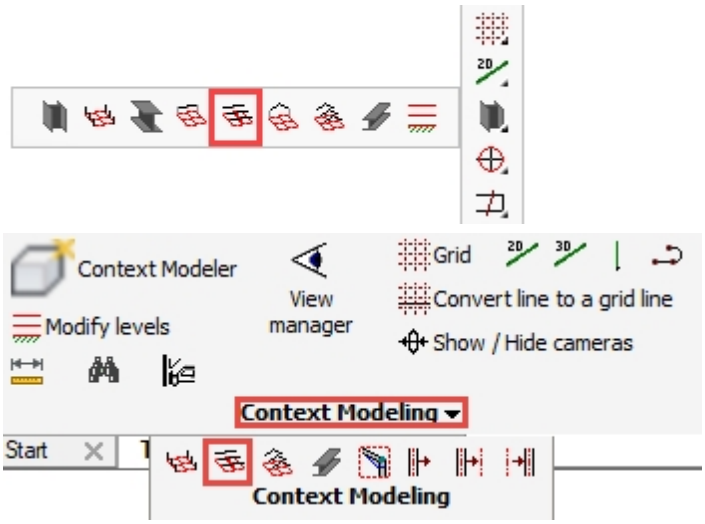
You will be prompted to **Select the first column**, followed by the **second column.**
Then press <**Enter**> and the beam will be drawn at the selected level or offset constrained between the two columns. Moving or adjusting the columns will automatically adjust the beam.

The beams cannot move independently of the columns, unless if the profile is disconnected from the macro.

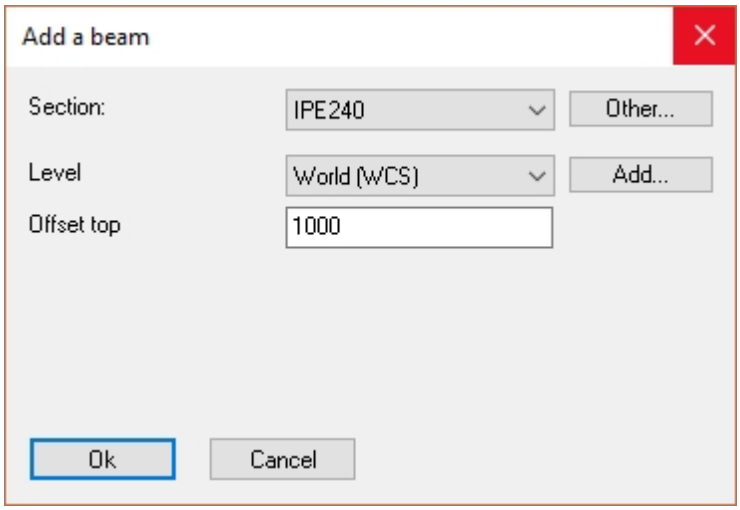
Beams with Grid Lines

Command - **PrB_Beams**

Draws beams between columns

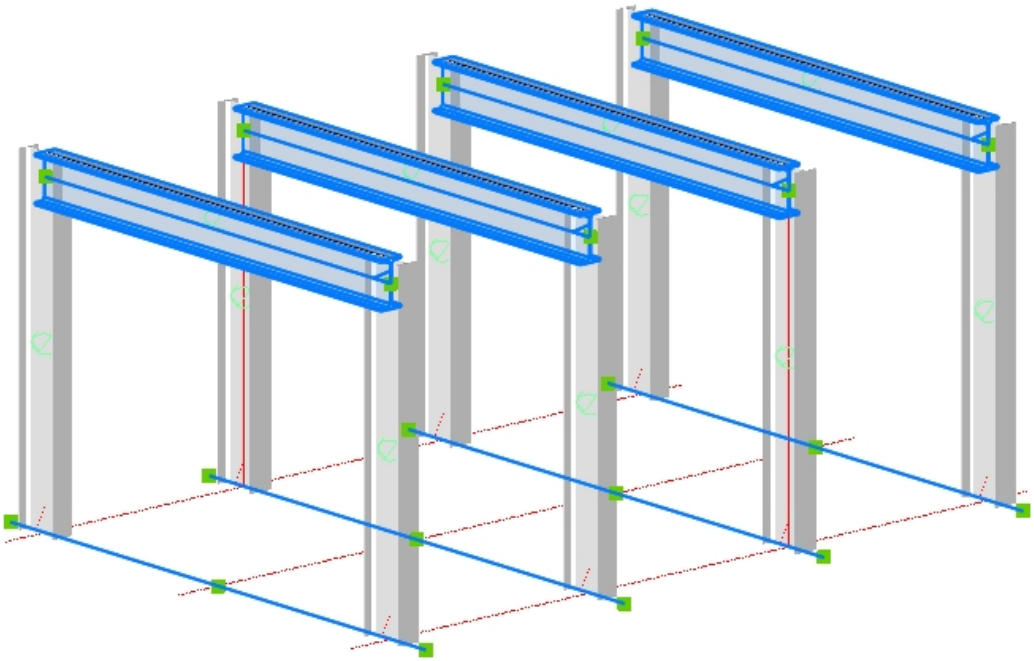


On activating this command, the **Add a beam** dialog will appear prompting you to select the profile to be drawn, together with the level for the new beam. If levels have already been established, they will be reflected here, additionally, you have the opportunity to create new levels.



You will then be prompted to **Select the grid lines,** when done press <**Enter**>.

In this example the blue beams are drawn automatically after selecting the 4 blue grid lines.



A condition is that the grid line that you selected needs to have at least 2 columns located on them.

Another condition is that the height that you choose should not be higher than the height of the columns.

Otherwise the tool will not draw any beams.

Moving or adjusting the columns will automatically adjust the beams.

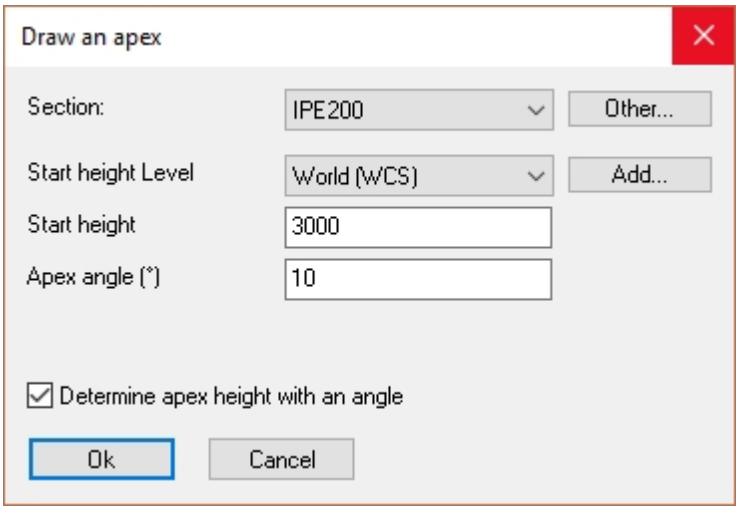
The beams cannot move independently of the columns, unless if the profile is disconnected from the macro.

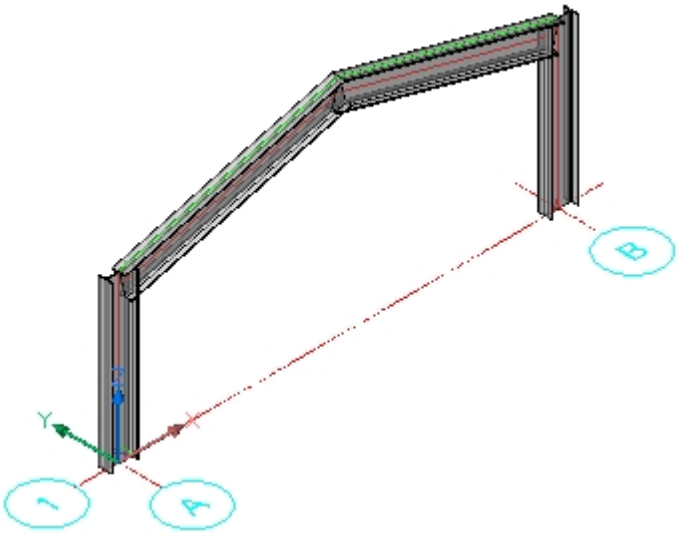
Apex between Columns

Command - **PrB_ApexBetweenCols**

Draws a portal rafter constrained between 2 columns

On activating this command, the **Draw an apex** dialog will appear prompting you to select the rafter profile to be drawn, together with the start height, which in most cases will be the top of the column. You have the opportunity to select the apex height (from the start height), or if the **Determine apex height with an angle** checkbox is checked you may enter the rafter angle.





You will then be prompted to **Select the first column**, followed by the **second column.**

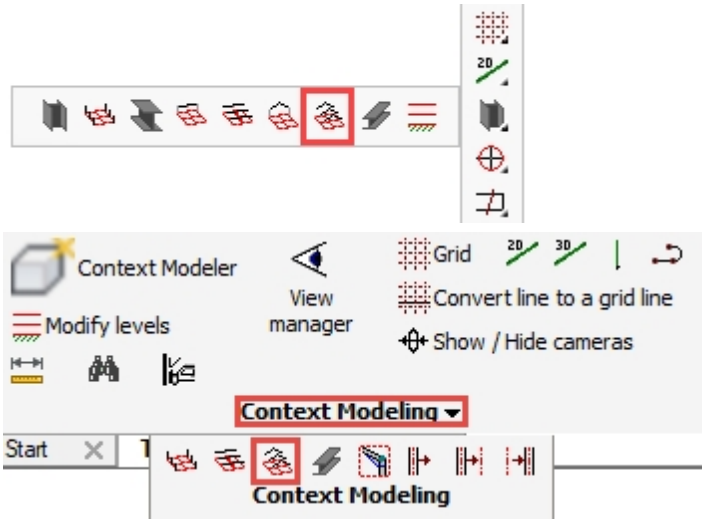
On completion, the rafter will be drawn at the selected level and either the apex height or angle constrained between the two columns. Moving or adjusting the columns will automatically adjust the rafter.

The rafter cannot be moved independently of the columns, unless if the profile is disconnected from the macro.

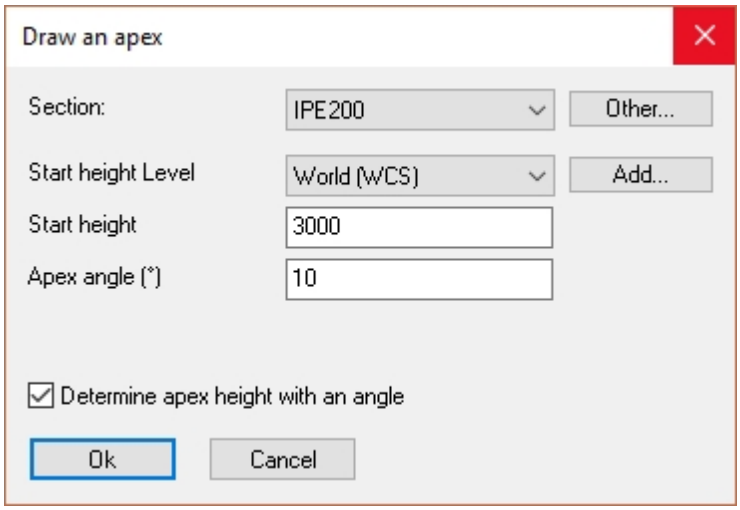
Apex with Grid Lines

Command - **PrB_Apexes**

Draws portal rafters constrained between 2 columns.

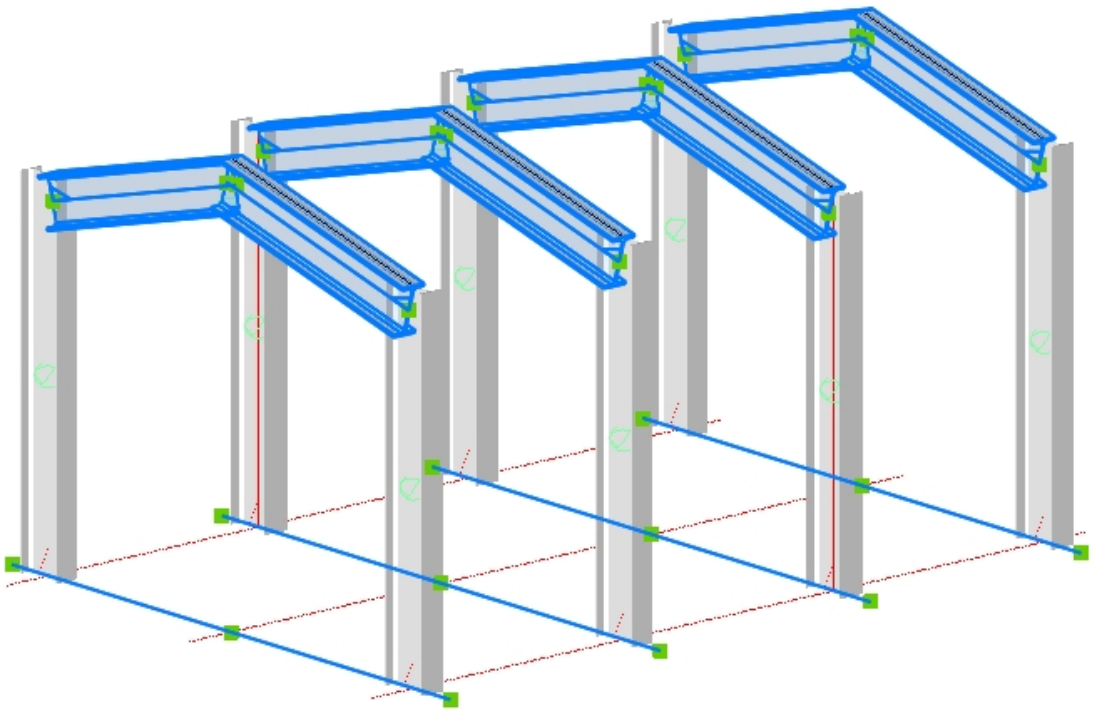


On activating this command, the **Draw an apex** dialog will appear prompting you to select the rafter profile to be drawn, together with the start height, which in most cases will be the top of the column. You have the opportunity to select the apex height (from the start height), or if the **Determine apex height with an angle** checkbox is checked you may enter the rafter angle.



You will then be prompted to **Select the grid lines,** when done press <**Enter**>.

In this example the blue rafters are drawn automatically after selecting the 4 blue grid lines.



A condition is that the grid line that you selected needs to have at least 2 columns located on them.

Another condition is that the height that you choose should not be higher than the height of the columns.

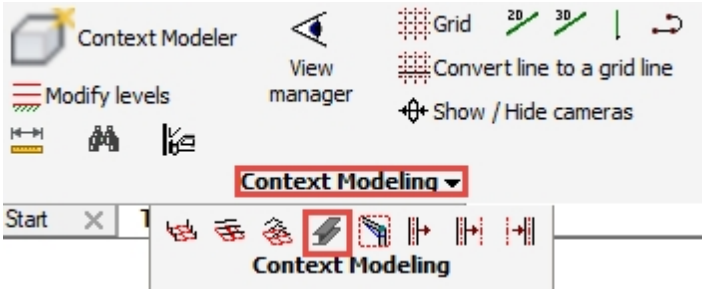
Otherwise the tool will not draw any rafters.

Moving or adjusting the columns will automatically adjust the rafters.

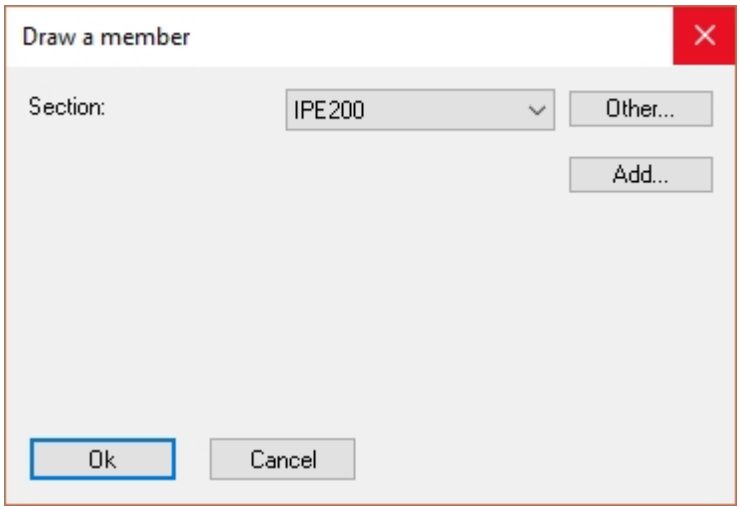
The rafters cannot move independently of the columns, unless if the profile is disconnected from the macro.

Free Member

Command - **PrB_FreeMember**



On activating this command, the **Draw a member** dialog will appear prompting you to select the profile to be drawn.

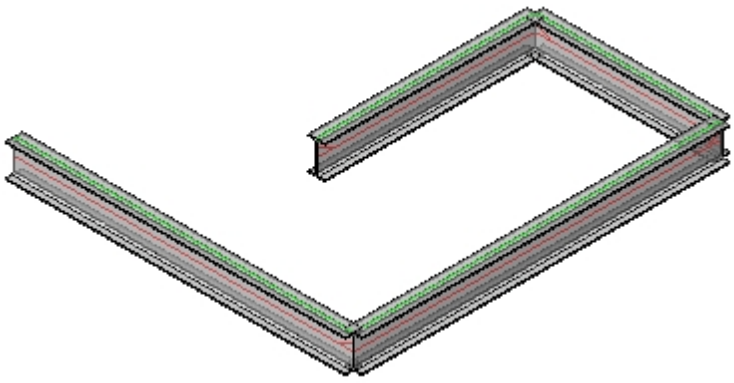


You will then be prompted to **Select the first point,** followed by the **next point**. Here you may enter as many points as you wish, on completion, press <**Enter**> and the profile(s) will be drawn.

There are some limitations to the points that you can use :

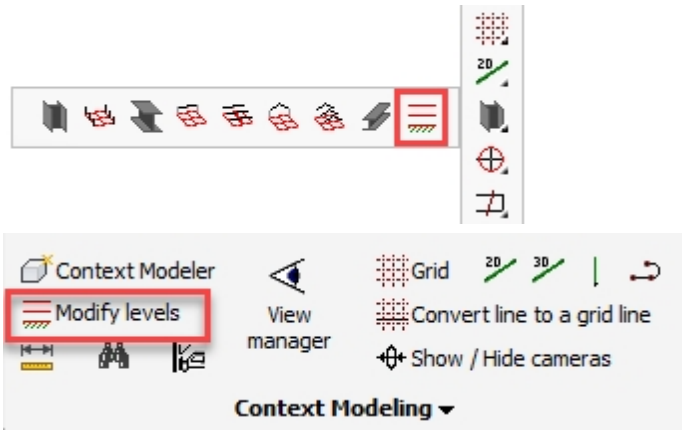
- When you choose a point in free space, the tool will project the point flat to the world XY plane, and will constrain the point to the closest grid line
- Choosing points in 3D is possible if you choose to use object snap on another entity.
  But then only endpoints will work. Midpoints will not work.
  This will only work on lines, plates, profiles and structures.
  The new profile will stay linked to the endpoint that you chose.

The profile(s) cannot be moved independently of the chosen points.



Modify Levels

Command - **PrB_ModifyLevels**



This command allows us to add, remove or modify levels.

See Modify Levels

Bolts / Holes

This Chapter covers the placing of bolts and holes, sub-chapters include:

- Bolt Standards
- Draw Bolt
- Bolts on a Plane
- Bolts on Touching surfaces

- Galvanizing Holes
- Check for New Holes
- Remove Holes and Cuts
- Slotted Hole
- Hole <-> Threaded Hole
- Hole <-> Countersunk Hole
- Hole <-> Blind Hole

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 options in the assembly, 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 parts 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 Properties for a detailed explanation of the bolt properties that can be changed.

Shop Bolts

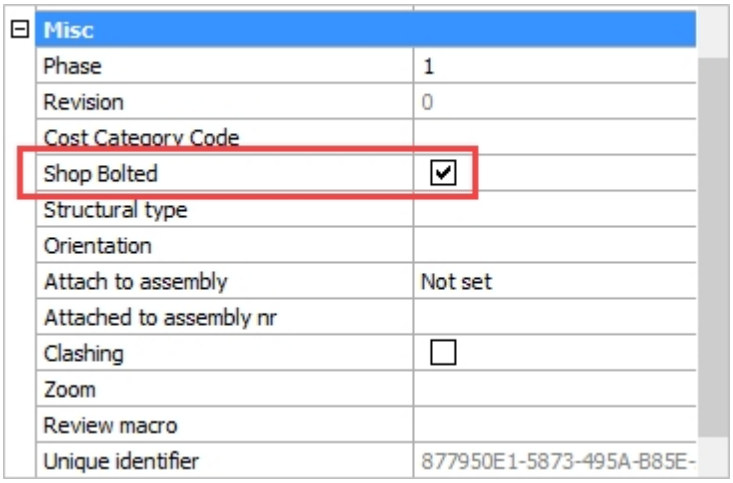
By default, all bolts are designated as 'Site Bolts'. To convert them to 'Shop Bolts' select from the top menu: **Modify** > **Properties**

Select the bolt(s) to be converted, and activate the **Shop Bolted** checkbox

The bolt(s) will be drawn as part of the assembly drawing, and will appear on the Bill of Materials.

This property will also influence the number of the assembly.

It's possible to influence which assembly the bolt is attached to.

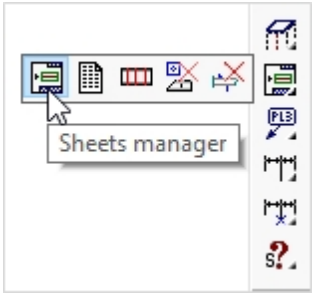


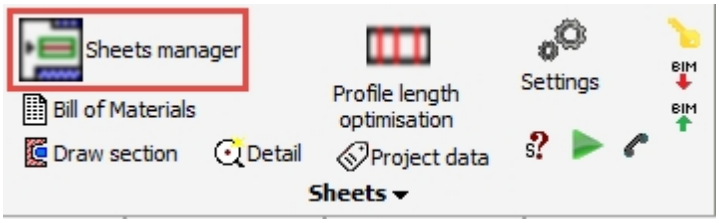
Influencing which Assembly a Bolt is attached to

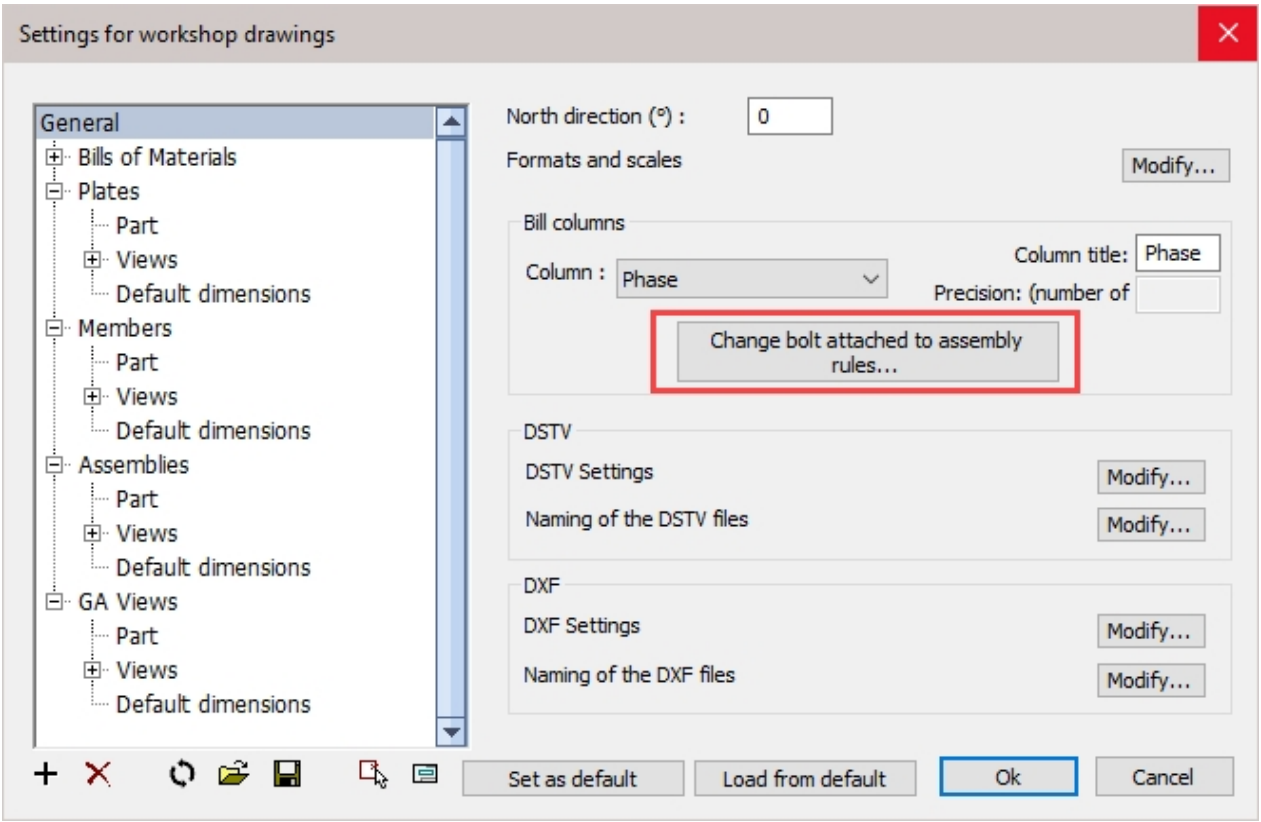
A bolt is always attached to one reference assembly, even if it connects 2 or more assemblies together.

We call this the reference assembly of the bolt. This reference assembly is important, because it will determine on which assembly drawing the bolt will appear. (a bolt will never appear on 2 assembly drawings to avoid counting the bolt more than once)

From the **Sheets Manager** / **Settings** / **General** / click on Change bolt attached to assembly rules - where you can influence which assembly the bolt is attached to.





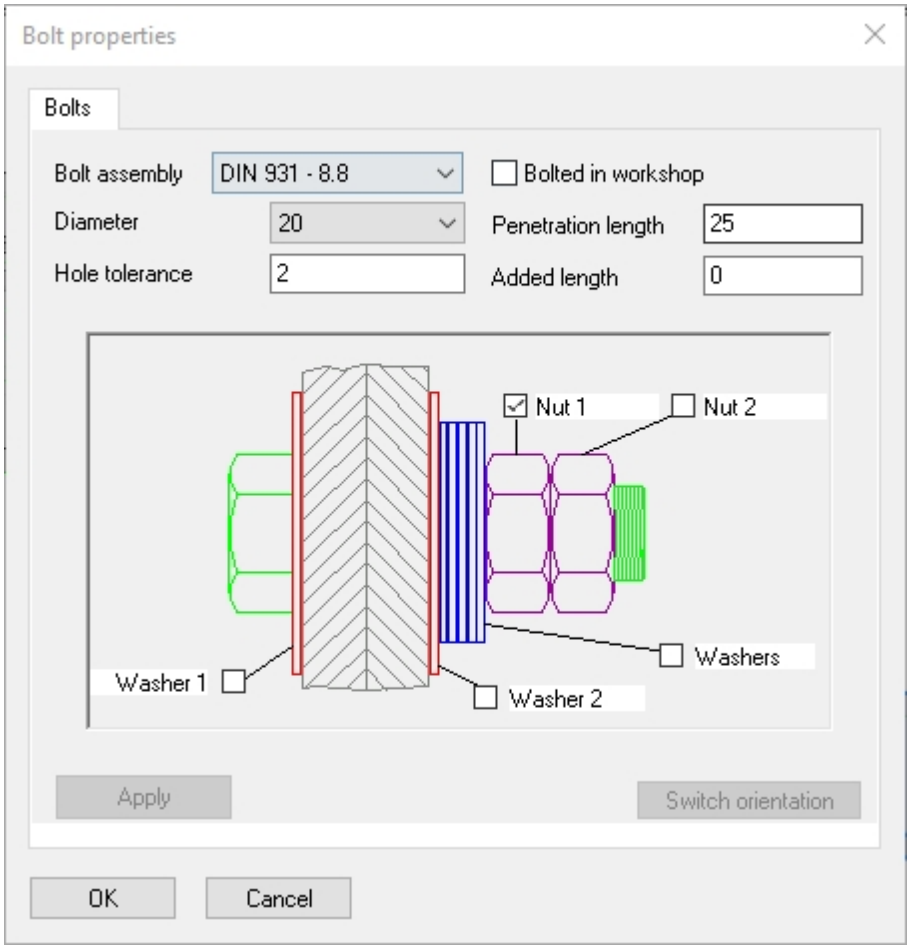


Draw Bolt

Command - **PrB_ArxBolt**

This command draws a single bolt through a plate or profile.

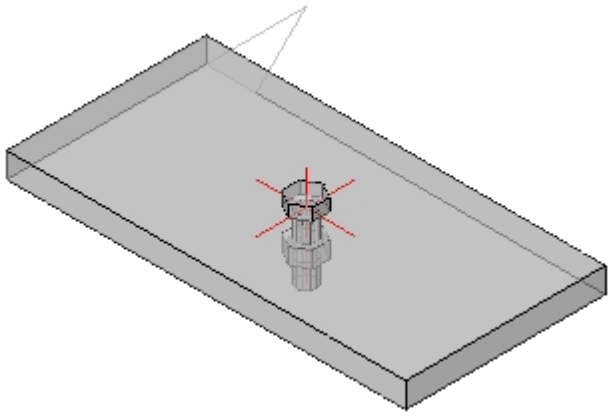
At the command line prompt, select a part, then the following dialog will appear:



Here you are able to select:

- The Bolt-assembly
- Bolt diameter
- Penetration length (Grip) = the plate thickness
- Hole tolerance (Bolt hole clearance)
- Added length - The bolt length is calculated as: (Length = Penetration length (Grip) + Washers + Nuts + Added length)

If the 'Bolted in workshop' checkbox is checked, then it is assumed that the assembly is at least partly bolted in the shop.



Bolts on a Plane

Command - **PrB_BoltsOnPlane**

This command draws a range of bolts on a plate or profile's surface.

On selecting this command, you will be prompted to select a plane on a plate or profile.

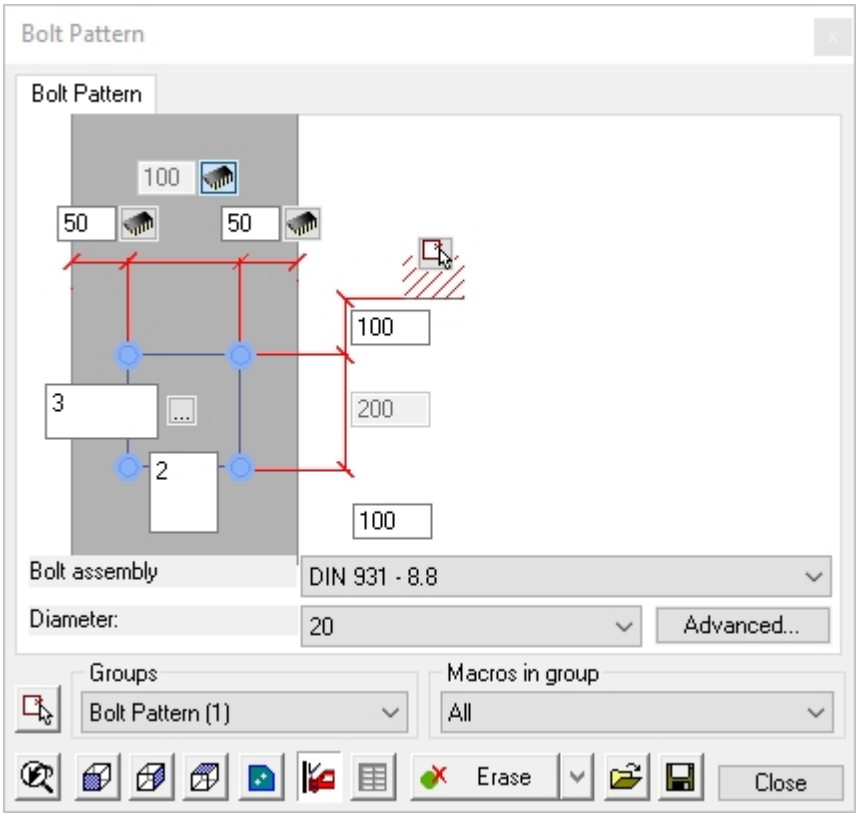
You can select this plane by moving the cursor inside a plane, and then clicking the left mouse button.

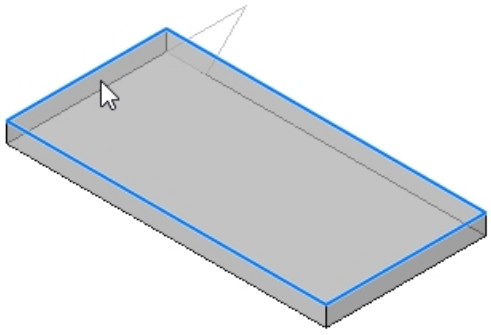
The circumference of the plane will be drawn thicker so that you can see the selected plane.

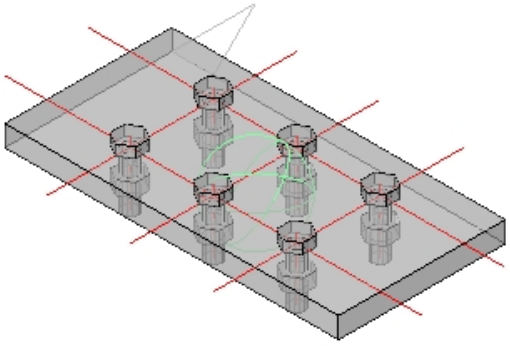
To select a different plane, you need to move the cursor and left-click the mouse again.

To accept the plane selection, press the **<Enter>** key or right-click the mouse.

A dialog will appear prompting you to enter the dimensions for the bolt group.


Bolt Pattern
Bolt Pattern
100
50
50
100
3
200
2
100
Bolt assembly
DIN 931 - 8.8
Diameter:
20
Advanced...
Groups
Bolt Pattern (1)
Macros in group
All
Erase
Close





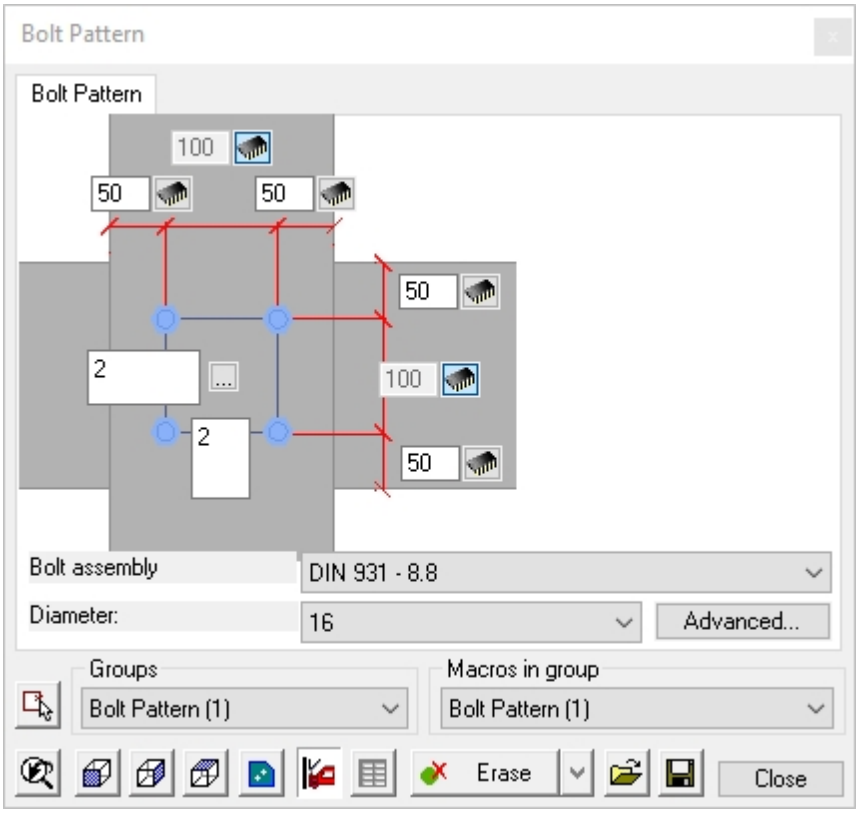
Bolts on Touching Surfaces

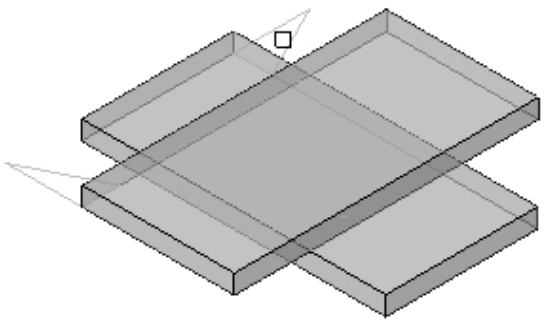
Command - **PrB_AutoBolts**

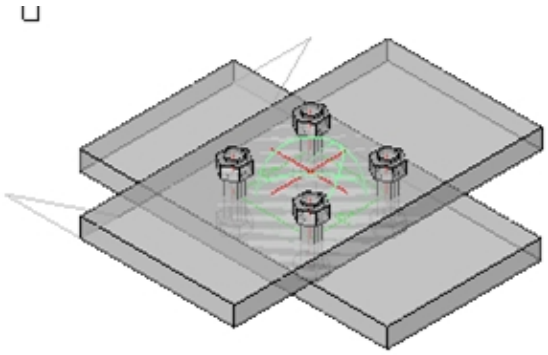
This command automatically draws bolts where plates or profiles are touching each other.

On selecting this command, you will be prompted to select the entities which are to be bolted.

A dialog will appear prompting you to enter the dimensions for the bolt group.







Galvanizing Holes

Command - **PrB_AutoGalvaHoles**

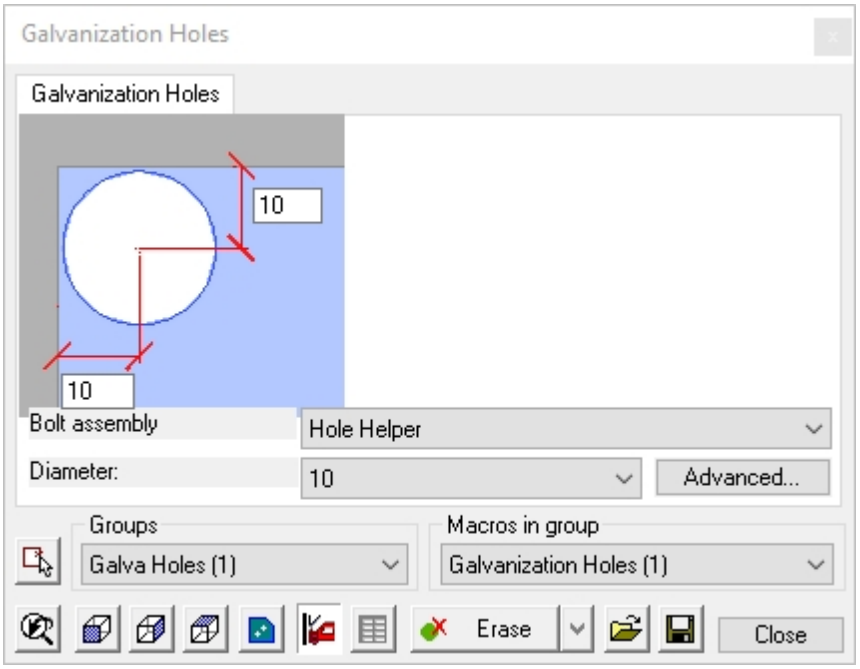
This command draws drain holes in end-plates of hollow sections or I shaped sections.

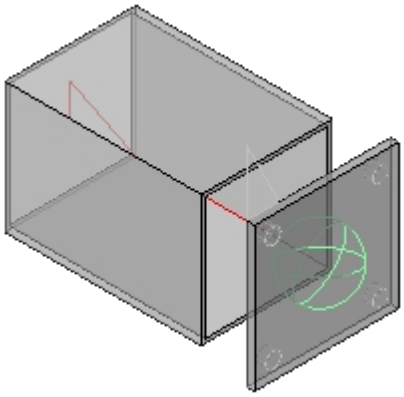
On selecting this command, you will be prompted to select the profile and the end-plate. You could also select the entire drawing.

Parabuild will search for all the 'corners' in which pockets of liquid zinc are possible.
This tool will zoom in on each group of galvanizing holes that it has identified. The holes will be automatically drawn and you will be prompted to either accept the proposed holes or not.

After all the groups have been iterated, the customization dialog will appear, prompting you to enter the hole diameter and location relative to the inner profile of the (hollow) section.

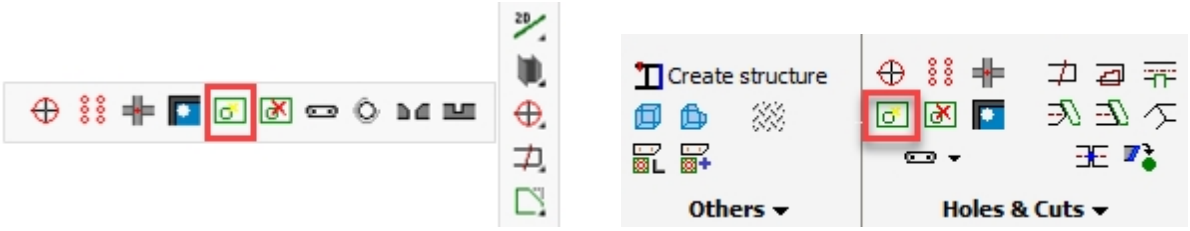
Parabuild uses bolts with the name *Hole helper* to draw these galvanizing holes - These bolts are not visible and will not be shown in the bill of materials.





Check for New Holes

Command - **PrB_HoleCheck**



This command will check for and draw missing holes where bolts are colliding with parts.

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

On selecting this command, you will be prompted to select the beams, plates, and bolts.

Any missing holes will automatically be identified and corrected.

Note!

If the bolt is too close to the edge of a beam or plate, or if it's too close to another bolt, it will be colored yellow. 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 Global Settings dialog box.

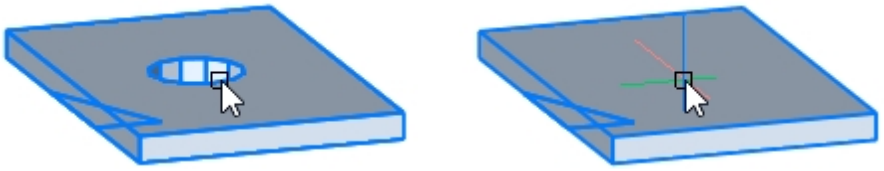
Bolts could also be shown in red when it was impossible to draw a hole. For more information about this see the Clash check.

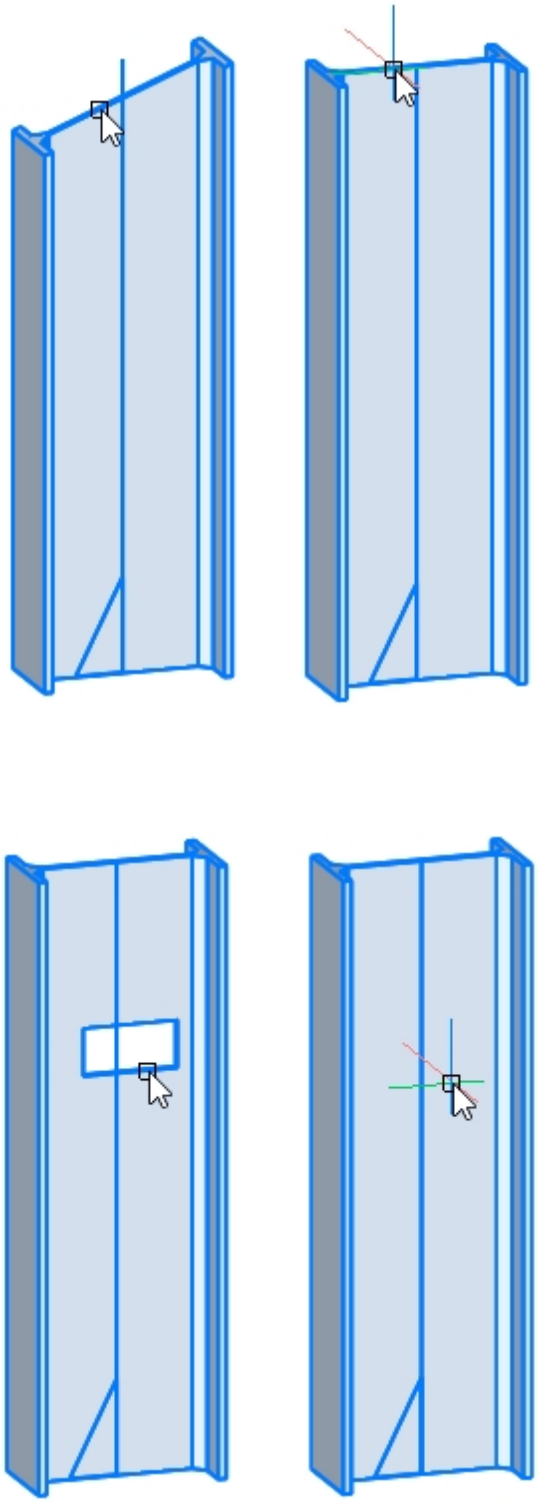
Remove Holes and Cuts

Command - **PrB_RemoveCutOrHole**

This tool will remove any hole or cut that you select.

On selecting this command, you will be prompted to select the edge of the hole or cut to be removed.





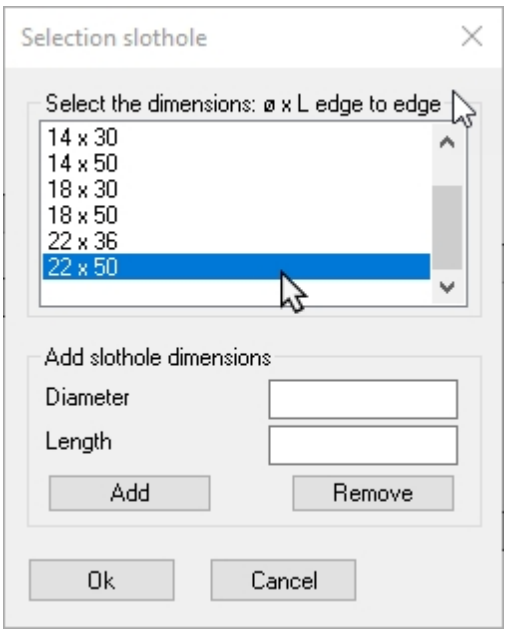
Slotted Hole

Command - **PrB_SlotHoleDlg**

This command converts a round hole to a slotted hole.

Tip It is easier and quicker to modify a hole from within the properties of the bolt instead of this command.

At first, Parabuild will ask you to select the hole you want to convert into a slotted hole.

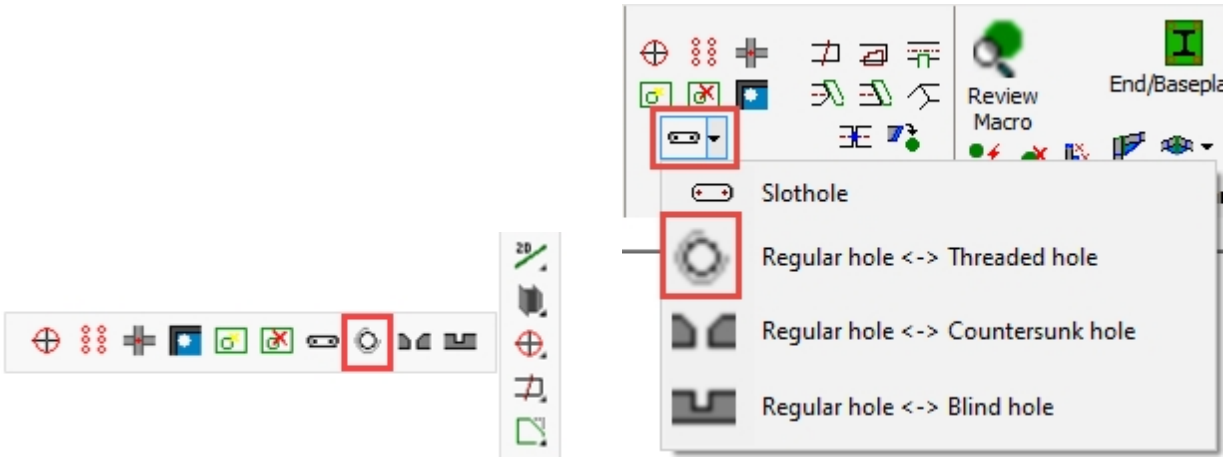


In the dialog, you can then select a specific measurement or with **Other** specify your own measurements.

After that, you need to specify the rotation angle by selecting a point relative to the axis of the slotted hole.

Hole <-> Threaded Hole

Command - **PrB_ChangeHoleThread**



Tip It is easier and quicker to modify a hole from within the properties of the bolt instead of this command.

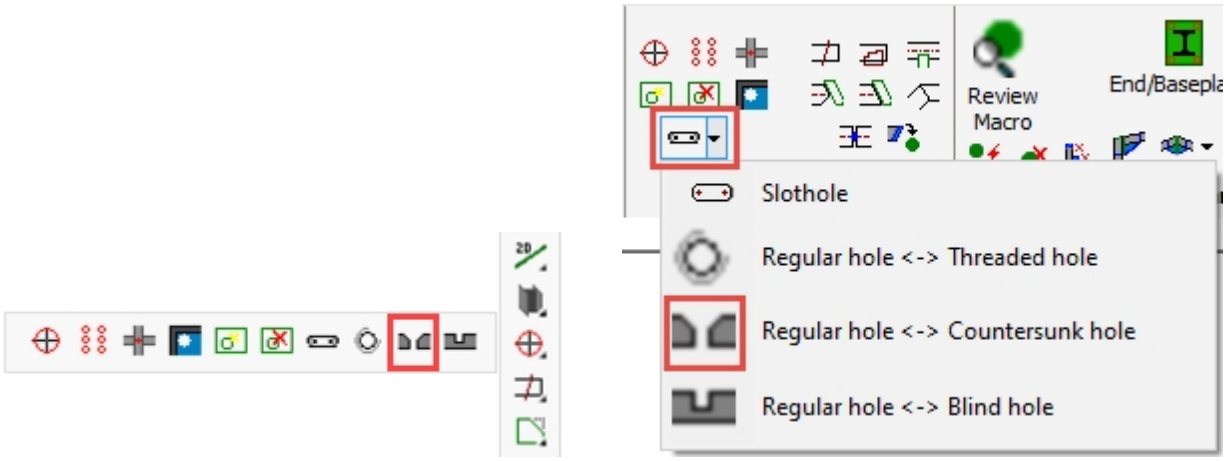
With this command you can convert a regular hole into a threaded hole and vice-versa. For illustration purposes, extra lines will be added to simulate the thread of the hole.

After starting this command you should select the edge of a hole. It will be converted instantly.

For more information about how Parabuild annotates threaded holes in the shop drawings, refer to the Global settings dialog.

Hole <-> C/Sunk Hole

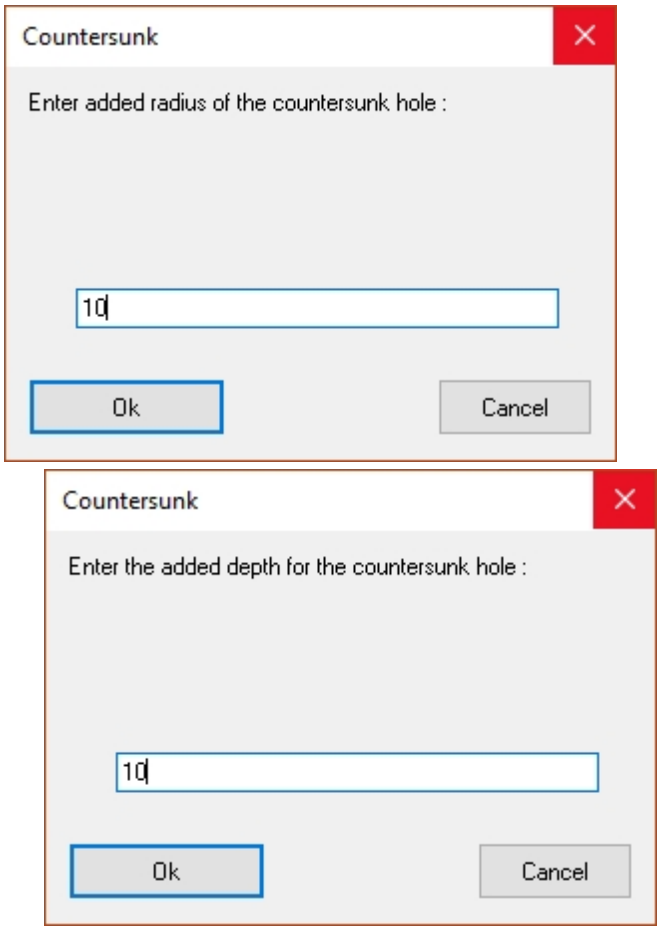
Command - **PrB_ChangeHoleCountersunk**



Tip It is easier and quicker to modify a hole from within the properties of the bolt instead of this command.

With this command you can convert a regular hole into a threaded hole and vice-versa.

At the command line prompt: **Select a hole or a bolt,** which will then bring up the following window:



Here you are prompted to Enter the *added radius* of the countersunk hole, press **Ok** and the second window will appear prompting you to Enter the *depth* for the countersunk hole - press **Ok**

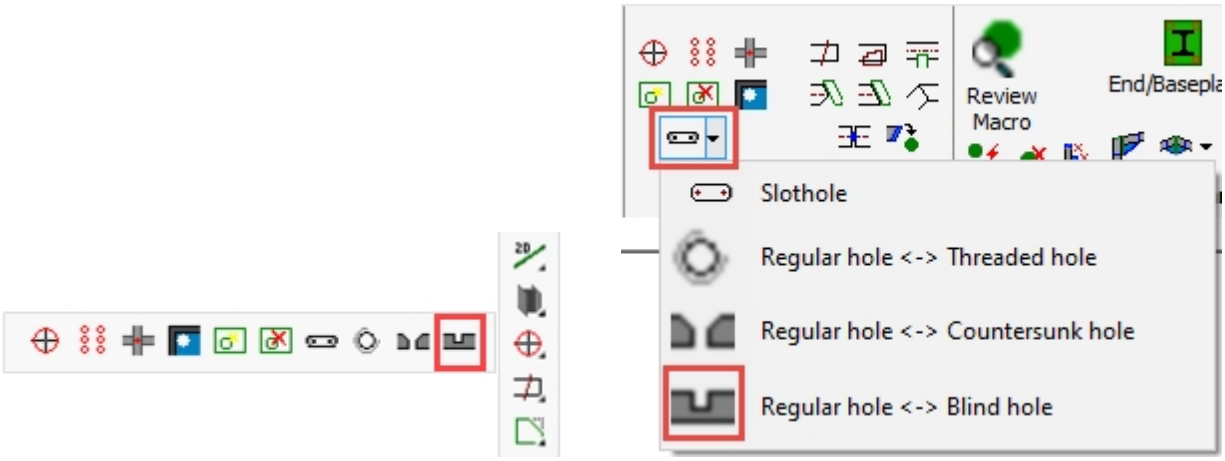
The countersunk hole is shown in 3D.

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 counter-bore 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.

Hole <-> Blind Hole

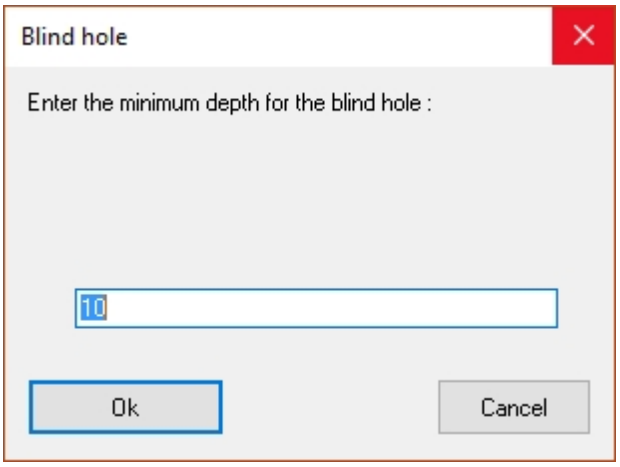
Command - **PrB_ChangeBlindHole**

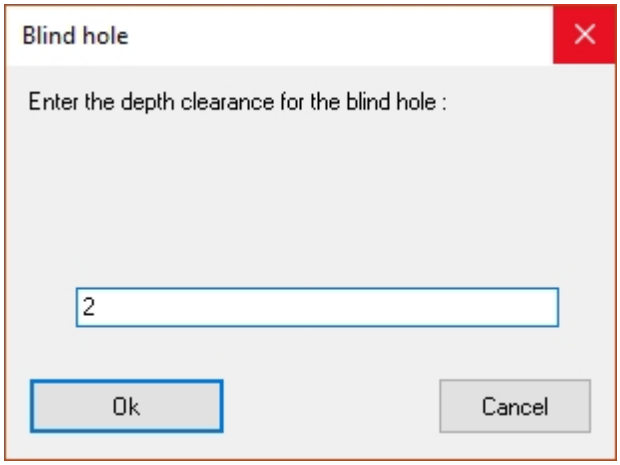


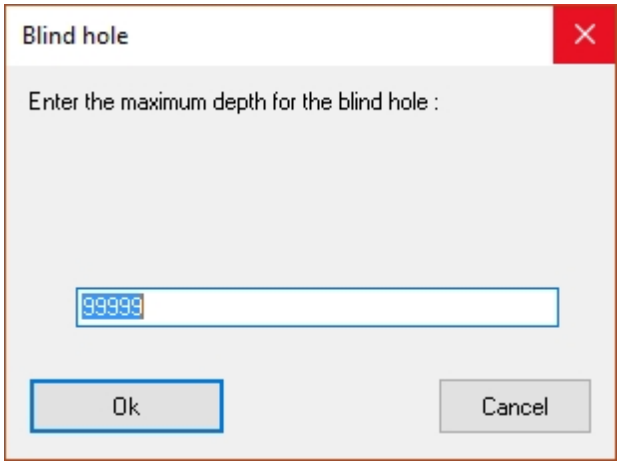
Tip It is easier and quicker to modify a hole from within the properties of the bolt instead of this command.

With this command you can convert a regular hole into a threaded hole and vice-versa.

At the command line prompt: **Select a hole or a bolt,** which will then bring up the following windows:







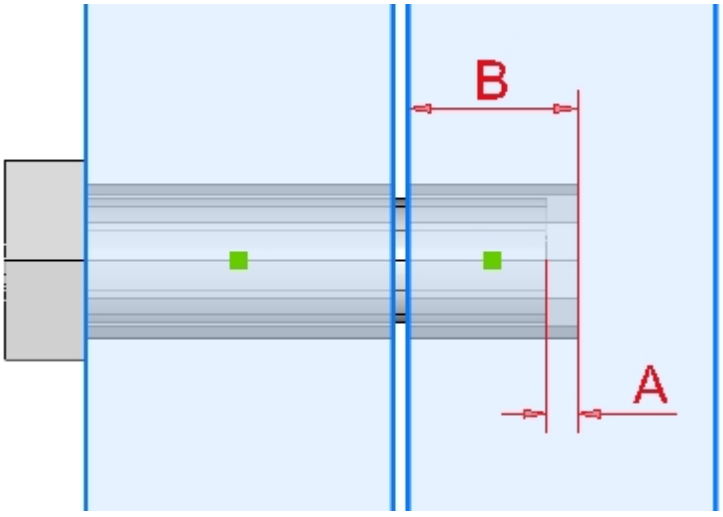
A blind hole is a hole with limited depth. In steel material, this hole should be threaded.

The bolt should not have a nut.

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).

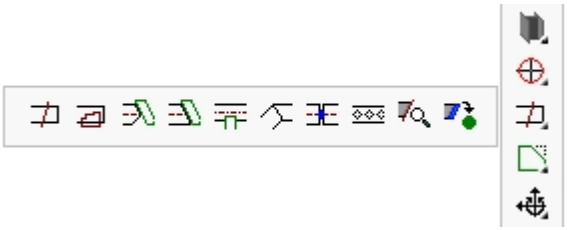
The minimum and maximum depth of the hole can be adjusted (B on the illustration).



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.

When trying out this tool, choose a bolt standard that has an adequate amount of bolt lengths. If the standard does not have the right bolt length then this tool can't draw the blind hole.

Cutting

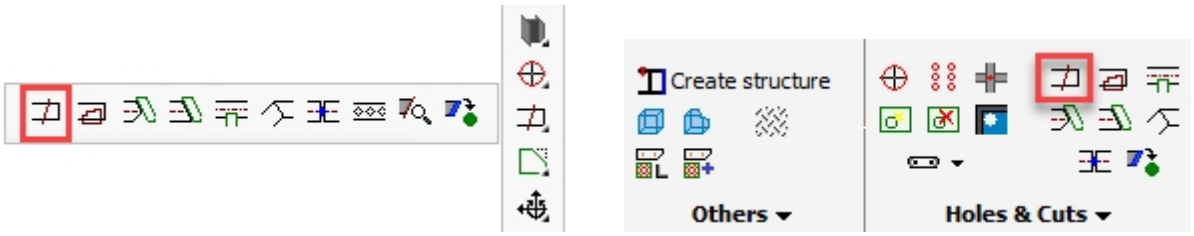


This chapter covers the modification of elements by cutting. The sub-chapters include:

- Cut on a Line
- Cut on a PolyLine
- Cut against an Element
- Cut out against a Profile
- Miter Joint
- Break Profile
- Hex. Holes (Castellated)
- Edit a Profile's Cuts
- Add Cut to macro

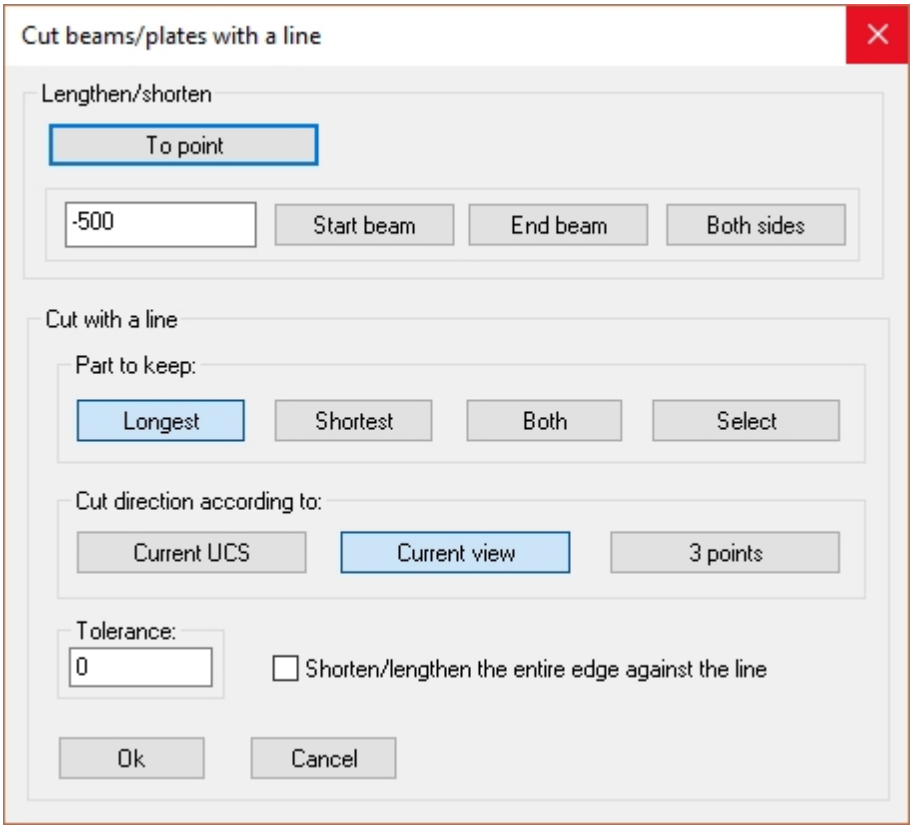
Cut by a Line

Command - **Prb_LineCut**



This command allows us to modify the start or end of a profile by shortening or extending it, but also by adding non-perpendicular cuts to it.

At the command line prompt, the following dialog will appear:



Here there are 2 options:

- To lengthen or shorten the plate or profiles
- To cut the plate or profile along a line

Shorten/Lengthen:

Start Beam *End Beam* *Both Sides*

First you enter the distance to be shortened or extended. A positive value will extend the entity, while a negative value will shorten it.

Choose from **To Point**, **Begin beam** and **End beam**.

To point - you have to indicate a reference point on screen and the beam will be cut to that point.

Start beam will shorten/lengthen the beam according to the entered value. The start beam position is the origin of the entity (where the triangle is located)

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

Both sides - will shorten/lengthen the beam at both ends

Cut using a line:

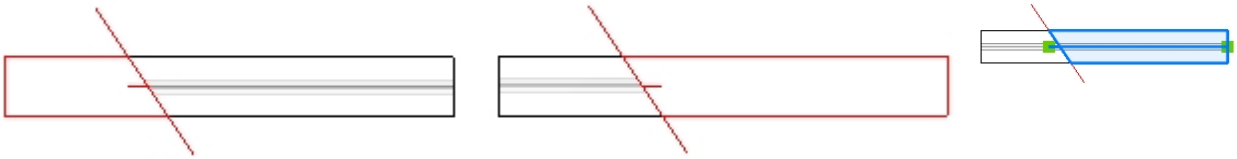
Part to keep - **Longest**: Will cut away the shortest section, keeping the longest measured from

the cut line

Shortest: Will cut away the longest section, keeping the shortest measured from the cut line

Both: Will cut the beam or plate at the cut line, keeping both sides

Select: Will allow you to manually select which side to keep



Longest *Shortest* *Both*

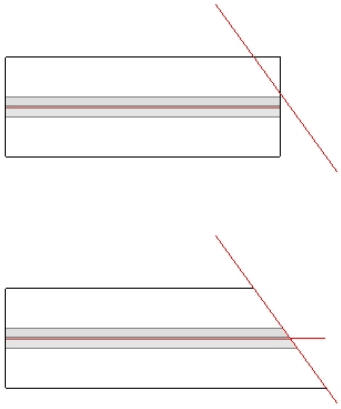
Orientation cutting plane:

- **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.

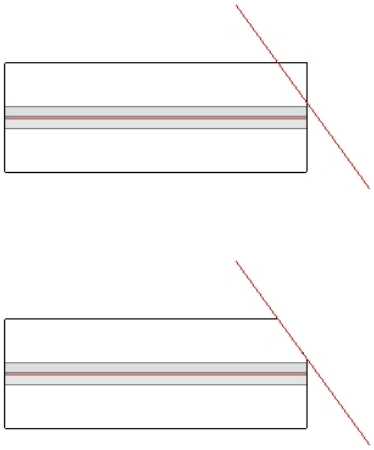
Tolerance: You can also set an offset between the line and the profile.

Shorten/lengthen the entire edge against the line when enabled this will shorten or extend the profile during the cut operation \

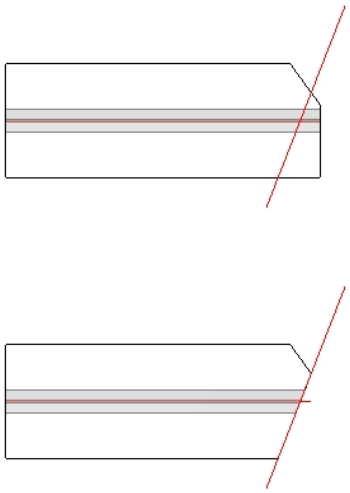
This is an example with the option **Shorten/lengthen** enabled :



This is an example with the option **Shorten/lengthen** disabled :



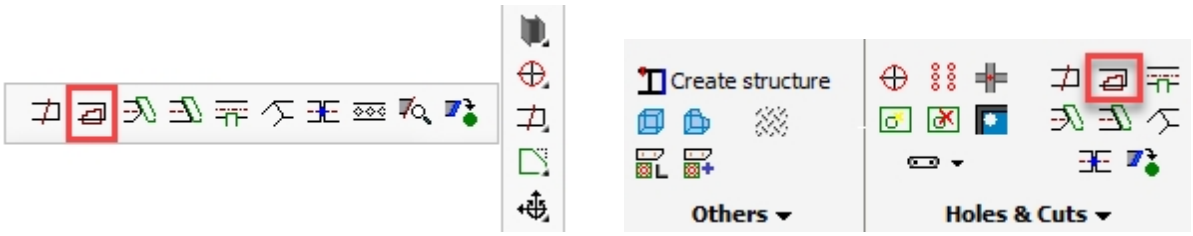
This is a second cut operation on the same profile with the option **Shorten/lengthen** disabled :



With the **Shorten/lengthen** enabled it would not be possible to achieve this miter.

Cut by a PolyLine

Command - **Prb_CutByPoly**



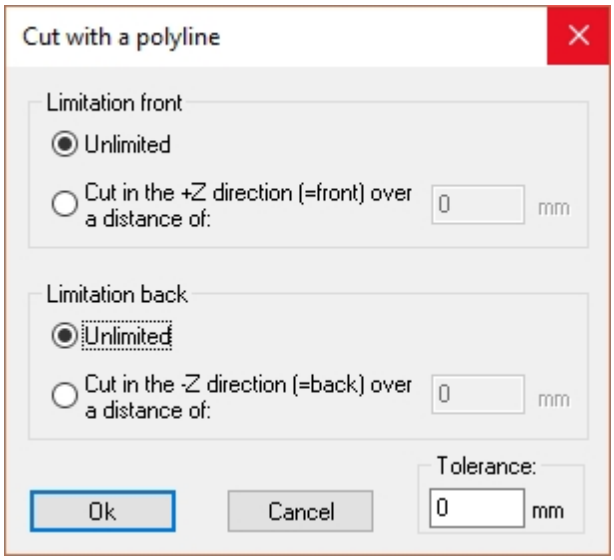
This command allows us to add internal copes to a profile or a plate.

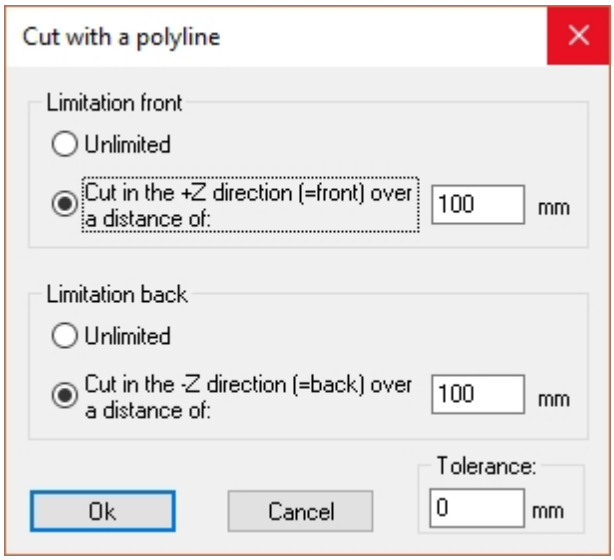
At the command line prompt, first select the plates and/or profiles to be cut, and then press <**Enter**>.

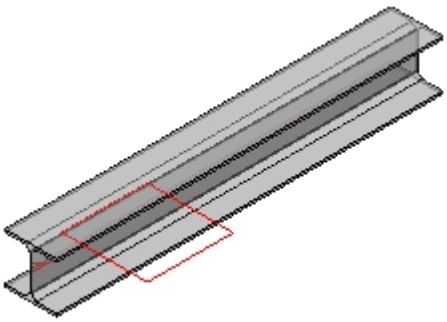
You will then be prompted to indicate the PolyLine.

The PolyLine may be a Circle, a Rectangle, or a user-defined PolyLine.

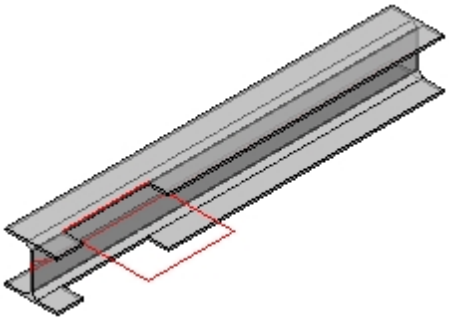
The following dialog will appear where you have the option of making the cut Unlimited - which will cut through the entity entirely - or alternatively, you may prescribe the depth of the cut.



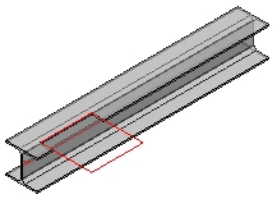




The PolyLine before applying the cut



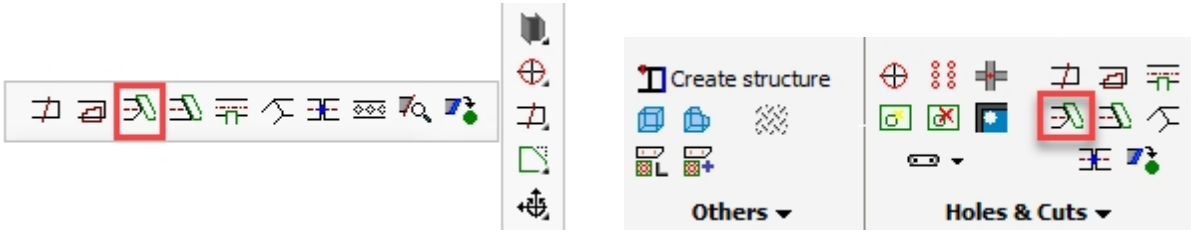
Cut result with unlimited depth



Cut result with limited depth in Z Direction

Cut against an Element (shorten only)

Command - **PrB_AddPrPlCut**



This command allows us to shorten a profile against another part. The new end cut of the profile

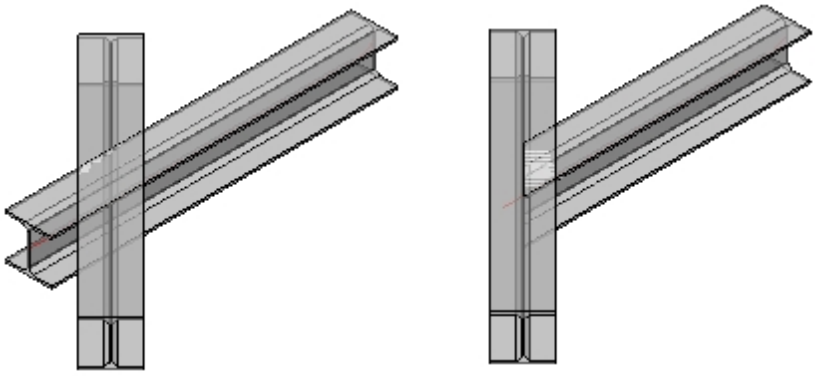
will be aligned to the other part.

On activating this command, you will be prompted to: **Select the plates or beams you would like to cut.**

You will be further prompted to indicate: **The cutting plate or profile.**

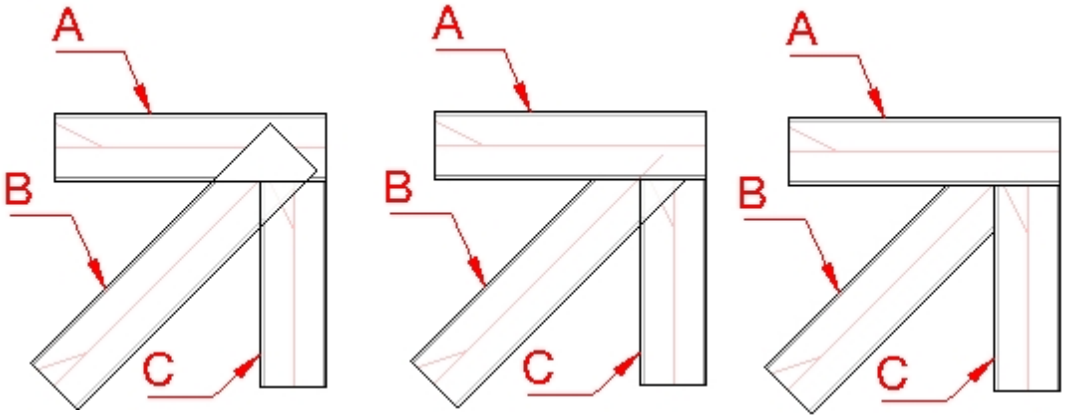
The profile or plate will immediately be cut to the main face of the cutting plate/profile.

This tool will cut against the web of I or U shapes.
If you need to cut to another face, then use the *Manually select cutting plane* option in the Add cut to macro tool.



The profiles before applying the cut *The profiles after applying the cut*

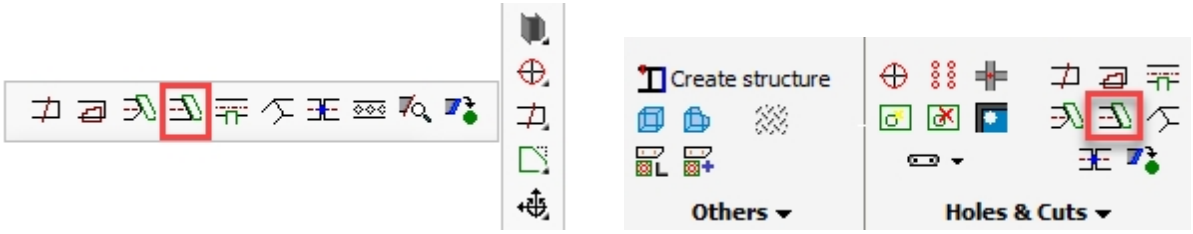
This tool will only shorten the profile. Therefore it becomes possible to combine multiple end-cuts on a profile at the same end :



The profiles before applying the cuts *The result after cutting profile B against profile A* *The result after cutting profile B against profile C*

Cut End against an Element (shorten and extend)

Command - **PrB_PrPlCut**



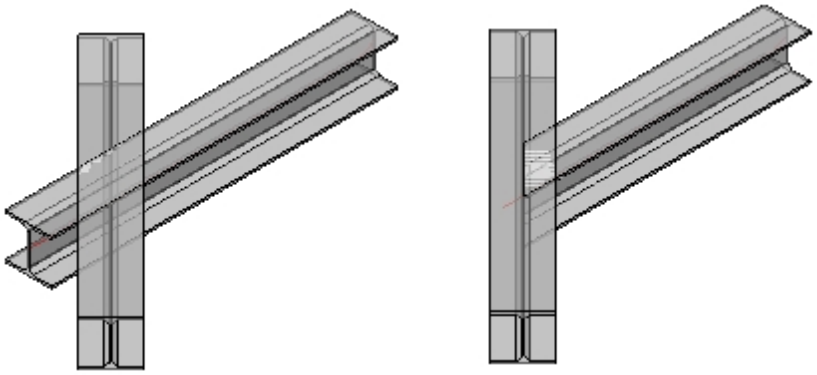
This command allows us to shorten or extend a profile against another part. The new end cut of the profile will be aligned to the other part.

On activating this command, you will be prompted to: **Select the plates or beams you would like to cut.**

You will be further prompted to indicate: **The cutting plate or profile.**

The profile or plate will immediately be cut to the main face of the cutting plate/profile.

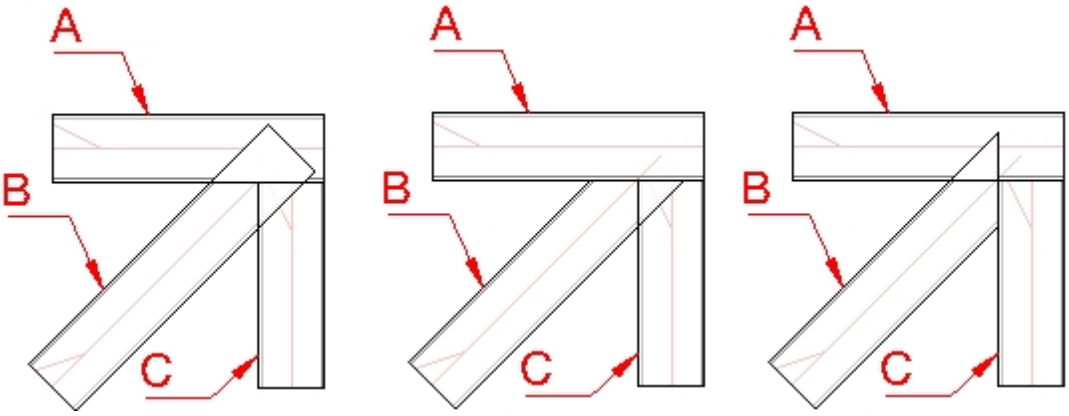
This tool will cut against the web of I or U shapes.
If you need to cut to another face, then use the *Manually select cutting plane* option in the Add cut to macro tool.



The profiles before applying the cut *The profiles after applying the cut*

This tool will shorten and extend the profile when needed. Therefore it is not possible to combine multiple end-cuts on a profile at the same end.

However, this command will automatically correct profiles that are too short :

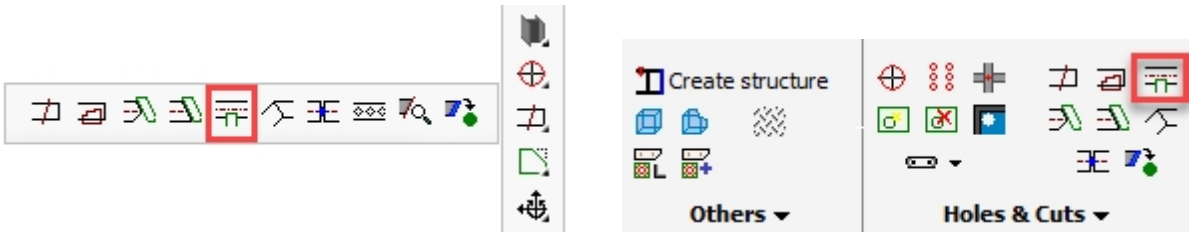


The profiles before applying the cuts | *The result after cutting profile B against profile A* | *The result after cutting profile B against profile C*

If you need to combine two end-cuts, then use this command first and then use the Cut against an element (shorten only) command for the second end-cut.

Cut out against a profile

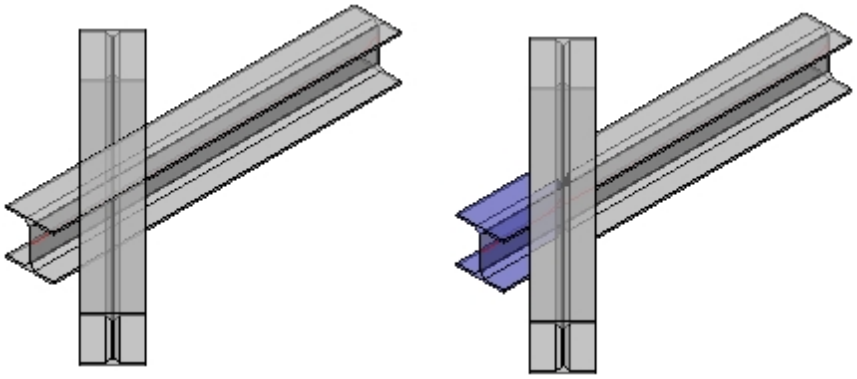
Command - **PrB_CutInProfile**

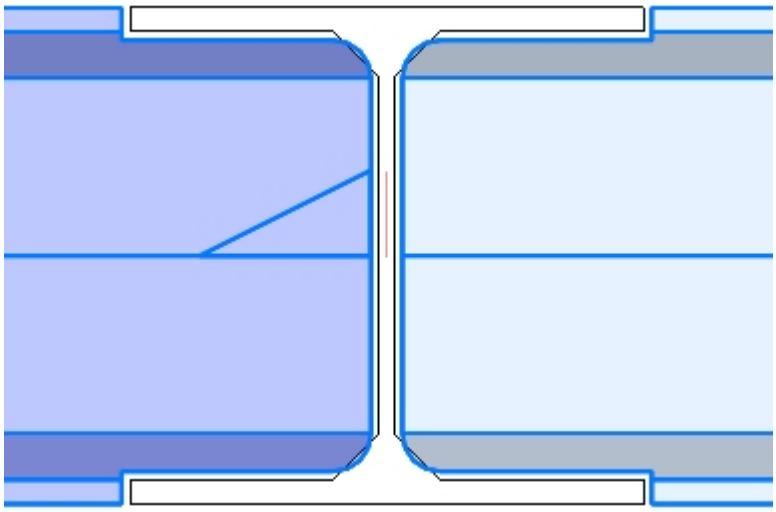


This command will draw a cut that matches the contours of the entity determining the cut.

The tool will ask for an offset value at the command line prompt. This value represents the clearance between the cut entity and the profile that determines the cut.

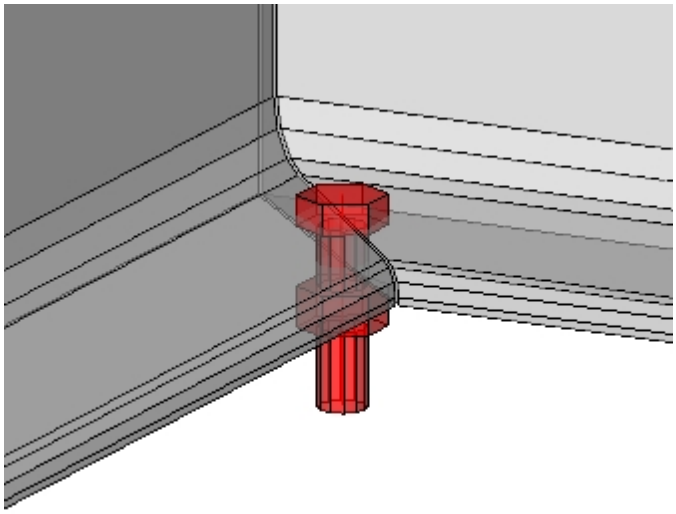
If the plate or beam passes through the entity that determines the cut, the plate or beam will be split up into 2 parts and both parts will be cut against the determining part.



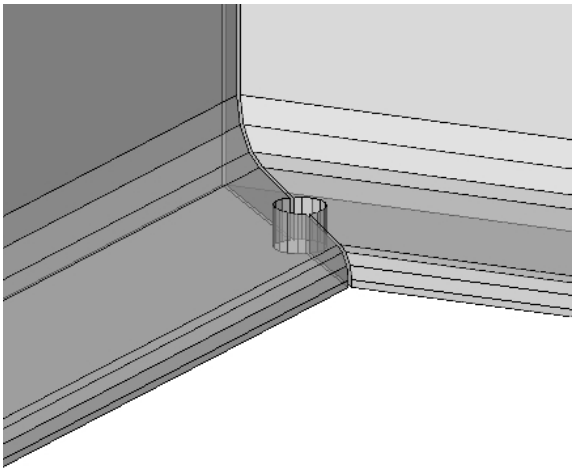


This tool will also accept a bolt for the cut definition

A practical use case for this is when a part of a hole needs to be drawn on a miter.
The Parabuild bolt/hole tools refuse to draw a part of a hole like this.
But with this cutting tool we can still draw this :



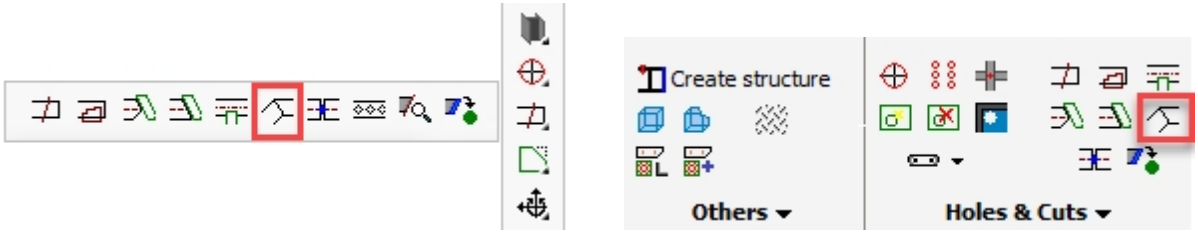
A bolt drawn in the middle of a miter.
The holes are not drawn automatically in such case.



By selecting the bolt as defining object, we can draw 2 half holes in the channels.

Miter Joint

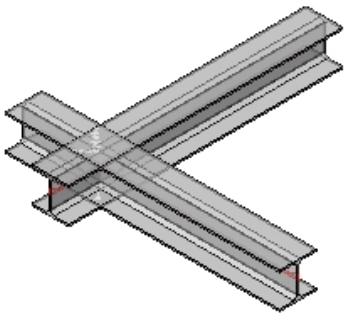
Command - **PrB_CutProfsVsEnds**



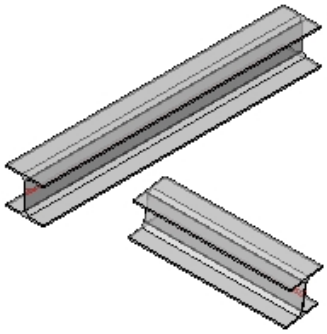
This command will insert a miter joint between two profiles. You will be prompted to select the two beams to be joined, after which, the joint will be placed. This command will repeat until you press <**Enter**> to stop.

This will work with all profiles, at any angle relative to each other. It will also join intersecting or non-intersecting profiles.

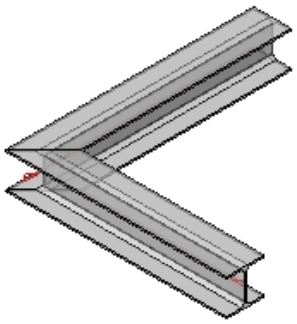
Profile of different sizes may also be mitered, but this practice is not recommended without an analysis of the result.



Intersecting Profiles



Non-Intersecting Profiles



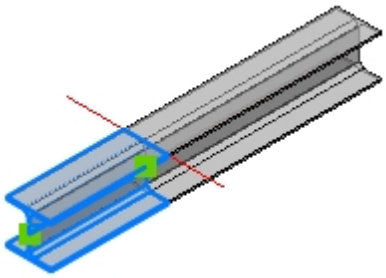
Resulting Mitered Joint

Break Profile

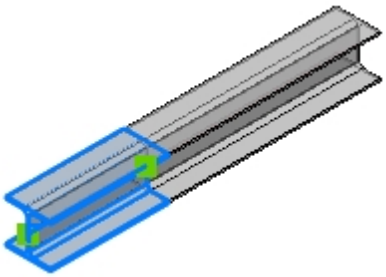
Command - **PrB_BreakProf**

This command will insert a break in a profile - select the beam you want to break and press <**Enter**> You will then be prompted to select either the break-point, or <**Enter**> to specify a distance:

The distance will be measured from the start of the profile (At the end where the triangle is located)



Select Break point

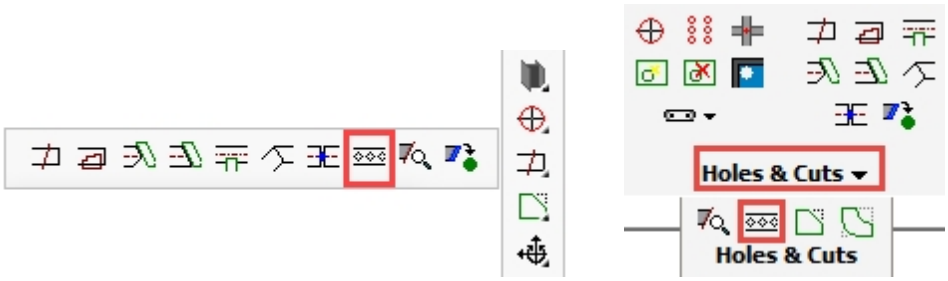


Specify a Distance

Hex Holes (Castellated)

Command - **PrB_Hex**

This command will draw a range of hexagonal holes over the length of a beam.

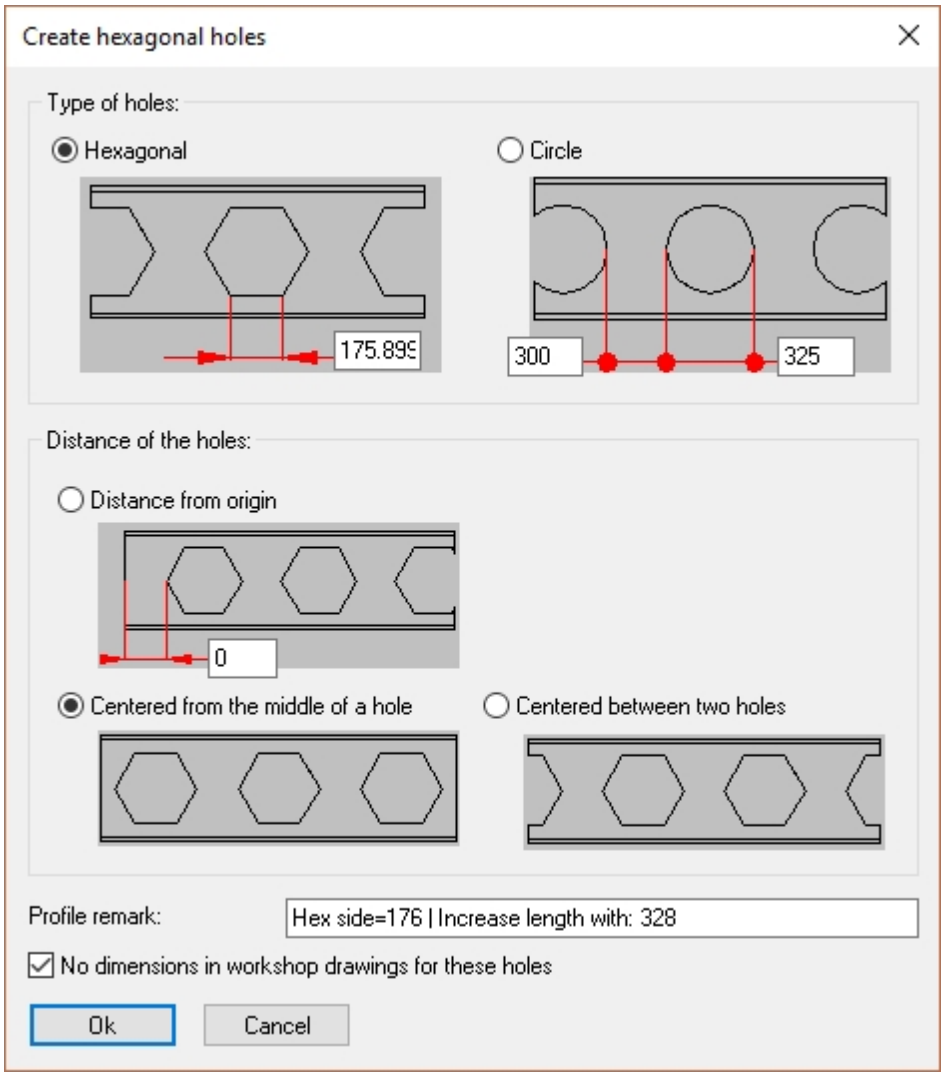


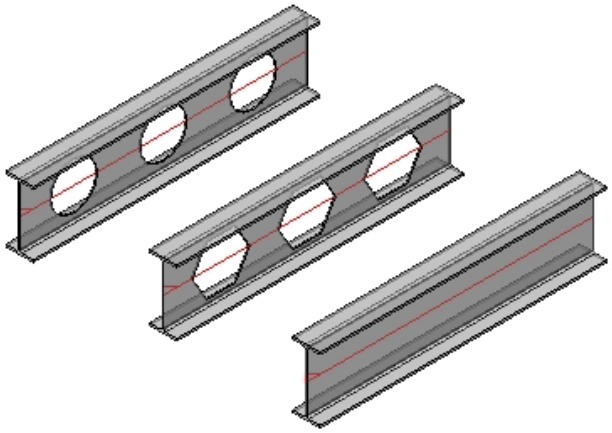
This command will create a Castellated or Cellular profile from a standard I section beam. Select the profile you wish to cut to open the following dialog

Here you have a choice of creating hexagonal holes to form a Castellated beam, or circular holes to create a Cellular beam

The options provided in the dialog are fairy straightforward - The spacing of the holes may be set from the ends, one hole centered on the center-line of the beam, or spaced equally about the center-line of the beam.

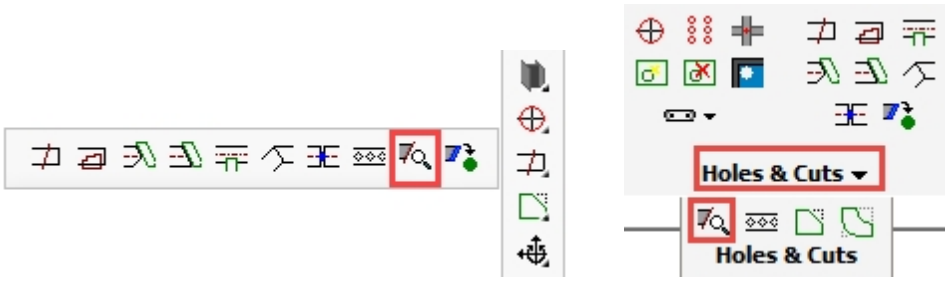
You also have the option of displaying the dimensions of the holes on the workshop drawings.





Edit a Profile's Cuts

Command - **Prb_EditProfCuts**

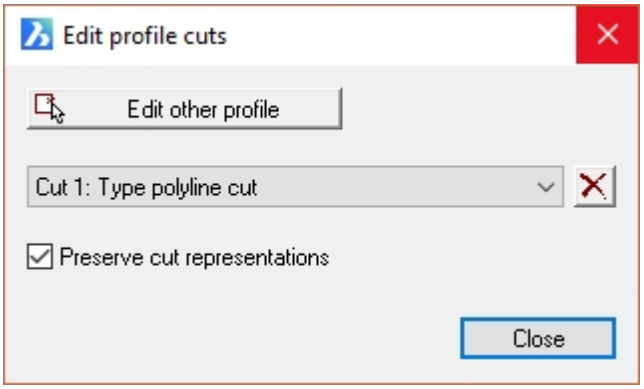


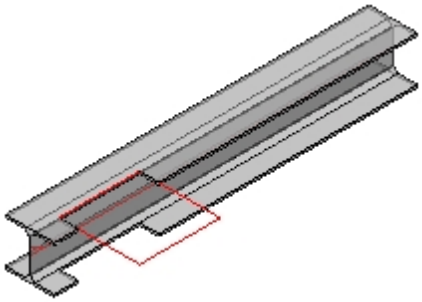
This command allows you to Edit / Remove a cut

Select the profile to be edited and press <**Enter**> - the following dialog will appear

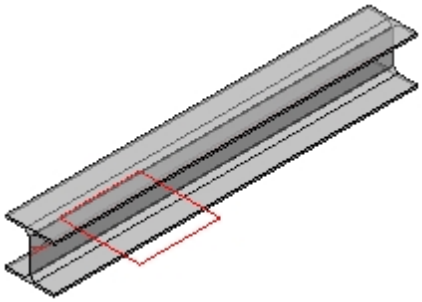
The cut(s) applied to the profile are indicated in the window - selecting one and pressing the **X** button will remove the cut.

Checking the **Preserve cut representations** checkbox will leave the original line or PolyLine upon which the cut was originally determined.





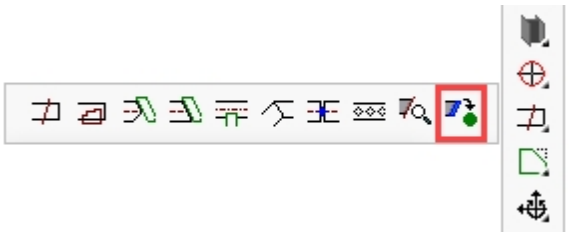
Original Cut

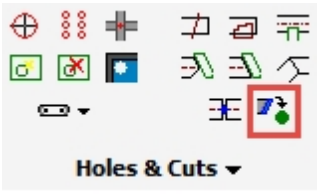


Cut was removed
but cut polyline was preserved

Add Cut to Macro

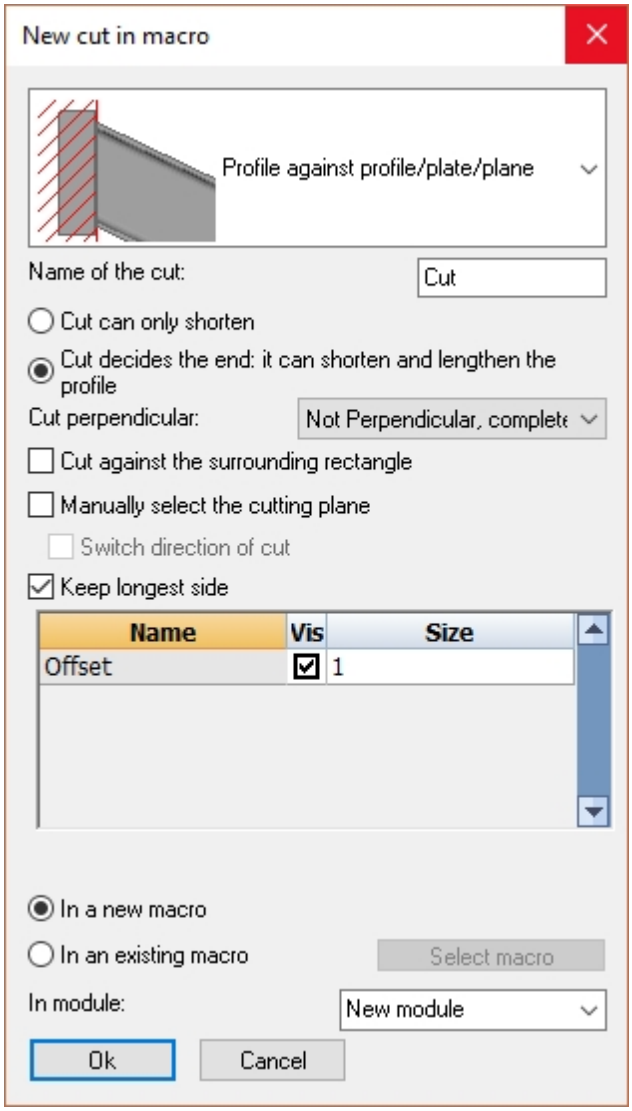
Command - **PrB_AddMacroCut**





With this command, you are 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.

Activating the command, the following dialog will appear:



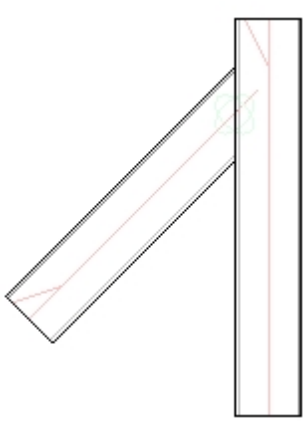
First of all at the top you must choose a type of cut, this will affect some options which are dependent on the type of cut chosen: The options include:

1. **Profile against profile/plate/plane**: This is a straight end-cut against a profile or a plate.
2. **Miter joint**: This is a miter joint between 2 profiles
3. **Cutout profile/plate**: This cut cuts out the form of one profile in another profile or plate.
4. **Against plane with 3 segments**: This cut is used for connections that have clearance cuts.
5. **Chamfer**: With this cut we cut a chamfer from a profile using two distances. You must select two planes as a basis.
6. **Fillet:** With this cut we cut a fillet from a profile using a radius. You must select two planes as a basis.

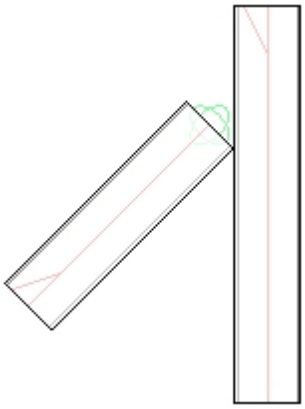
This is an explanation of the most important options :

- **Cut can only shorten** : This is explained extensively in the Cut against an element (shorten only) topic.

- **Cut decides the end** : This is explained extensively in the Cut against an element (shorten and extend) topic.
- **Cut perpendicular** : With this option you can force the cut to be fully or partially perpendicular to the profile's axis. Consider this example :

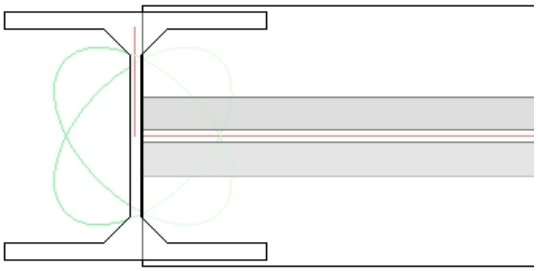


*An end-cut drawn with the option **Not perpendicular***

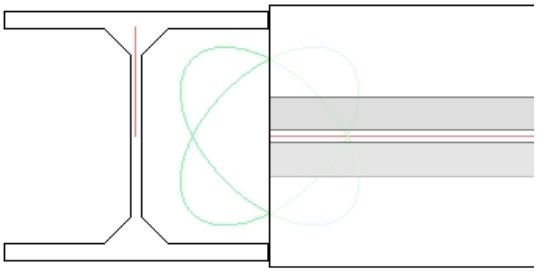


*An end-cut drawn with the option **Completely perpendicular***

- **Cut against the surrounding rectangle** : Enable this for drawing a cut outside of the bounding box of the profile's section shape.

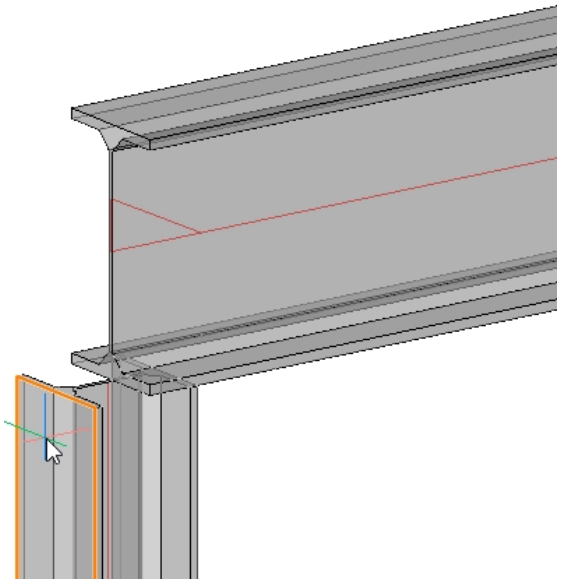


*With **Surrounding rectangle** disabled, the tool will find the main face, which is the web of I and U shapes.*

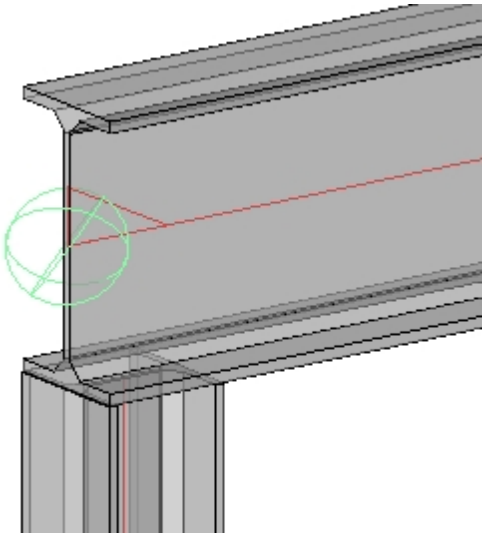


*With **Surrounding rectangle** enabled, the tool will create a bounding box around the section and will draw a cut against that box.*

- **Manually select the cutting plane** : With this option you can manually select the cutting plane. This allows you to choose a plane that would not be possible to choose with the other options in this dialog box.
  This is an example case for which this option can be used :



In orange you can see the plane that is being selected manually.



The end result.

The desired result in this case is a cut that extends as far as the farthest plane of the determining profile. We can only achieve this by selecting this plane manually.

- **Switch direction of cut** : Do not use this unless if you are designing a new connection with geometric rules.
- **Keep longest side** : If you enable this, then the longest side will be kept in case the profile intersects the determining profile. Disable this if you are designing a new connection with geometric rules, and instead use the *Switch direction of cut* option which is safe for connections that will be smart-copied and arrayed.

At the bottom you will find the options **In a new macro** and **In an existing macro**.

Leave this on **In a new macro**, unless if you are designing a new connection with geometric rules.

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.

Modify Profiles and Plates

This chapter covers the modification of elements. The sub-chapters include:

Chamfer

Command - **Prb_Chamfer**

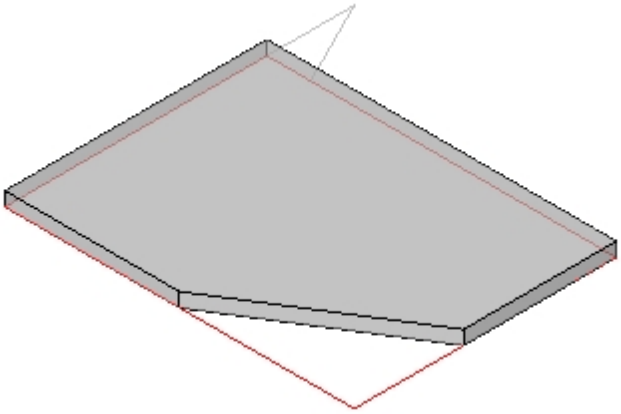
This command will work on almost all edges of plates and profiles.

It will also work on the axis of a profile if you use it on a profile with at least 2 axis line segments.

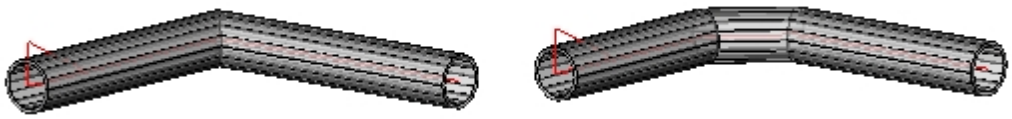
At the command line prompt:

1. Enter the first distance
2. Enter the second distance
3. Select the first line
4. Select the second line

Note that the first line will be equal to the first distance entered, the second line will be equal to the second distance that was entered.



The result after applying a chamfer on a plate



Before and after applying a chamfer on the axis of a profile

Fillet

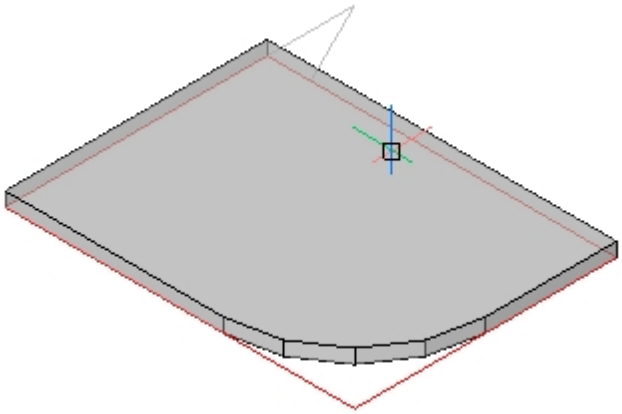
Command - **Prb_Fillet**

This command will work on almost all edges of plates and profiles.

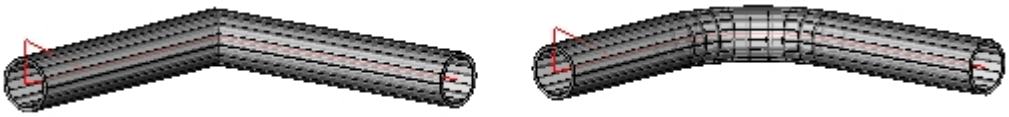
But it will also work on the axis of a profile if you use it on a profile with at least 2 axis line segments.

At the command line prompt:

1. Enter the fillet radius
2. Select the first line
3. Select the second line



The result after applying a fillet on a plate



Before and after applying a fillet on the axis of a profile

Modify Plate Vertices

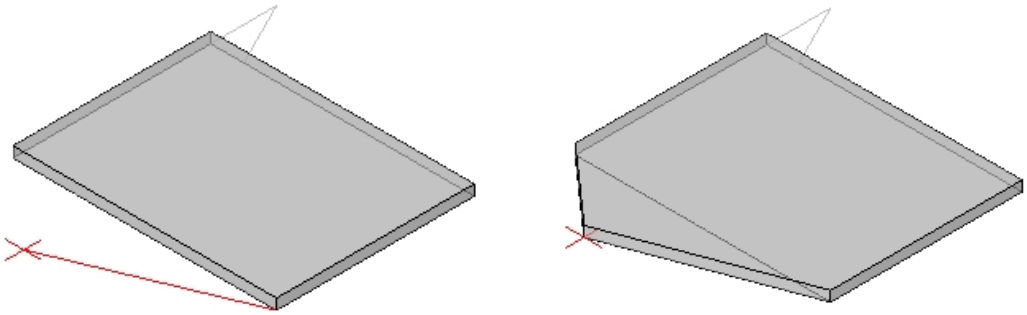
Command - **Prb_EditPlate**

A vertex is a corner point of a polygon formed by the intersection of edges of the object.

Activating this command you will be prompted as to whether you wish to add or remove a vertex.

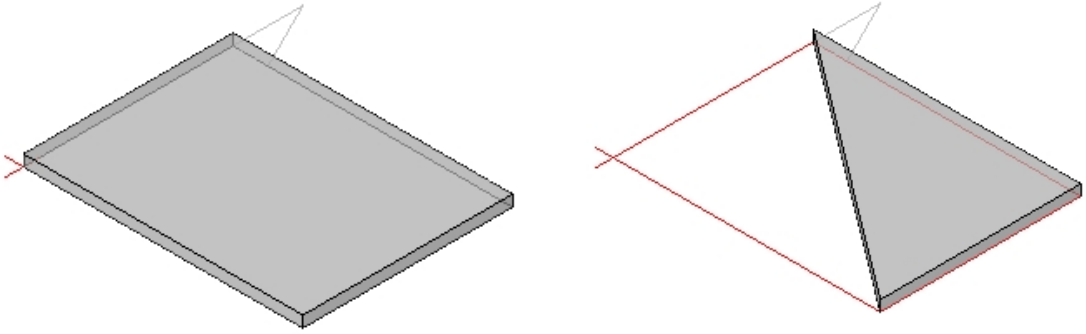
To add a vertex:

At the command line prompt, type '**A**' (Add) and then the location for where to insert a new point.



To remove a vertex:

At the command line prompt, type '**R**' (Remove) and then select the plate near the point you wish to remove



Convert Strip or Plate

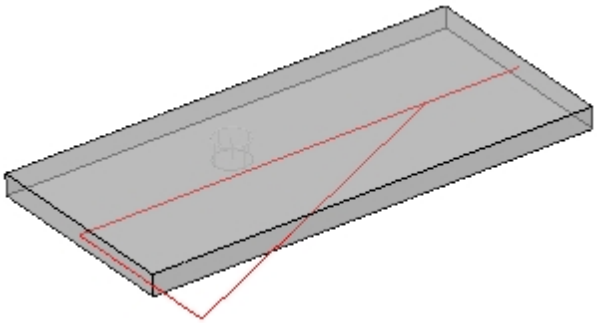
Command - **PrB_ConvertStripOrPlate**

This command allows you to automatically convert a strip to a plate, and vice versa.

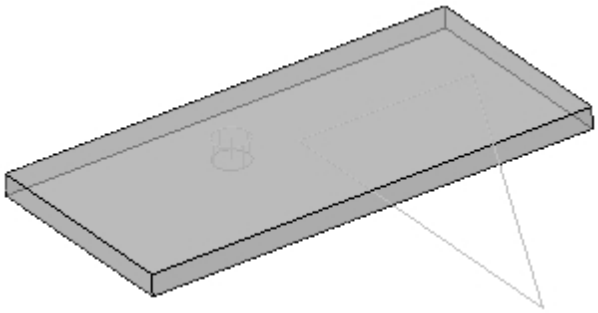
We will explain the 2 different object types, and why there are 2 different object types for basically the same thing.

One can always recognize the type of object (strip or plate) with the help of the axis.

These are the visual difference between a strip and a plate :



The strip has a red axis and a red triangle at the origin of the object, just like any other profile.



The plate only has a triangle in the same color as the plate itself. The location of this triangle is not important, but the rotation is.

The triangle of the plate can be rotated freely. The orientation of the triangle of a plate determines the direction in which the width and length of the plate are calculated.

For strips, the length is always the axis and it can't be rotated.

Plate

Plates are usually drawn automatically by connections. But they can also be drawn manually with the Plate with polyline command.

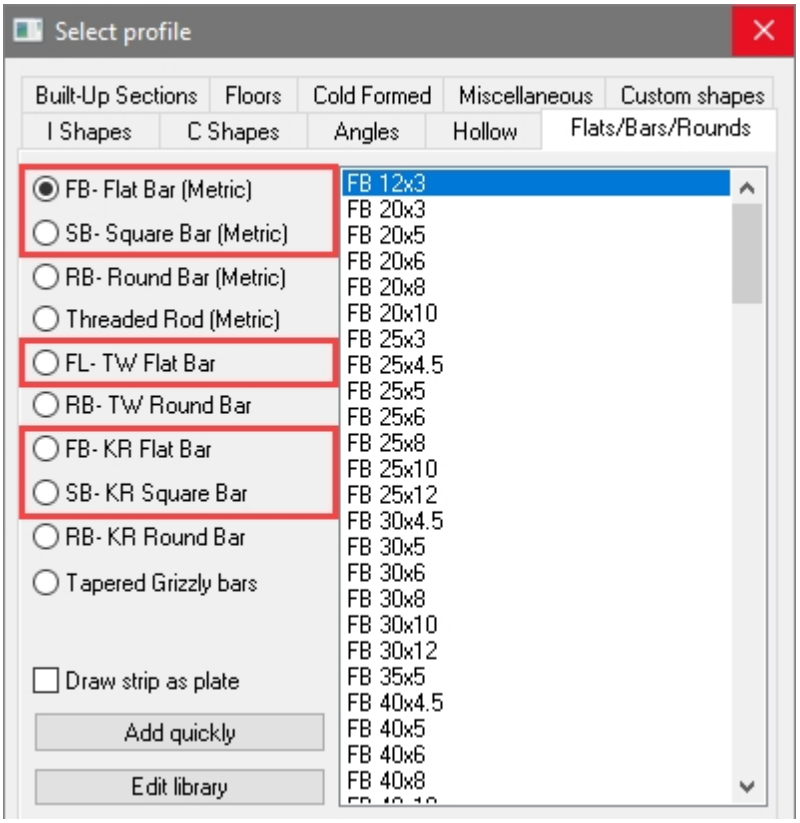
The plate is a unique object in Parabuild because it has a geometric limitation.

We can't cut the plate oblique in it's thickness. So it is actually a 2D circumference with a thickness.

Strip / Flat

A strip is a profile that happens to have the section shape of a plate.

It is drawn with the help of the Select Profile dialog box on the Flats/Bars/Rounds tab :



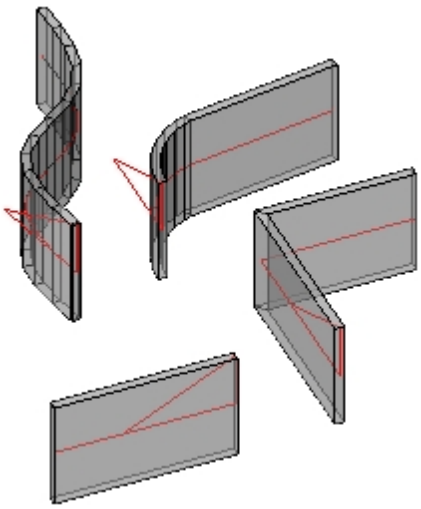
The advantage of strips over regular plates is the fact that these are fully flexible 3D objects.

We can draw bent plates and curved plates if we use a polyline for the model line of the strip.

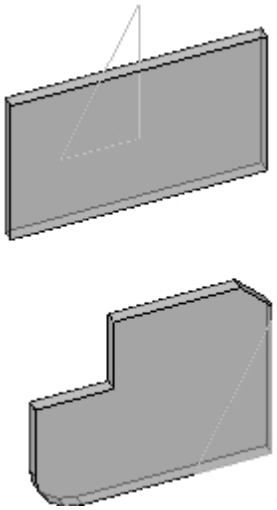
We can also do more cutouts than with a plate. More specifically we can do changes to the thickness of the plate.

These are the geometric limitations of plates compared to strips :

- Strips can be drawn bent and on arcs with polylines, on splines and on 3D polylines :



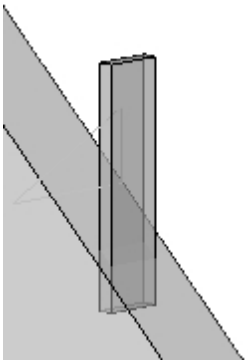
All supported shapes of the strip



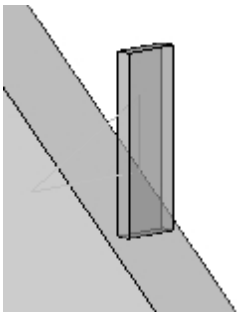
The plate only supports contours

- Strips will follow the exact contours when cutting the thickness. Whereas the plate's thickness will always be drawn perpendicular.

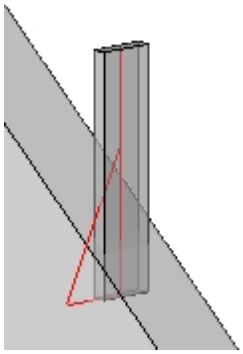
Consider these differences when using the Cut against an element tool on a strip vs a plate:



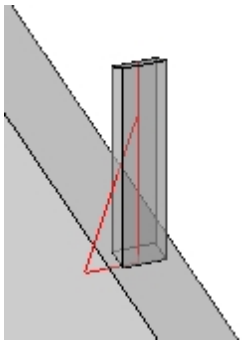
The plate before the cut was applied



The plate after the cut was applied



The strip before the cut was applied



The strip after the cut was applied

Conclusion

This tool solves the inconvenience that arises due to there being 2 different object types for plates/strips in Parabuild. We can convert one to the other at any time.

At the same time, having the 2 different object types gives us greater flexibility and convenience for modeling in 3D. Because the plate's geometric limitations are exactly the same limitations that a Plasma/Burning table has.

Therefore while drawing a plate in Parabuild, you're sure that this plate will automatically get a DXF file generated and the plate that you see in 3D is exactly the plate that will be produced.

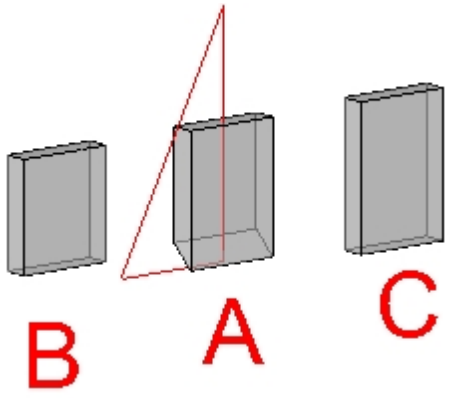
For strips, Parabuild can't generate the DXF file unless you convert it to a plate first.
During the conversion process we will see the modifications that are needed to the 3D model to make it producible on a Plasma/Burner table.

The conversion

The conversion is straightforward and automatic.

However when converting a strip to a plate, the tool will ask whether it needs to remove material or add material.

The difference between the 2 options is explained in this example :



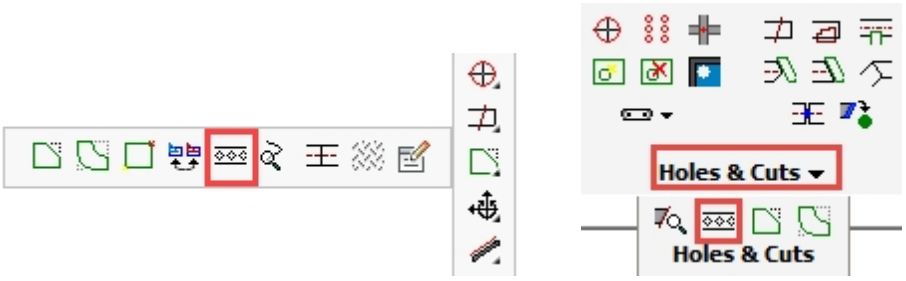
A. is the strip being converted

B. is the resulting plate after choosing the minimize material option.

C. is the resulting plate after not choosing the minimize material option (which implies maximize material).

Hex Holes (Castellations)

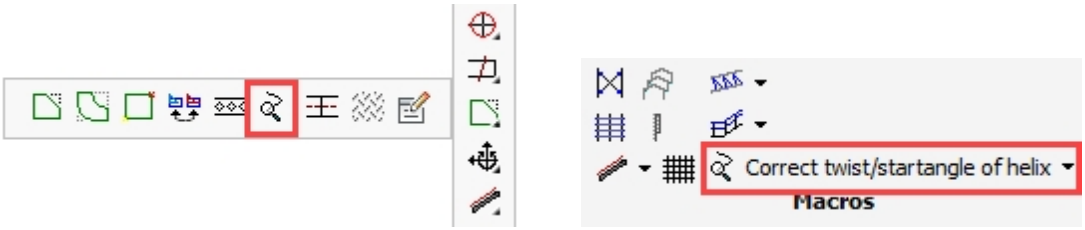
Command - **PrB_Hex**



This tool is already explained in another topic : Hex Holes (Castellations)

Correct Twist of Helix

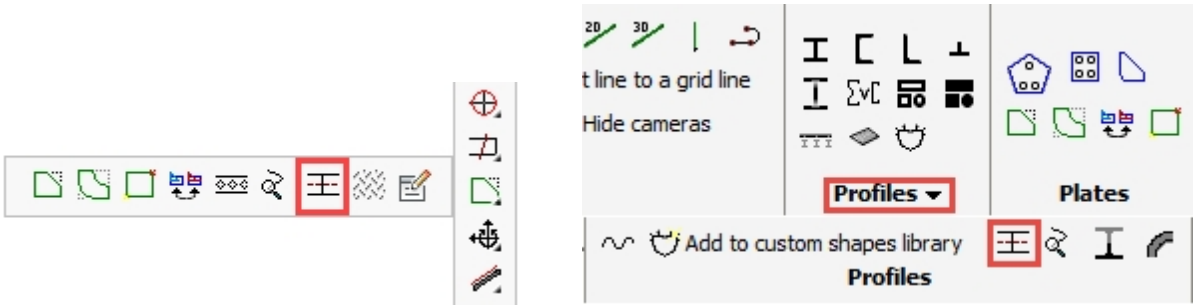
Command - **Prb_ResetTwistAngle**



This tool is already explained in another topic : Correct twist of Helix

Modify Axis of Profile

Command - **PrB_EditProfPath**

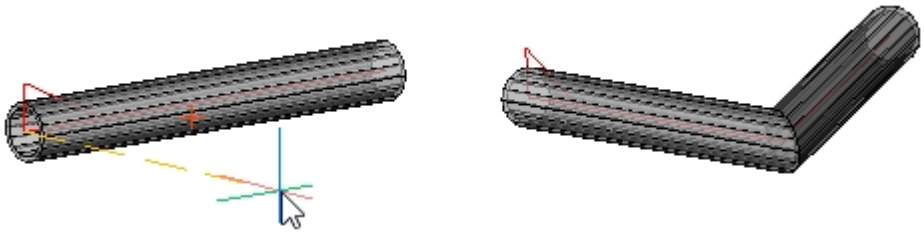


This command can be used to add to or remove bends from the axis of a profile.

Add a new vertex

In order to add a new bend to the profile, one must first select the red axis of the profile.

After that, the location of the new vertex is requested.



This is the profile before and after adding a new vertex to the axis

Remove a vertex

Select the profile close to the 'bend' in order to remove the vertex at that location.

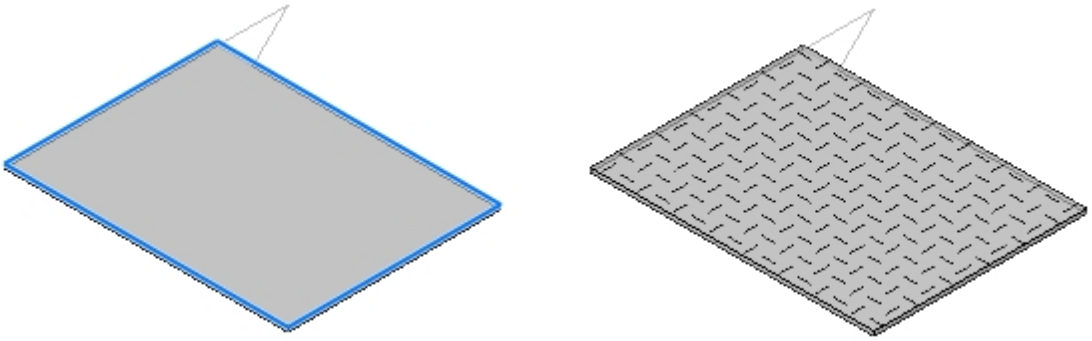
This is the profile before and after removing a vertex from the axis

Modify Surface Treatment

Command - **PrB_ModifySurfaceTreatment**

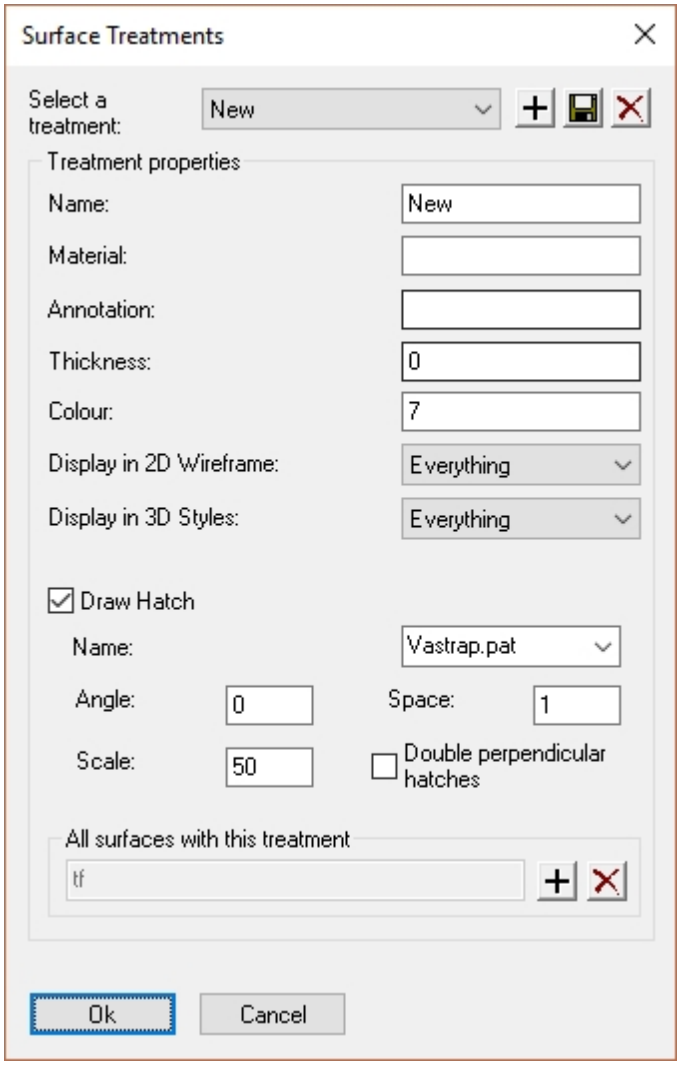
This command allows you to apply surface treatment to any surface of a profile or plate.

This treatment will influence the numbering, so if the treatment of all the surfaces are not the same then the parts will get a different pos/mark number.



Before and after applying a surface treatment

The following dialog allows you to change some properties of the treatment :

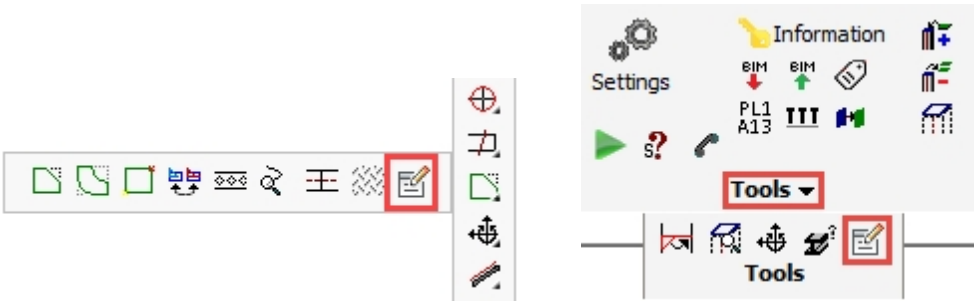


Some example values for a treatment could be :

- **Name** : Galvanizing
- **Material** : Zinc
- **Annotation** : Galv.
- **Thickness** : 0
- **Color** : 7 (index number of the color)
- **Display in 2D wireframe/3D Styles** : Display of the treatment can be limited per display style
- Activating the **Draw Hatch** checkbox will enable you to select a hatch pattern from the drop-down which will be drawn onto the surface of the plate or profile
- The **Hatch Angle**, **Scale**, and **Spacing** all refer to the display style of the hatch
- **All surfaces with this treatment** : Use this to assign the treatment to more than 1 surface, or to remove the treatment from a surface

Edit Entity

Command - **PrB_EditEntity**



This is the command that is used when you would double-click an entity.

But with this icon we can call this command directly.

This tool will first check if entities are already selected.

If none are selected, then it will ask you to select some entities.

After acquiring the entity or entities, it will start the Review macro dialog box automatically if :

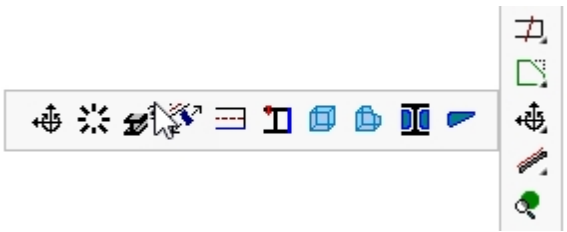
- A macro sphere was selected
- An entity owned by a macro was selected

For other Parabuild entities it will open the respective editing dialog box for that entity.

For example the Weld symbol dialog box for a weld symbol.

For all other entities that do not have such a dialog box (or in case a non-Parabuild entity was selected), the Properties dialog will be opened.

Tools



This chapter covers the manipulation of elements, structures, and stiffeners. The sub-chapters include:

- Move / Rotate
- Copy / 3D Rotate
- Show / Modify Properties
- Move along a Line

- Draw Line on an Edge
- Create Structure
- Draw a Body
- Manipulate a Body
- Web Stiffeners
- Flange Stiffeners

Move / Rotate

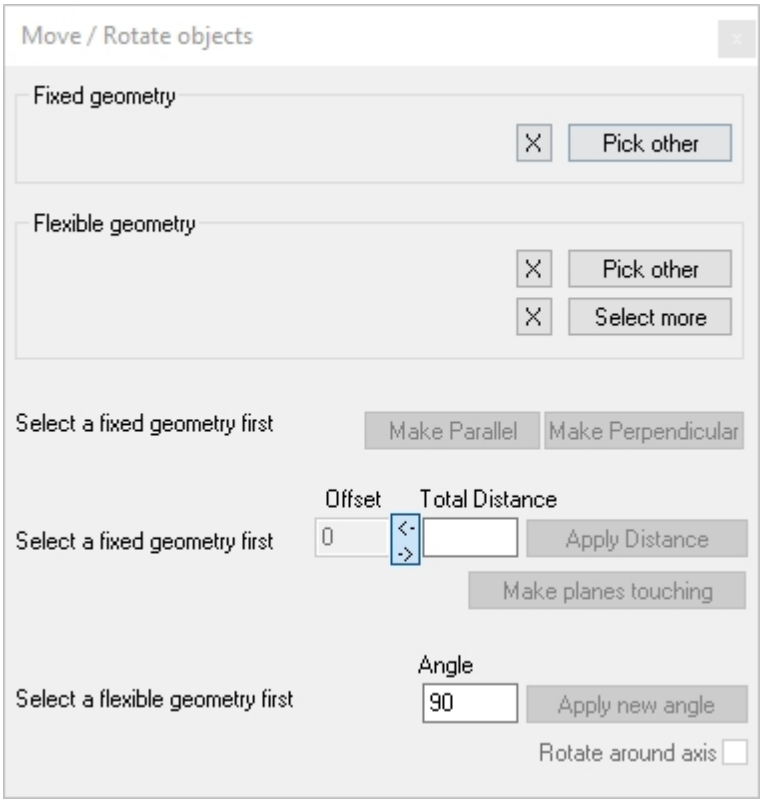
Command - **Prb_MoveRotate**

This tool allows one to directly change the distance between 2 parts, or to set a rotation between 2 parts.

The uniqueness of this tool is that it works on sub geometries for the input.

Counterpart commands such as move and rotate will work on coordinates instead of geometries.

In order for this dialog box to work, we need to select at least 1 geometry under the fixed geometry, and select 1 geometry under the flexible geometry.



A short description for each option in this dialog :

Fixed Geometry

This is the reference geometry that will not be moved or rotated.

This would be the geometry to which you may want to align the flexible geometry to.

This geometry can be a point, a line or any planar surface of a Parabuild part.

Flexible Geometry

This is the geometry that will be moved and/or rotated to satisfy the distance or angle action (see below).

This geometry can be a point, a line or any planar surface of a Parabuild part.

You can use the Select more button to add more objects to the selection set. All of these extra objects will join in the same move/rotate operations.

Make parallel and Make Perpendicular

These buttons will only do a rotate operation to make both flexible geometry parallel or perpendicular with the fixed geometry

Offset

Use this only if you want to move the part(s) in the direction of the *Fixed geometry, while ignoring the current distance between the parts.*

Total distance

This value will be filled with the current distance between the parts.
This value can thus be seen as a measurement.

However this value may be misleading when the fixed and flexible geometries are not parallel.

Apply Distance

This action will move and rotate the *Flexible geometry* to satisfy either the *Offset* value or the *Total Distance* value (whichever value is active).

- In case of the *Offset* value a simple move operation will be done in the direction of the Fixed geometry
- In case of the *Total Distance* value the flexible geometry is moved and rotated so that the gap between the parts is the distance entered here.

Make planes touching

This does almost the same as Apply Distance, but the distance will be 0, and the flexible part is rotated so that the material directions are opposed so that the parts are not colliding.

Apply new angle

This will do a rotation only.

It will rotate the Flexible geometry around the fixed geometry.

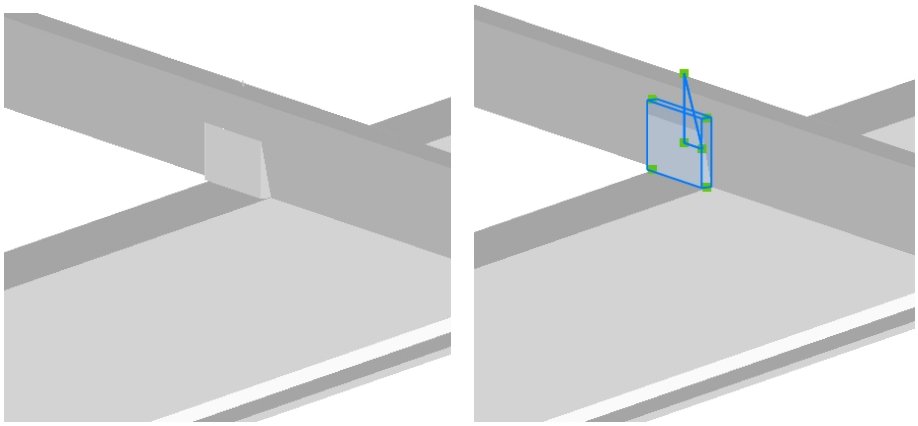
If you selected planes for both the fixed and the flexible geometry, then the flexible geometry will rotate with the plane's normal as the axis of the rotation.

The value in the angle field is not filled in automatically so this shouldn't be seen as a measurement.

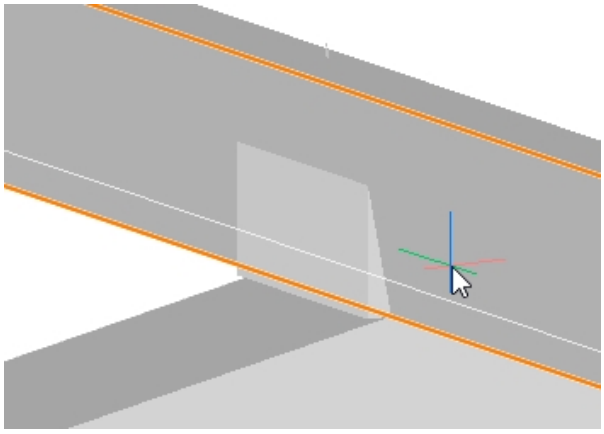
Example

This is an example move operation that can be achieved with the *Apply distance* button.

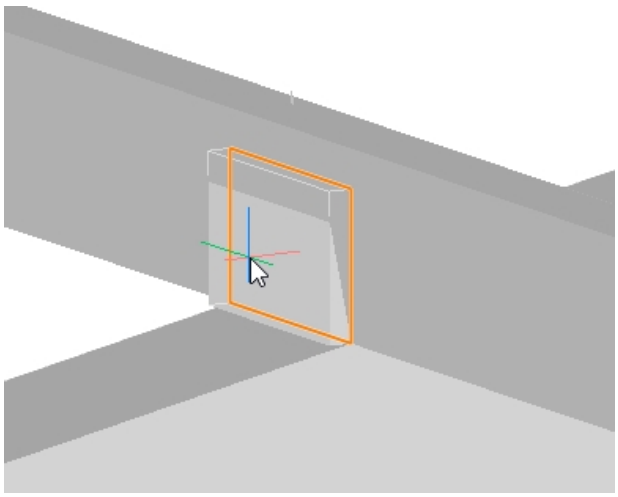
This is the issue that needs to be solved : the plate was drawn relative to World but instead it should follow the roof's pitch :



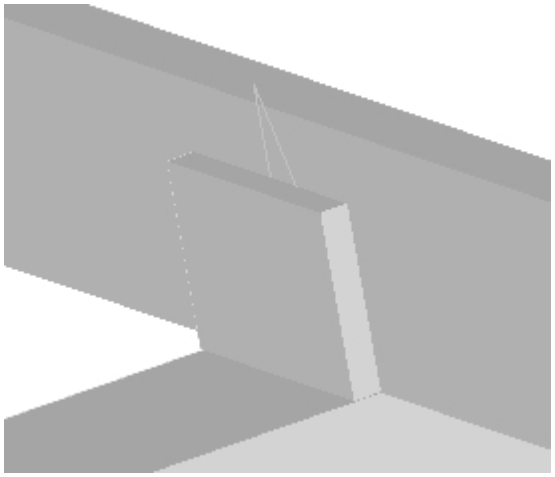
Selecting the fixed plane :



Selecting the plane that needs to be rotated :

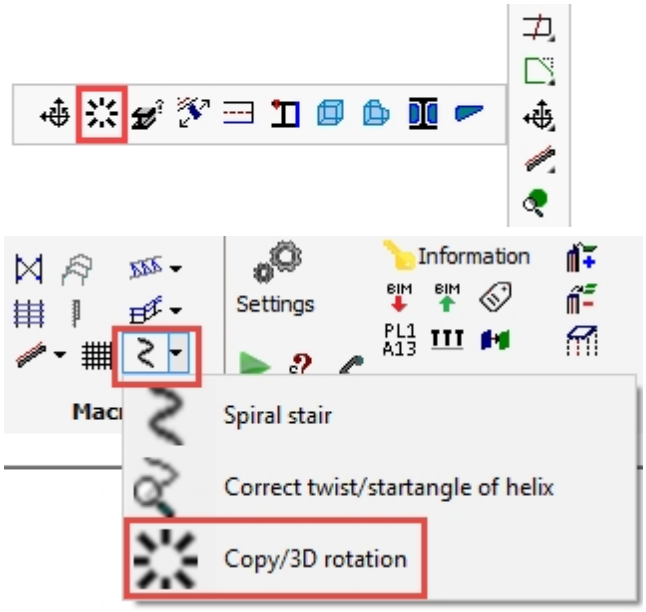


The end result after applying a *Total Distance* of 0 :



Copy / 3D Rotate

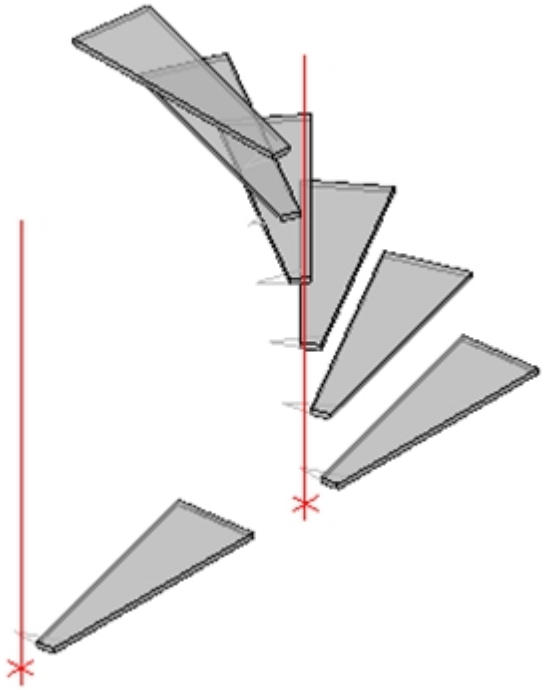
Command - **PrB_ArrayRotate**



This tool can create copies of a set of objects, while rotating the new entries incrementally.

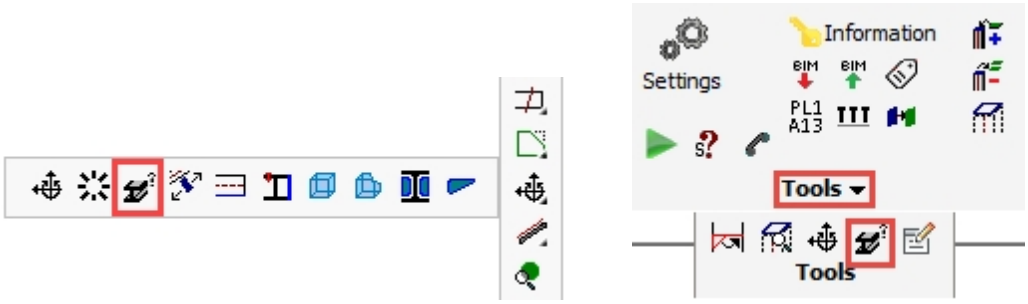
This tool was created for copying the steps of a spiral stair.

In this image, the red cross is the rotation base point which should be the center of the spiral stair :



Show / Modify Properties

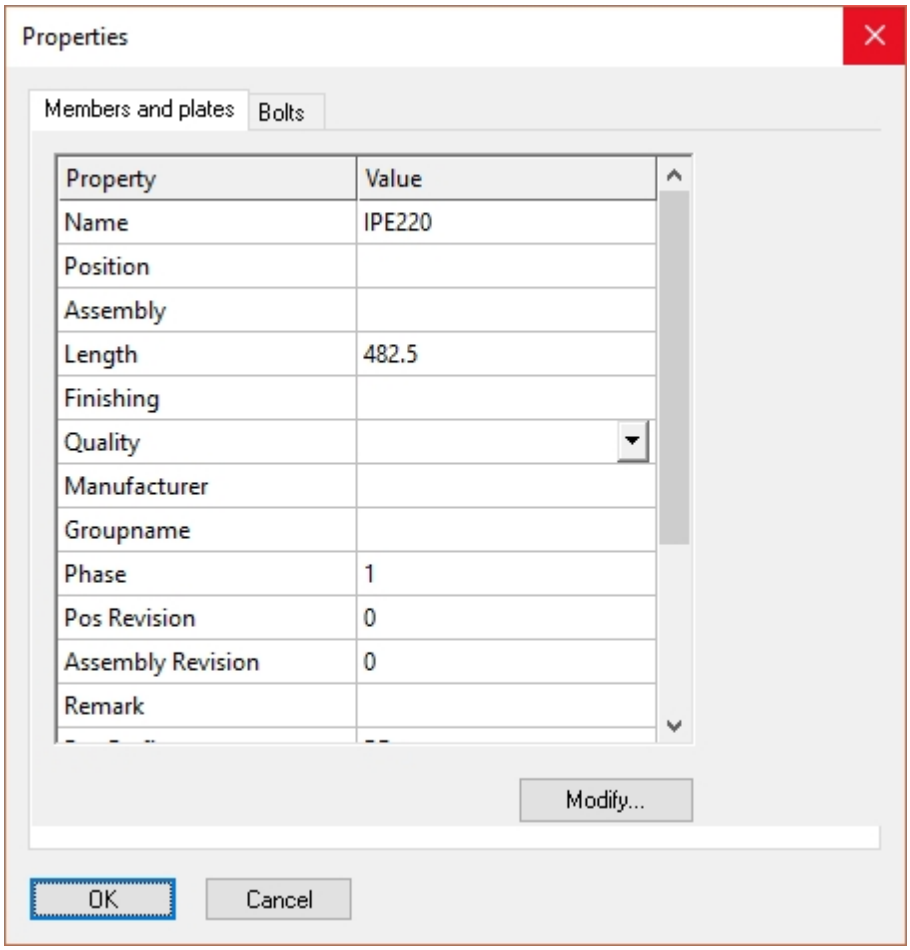
Command - **Prb_Properties**



This command allows you to review and edit the properties of the selected item.

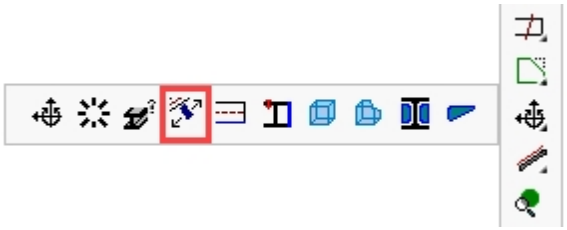
Compared to the AutoCAD/BricsCAD properties panel, it provides an alternative user interface and an alternative selection method.

The properties panel has many more capabilities so therefore it is recommended to use the panel instead of this command.



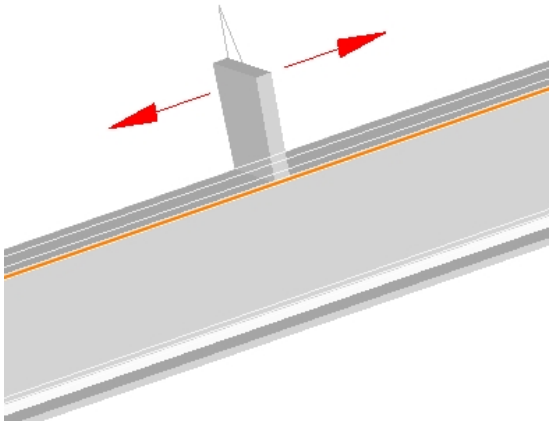
Move along a Line

Command - **PrB_LineMove**



This tool allows you to move a part in the same direction as another part's edge.

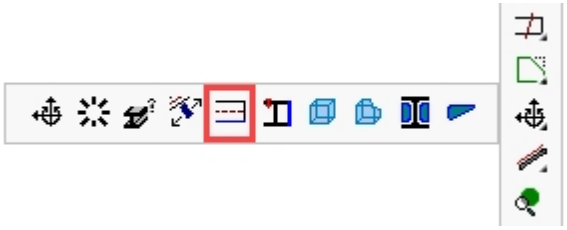
So in the below case the orange line is selected, and this allows us to move the plate along the inclination of the profile.



The advantage of this tool is that it replaces 2 handlings if we wanted to do the same thing with a standard move command : First set the ucs according to the profile, and then do the move command.

Draw Line on an Edge

Command - **PrB_SubGeomPoly**



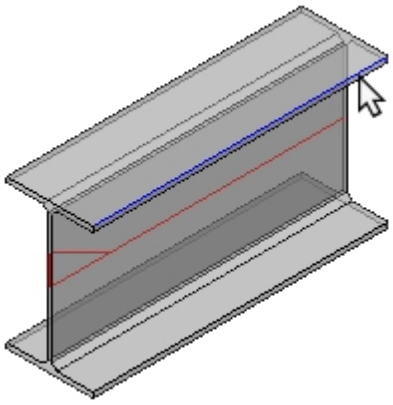
This tool will draw a line or polyline on the edge or on a surface of a profile or plate.

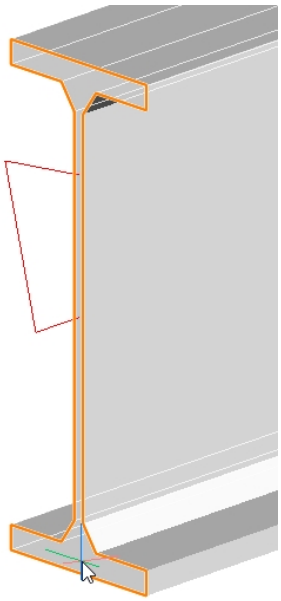
The command prompts you to select an edge or a surface.

A line will then be drawn on top of the edge(s) of the selected geometry.

The resulting lines will not always be visible as they are lying on top of the profile lines.

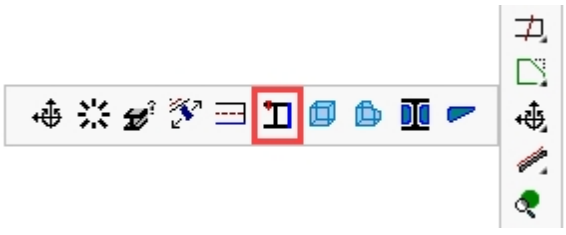
This function can also be used to select the edges and faces of cuts.





Create Structure

Command - **PrB_Struct**



A structure is a group of elements that are grouped into one part and will therefore appear as one element in the part lists and workshop drawings.

A good example use for the structure is a stair tread.

The alternative to this tool is to leave the items as separate component parts, i.e. Profiles and Plates but the disadvantage is that these components will appear as separate parts in the workshop drawings and parts lists, which may defeat the objective, especially if the part is intended to be bought-out as one unit.

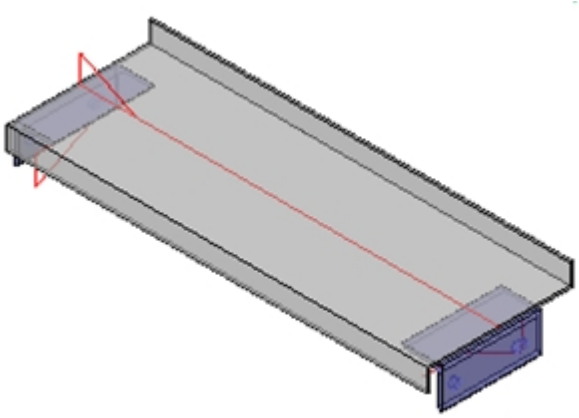
Another option is to draw the part using lines / solids, but here the component parts will not be added to the parts lists, and a variety of advantages will be lost i.e. No bolt-check, No clash-check ....

The solution is to create a **Structure** object.

To Create a Structure

In this example we will create a simple solid stair tread which comprises of a bent plate with a supporting angle welded to each end.

The tread plate has been created using the Custom Shapes command, while the angles have been selected from the Parabuild profiles library



After drawing the tread as individual parts, there are 2 methods for converting them into a single structure :

- Use the command PrB_Struct to create a structure of the parts, directly inside the drawing.
- Save the elements together into 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 stair will appear on the screen, but the objects are now merged into 1 object.

The tread will now be added to the part list as a single part (*under the name selected by the user*).

If this element is selected, and the properties are requested, the element will be recognized 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 parts list.

What exactly are structure objects?

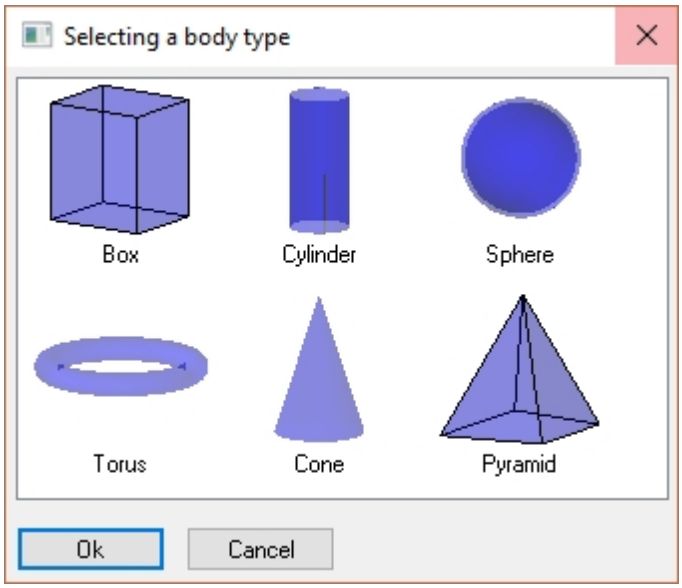
A structure is a reference to a block.

This Block Reference is labeled 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.

Draw a Body

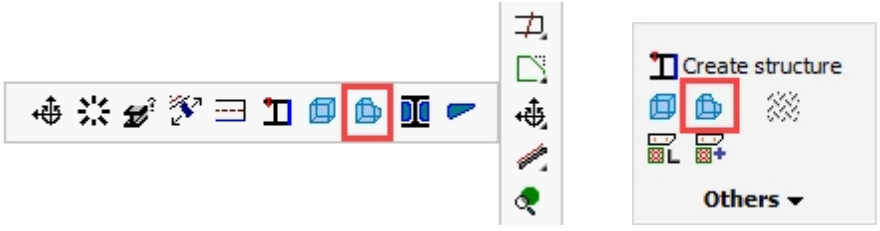
Command - **PrB_CreateBodyDlg**

This command will create any of these volume types :

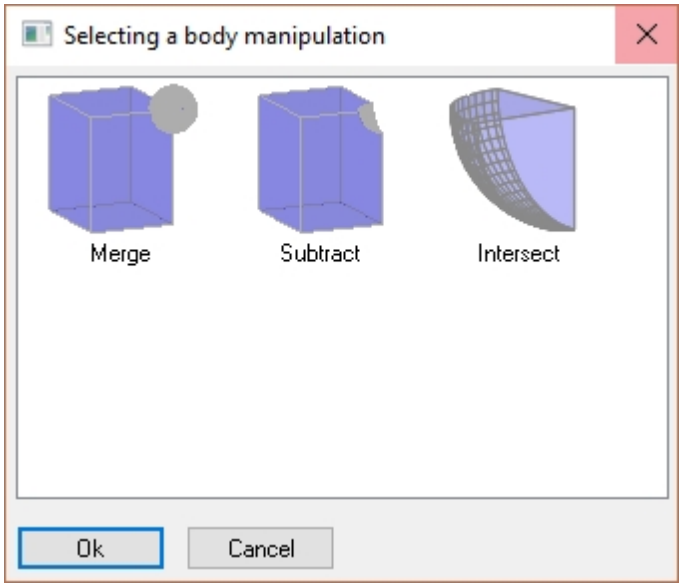


Manipulate a Body

Command - **PrB_ManipulateBody**

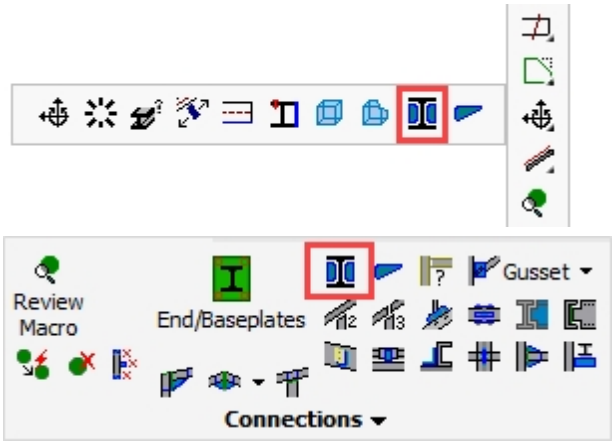


This command can perform 3 actions on 2 volumes :



Web Stiffeners

Command -



This command will add web stiffeners to any **I** or **C channel** section profile

After starting this command you will be prompted to select the profile to which the stiffeners are to be added.
After this selection, a dialog will open offering the following options:

Reinforced cut out

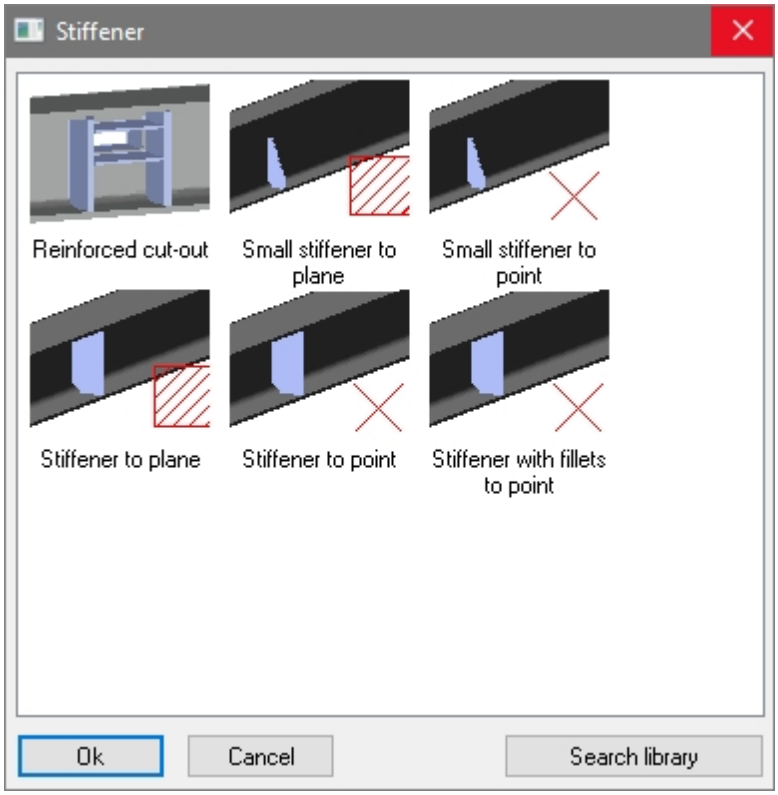
Small stiffeners to plane

Small stiffeners to point

Stiffeners to plane

Stiffeners to point

Stiffeners with fillets to point



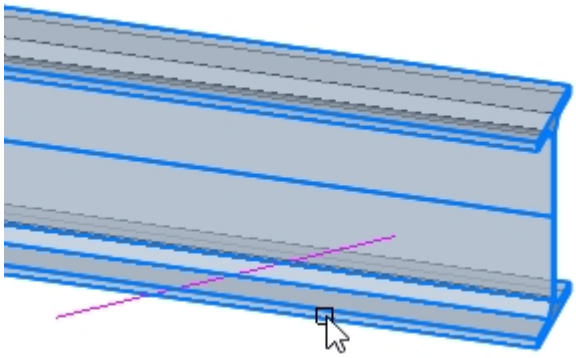
For many stiffeners there are 2 variants, ***to point*** or ***to plane***.
We will explain the difference here :

To point

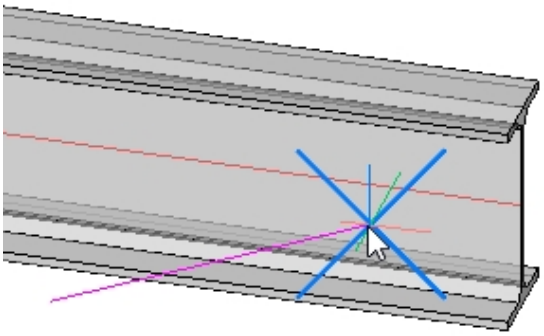
When you choose this method, Parabuild will prompt you to select a point sub-geometry. However by pressing **<Enter>** instead of selecting the point, you will be able to select a coordinate using the Object Snap tools.

The point on the axis of the profile that is closest to your selected point will determine the position of the new stiffeners.

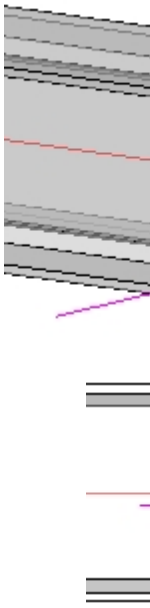
An example of using the point sub-geometry :



Selecting the profile that needs a stiffener attached to it

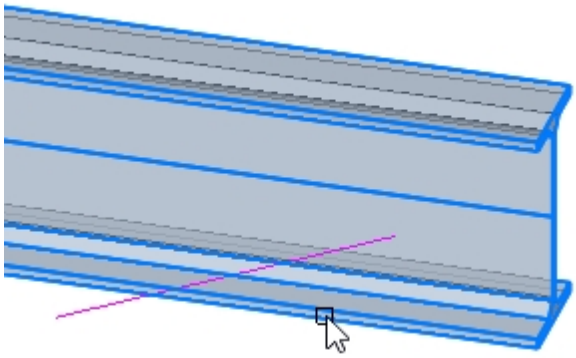


Selecting an endpoint of a line. We could select any point of any profile, plate, structure, volume, line, or polyline.

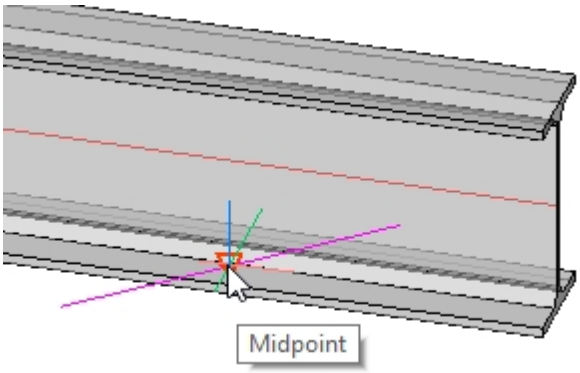


The end po

An example of using the point coordinates :



Selecting the profile that needs a stiffener attached to it



After pressing <Enter> for not wanting to select a sub geometry : Selecting the midpoint of the line with the help of Object Snapping. This is just an example.

We could snap to any point on any object. This can be a point on an object unknown to Parabuild such as 3D Solids or 3rd party software objects.

The end result is positio

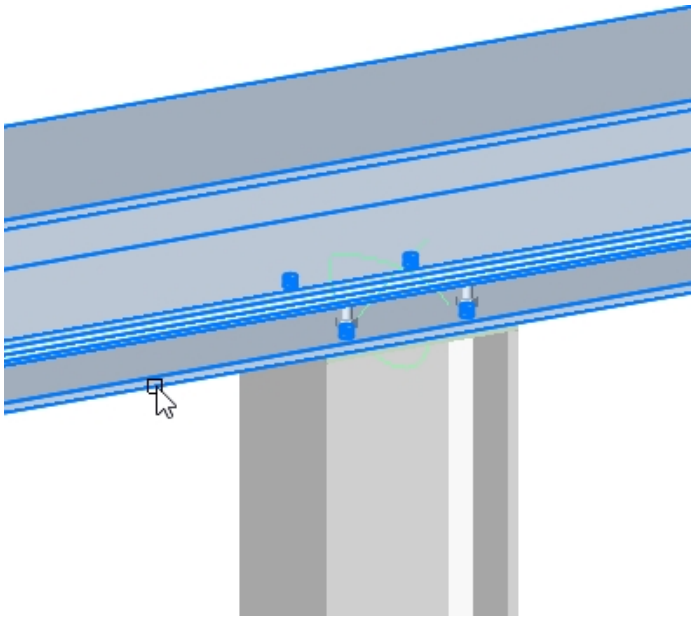
To plane

When you choose this method, Parabuild will prompt you to select a planar surface sub-geometry.

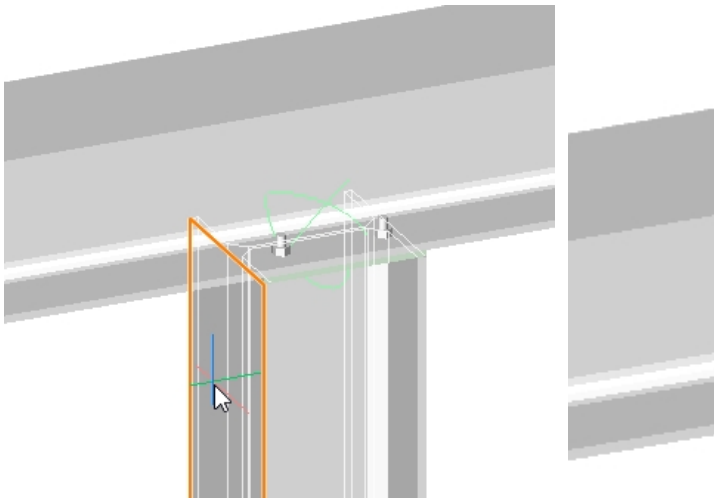
The intersection of the selected surface with the point on the axis of the profile will determine the position of the new stiffeners.

The selected surface should not be parallel to the selected profile because otherwise, an intersection cannot be found.

A practical example of using the planar surface method :



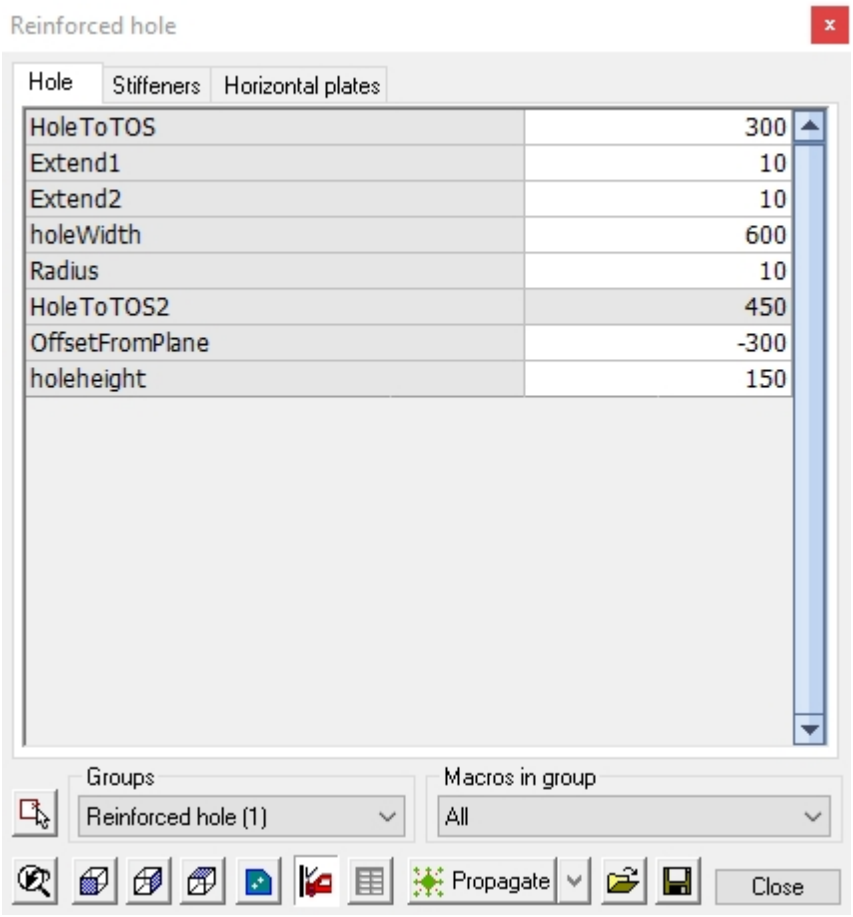
Selecting the profile that needs a stiffener attached to it

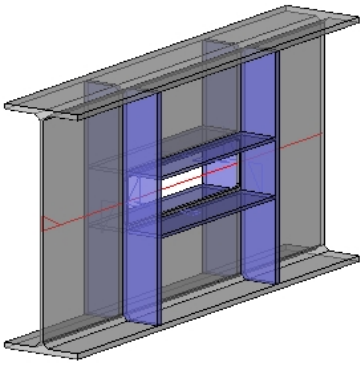


Selecting a surface of another object. The selected surface is highlighted on screen. We could select any planar surface of any profile, plate, structure, or volume.

Reinforced cut-out

This macro only exists with the *to plane* method.





This edit dialog has 3 tabs:

Hole - where you can edit the size and position of the hole

Stiffeners - where you can edit the size and thickness of the vertical stiffeners

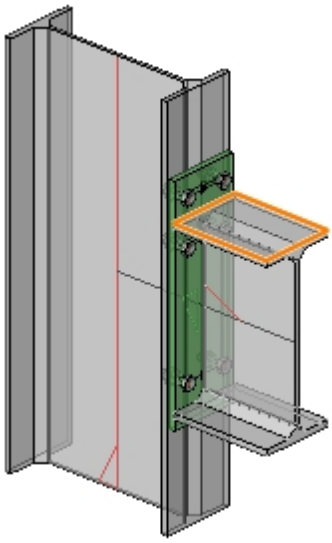
Horizontal plates - where you can edit the size and offsets of the plates

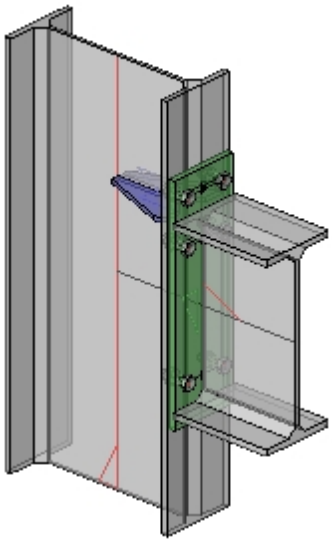
Small stiffeners to plane

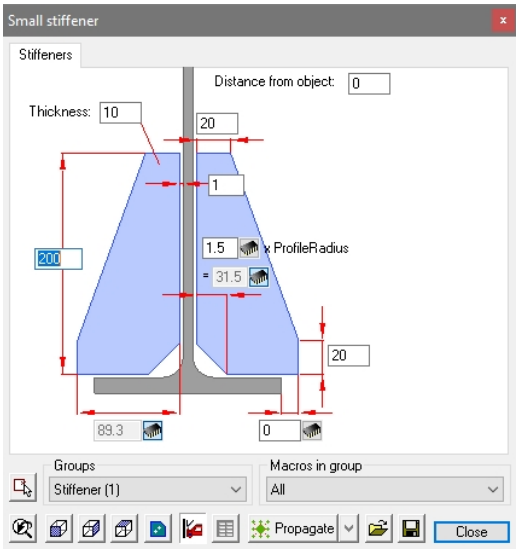
The stiffener will be placed adjacent to the plane of an incoming column or beam.

The stiffeners are placed on each side of the profile's web, with the top of the plate in-line with the selected plane.

Here, the top flange of the incoming beam was selected :







Select a Plane Stiffener placed The macro edit dialog

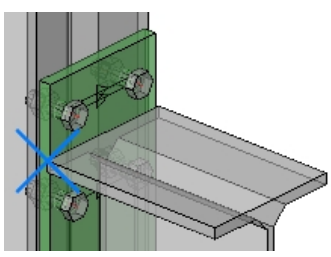
If the stiffener is to be placed at the bottom flange of the incoming beam, it may be necessary to adjust the **Distance from object** variable by the thickness of the plate. This will ensure the bottom of the plate is in-line with the plane of the bottom flange.

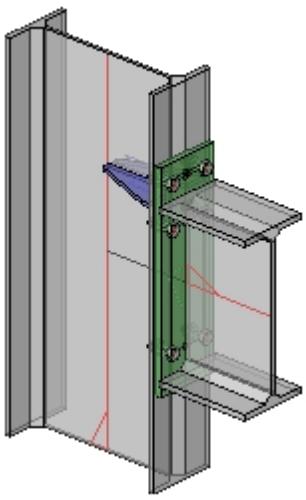
Small stiffeners to point

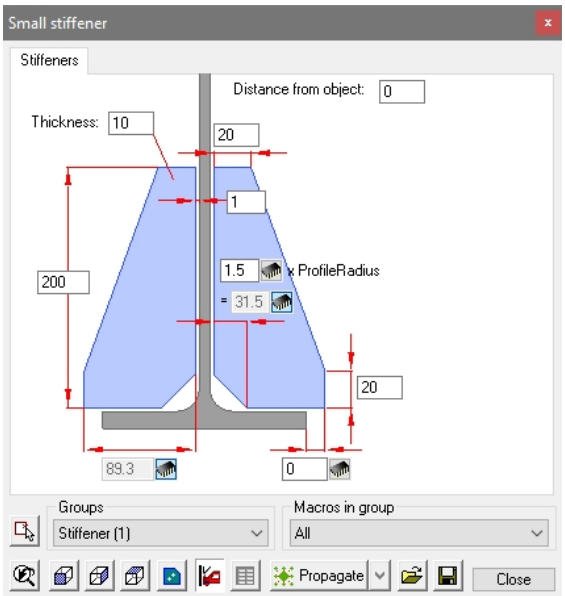
The stiffener will be placed adjacent to the point of an incoming column or beam

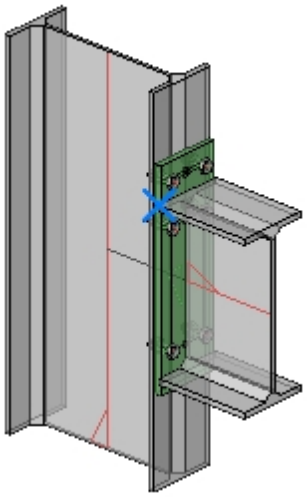
The stiffeners are placed on each side of the profile's web, with the top of the plate in-line with the selected plane.

Here, the top flange of the incoming beam was selected :









Select a Point Stiffener placed The macro edit dialog

If the stiffener is to be placed at the bottom flange of the incoming beam, it may be necessary to adjust the **Distance from object** variable by the thickness of the plate. This will ensure the bottom of the plate is in-line with the plane of the bottom flange.

Stiffeners to plane

This works exactly the same as Small stiffeners to plane, except a full-depth stiffener is drawn

Stiffeners to point

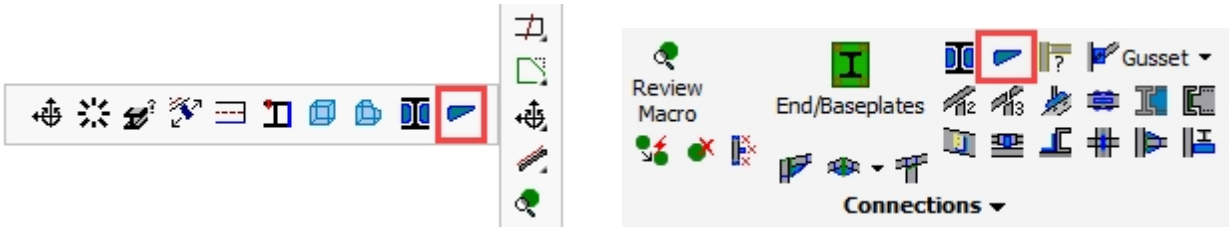
This works exactly the same as Small stiffeners to point, except a full-depth stiffener is drawn

Stiffeners with fillets to point

This works exactly the same as Small stiffeners to point, except a full-depth stiffener with fillets is drawn

Flange Stiffeners

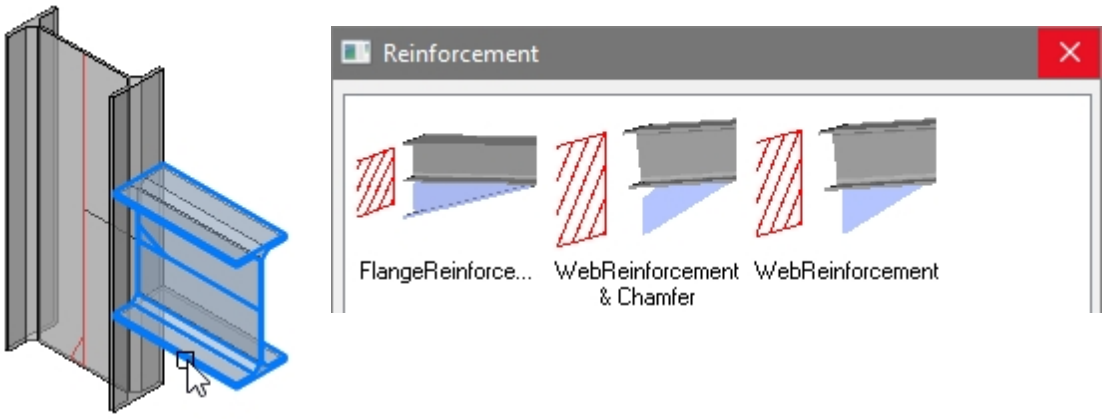
Command -



This command allows us to add Flange Stiffeners to a profile.

Activating this command, you will be prompted to select the profile to which the stiffeners are to be added. This may be either a beam or column.

After selection, the following dialog will open offering the following options:

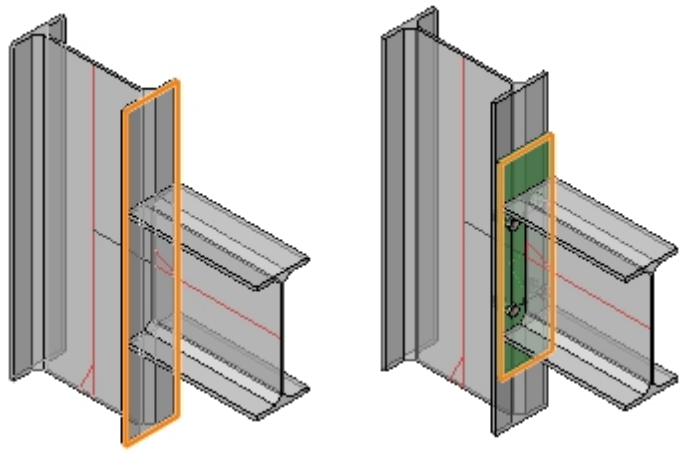


Selecting the profile

Note! Selecting the top of the beam will place the stiffener above the top flange - likewise, selecting the bottom of the beam will place the stiffener under the bottom flange.

After making the selection, you will be prompted to **Select a plane.** This plane will determine the face to which the selected member will be stiffened.

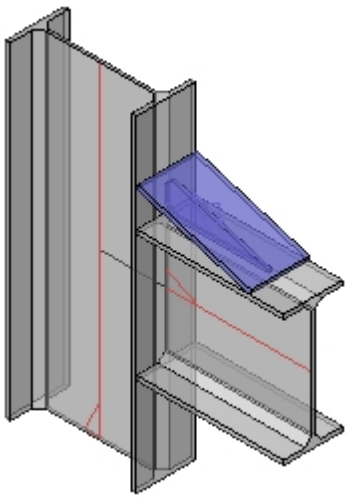
This may be the flange or web of a profile or plate:



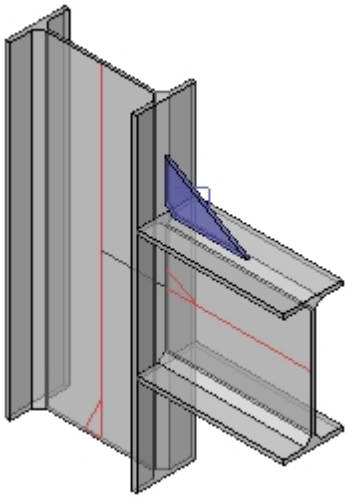
Selecting a plane of a profile

Selecting a plane of a plate

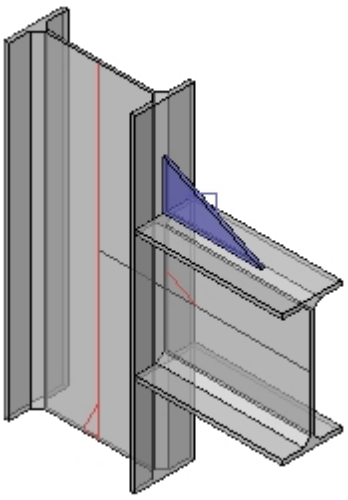
The default stiffener will be drawn :



Flange Reinforcement



Web Reinforcement with Chamfer



Web Reinforcement

Stairs

- Stair
- Handrail
- Ladder
- Spiral Stair
- Correct twist of Helix
- Railing End-Connections
- Railing Intersection-Connections
- Connect Post
- Railing to Post - 2 Base
- Railing to Post - 3 Base
- Weld Connections
- Draw Grating

Stair

Command - **Prb_Stair**

This command will automatically draw 2 stair stringers with accompanying treads and bolts.

There are 3 basic stair models covered by this command:

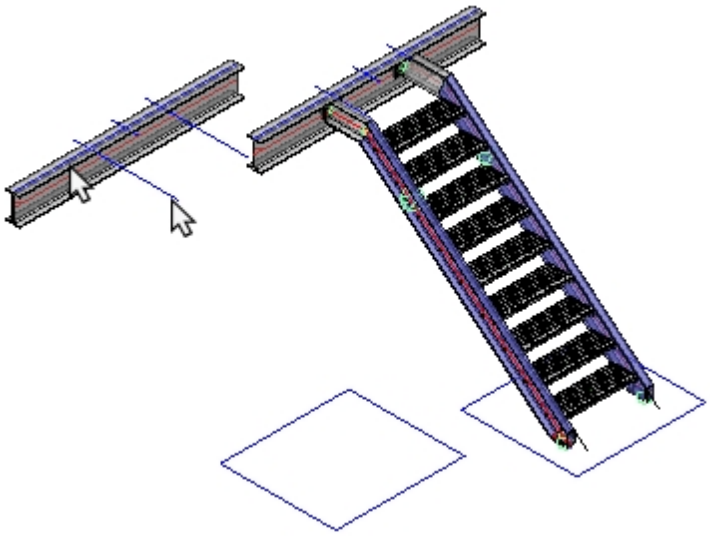
1. Draw a stair from the ground level (UCS World) to a beam
2. Draw a stair - from beam to beam
3. Draw a stair on a line

Connections between the stair stringers and the supporting steelwork may be selected from the standard range of Connections

All 3 models can be drawn with the same command.

The types of objects that you select will determine which of the 3 stair types will be drawn.

Draw a stair from the ground level (UCS World) to a beam



At the command line prompt:

Select the near side face of the top beam. Note! this selection will determine the direction of the stair.

As there is no bottom beam press <**Enter**>

Next, select either a line or profile to indicate the horizontal placement of the stair. Note! that the horizontal geometry will determine the inside face of the left hand stringer. Press <**Enter**>

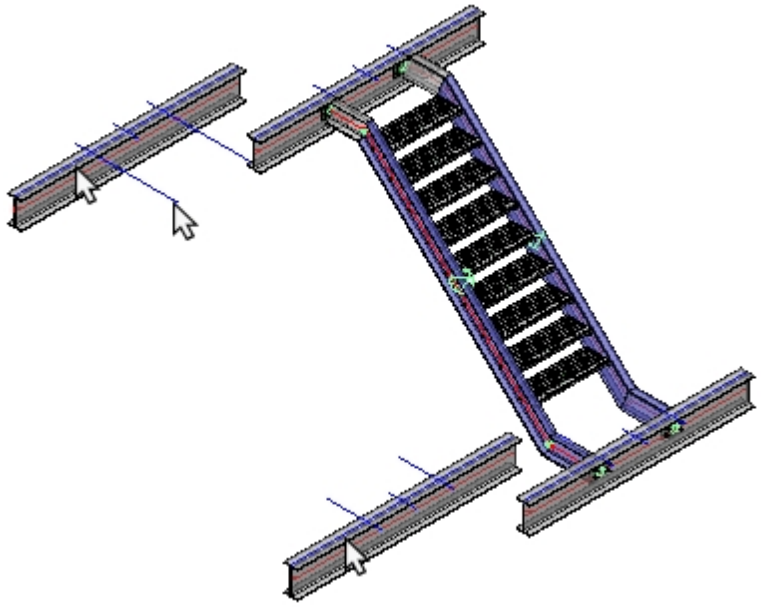
The Stair configuration dialog will appear prompting you to select the desired stair type.

The default stair will be drawn accompanied by the Stair macro edit dialog. Note! that the stair will be drawn with the top stringer aligned with the top flange of the beam

You can review the stair at any time by selecting the large macro sphere and activating the Review macro command, or you can double-click the sphere - both options will open the Edit dialog.

You can also edit components of the stair, for example the base connection, by selecting the smaller macro sphere

Draw a stair - beam to beam



At the command line prompt:

Select the near side face of the top beam. Note! this selection will determine the direction of the stair.

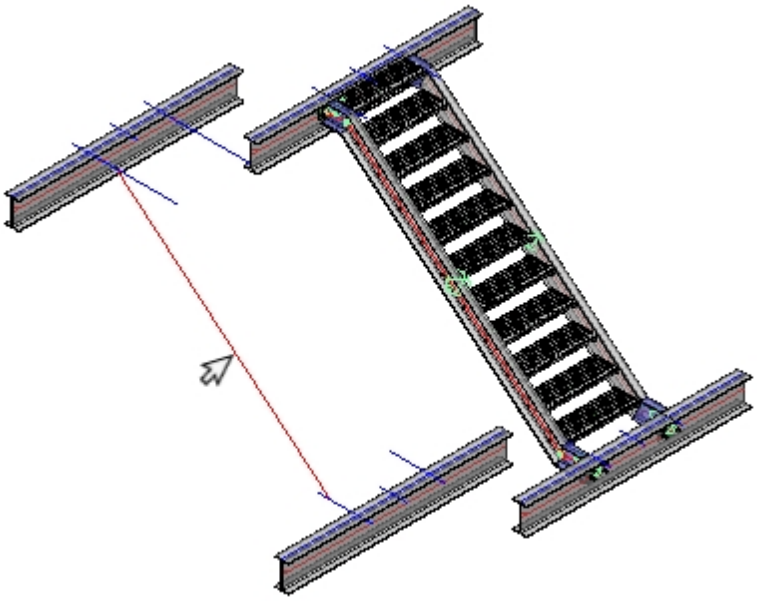
Then select the bottom beam and finally the horizontal geometry, which again, may be either a line or an incoming profile. press <**Enter**>

The Stair configuration dialog will appear prompting you to select the desired stair type.

The default stair will be drawn accompanied by the Stair macro edit dialog. Note! that the stair will be drawn with the top and bottom stringers aligned with the top flanges of the beams

You can review the stair at any time by selecting the large macro sphere and activating the Review macro command, or you can double-click the sphere - both options will open the Edit dialog.

Draw a stair on a line



The line drawn will define the geometry of the stair, i.e. the angle and stringer length.

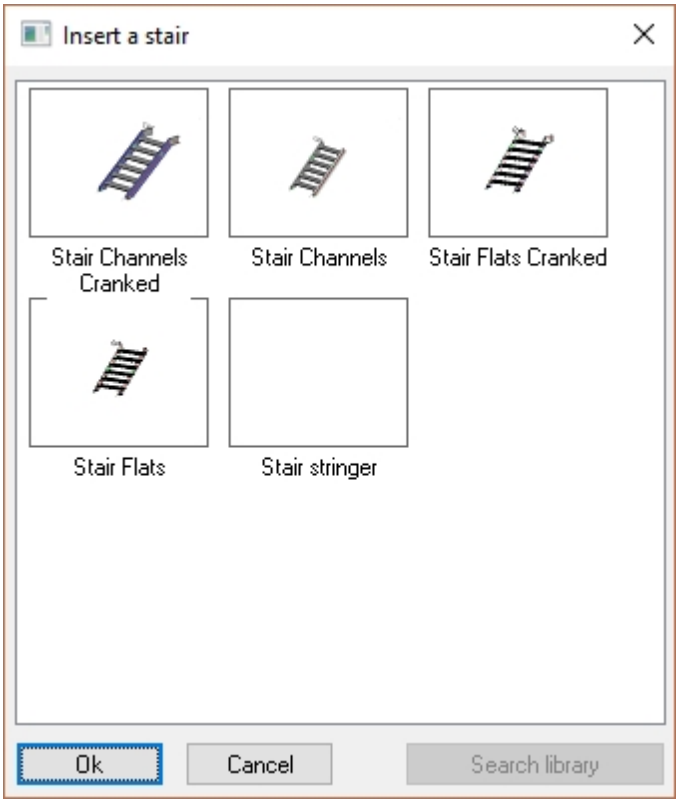
At the command line prompt: Select the line for the stair - this will open the Stair configuration dialog prompting you to select the desired stair type.

Note! that the line will determine the inside face of the left hand stringer

When done, press **OK** and the default stair will be drawn accompanied by the Stair macro edit dialog.

The stair will be constrained to the line, my moving the line, the stair will move and the stringers,cranks, and connections will automatically update.

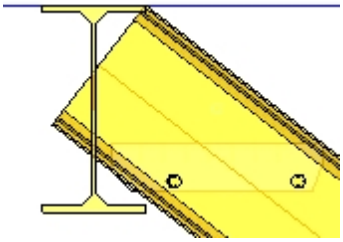
The Stair configuration dialog



The stair configuration dialog will prompt you to select the stair type - the main options include:

1. **Stair Channels Cranked** - This selection will draw a stair with channel stringers, cranked at the top and/or bottom.

2. **Stair Channels -** This selection will draw a stair with channel stringers but not cranked. By default, Parabuild will draw the top of the stringer intersecting the toe of the beam. This may be modified using the Stair macro edit dialog

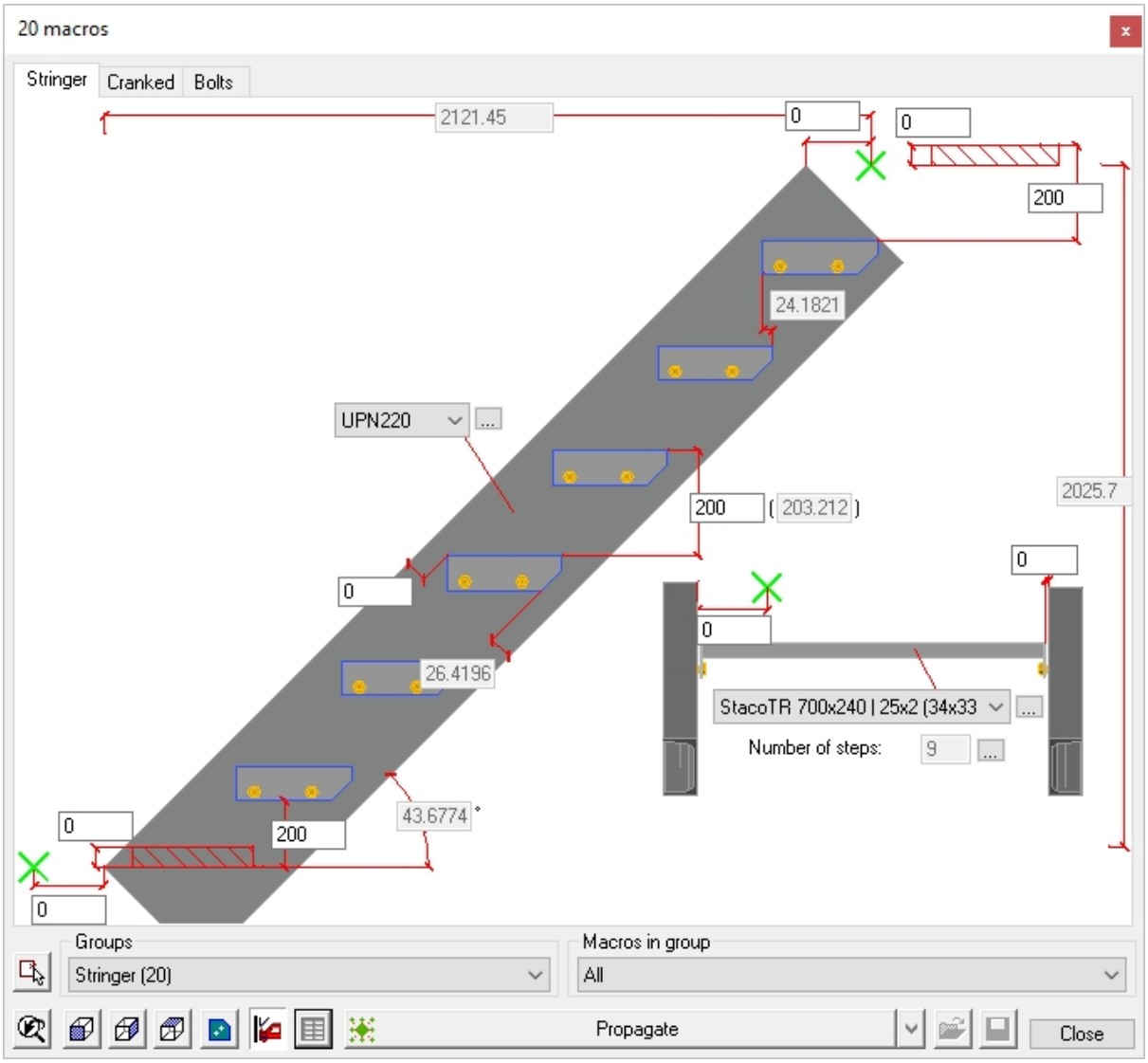


3. **Stair Flats Cranked** - This selection will draw a stair cranked at the top and/or bottom with flat bar stringers.

4. **Stair Flats** - This selection will draw a stair with flat bar stringers without cranks. The geometry will be the same as for **Stair Channels.**

The Stair macro edit dialog

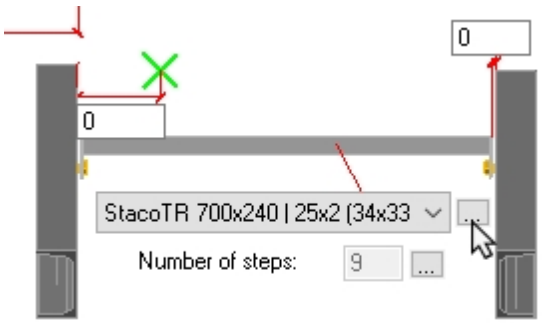
The default stair is accompanied by the customization dialog:



This dialog enable you to revise the geometry of the stair within the boundaries of the selected supporting steelwork. The dialog has 3 tabs:

1. / Stringer - 2. / Cranked - 3. / Bolts

The stringer tab enables you to set the profile size, stair tread risers, and the stair slope. Note! that the stair slope cab be modified directly if the stair is drawn from ground level. If the stair is constrained between a top and bottom beam, the angle can only be changed by revising the geometry of the horizontal component of the crank (Refer to the **cranked** tab). Or moving one of the beams up or down would also change the stair's slope.

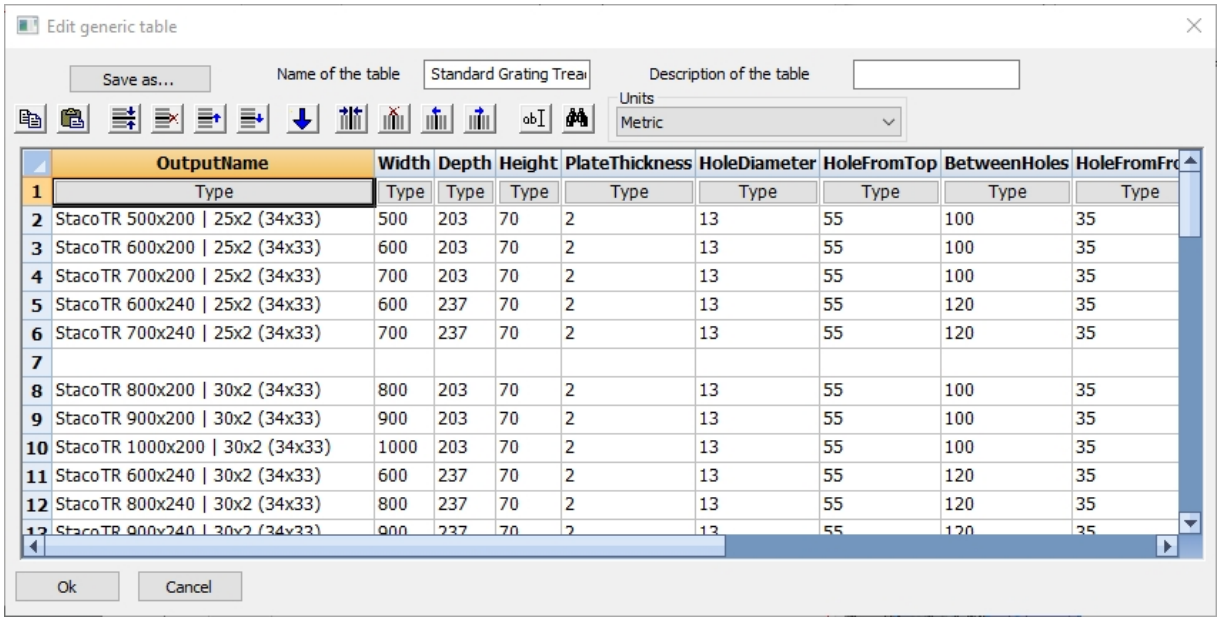


The stair treads may be set from the drop-down options - where the default options are drawn from the 'Staco' standard range. Note! that the width of the tread will determine the width of the stair.

For stair treads not included in this range, you may include standards of your choice by editing the treads table

To open the tread table, press the button ... on the right-hand side of the stair tread drop-down.

Here you will see a table wherein each row represents a tread.
The columns determine the geometry of the treads, more specifically:



	OutputName	Width	Depth	Height	PlateThickness	HoleDiameter	HoleFromTop	BetweenHoles	HoleFromFr
1	Type	Type	Type	Type	Type	Type	Type	Type	Type
2	StacoTR 500x200 | 25x2 (34x33)	500	203	70	2	13	55	100	35
3	StacoTR 600x200 | 25x2 (34x33)	600	203	70	2	13	55	100	35
4	StacoTR 700x200 | 25x2 (34x33)	700	203	70	2	13	55	100	35
5	StacoTR 600x240 | 25x2 (34x33)	600	237	70	2	13	55	120	35
6	StacoTR 700x240 | 25x2 (34x33)	700	237	70	2	13	55	120	35
7									
8	StacoTR 800x200 | 30x2 (34x33)	800	203	70	2	13	55	100	35
9	StacoTR 900x200 | 30x2 (34x33)	900	203	70	2	13	55	100	35
10	StacoTR 1000x200 | 30x2 (34x33)	1000	203	70	2	13	55	100	35
11	StacoTR 600x240 | 30x2 (34x33)	600	237	70	2	13	55	120	35
12	StacoTR 800x240 | 30x2 (34x33)	800	237	70	2	13	55	120	35
13	StacoTR 900x240 | 30x2 (34x33)	900	237	70	2	13	55	120	35

1. **Output name** - Enter a unique name for the stair tread
2. **Tread width** - Measured over the side plates
3. **Tread depth** - Measured from the nosing to the back of the tread
4. **Plate thickness** - Side-plate thickness
5. **Hole diameter** - Bolt-hole diameter
6. **Hole from top** - Measured from the top of the tread to the center of the hole
7. **Between holes** - Measured as the horizontal distance between the holes
8. **Hole from front** - Measured from the front nosing to the center of the first hole
9. **Chamfer front1** - Side plate front chamfer
10. **Chamfer front2** - Side plate front chamfer
11. **Chamfer back1** - Side plate back chamfer
12. **Chamfer back2** - Side plate back chamfer

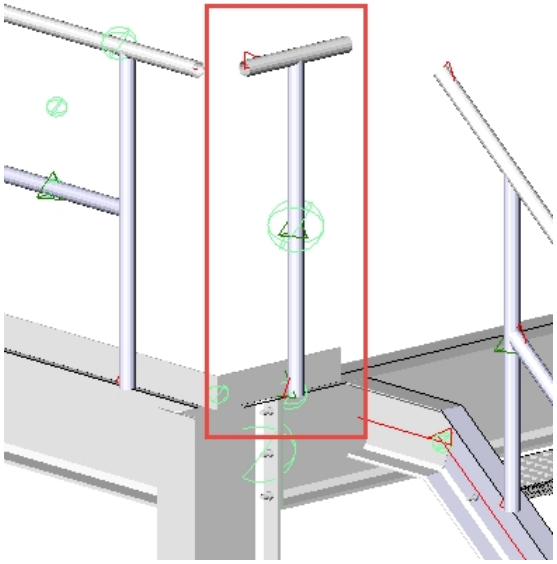
Railing

Command - **PrB_Railing**

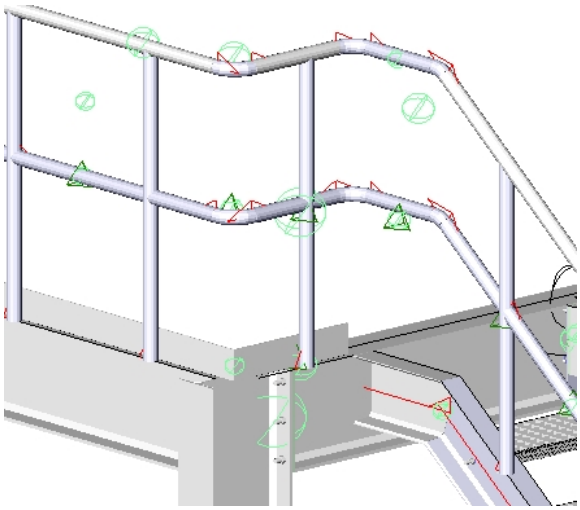
This command can draw railing segments on top of lines or profile edges. It will also connect these railing segments wherever possible.

After activating this command the **Insert a railing** dialog will appear, giving you the following options:

- **Railing**: This option will draw both handrail / knee-rail complete with a flexible amount of stanchions. By default it has a minimum of 2 stanchions. The number of stanchions will grow depending on the railing segment's length.
- **Railing 1 post**: This option will draw the Hand and Knee-rail only with just 1 vertical stanchions. This option would be used when the railing segment is so small that only 1 stanchion is needed. An example of such situation :

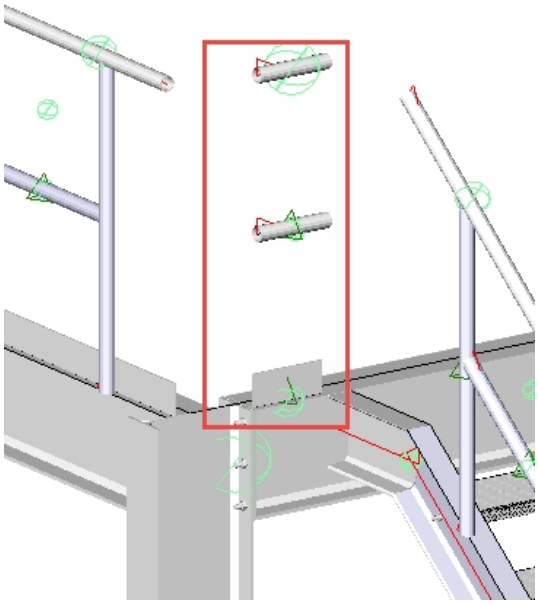


The first result after drawing the railing segment with just 1 post

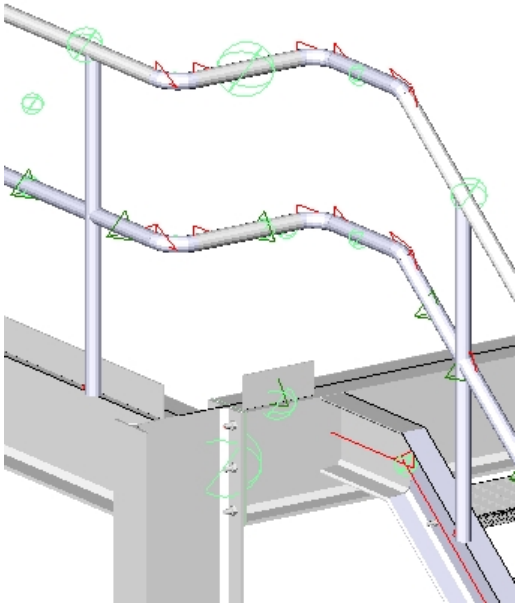


The end result after using the Railing intersection connections on it

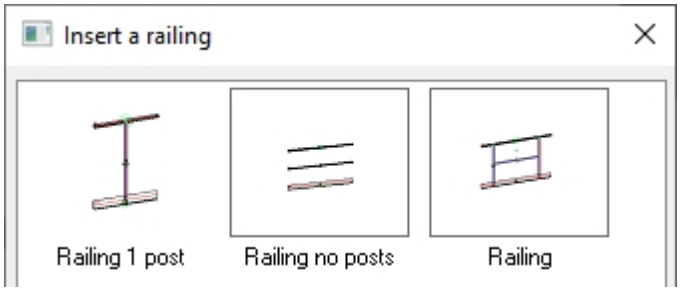
- **Railing - no posts**: This option will draw the Hand and Knee-rail only without vertical stanchions. This option would be used when the railing segment is so small that no stanchions are needed. An example of such situation :



The first result after drawing the railing segment without posts



The end result after using the Railing end connections and Railing intersection connections on it

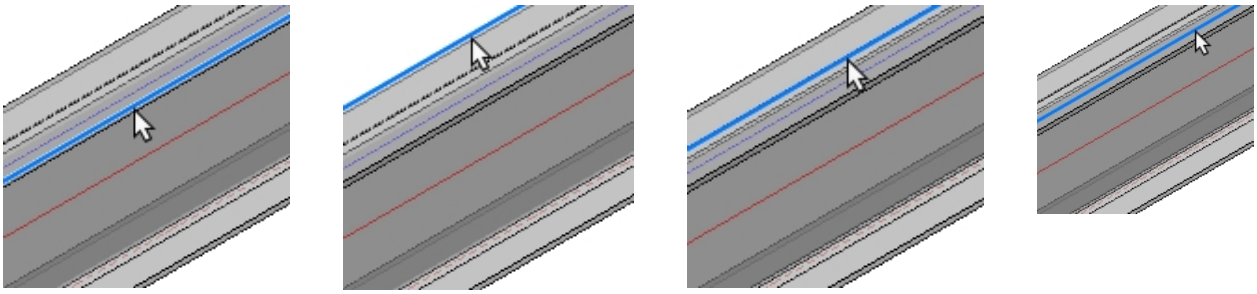


Select the desired option and press **OK** - then select the profile on which to place the railing

Here, it's important to note, that the selected edge of the beam will determine the placement, which may be either of the top edges, the center of the beam, or a user drawn line.

Note that the selected edge will become the center of the handrail. This location can still be offset afterwards.

The handrail may be edited at any time by means of the Review macro command.

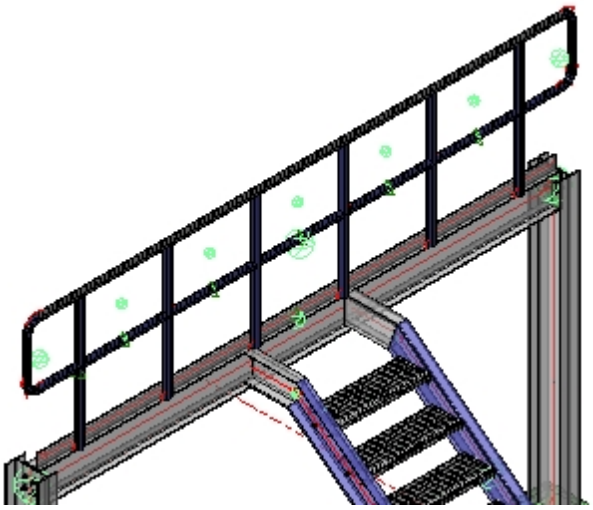


Near Side Edge *Far Side Edge* *Center* *User defined line*

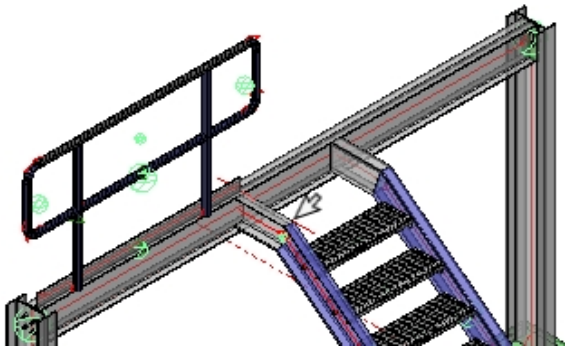
Next you will be prompted to select a geometry to shorten the railing

Pressing <**Enter**> will draw the handrail the full length of the selected member or line, disregarding any incoming stairs or platforms (*See Fig. 1*)

Alternatively, you may select an incoming beam or stair stringer to shorten the railing (See Fig. 2)



*Pressing <**Enter**> will draw the handrail the full length of the selected member or line*



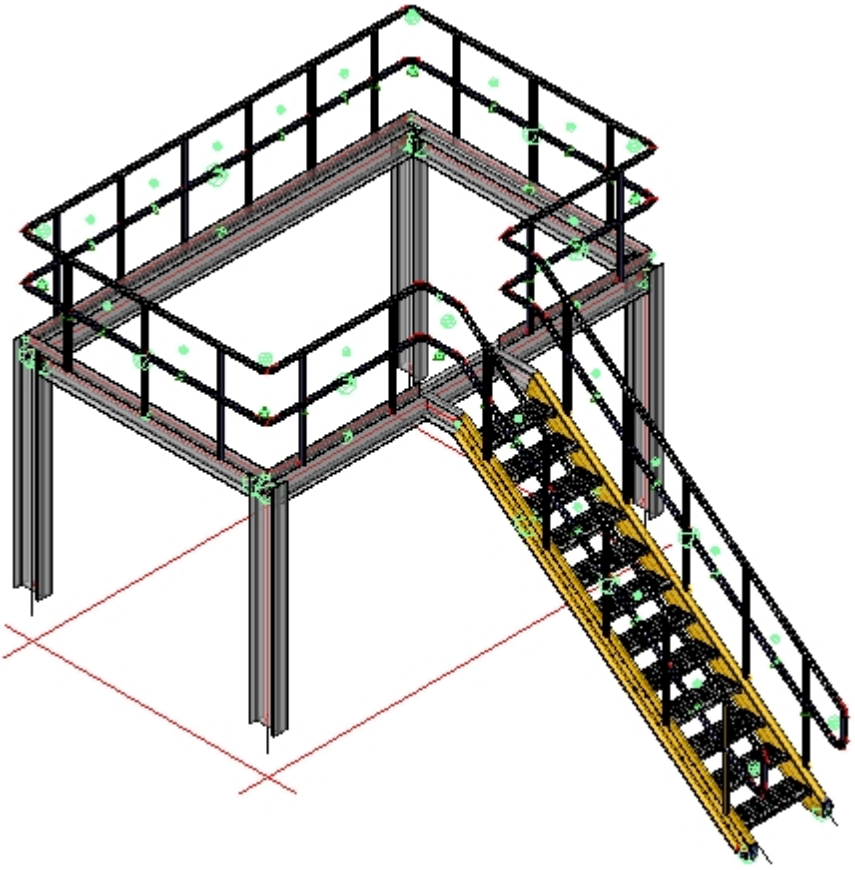
Indicating a geometry to shorten the railing - in this case - the inside face of the incoming stair stringer

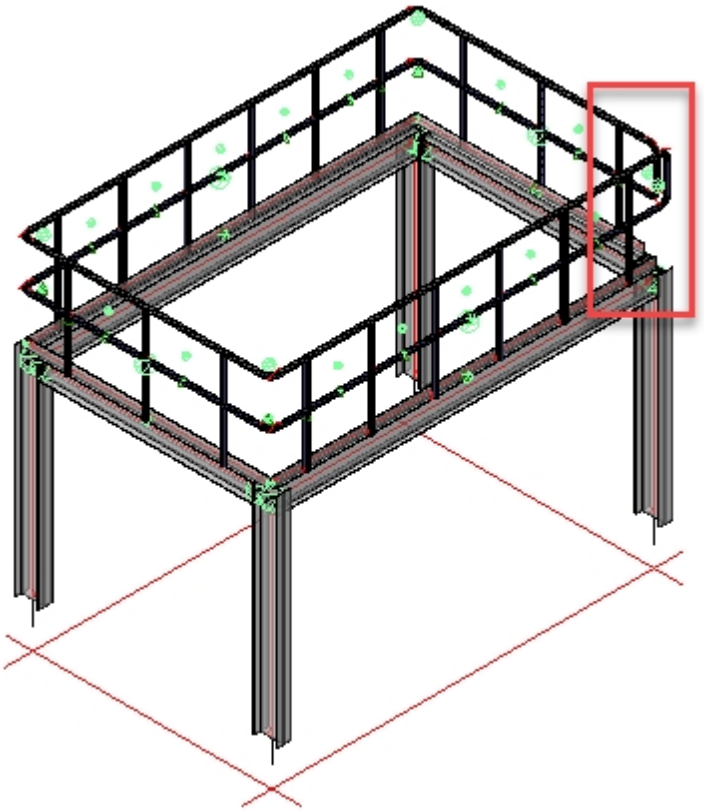
Chaining the handrail segments

After drawing the first handrail segment - the prompts will repeat allowing you to continue to trace the handrail route. Parabuild will automatically join the handrail segments together. To enable this sequential operation to work at its best, the handrail segments should be selected contiguously.

Parabuild will automatically insert a closure at non-contiguous joints. In the example illustrated below, the handrail was begun and ended at the stair - in which case the closures at the bottom of the stair are appropriate.

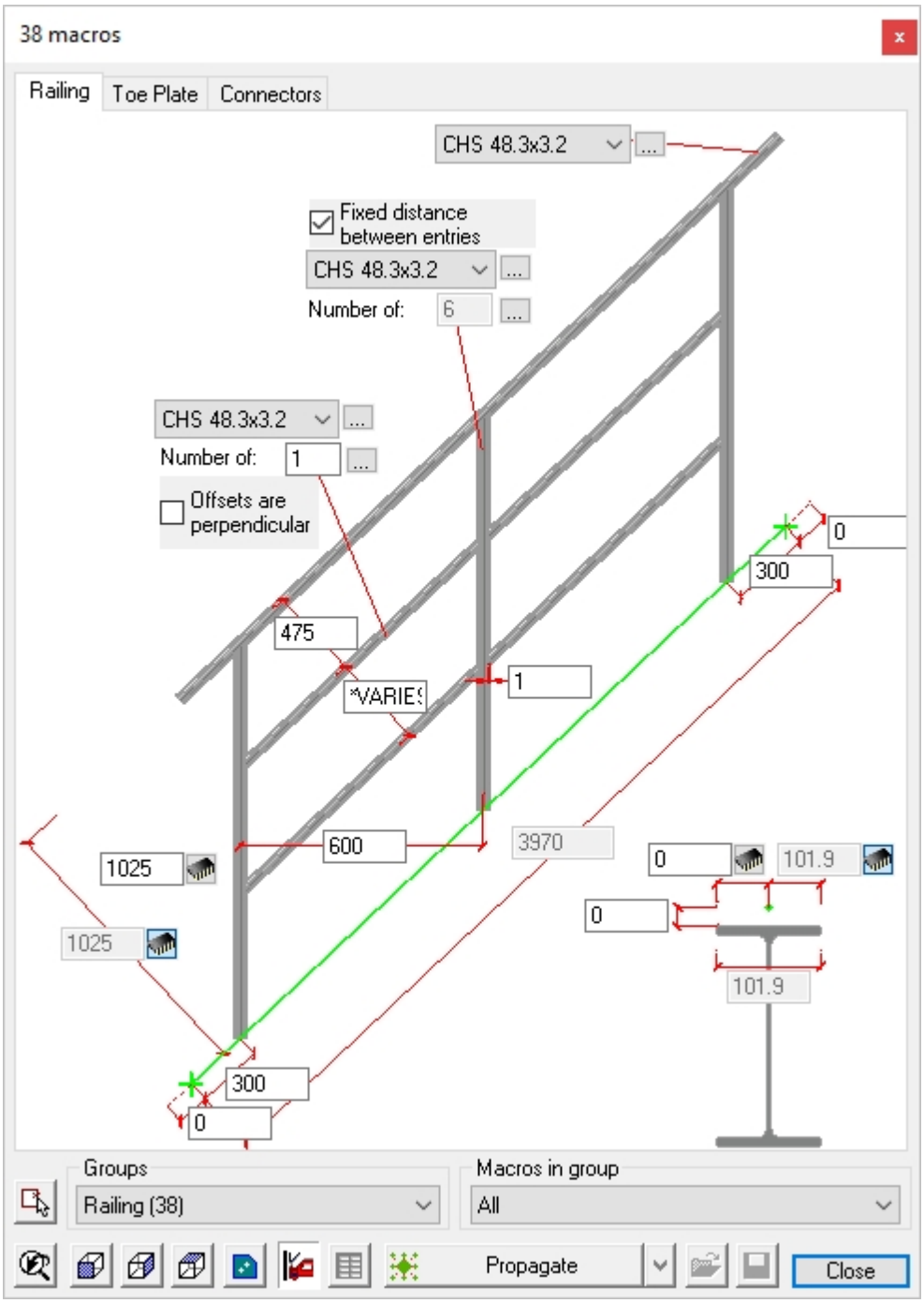
If the handrail is placed around a platform with no shortened segments, the closures will still be placed at the beginning and end of the chained handrail segments. To rectify this, delete the closure segments using the Delete macro command and activate the Railing intersection connections command and select the handrail stanchions - Press <**Enter**> and the returns will be automatically drawn.





Editing the Handrail Dialog

As the handrail is drawn, the handrail edit dialog will appear. This dialog has 3 tabs: Railing / Toe Plate / Connectors



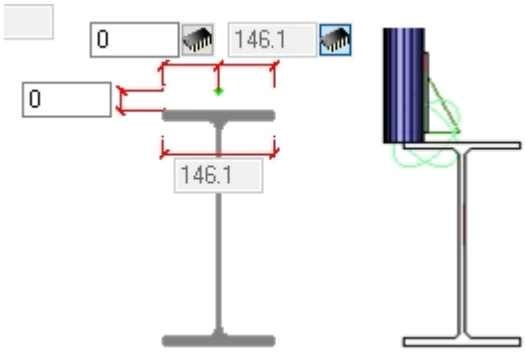
Railing:

Here, you may edit the handrail, knee rail, and stanchion section sizes. By default they are drawn as standard Circular Hollow Sections - by activating the drop down options menus adjacent to the default size, you may select another size.

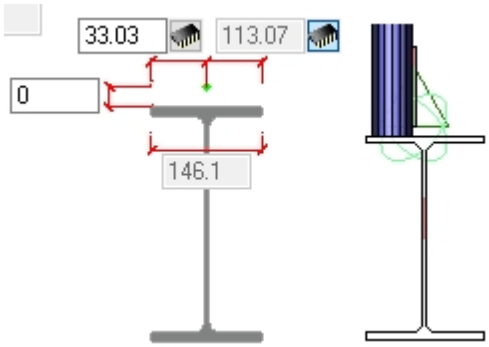
Hitting the button to the right will open the Profile placement dialog where you are able to select a section from another group. (*Be careful when doing this as it may require significant editing of the various handrail joints and connections*)

It is also possible to edit the stanchion and handrail spacing.

Placing the handrail relative to the beam or stringer can be done by revising the dimensions at the bottom/right of the dialog

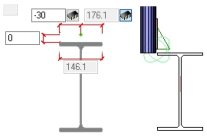


Handrail located on edge of beam



Handrail located on beam flange

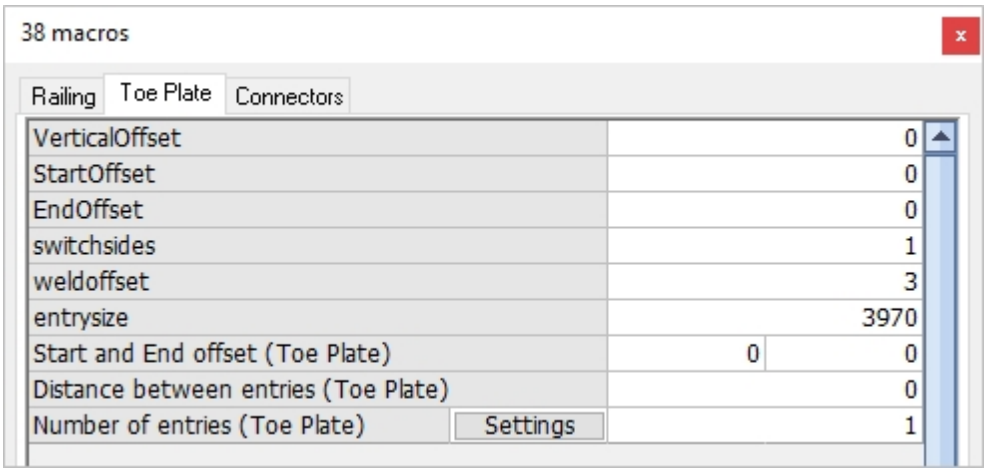
For base-plate mounting



Handrail located to clear beam

For side-plate mounting

Toe Plate:



The options for editing the **Toe Plate** or **Kick Plate** include:

Vertical offset - the distance between the bottom of the toe plate and top of steel

Start offset - The horizontal offset at the beginning of the toe plate

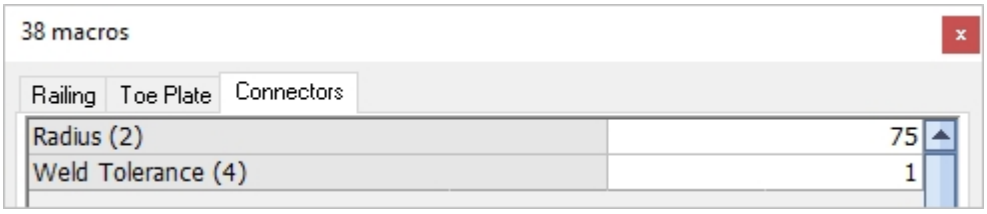
End Offset - The horizontal offset at the end of the toe plate

Switch sides - Is a boolean variable, a value of **0** will place the toe plate on the inside of the handrail stanchion, while a value of **1** will place it on the outside

Weld offset - refers to the weld gap between the stanchion and the toe plate, the default setting is 3 mm

Connectors:

The Connectors tab contains 2 options:

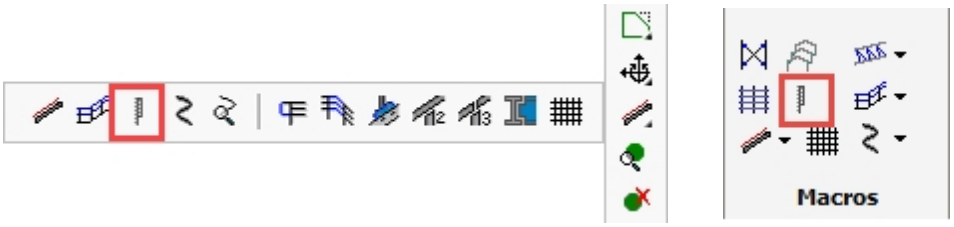


Radius - Here you may edit the radius of the bends

Weld Tolerance - Here you may edit the weld gap between the bends and the horizontal rails.

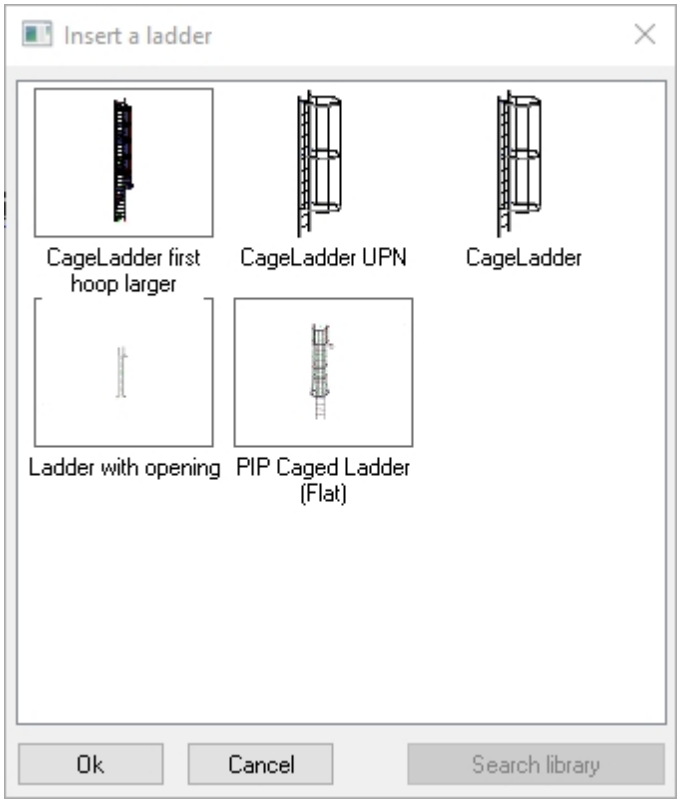
Ladder

Command - **PrB_Ladder**



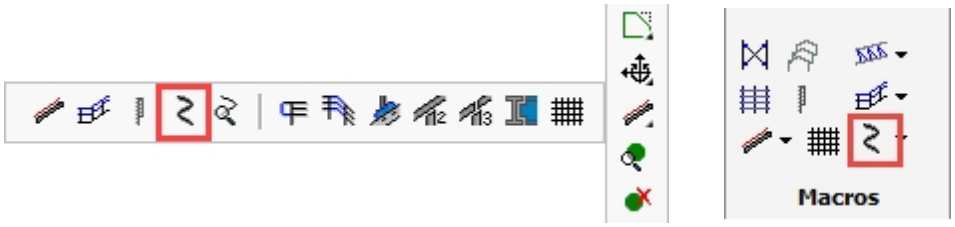
This command allows you to draw a variety of ladders, among which :

- Cage Ladder - First hoop larger
- Cage ladder UPN
- Cage Ladder
- Ladder with Opening
- PIP Caged Ladder



Spiral Stair

Command -



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.

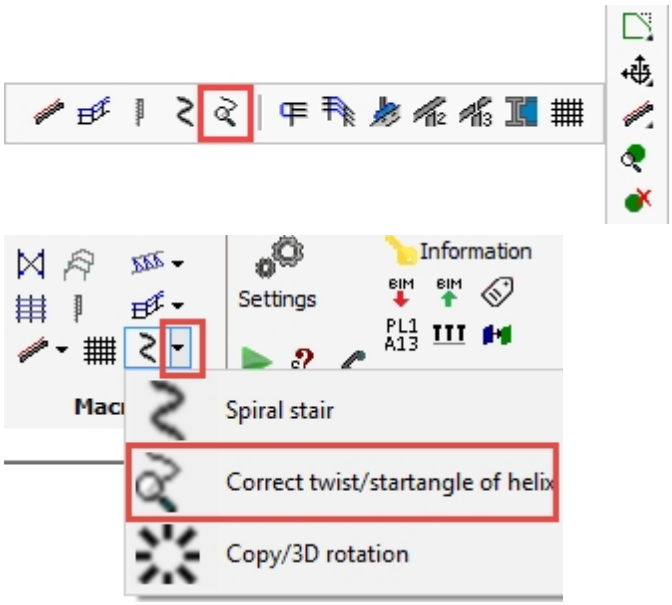
This command will draw the following entities :

- **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.

Correct Twist of Helix

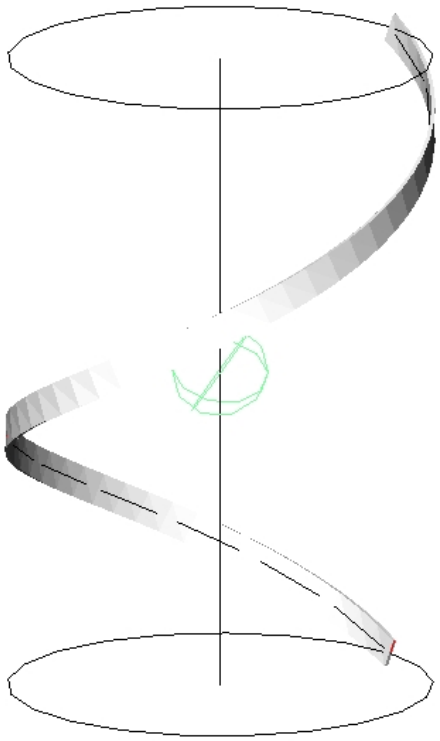
Command - **Prb_ResetTwistAngle**



This command will recalculate the twist angle of a profile with a helix axis.

The twist angle is what keeps a flat bar 'upright' on the helix path.

The current twist angle value can be found in the properties of the profile.

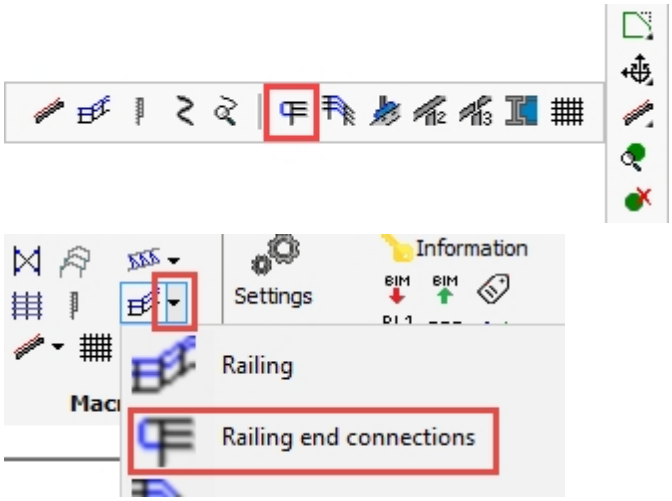


Moreover, when a profile is linked with a macro to the helix, then the macro will maintain the twist angle so that this command is not needed.

A Circular Hollow Section does not need to have a correct twist angle because the section shape is fully symmetric.

Railing End Connections

Command - **PrB_AddRailingEndConns**



This command will add a closure to a railing segment.

The railing segment command already does this automatically.

But there may be instances where a handrail closure must be added manually.

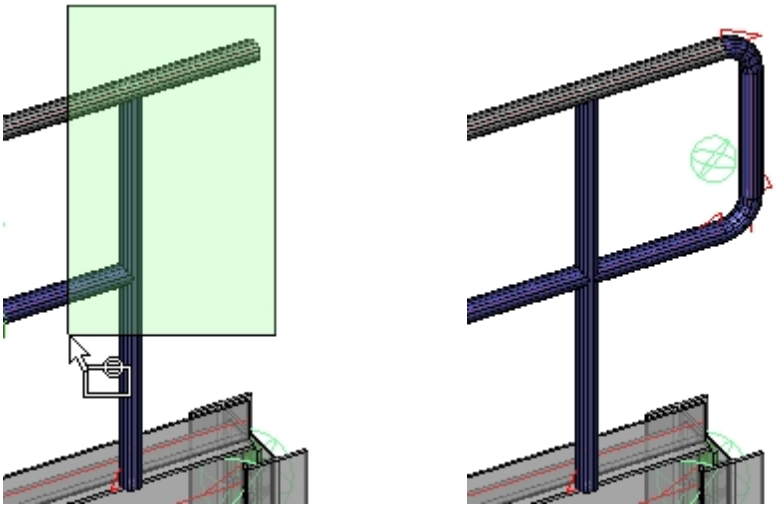
This command has 2 distinct methods :

Automatic Method

This method is unique, because you just need to select the handrails and the last posts.

It does not matter if you select more posts or knee-rails. The sequence is also not important.

Block-select the handrail end that needs to be connected and press <**Enter**> The closures will be automatically placed.



Automatic Method

Manual Method

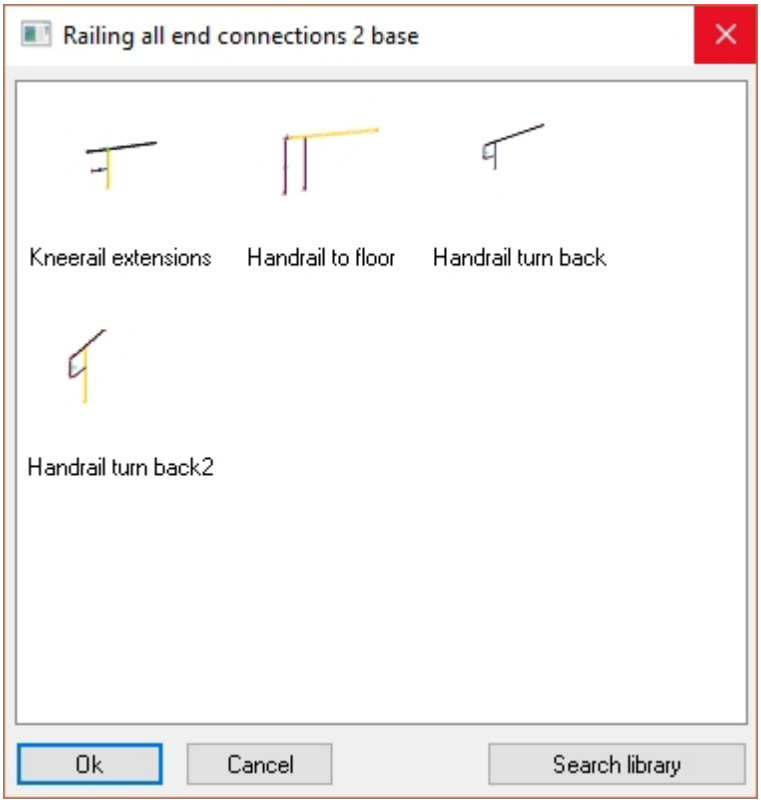
Right after starting this command, press the **<Enter>** button without selecting anything.

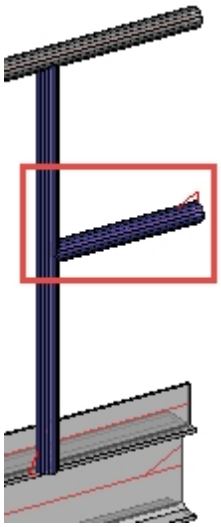
This brings us to the manual method.

Now you will have to first select the handrail close to it's end.

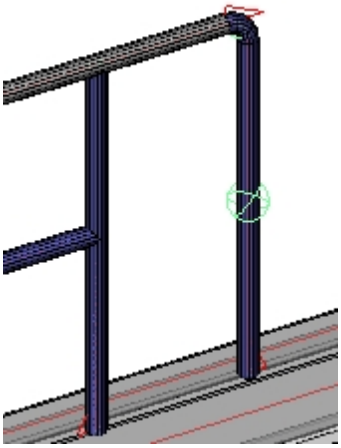
Note : Parabuild will use this handrail's side as an orientation marker to decide on which side of the post to place the connection.

After that, select the post and you will have the option to choose any of the connections in the group :

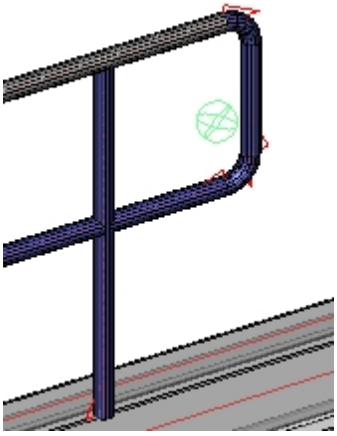




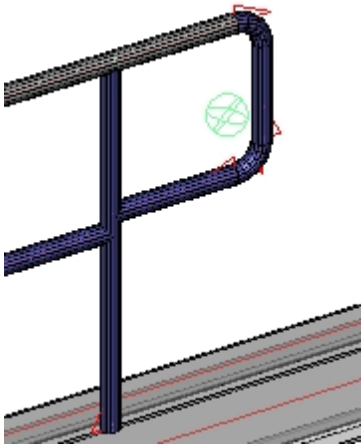
Knee rail Extensions. These are needed when you want to connect a stanchion to a railing segment without stanchions. An example case is shown in the Handrail section.



Handrail to floor



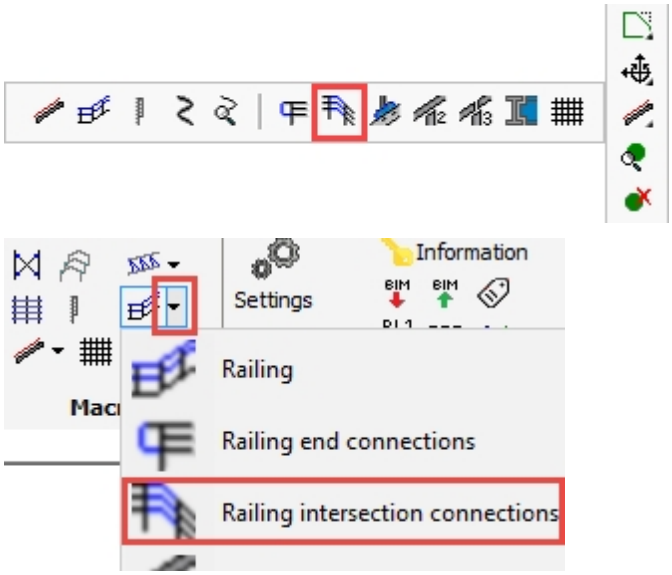
Handrail Turn-back



Handrail Turn-back 2
This is exactly the same macro, but it has a different orientation.
In this command it does not matter which one you choose, but it does matter for automated tools that draw these end connections automatically.

Railing Intersection Conns.

Command - **PrB_AddRailingInterConns**



This command can connect 2 railing segments.

The railing segment command already does this automatically.

But there may be instances where a handrail closure must be added manually.

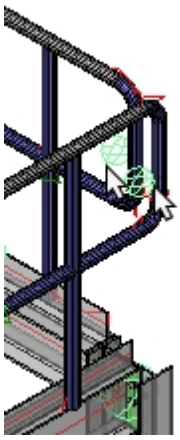
This command has 2 distinct methods :

Automatic Method

This method is unique, because you just need to select the handrails and the last posts.

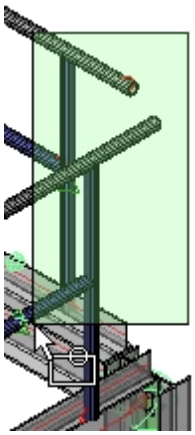
It does not matter if you select more posts or knee-rails. The sequence is also not important.

Parabuild will try to automatically connect all the members in the selection using the connections in the chosen library group.



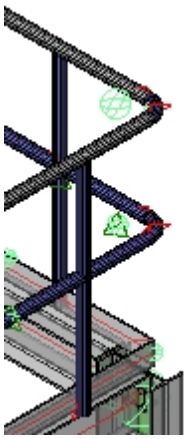
1

Select and delete the unwanted macros



2

Block select the handrail stanchions - Press <**Enter**>



3

The returns are automatically drawn

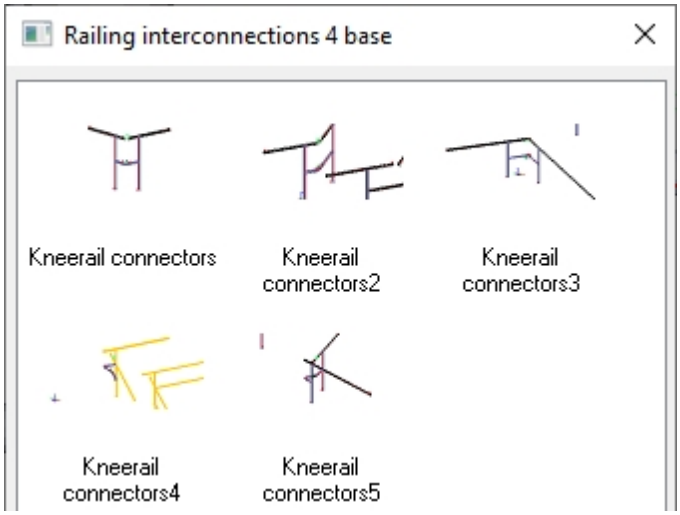
Manual Method

Right after starting this command, press the **<Enter>** button without selecting anything.

This brings us to the manual method.

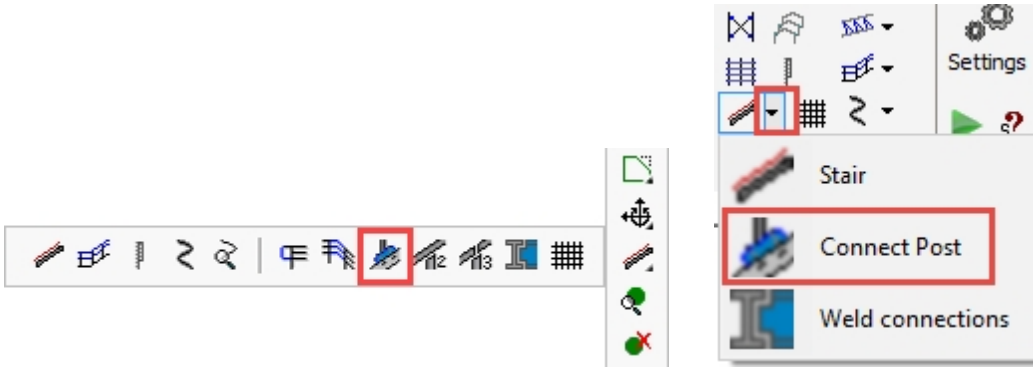
Now you will have to select both the handrails and the posts.

After that you will have the option to choose any of the connections in the group :



Connect Post

Command -



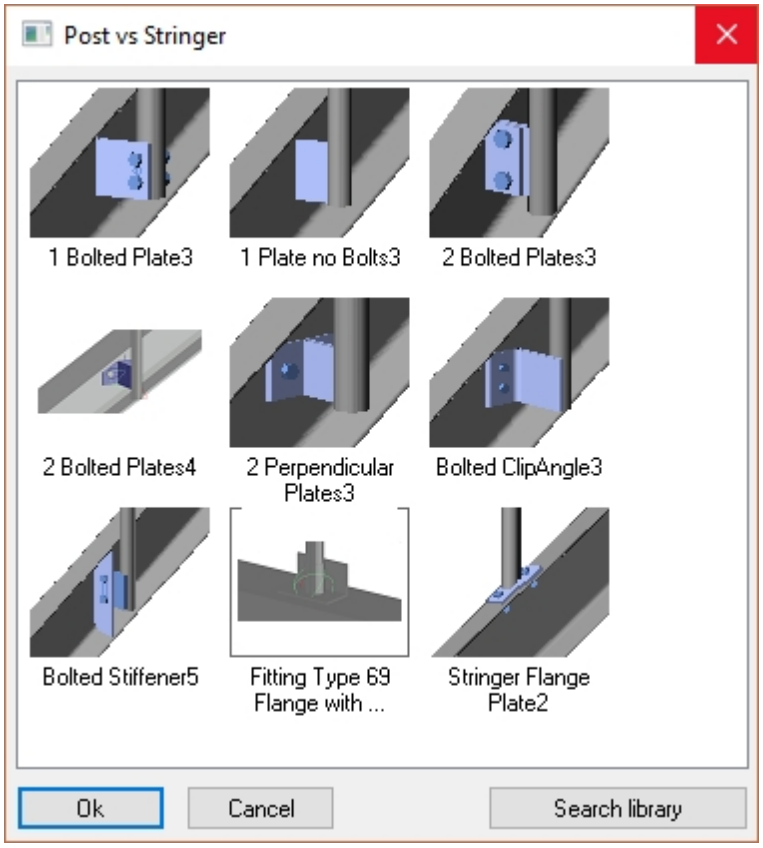
This command will apply handrail stanchion connections to a beam of stair stringer

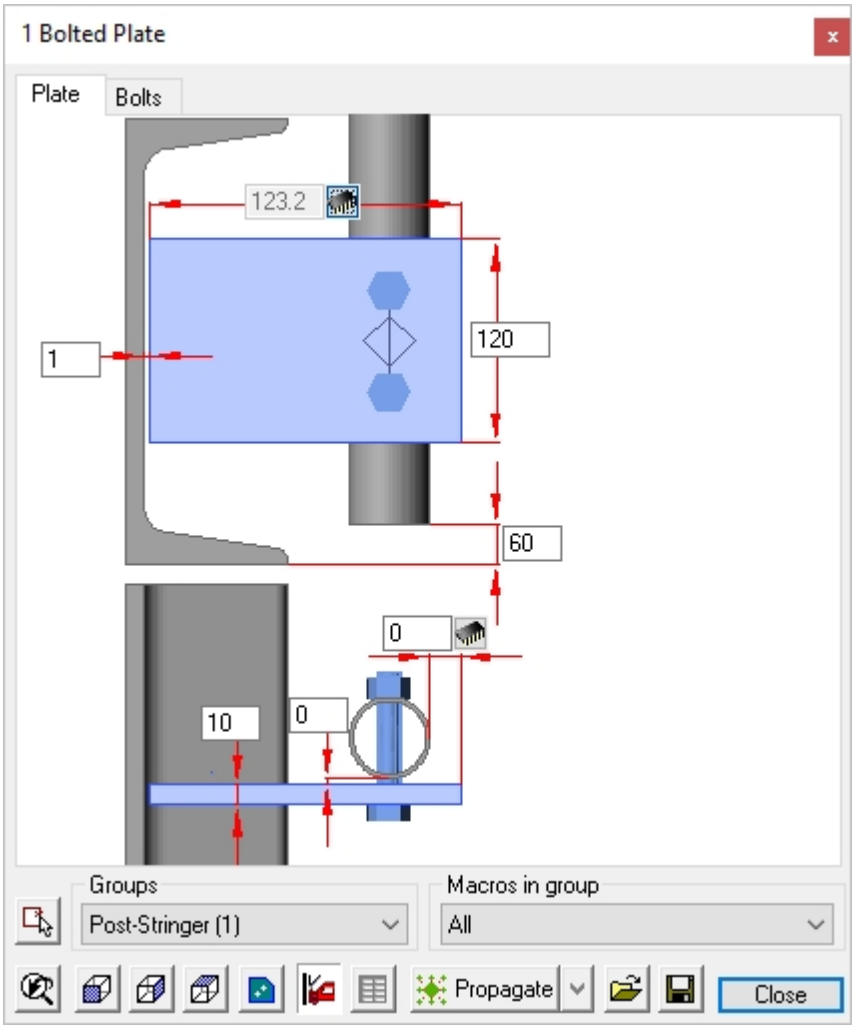
Select the beam or stair stringer and Parabuild will automatically detect the angle of the stair stringer and adjust the connection orientation to suit.

Then select the post and press **Ok** - The default connection will be drawn - Pressing <**Enter**> will open the connection edit dialog.

Note! Ensure that the handrail post is properly located on the stringer to accommodate the selected connection. For more information on this go to - Stairs / Handrail / Railing

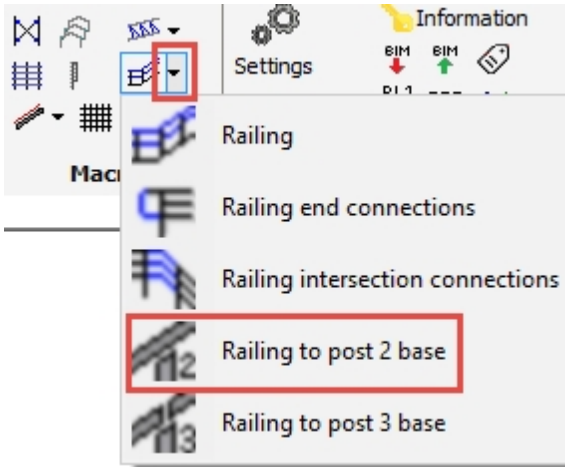
For more information on editing macro dialog boxes see the Review macro topic.


Post vs Stringer
1 Bolted Plate3
1 Plate no Bolts3
2 Bolted Plates3
2 Bolted Plates4
2 Perpendicular Plates3
Bolted ClipAngle3
Bolted Stiffener5
Fitting Type 69 Flange with ...
Stringer Flange Plate2
Ok
Cancel
Search library



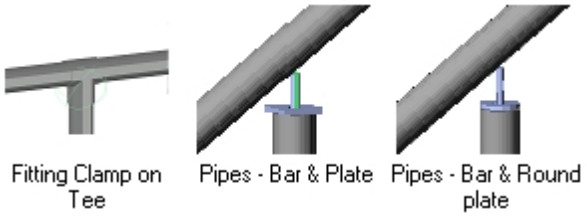
Connect Post to Handrail - 2 Base

Command -



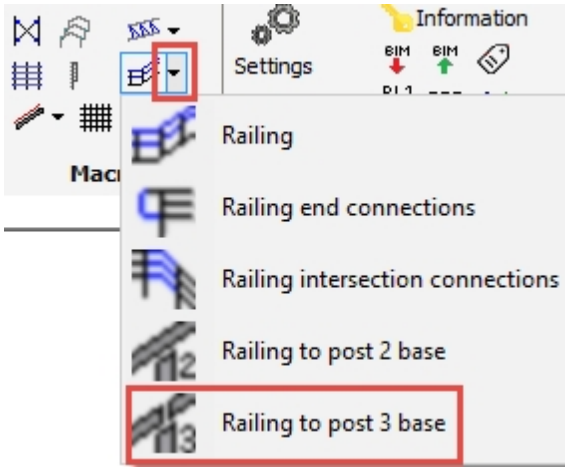
This command can connect a post to a railing.

Some example connections are :



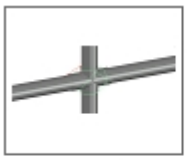
Connect Post to Handrail - 3 Base

Command -



This command can connect a post to 2 railings when they are spliced.

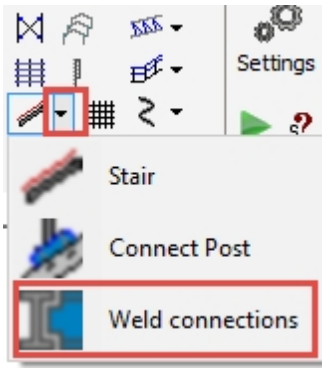
For now the only application for this command is for fitting connections :


Fitting 2 Socket Cross


Fitting 3 Socket Tee

Weld Connections

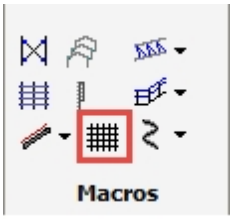
Command -



For more information about these connections, see the Weld connections topic in the connections chapter.

Draw Grating

Command - **PrB_Grating**



This command allows you to draw floor grating or raised pattern floor plates over a defined surface area. The flooring options include:

PLBG - Press lock bar grating

WBG - Welded bar grating

SLBG - Swage lock bar grating

RPFP - Raised pattern floor plates

These options are explained in more detail under Floors

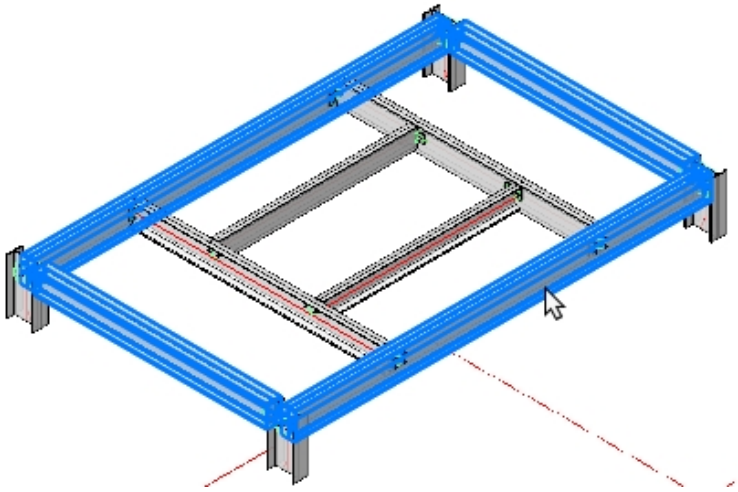
You will be prompted to select the entities that define the boundary for the floor, when don, press <**Enter**> The floor will be automatically drawn accompanied by the edit dialog.

The floor panels will be placed as an array at the extremities of the selected entities with the bearing bars running in the direction of the shortest span. The floor panels are set according to the selected standard width. These widths have been established by Parabuild at 800 mm, 900 mm, 1000 mm, 1100 mm, and 1200 mm.

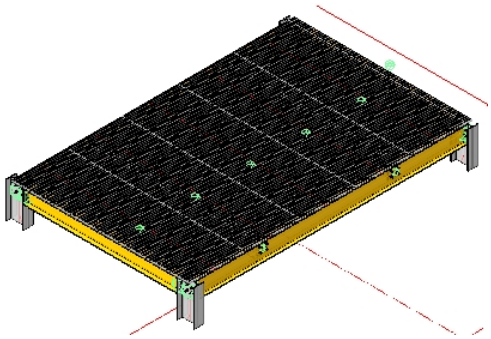
At the time of writing, the command only supports flooring patterns of which the contour is described by 4 profiles.

Note! Standard widths, maximum panel lengths, bearing bars, transversal bars, may be customized to suit regional or manufacturers standards by editing the Flooring Profile Library. The same library allows you to create new libraries to meet the above criteria.

The floor panels will be placed, with the width of the last panel being non-standard to accord with the total overall width of the defined surface area. This may be edited in the flooring edit dialog.

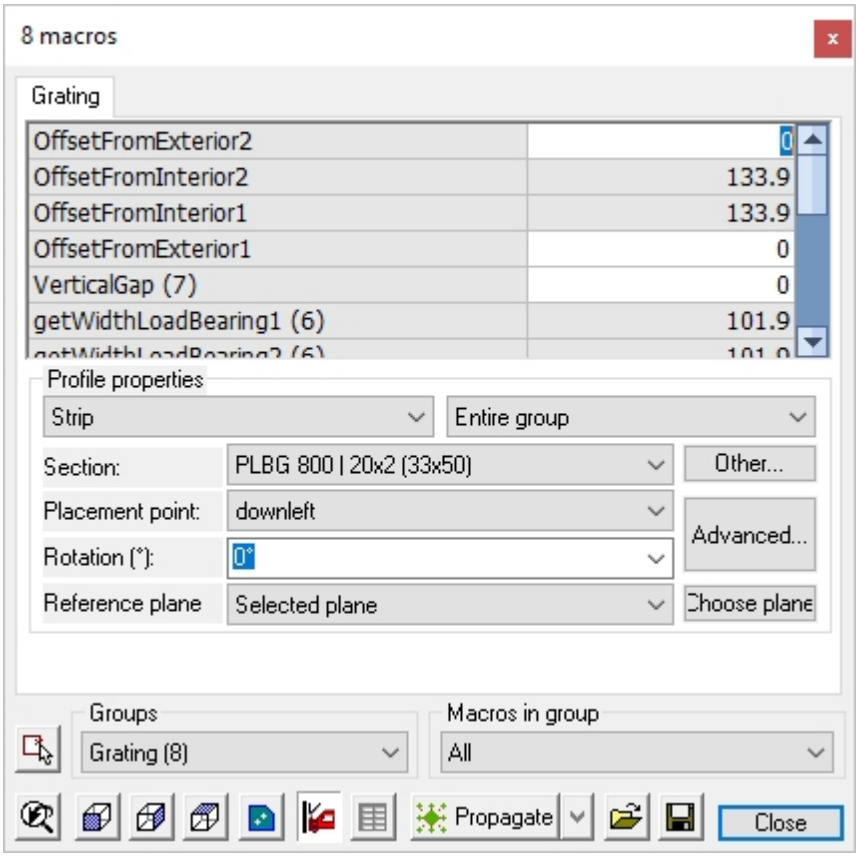


Select Entities



Draw Grating

The initial flooring edit dialog will edit the entire flooring array providing the following options: Individual floor panels may be edited by using the Review macro command.



OffsetFromExterior2 - (Variable) - The offset between the floor panel and the right side extreme edge of the defined surface area

OffseFromInterior2 - (Fixed) - The value given is representative of the width of the top flange of the beam. This value will update as the former is edited.

OffsetFromExterior2 - (Variable) - The offset between the floor panel and the right side extreme left of the defined surface area

OffsetFromInterior2 - (Fixed) - The value given is representative of the width of the top flange of the beam. This value will update as the former is edited.

VerticalGap - Is the distance between the top of the floor beams and the underside of the flooring panels

GetWidthLoadBearing1 - Is the flange width of the top load bearing member

GetWidthLoadBearing2 - Is the flange width of the bottom load bearing member

OffsetFromLoadBearingExterior1 - (Variable) - The offset between the floor panel and the top side extreme edge of the defined surface area

OffsetFromLoadBearingInterior1 - (Fixed) - The value given is representative of the width of the top flange of the beam. This value will update as the former is edited.

OffsetFromLoadBearingExterior2 - (Variable) - The offset between the floor panel and the top side extreme edge of the defined surface area

OffsetFromLoadBearingInterior2 - (Fixed) - The value given is representative of the width of the bottom flange of the beam. This value will update as the former is edited.

GapBetweenGratings - Is the gap between the grating panels

TotalLength - This is the total length of the area from first to last beam.

Start and End offset (**Grating array**) - This will set the start and end offset of the entire grating array measured from the defined surface area

Distance between entries (**Grating array**) - This value is derived from the floor panel width and the gap between panels. This value will automatically update as the gap between panels is changed.

Number of entries (**Grating array**) - The number of panels forming the flooring array

Profile Properties

From the drop-down menu - **Strip** - All the grating panels are grouped under the name *Strip.* In other macros, more or other group names might be listed here.

From the drop-down menu - **Entire group** - Select this to edit all of the grating panels

Section - From the drop down menu you may select a different flooring type e.g. panel width, depth, bearing bar .....

Pressing the **Other** button will open the Select profile dialog, where you may select flooring from another group (PLBG, **WBG**, **SLBG**, **RPFP**)

Placement point - From the drop-down menu, you may select the placement points for the individual floor panels in the array. pressing the **Advanced** button will open the Profile placement dialog - where you may edit all the profile properties and placement.

Rotation - where you may select a vertical rotation angle from the drop-down options menu.

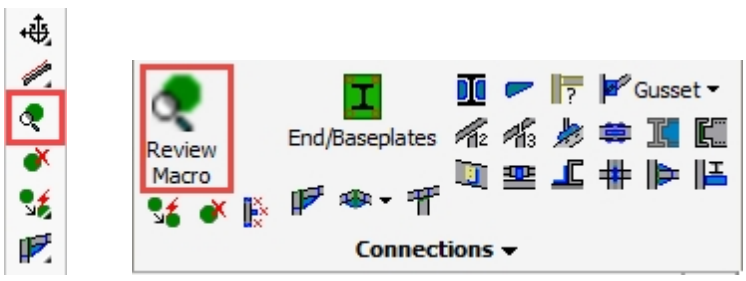
Reference plane - where you may select from the drop-down options menu the reference plane options, here you have the option to select a plane by pressing the **Choose plane** button. By default Parabuild will place the floor panels on top of the selected defined surface.

Review / Delete Macro

- Review Macro
- Delete Macro

Review Macro

Command - **PrB_DlgMacro**



A Macro is identified by a sphere, which is a collection of geometric rules that holds the 'intelligence' of the elements making up the connection.

When a base profile of a connection is modified, the rules in the sphere will ensure that the elements in that connection adapt automatically to whatever modifications are made.

Using this command you can review and edit multiple macros.

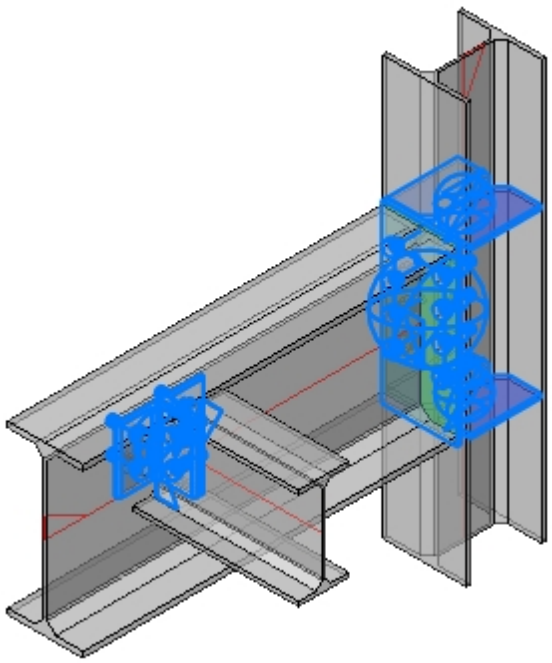
Here you may select one macro sphere using the mouse pointer, or any number individually, or using a frame.

When done, press <**Enter**> to open the macro edit dialog

For opening the the overall macro dialog of arrays of macros such as railings and trusses, do the following :

- If you are working with AutoCAD - holding the **ALT** key while double-clicking the **part**.
- If you are working with BricsCAD - the same can be achieved by holding the **CTRL** key while double-clicking the **part.**

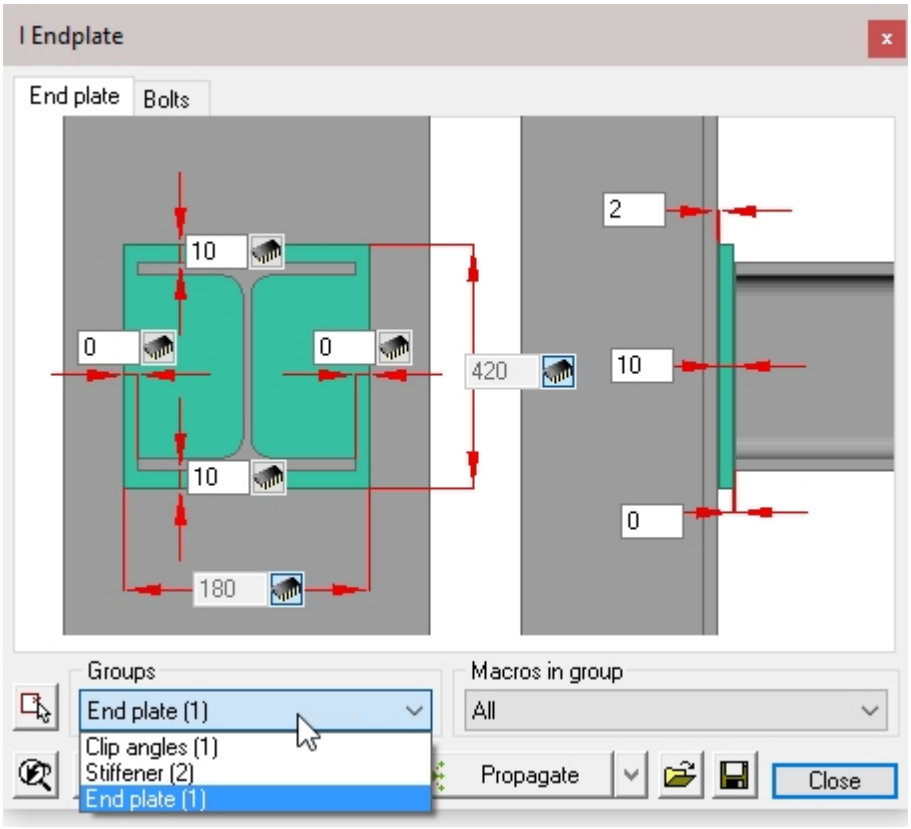
In this example we'll be working with a drawing fragment which is comprising of two connection types and two column web stiffeners.



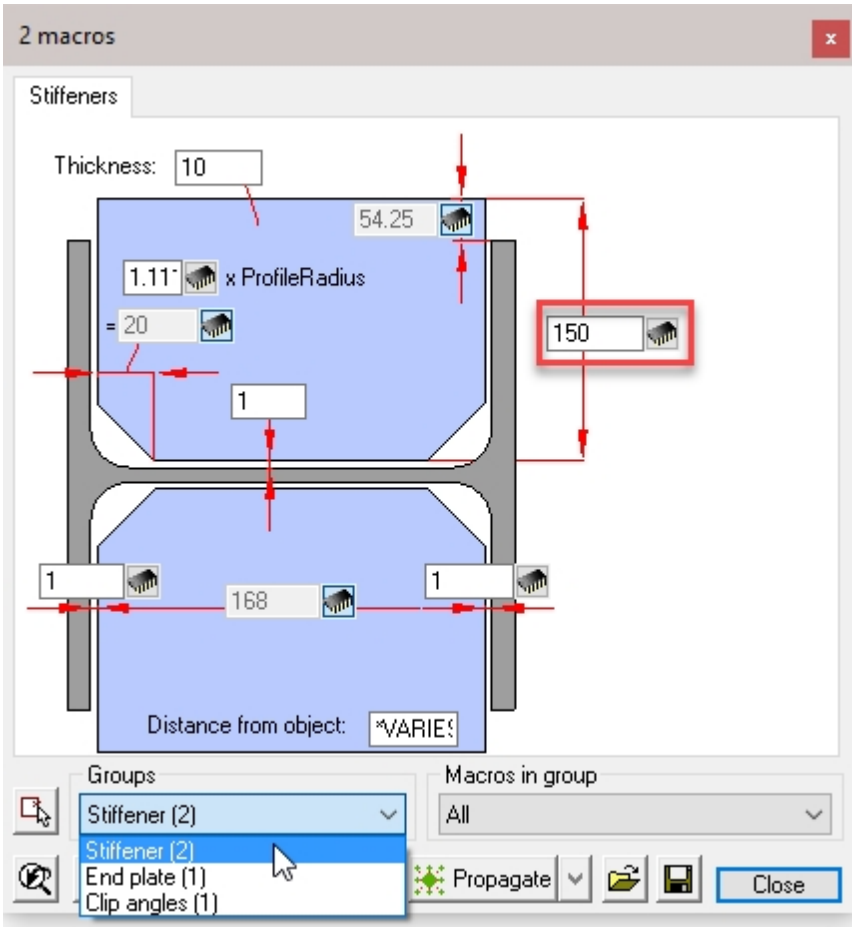
Clicking the **Groups** drop-down menu will show which macros have been captured by the selection (the number of each of the macros is shown in parenthesis).

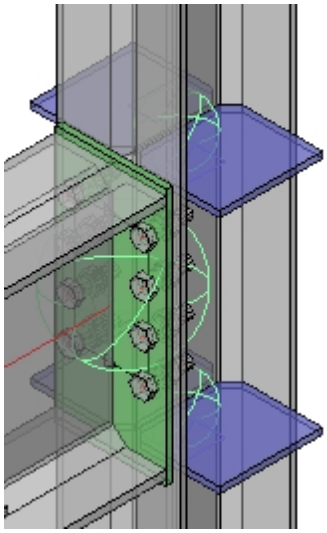
Clicking on one of these macros will open the specific edit dialog (For more information on editing the dialog go to The macro edit dialog).

Should there be more than one macro in the selected group, any changes made will reflect in all macros within that group. An example, in this case, are the stiffeners.



Select **Stiffener (2)** from the drop-down menu to open the edit dialog and change one of the dimensions, both of the stiffener macros will update

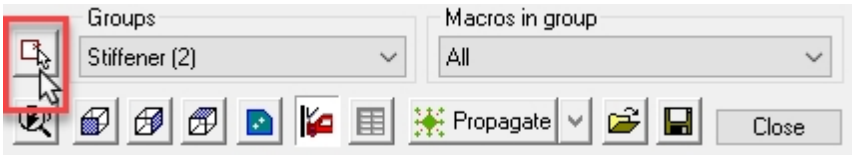


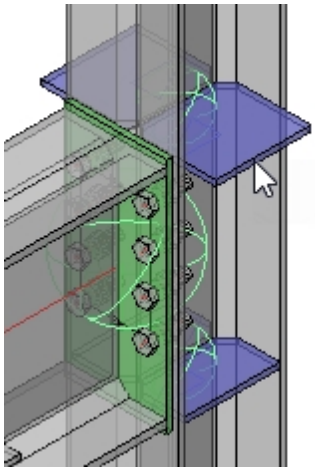


To change only one of the macros, click on the **Select other macros** button, and following the command line prompt: **Select Entities**.

Here you may select the stiffener macro sphere, or the stiffener itself and press <**Enter**>. Now edit the macro dialog as before, but this time only the selected macro will be updated.

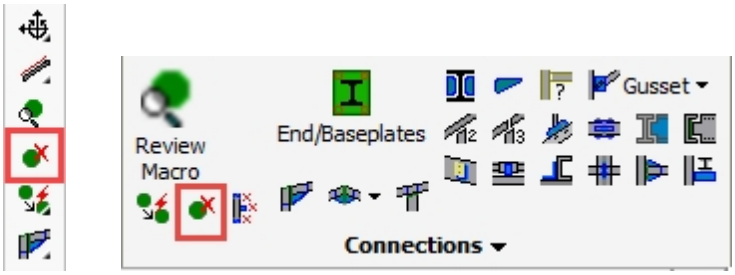
This function will work with all macros.



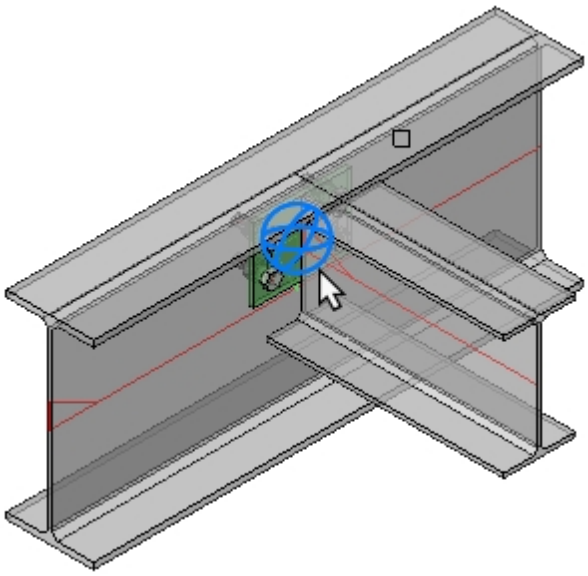


Delete Macro

Command - **PrB_EraseMacro**



This command will erase all parts of the macro and all holes or cuts that were determined by the macro.

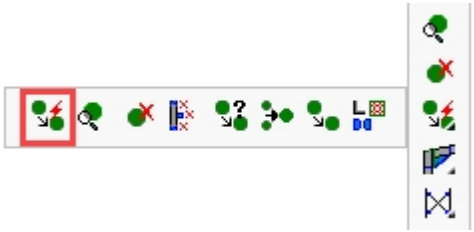


Modify Macros

- Smart copy
- Review macro
- Erase macro
- Erase objects and dependent macros
- Smart copy settings
- Merge macros
- Copy a macro
- Connection components library

Smart Copy

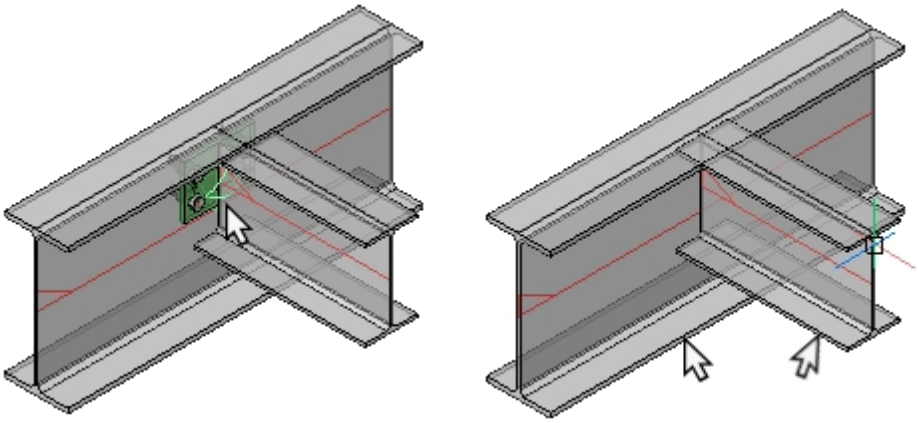
Command - **PrB_AutoApplyMacro**



This command will copy an existing macro to other base profiles in the same drawing.

Select the macro and then the base profiles for the new macro.
You should select the profiles in the same order they were selected when the original macro was inserted.

The macro will be copied in its entirety to the new profiles. You may modify or remove components of the new macro without affecting the original macro.



Smart Copy and macro mirroring

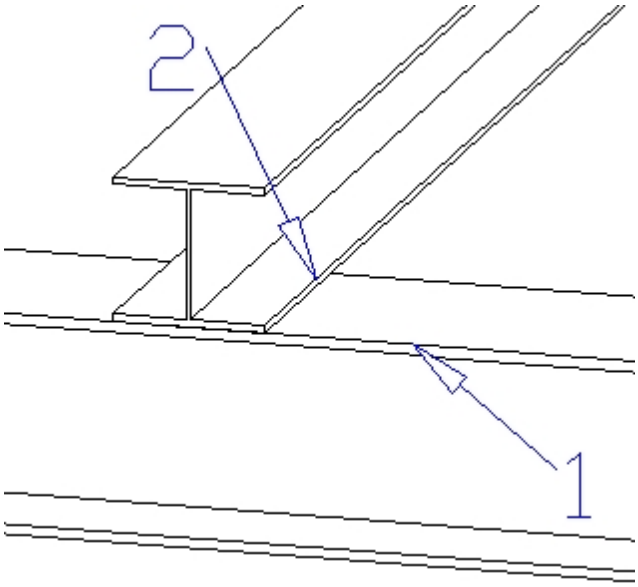
Some macros can be mirrored by the smart copy command. Other can only be rotated.

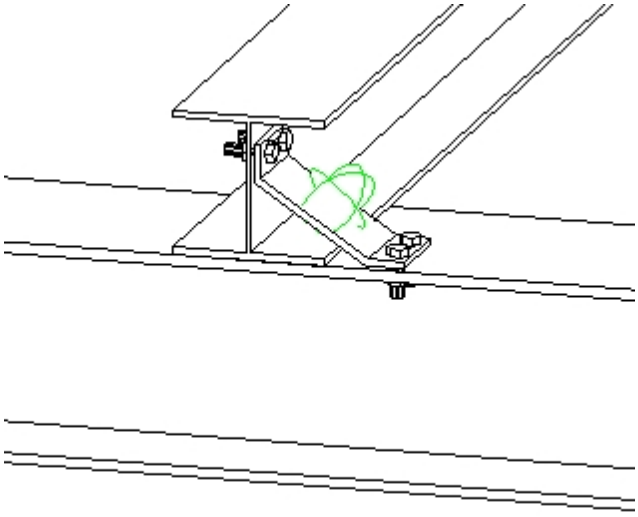
The Smart Copy command uses the location where you select the base profiles to know how you want the new macro to be rotated and mirrored.

With the following four examples we illustrate what happens with a connection that can be rotated and mirrored.

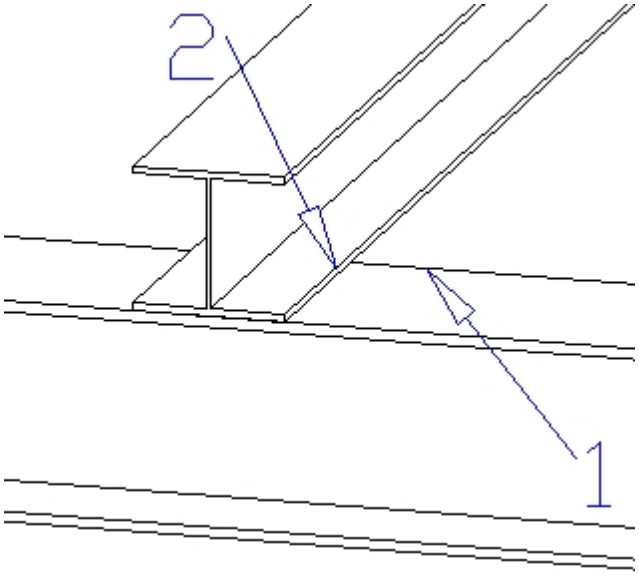
For each example the left image shows the selection that was during the smart copy. The right image is the result.

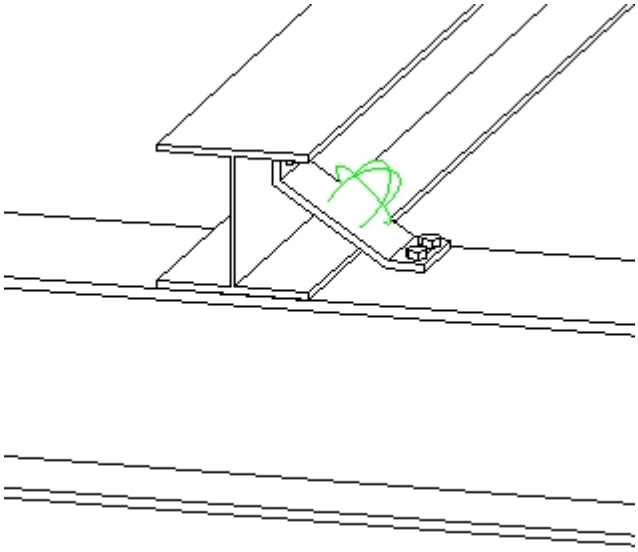
Example 1





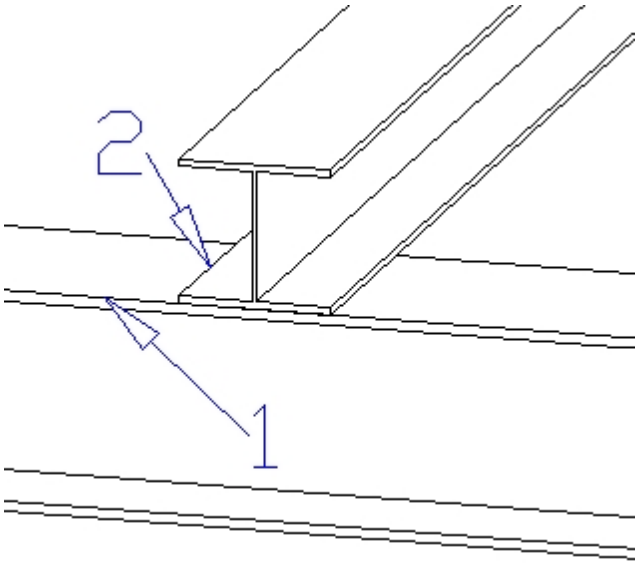
Example 2

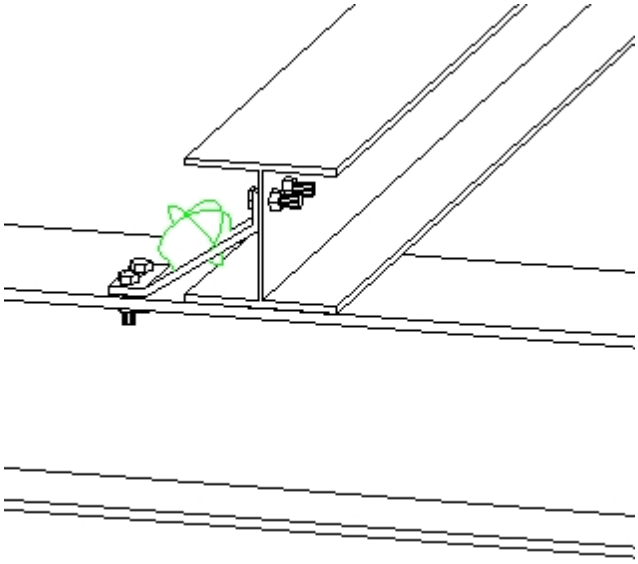




In this example the first base profile was selected on another spot, on the other side of the upper flange. The result is that the bent plate is also placed on that side of the flange. The bent plate was mirrored over 1 axis.

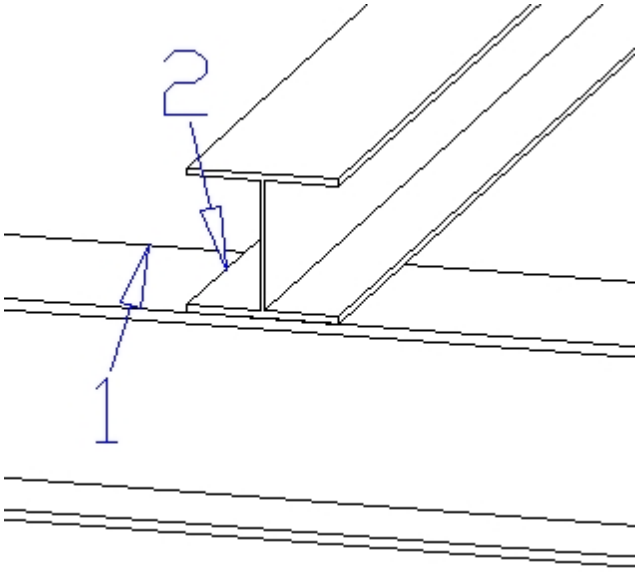
Example 3

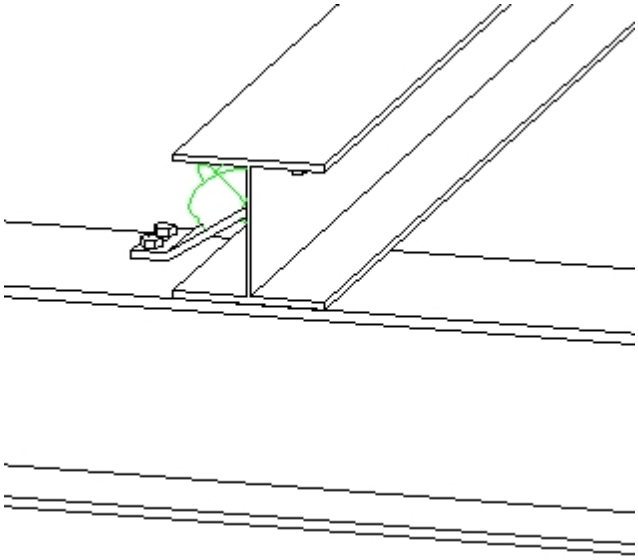




In this example the second base profile was selected on another spot, on the other side. Again we get a mirrored plate but at the opposite side of the second profile.

Example 4:



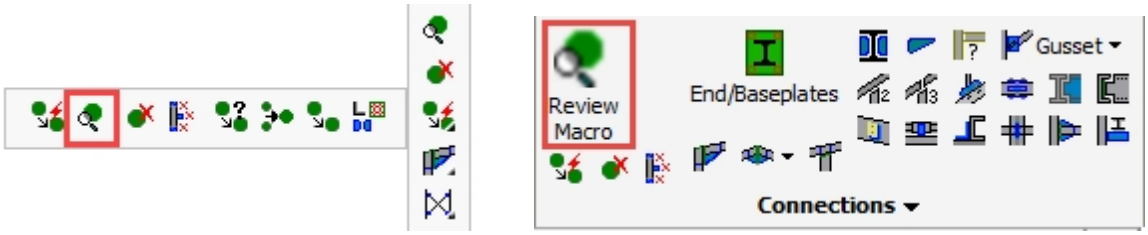


In this last example both the first and the last base profile were selected on another spot. The result is a double-mirrored plate.

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.

Review Macro

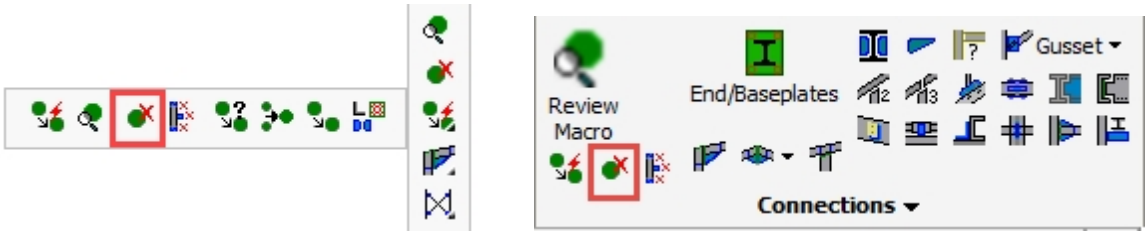
Command - **PrB_DlgMacro**



For more information, see Review Macro

Erase Macro

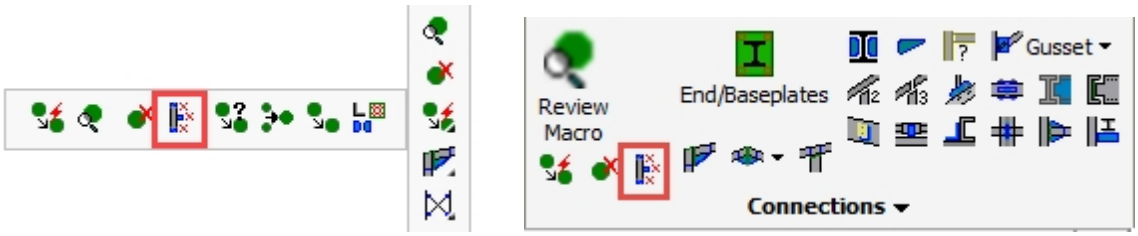
Command - **PrB_EraseMacro**



For more information, see Delete Macro

Erase Objects and dep. Macros

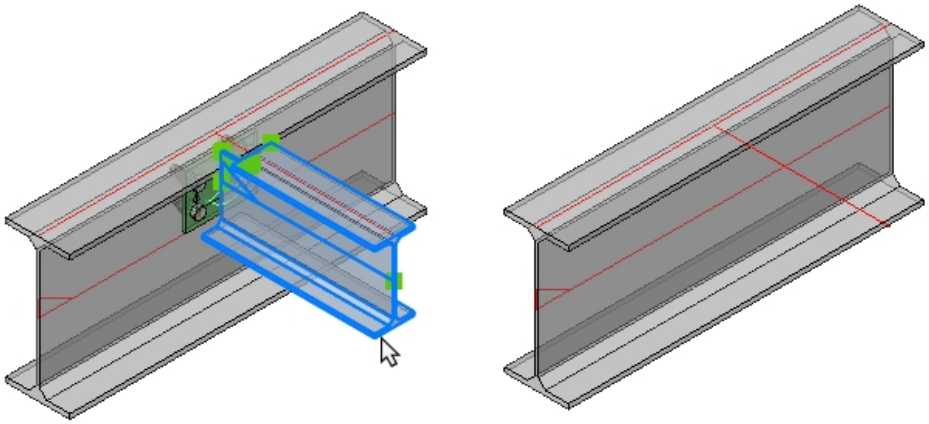
Command - **PrB_EraseObjectsAndDependentMacros**



This command will not only delete the objects that you select, but it will also clean up all the macros that are attached to the objects.

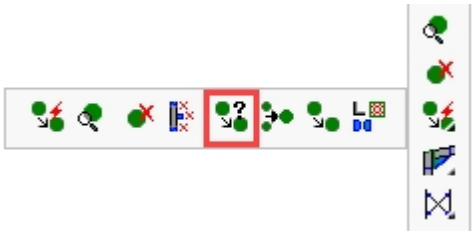
Indicate the object. Here, you may select one or multiple objects.

Press <**Enter**> and the objects and their dependent macros will be deleted



Smart Copy Settings

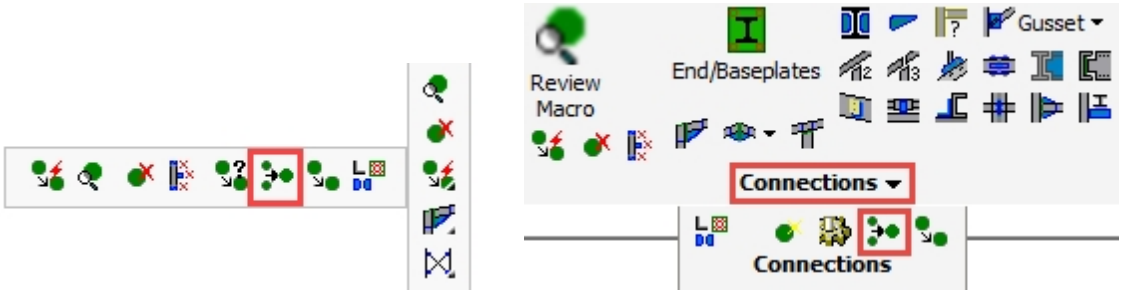
Command - **PrB_AutoApplySettings**



For more information, see Smart copy settings

Merge Macros

Command - **PrB_MergeMacros**



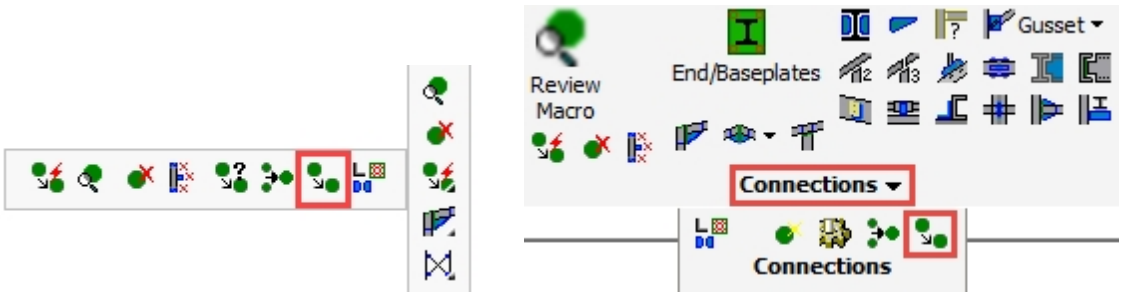
With this command you can **merge** two or more macros into one macro. There will be no loss of intelligence.

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

Select the first macro, which will be preserved - followed by the second macro - press <**Enter**> and the macros will be merged into one.

Copy a Macro manually

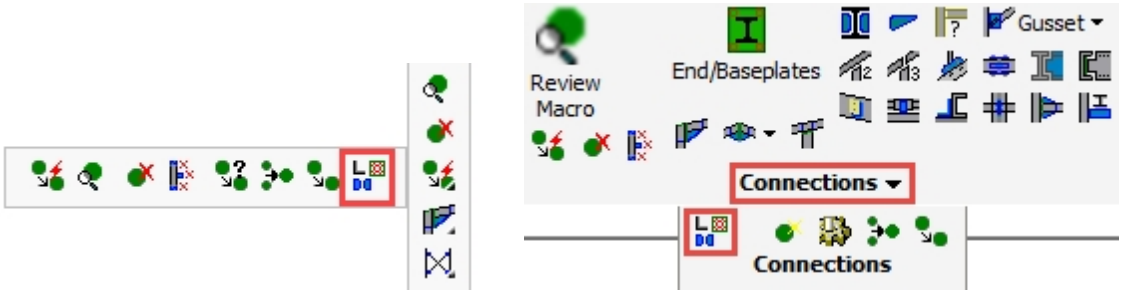
Command - **PrB_ApplyMacro**



For more information, see the topic Manually copy a Macro

Connection Components Library

Command -



This tool allows us to store and retrieve component macros.

A typical macro is based on one or more base profiles.
The macros found in this command could have a plate or line as base element.

The purpose of this command is to store and later retrieve partial macros as components.
These components would contain objects that are frequently recurring in connections.
Saving them with this command allows you to retrieve the components in any other drawing for reuse.

At the time of writing, this command only works in AutoCAD.

Connections

- Standards for Connections
- Applying Connections
- The Review macro dialog
- Haunch
- Apex
- Apex with Column
- Column under Beam
- Beam Offset to Column
- Beam to Column
- Adjacent Beams to Column
- Purlin on top of Beam

- Purlin Brackets
- Purlin to Rafter - 3 base
- Beam to Beam
- Beam / Column splice
- Connect Railing Post
- Connect Railing Post - 2 Base
- Connect Railing Post - 3 Base
- Welded Connections
- New Connection from a Group
- Web Stiffeners
- Flange Stiffeners
- Connection Components Library
- End / Base Plates

Standards for Connections

The connection standards is a collection of rules that can automate filling the values of dimensions and components in connections.

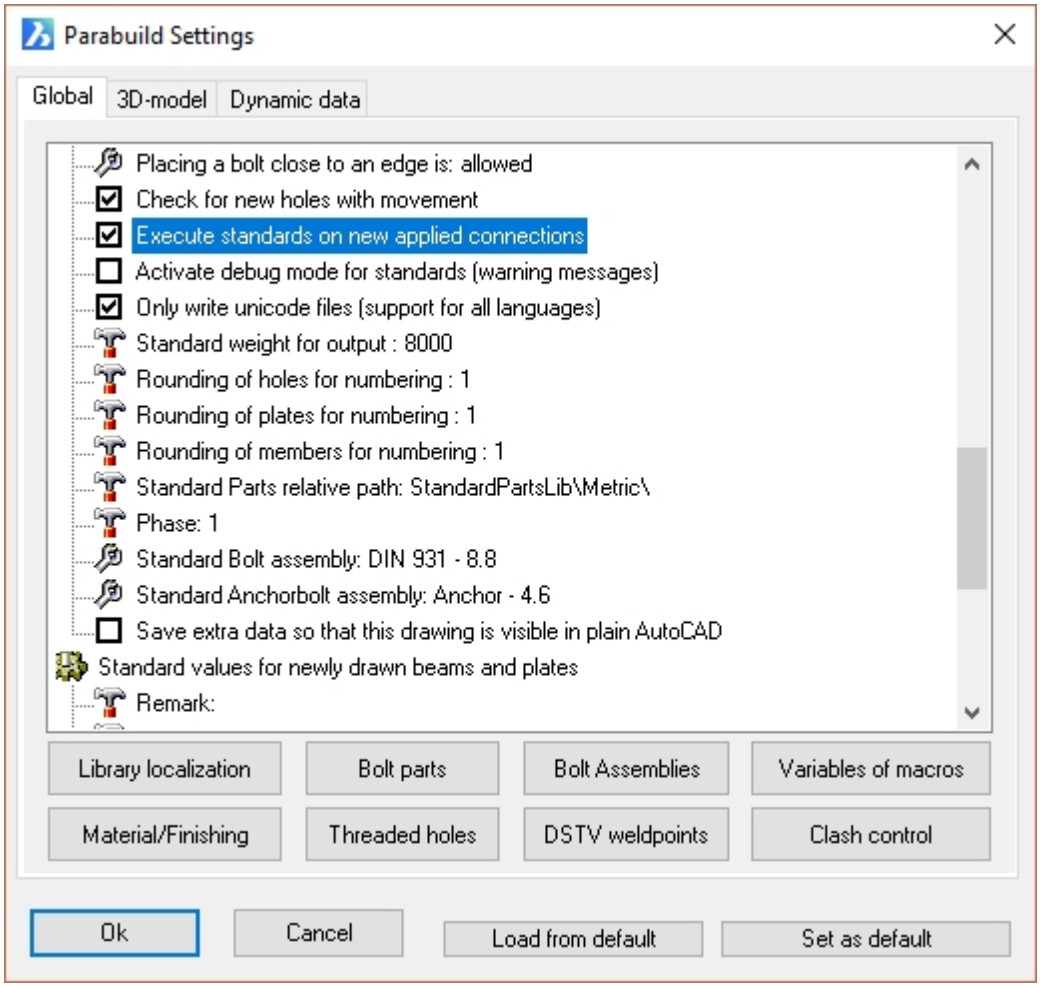
Below are some examples of things that can be automated:

- 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 Global Settings dialog box using the checkbox: **Execute standards on new applied connections**:



Applying Connections

The connections from the library are sub-divided in groups, so that selecting the correct connection from the extensive library is made simpler and more intuitive. There are also a geometric filter that reduce the number of connections to choose from.

Some examples of groups are Haunch connections, Apex connections, End plates, Base 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 determines that all connections of the same group must have the same amount of base profiles.

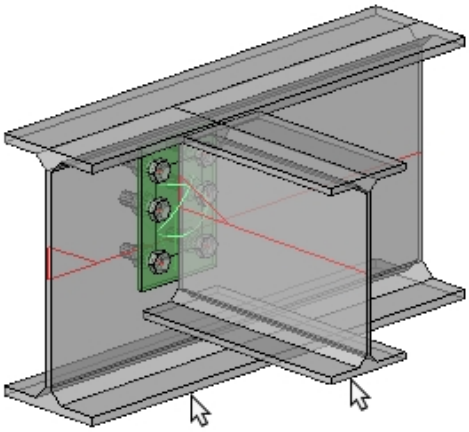
The order in which you select the profiles is important!

Follow the command line prompts, when selecting the profiles. Generally, you will be prompted to select the supporting member first, followed by the connecting member. In cases where this is not clear, we can follow the rule of first selecting the profile that should not be modified in length, and

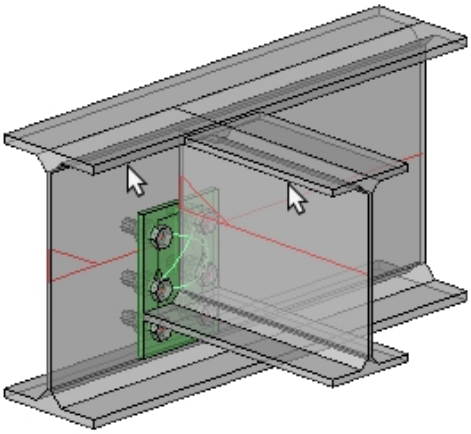
afterwards the profile that will be lengthened.
If you select in the wrong order, the orientation of the resulting connection will be wrong.

Its possible to inadvertently mirror connections by not being consistent when selecting the sections using the mouse pointer. To avoid surprises you should always select the underside of the members, thus ensuring the connection will be drawn correct.

Selecting the tops of members may mirror the connection (upside-down) as illustrated in the example below.

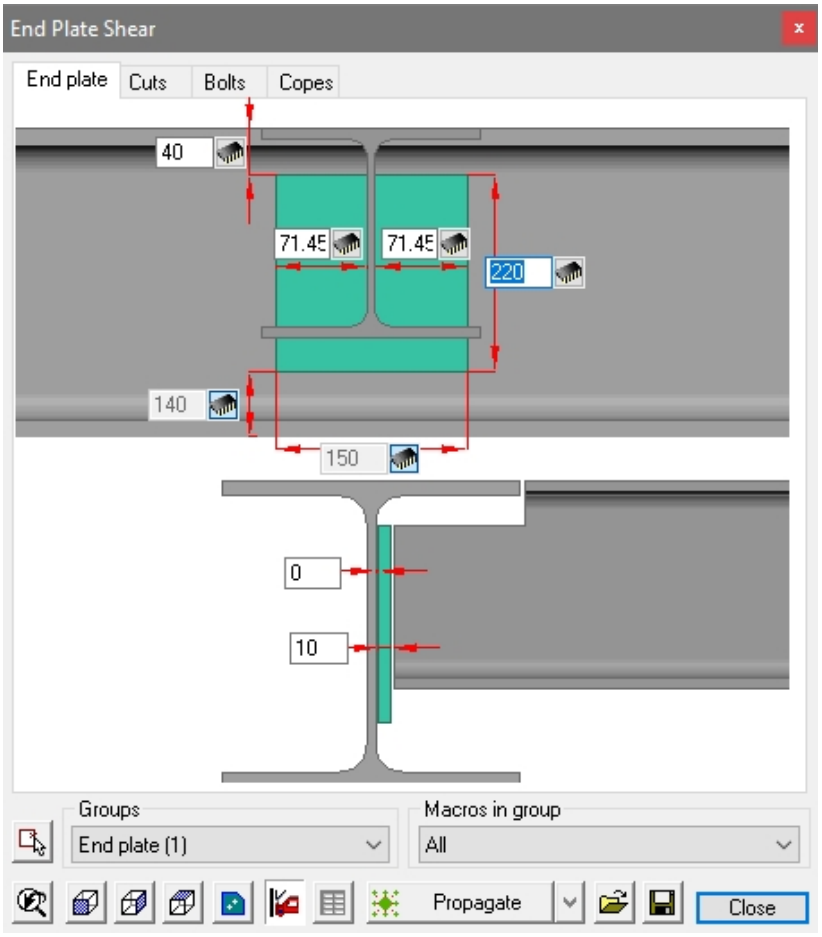


Result after selecting the underside of the members

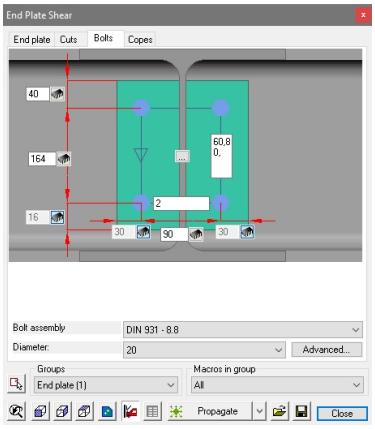


Result after selecting the tops of the members

This 'mirroring' can be a nuisance because its not evident when editing the connection macro dialog - the edited dialog (Below) is identical for both the options shown above - but you can clearly see the difference with the end result - these differences are based solely on the manner in which the members were selected.



The edited End Plate tab



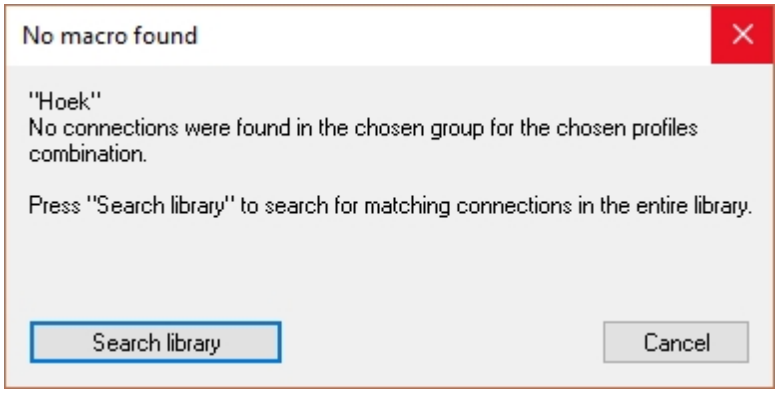
The edited Bolts tab

Geometric Filters

The base profiles selected are checked against those in the library to determine which connections are applicable. This is a filter that happens automatically.

The connections that are not compatible with the chosen profiles will not be visible.

For example: a typical haunch connection would imply that the beam connects to the column flange, should the orientation be such that the beam is connecting to the column web, Parabuild will check the library, and if no suitable connection is available the following window will be displayed offering Parabuild to check the entire library for a suitable alternative.



Inserting a macro from the library

After you have chosen a connection and clicked on **Ok** the connection is inserted and a dialog box will be opened that allows you to adapt the dimensions of the connection. *(The chapter Macro Edit Dialog box discusses this dialog box)*

Deleting unwanted parts of a connection

You are free to remove a part of the connection (components such as plates or profiles). The macro will not redraw the elements that you have removed. All other elements of the macro will usually 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 other parts of the connection.

The only parts that we should avoid removing is the base profiles themselves. Once 1 or more base profiles have been removed, Parabuild does not have enough information to calculate the location of the objects in the connection.

As a consequence, the macro sphere may become red and the objects in the macro might move to unwanted locations.

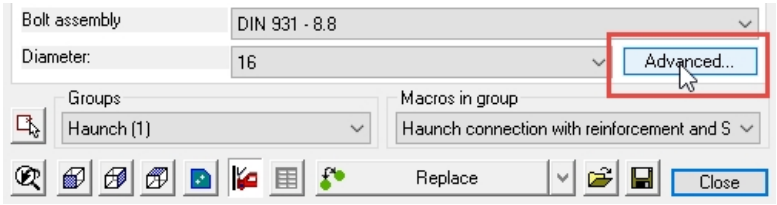
In this case it is advised to erase the macro.

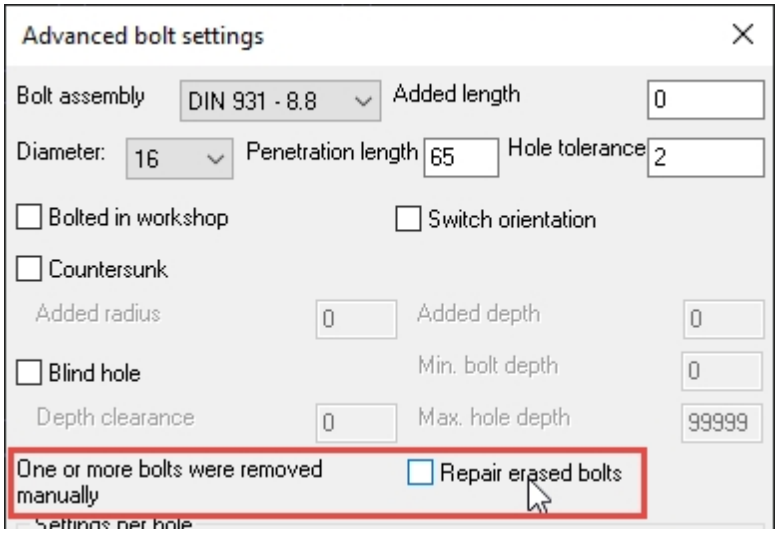
Deleting bolts in a macro

We can delete bolts that were drawn by a macro bolt pattern.

When a bolt has been manually deleted, Parabuild will not try to restore the erased bolt.

If you do want to restore the deleted bolt(s), then use the checkbox in the *Advanced bolt* dialog box :





Deleting the macro sphere

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 part sizes can no longer be adapted. The connections elements become "Static parts" as if they were drawn manually.

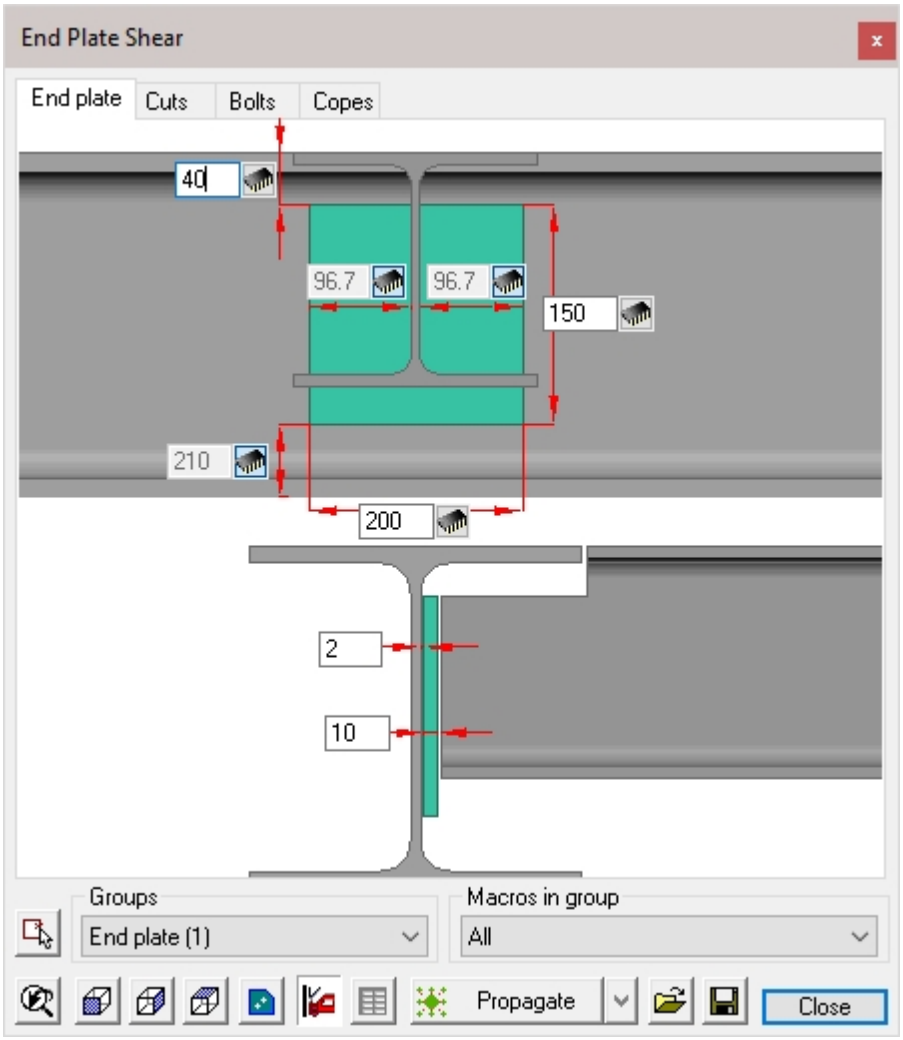
Macro Edit Dialog

The Macro edit dialog is available for all connections and will appear whenever a new connection is drawn.

Alternatively, the edit dialog may also be accessed by the Review macro command.

Each dialog is divided into 5 sections:

1. **The Title bar** - which displays the current macro name (or the number of macros if more than 1 is being edited)
2. **The Tabs bar** - which indicates the number of modules making up the macro. Selecting one of the tabs will open the appropriate edit dialog window. (Note! that the tabs will vary according to the selected connection type)
3. **The Dialog edit window** - where the dimensions may be edited. This is explained in detail later in this chapter
4. **Bolts** - (**Under the Bolts tab**) where you may select the bolt assembly and diameter from the drop-down menus
5. **The functions bar** - where you may select, view, copy, and adjust macro settings. This will also be explained in detail later in this chapter.



The Dialog Edit Window

There are 4 different types of dimension controls:

1. **Switchable** - This dimension type will be found where a dimensions is part of a string. Activating the switch will either enable or disable the dimension making it editable or responsive. A responsive dimension will automatically update when other dimensions in the string are changed.

Active Responsive

2. **Active** - an active dimension may be edited at any time

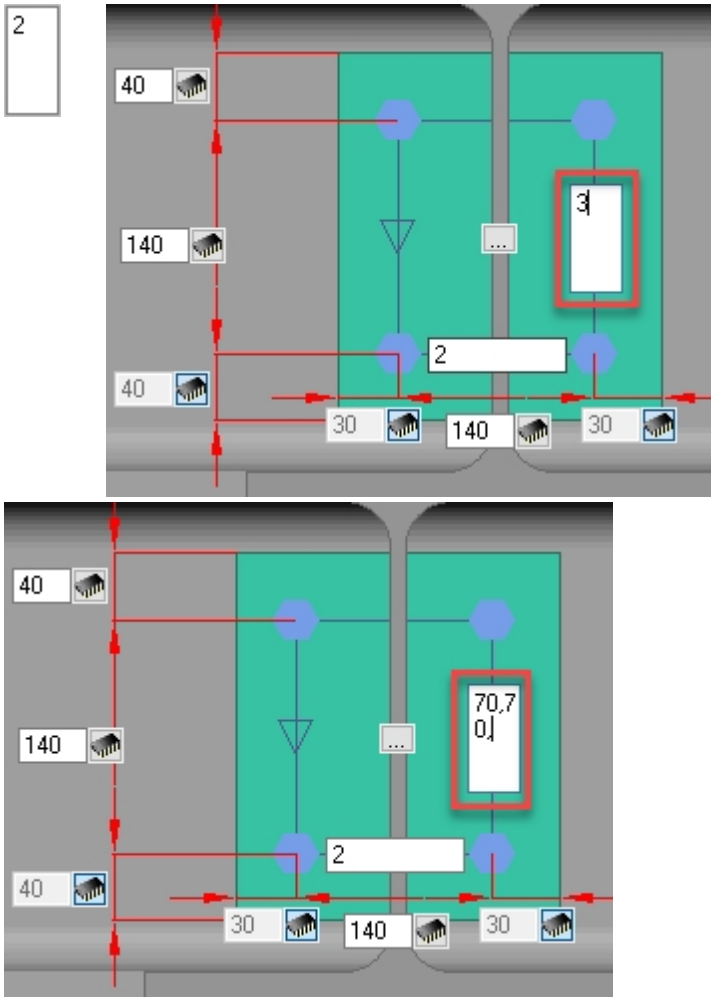
3. **Responsive** - A responsive dimension is shown 'greyed' and cannot be directly edited, it will automatically update when other dimensions in the macro are changed, or when a dependent object in the drawing has changed. These dimensions are to be considered 'Reference dimensions'.

4. **Bolt spacing** - These dimensions are found with bolt groups and can be identified by their elongated nature. There are two ways in which these dimensions are used. A single value will indicate the number of bolts to be equally spaced within the overall centers.

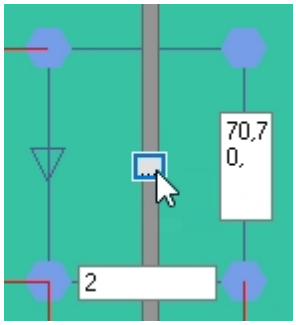
   In the first example, there will be 3 bolts, equally spaced inside the overall centers of 140 (2 x 70)

   In the second example, the hole centers are specified separated by a comma (70 x 70). Any number of bolt centers may be specified.

   This is particularly useful when the bolts are not equally spaced.

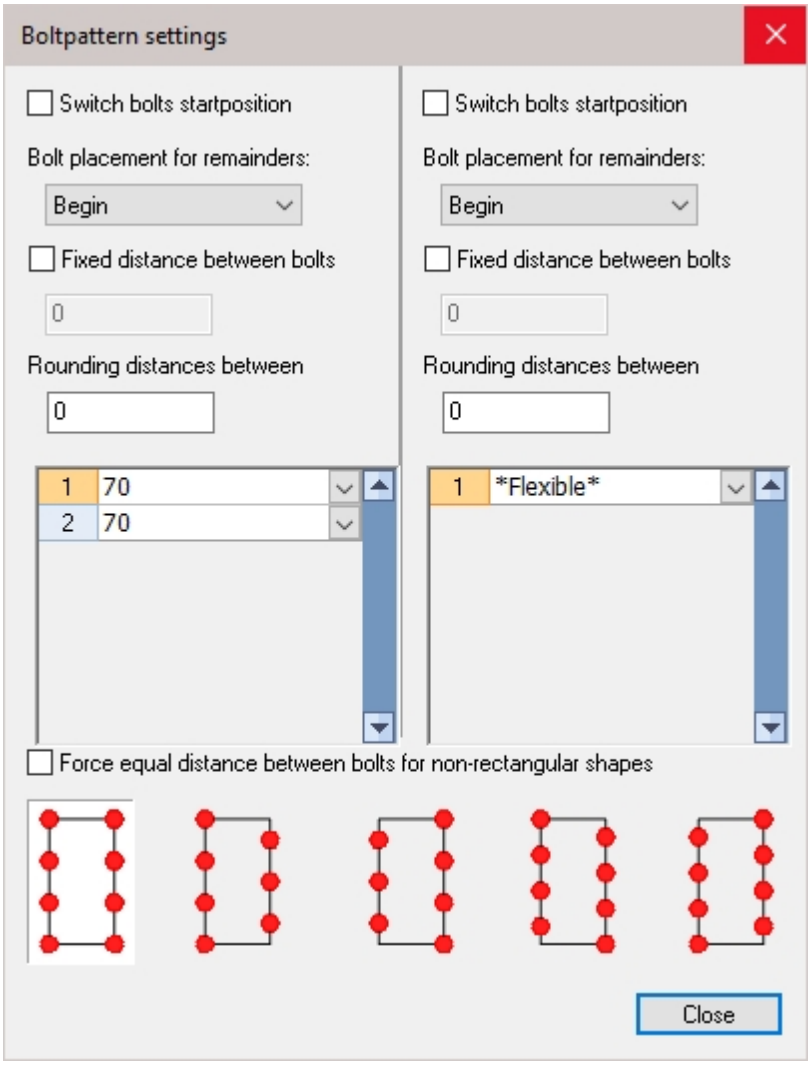


Pressing the button at the center of the section will open the **Bolt pattern settings** dialog, where you can perform the following actions:



Bolt Pattern Settings

This dialog comprises of 2 columns, the left hand column applies to bolts in the horizontal, while the right column applies to the vertical direction.



Switch bolts start position: Activating this check-box will reverse the order of the bolts, either from top to bottom, or side to side. This checkbox will only have an effect when the manual bolt spacing has been used.

Bolt placement for remainders: This option works in combination with the *Fixed distance between bolts* option because with it you stipulate at which side the remaining distance will be placed.

Fixed distance between bolts: The distance that you enter will always be respected.

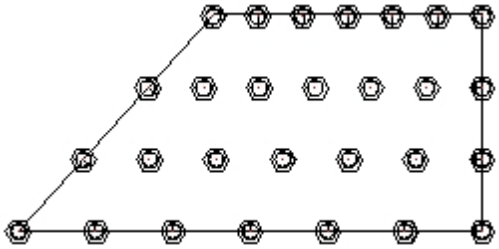
Rounding distances between: This is an important option to avoid undesirable distances. Imagine that we have a pattern with a length of 100 mm. You choose 3 bolts for this pattern. The bolts are divided equal and as a result we get a distance between the bolts of 33.33 mm. Later that distance (rounded 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

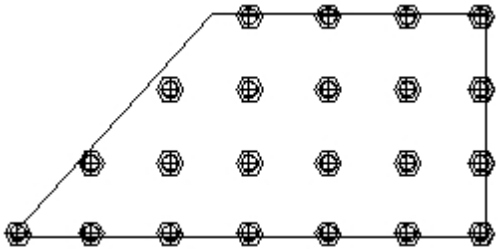
choose.

Force equal distances between bolts for non-rectangular shapes:

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

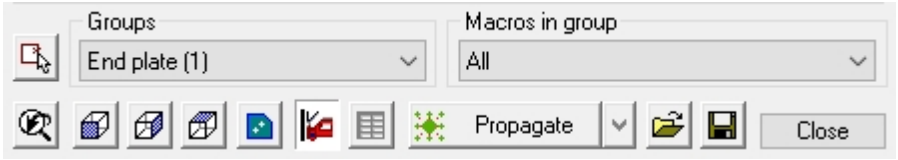


Example with force equal distances disabled



Example with force equal distances enabled

The Functions Bar



Select other macros  Activating this command will prompt you **Select entities**, enabling you to modify a different selection set of macros.

Groups - The drop-down menu will list all the selected macros, the number in the (parenthesis) will indicate the number of selected macros in the group. Selecting a macro will open the macro edit dialog.
Any changes that you do in the dialog will be applied to all the macros within the group. If you wish to edit only one of the macros, use the **Select other macro** function below.

Macros in group - From the down menu 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.

View buttons

 Restore original 3D view of the selected macro

 Front view of the selected macro

 Side view of the selected macro

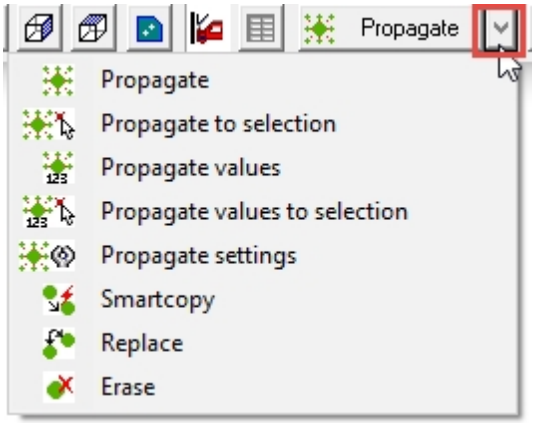
 Top view of the selected macro

 Front view of the current part. This button will give you a view of the parts of the current module tab : If the Bolts module is active, you will see a top view of the bolts.

 When you modify a dimension of the connection, 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 are editing many connections simultaneously and the clash control demands too much calculation time.

 This will execute the standards on the connection again. For more information about this see the chapter Standards for connections.

Next to the propagate button, there is a drop-down button that gives us access to a range of tools that will do something with the currently active macro(s) :



These are the purposes of all the tools :

 This tool will automatically copy the macro to all similar geometric locations in the drawing, potentially saving you a lot of time. For more information about this automation tool, see chapter Propagate.

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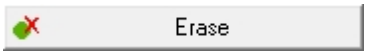
 This tool allows you to copy the macro to other profiles. For more information about this tool, see chapter SmartCopy.

This tool will replace the current macro with a new one from the library.

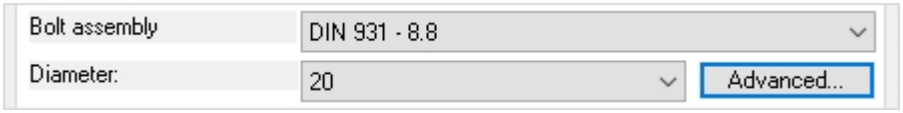


This tool will erase the macro and all of it's components.


With these tools you can save all values of the current selected connection(s) under a name and recall them at a later time.

Bolts

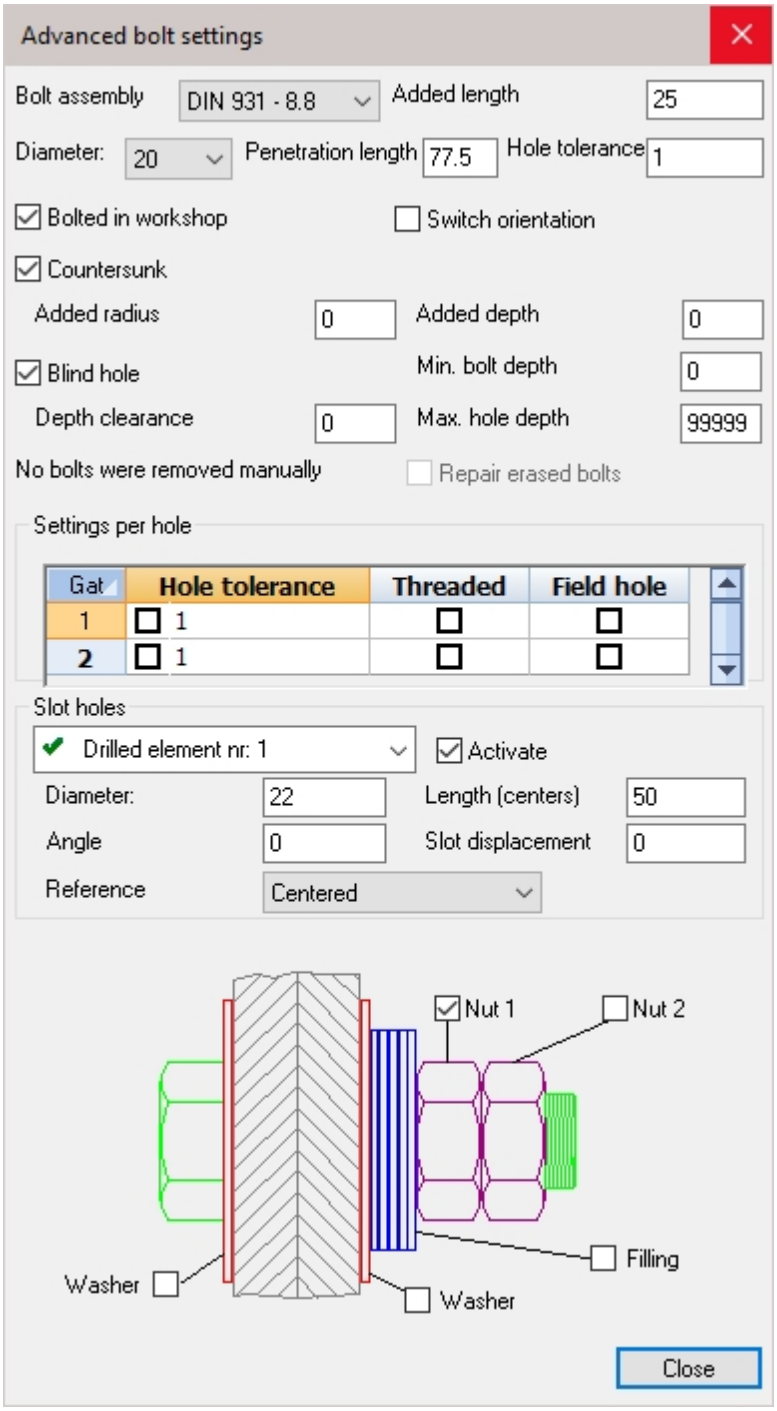
The bolt options are only available from the **Bolts** tab



Bolt assembly - Select from the drop-down menu - The bolt assemblies can be configured, see chapter Bolt Assemblies

Diameter - Select one of the available bolt diameters from the drop-down menu

Advanced - Will open the Advanced bolt settings dialog - the settings entered here will only apply to the selected macro(s)



Bolt assembly - Select a new bolt assembly from the drop-down menu

Added length - increase the distance between the nut and the plate, the bolt length will increase accordingly

Diameter - Select a new bolt diameter

Penetration length - This length is calculated automatically : it is the total thickness of all the parts that are bolted together

Hole tolerance - Changes the bolt hole diameter. The value = Bolt diameter + Hole Tolerance

Bolted in workshop - Enabling this will make the bolt a shop-bolt, which will then be shown in the Position and Assembly sheet

Switch orientation - Enabling this will switch the orientation of the bolt - reversing the orientation

of the nut and bolt-head

Countersunk - will change an ordinary bolt-hole to countersunk. This will enable the options to enter the **countersink diameter** and **depth**

Blind hole - This will change an ordinary bolt-hole to a blind hole. This will enable the options to enter the blind-hole depth.

Repair erased bolts - If a bolt was manually deleted, the prompt will read **One or more bolts were removed manually**. Checking this option will reintroduce the bolts.

Settings per hole - Here you can change the individual settings for the bolt-hole by selecting a new hole tolerance, making the hole threaded, and indicate whether its to be shop or site drilled.

Slot hole - Will change an ordinary hole into a slotted hole in one of the joined elements. Activating the checkbox will enable you to enter a **hole diameter**, **slot length**, the **angle**, and the **slot displacement**.

Finally - you can add or remove **nuts**, **washers**, and **filling** by checking or un-checking any of the checkboxes.

Propagate

This tool is accessible from within the Macro Edit dialog :

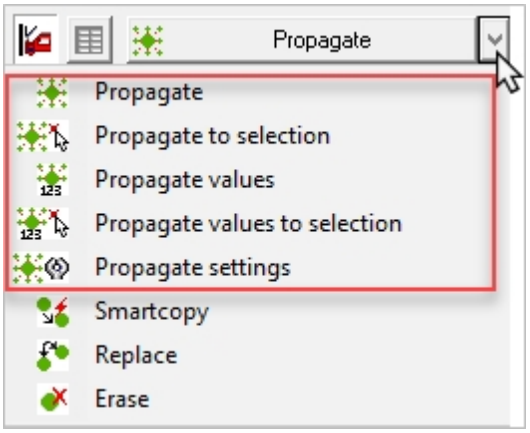
Propagate is much more than a 'copy' command.

This tool will automatically copy the macro to all similar geometric locations in the drawing, which can potentially save you a lot of time.

This will work for all connections including, haunches, apex, splices, base-plates, bracings etc.

After the connection has been 'Propagated' the Macro Edit dialog will appear. At this point the connection macro may be edited which will be reflected on all propagated macros. After this dialog is closed, the individual connection macros will be edited individually.

Propagate Options



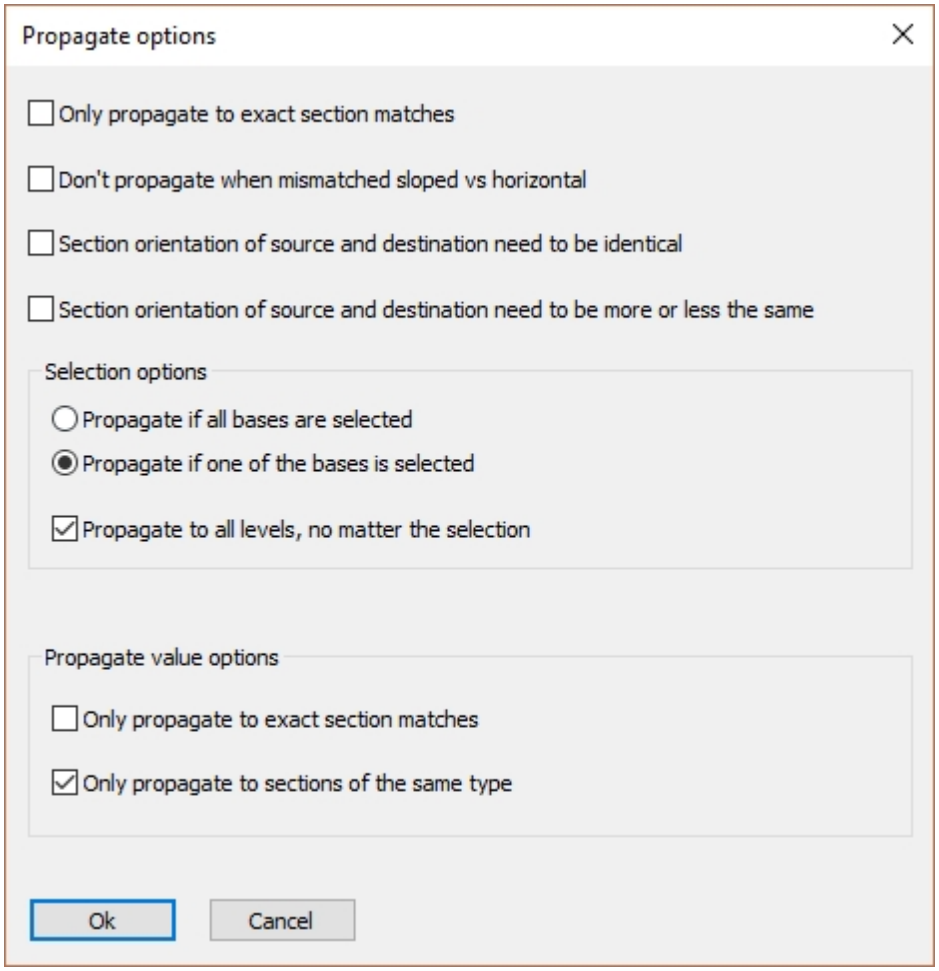
This will copy all macros to matching profiles in the entire drawing.

This works the same as the normal propagate, but it will only propagate to a selection of profiles, instead of the entire drawing.

This tool will not propagate any geometry. Instead it will only propagate the sizes of the macro that is currently being edited. The sizes will be copied to the macros of the same type, and with roughly the same geometry.

Works the same as the *Propagate values*, but it will only propagate to a selection of profiles, instead of the entire drawing.

This will open the **Propagate options** dialog**.** With these options, it is possible to influence the behavior of the propagate tools :



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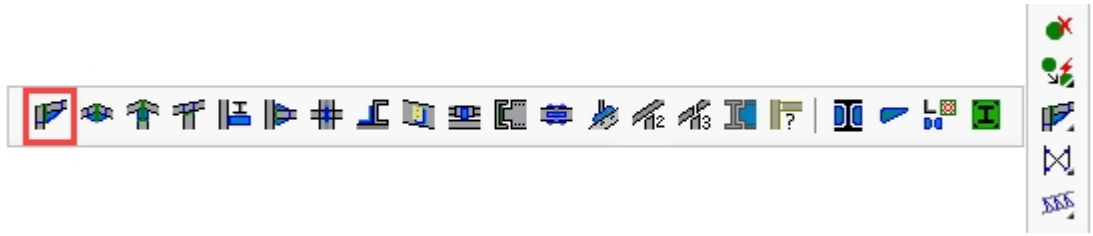
Haunch

Command -



The Haunch connection is a roof rafter connection to a column flange. The rafter may be set to the horizontal or at any angle specified by the design.

Selection order

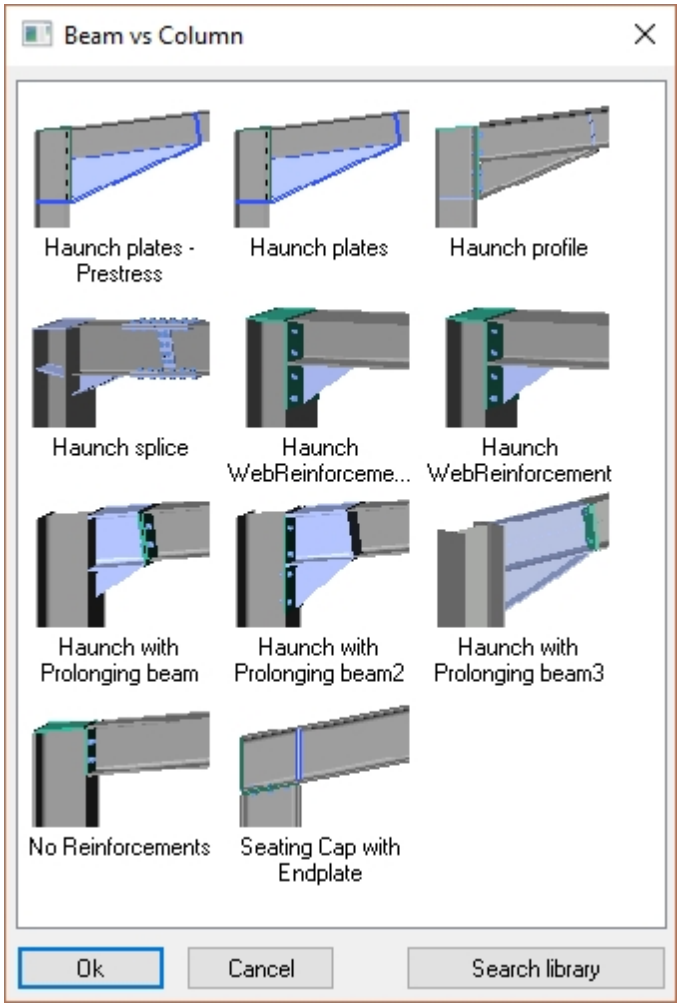
First select the column, and then the beam.

Note!

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

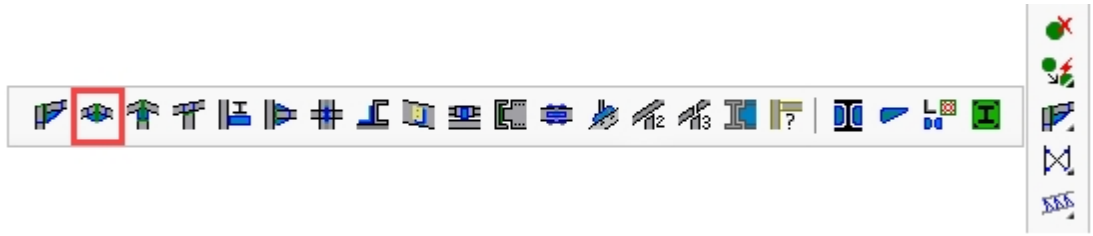
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used to filter out the incompatible connections in the selection dialog.

Therefore, the presented options may differ from those illustrated here.



Apex

Command -



The Apex connection is a roof rafter connection at the apex of the roof. The rafter may be set at any angle specified by the design.

Selection order

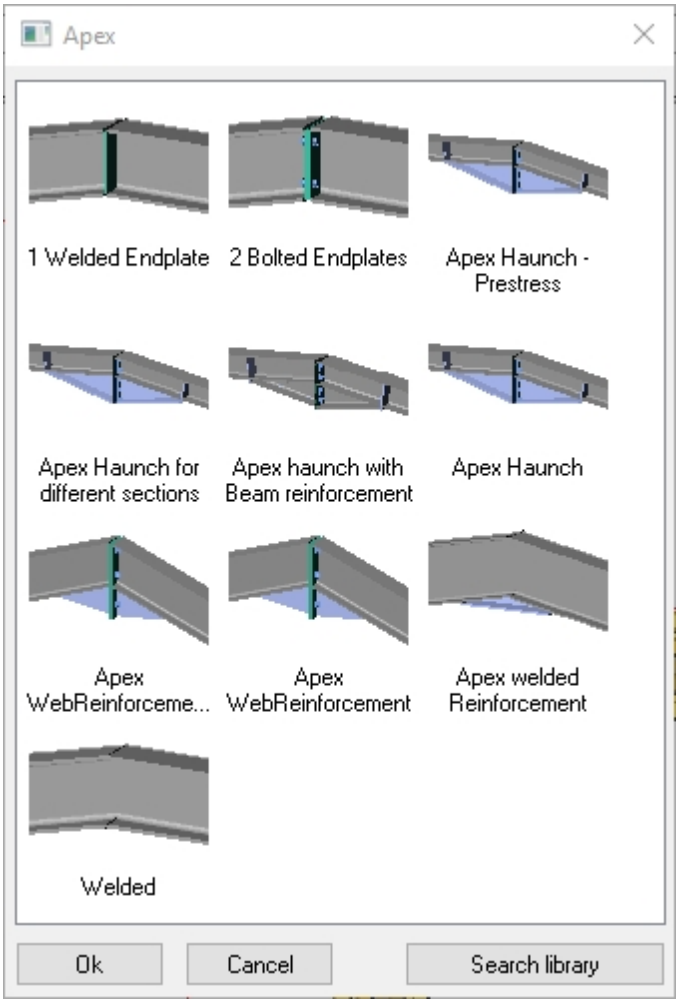
The order of the rafters does not matter.

Note!

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

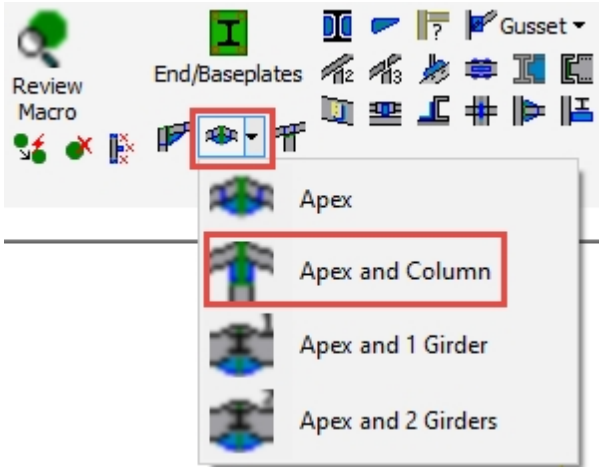
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Apex with Column

Command -



The Apex with column connection is a roof rafter connection at the apex of the roof. The rafter may be set at any angle specified by the design.

Selection order

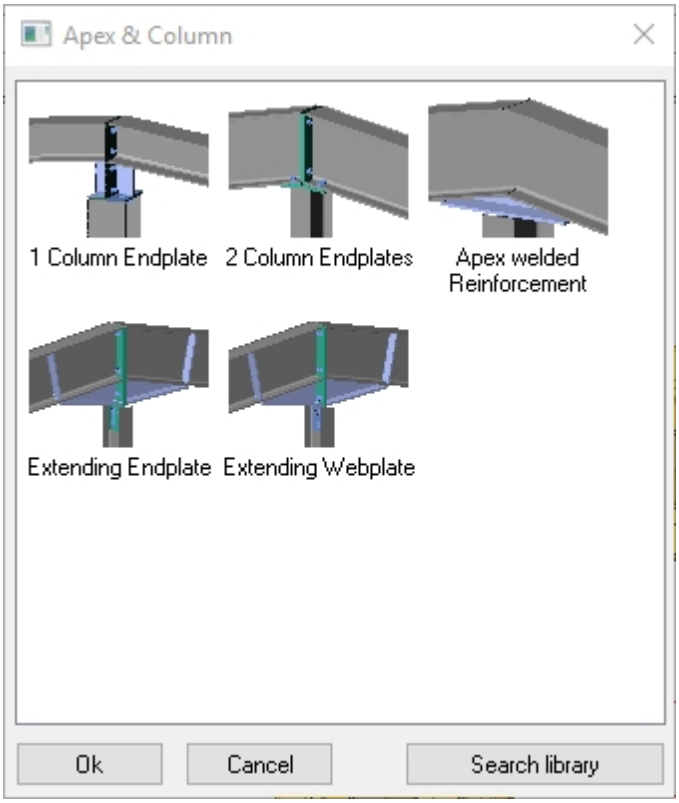
First select rafter 1, then rafter 2, and then the column.
The order of the rafters does not matter.

Note!

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

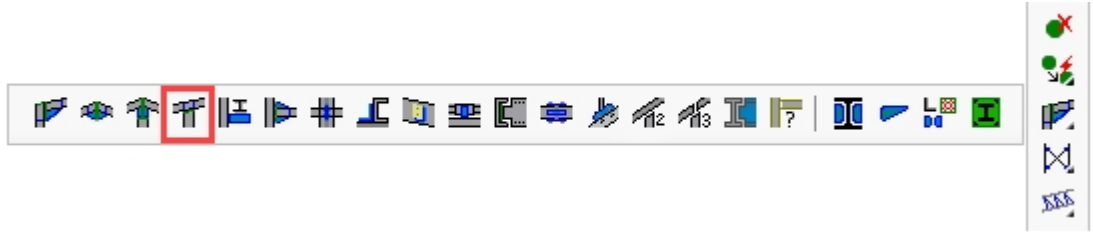
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

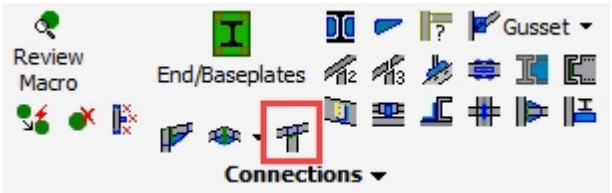
Therefore, the presented options may differ from those illustrated here.



Column under Beam

Command -





The Column under Beam connection is a post connection connecting a column to the underside of a beam. The beam may be horizontal or set at any angle specified by the design.

Selection order

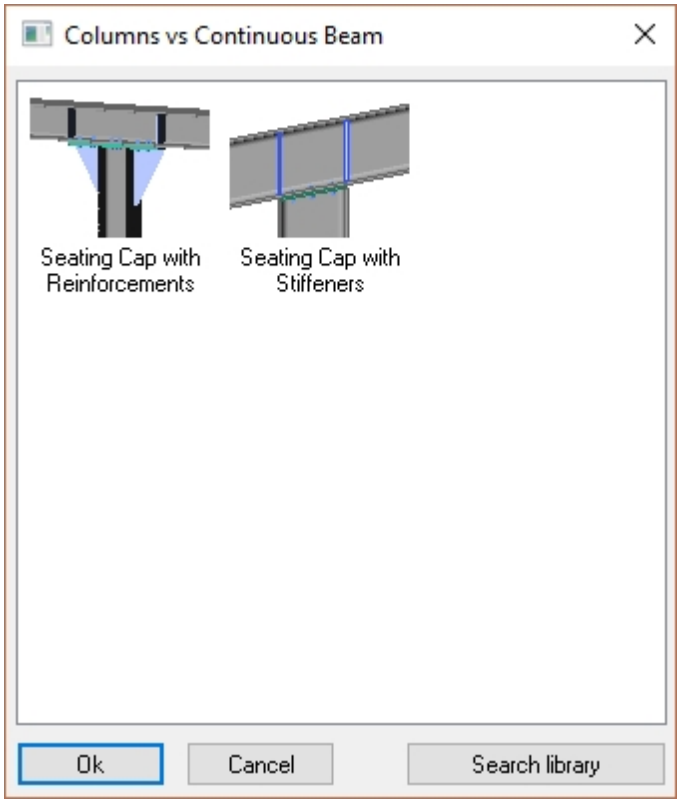
First select the beam and then the column.

Note!

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

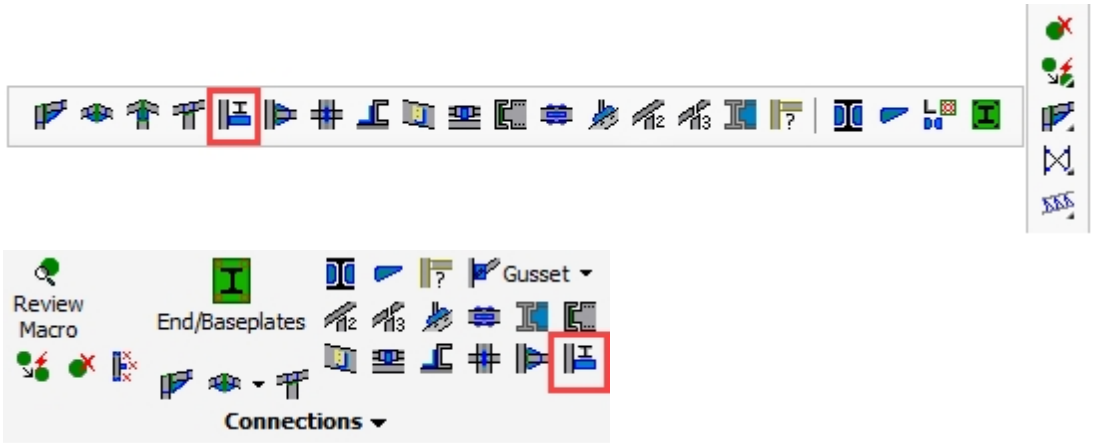
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Beam Offset to Column

Command -



The Beam Offset to Column is a beam connection connecting to the side of a column. This connection is usually applied where the column is supporting some form of runway or crane beam.

Selection order

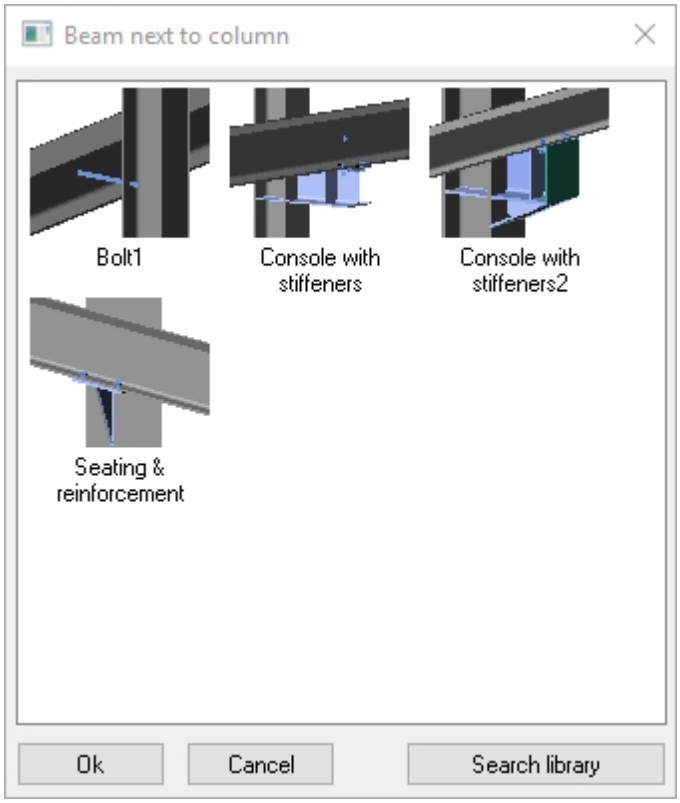
First select the column and then the beam.

Note

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Beam to Column

Command -

The Beam to Column is a beam connection connecting to the web or flange of a column. This connection may be either 'Moment-resisting' or 'Flexible' (flexible connections are also referred to as 'Shear' or 'Simple' connections).

As a general rule - a Moment connection is identified when the flanges of the incoming beam connect in some form to the supporting column - conversely, a flexible connection is identified when the beam is connected to the column by its web only.

Selection order

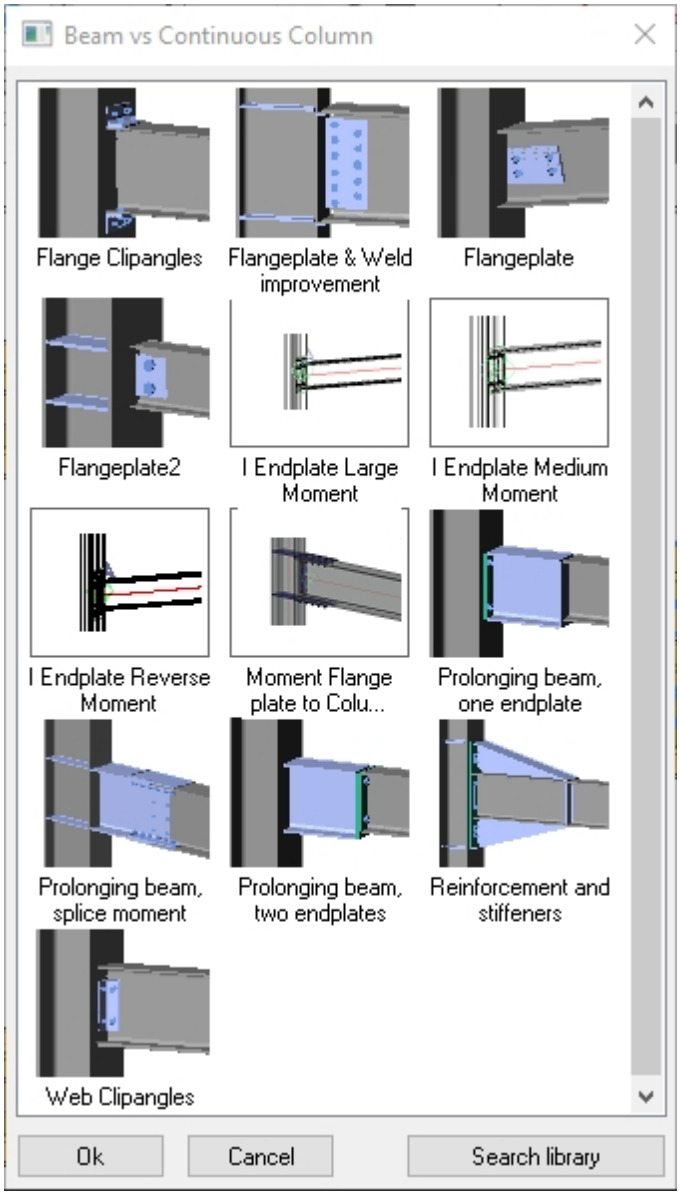
First select the column and then select the beam.

Note

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

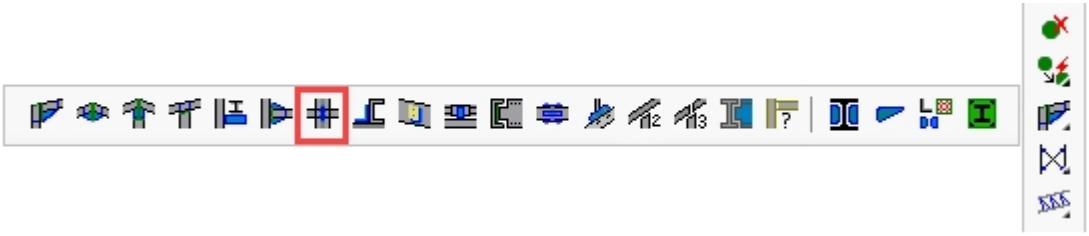
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Adjacent Beams to Column

Command -



This connection group can be seen as the merger of 2 Beam to Column connections : When 2 beams both connect to the web of the column, then the 2 end plates will have to share the same bolts.

Or it might be a merge of 2 Beam Offset To Column connections : When 2 beams are spliced on top of the seat connection.

Selection order

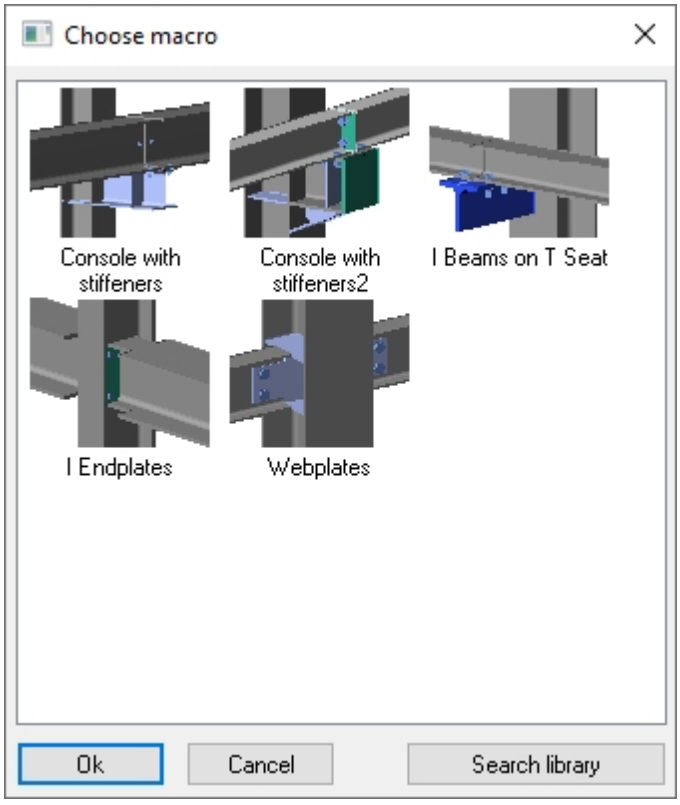
First select the column, then beam 1 and then beam 2.

Note!

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

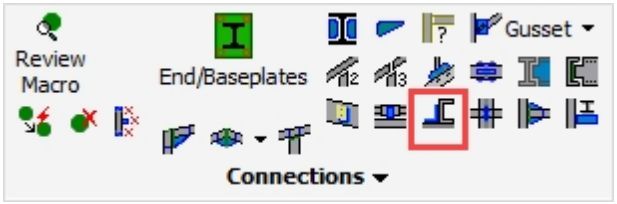
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Purlin on top of Beam

Command -



The **Purlin on top of Beam** connection connects the purlin to a roof beam or rafter.

Selection order

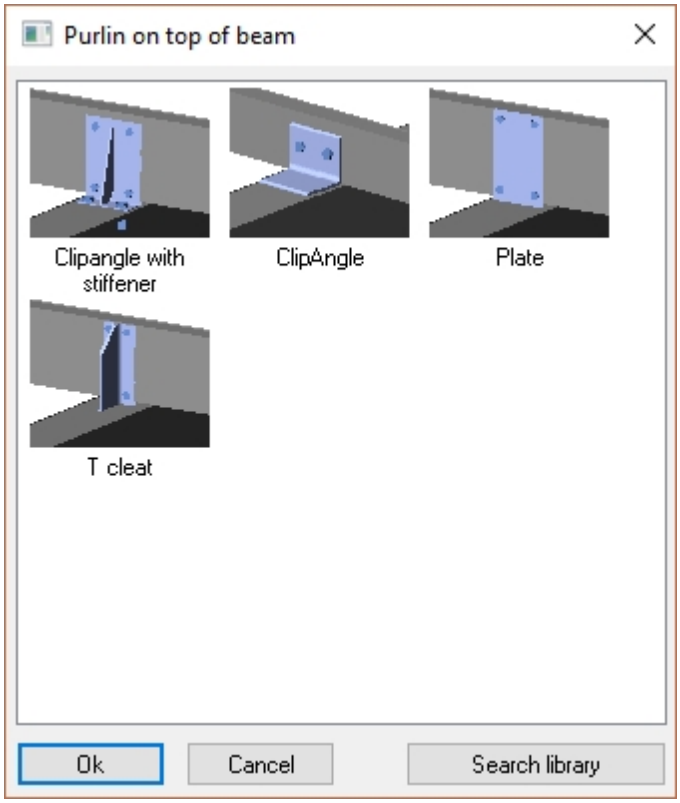
First select the beam, then select the purlin on the side where the bracket is to be placed.

Note

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

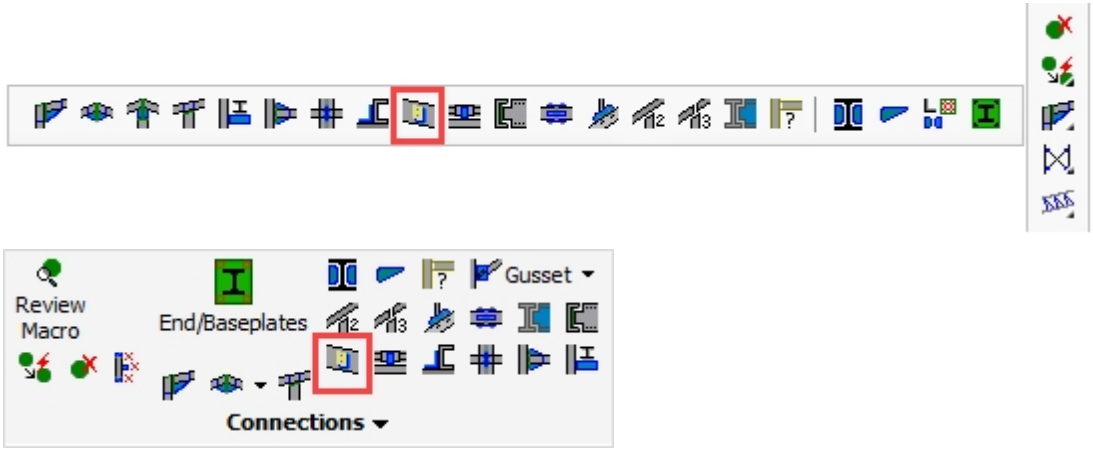
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Purlin Brackets

Command -



The **Purlin bracket** connects the purlin to a roof beam or rafter.

Selection order

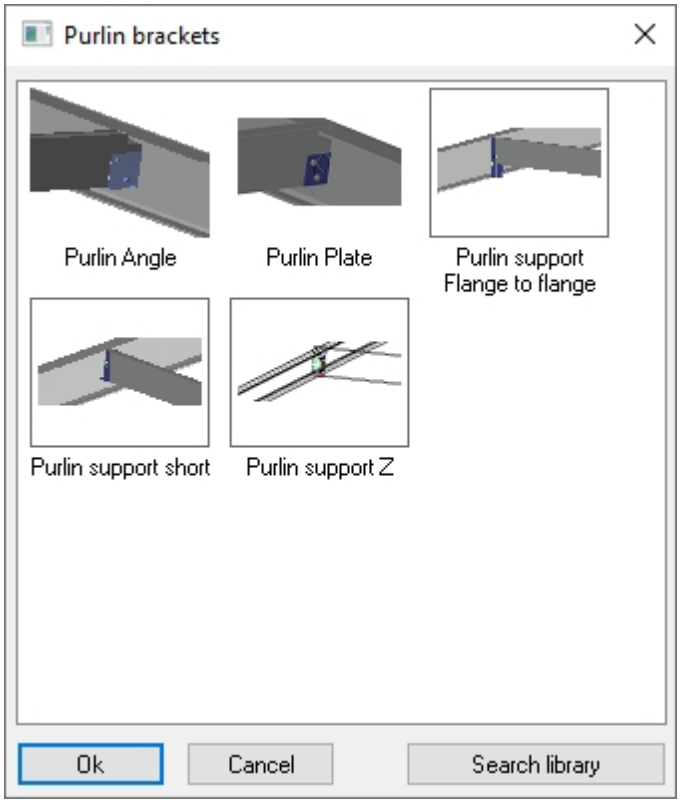
First select the beam, then select the purlin on the side where the bracket is to be placed.

Note

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

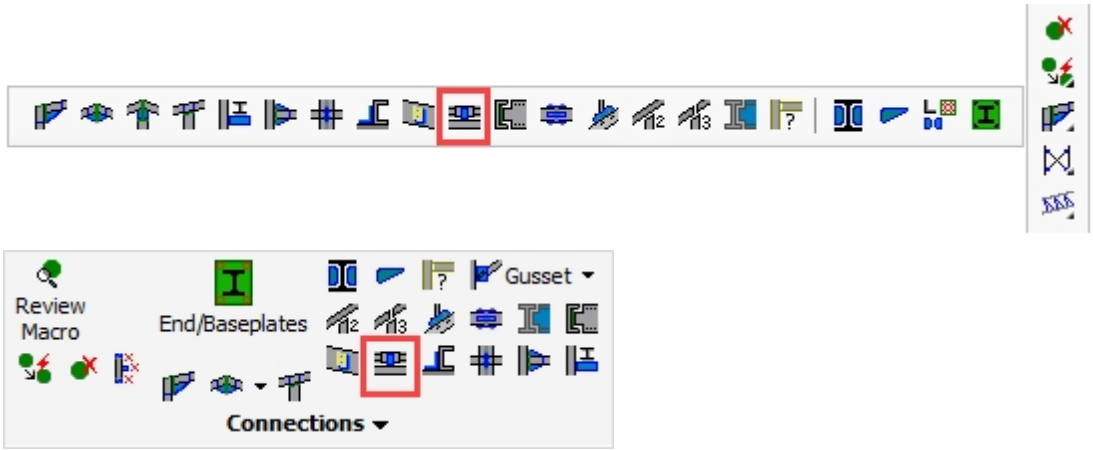
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Purlins on Beam / 3-bases

Command -



The **Purlin on Beam 3-base** connection joins 2 purlins at the center of the supporting beam or rafter.

Selection order

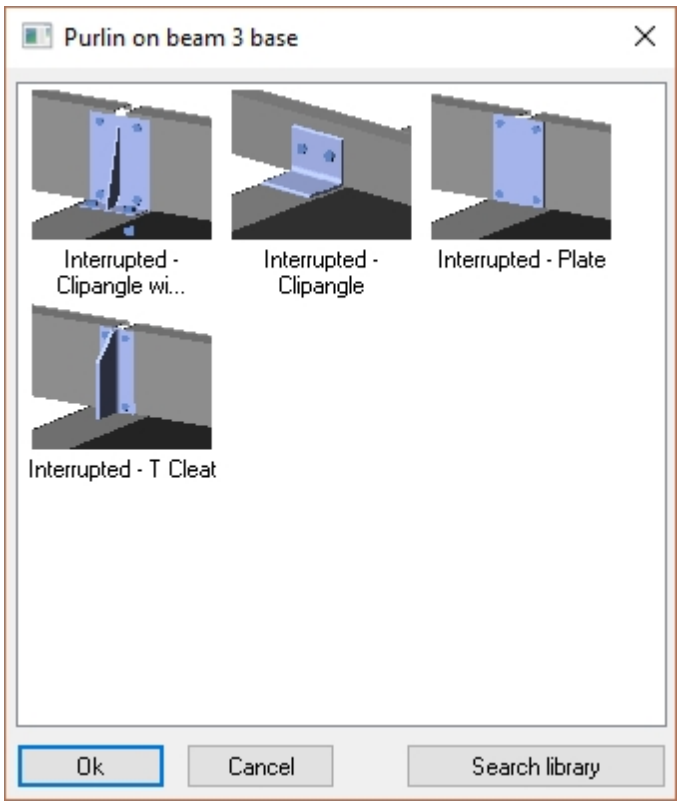
First select the beam, then select the purlin 1 and then purlin 2.

Note!

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Beam to Beam

Command -

The **Beam to Beam** connects a beam to the web of a supporting beam.

Selection order

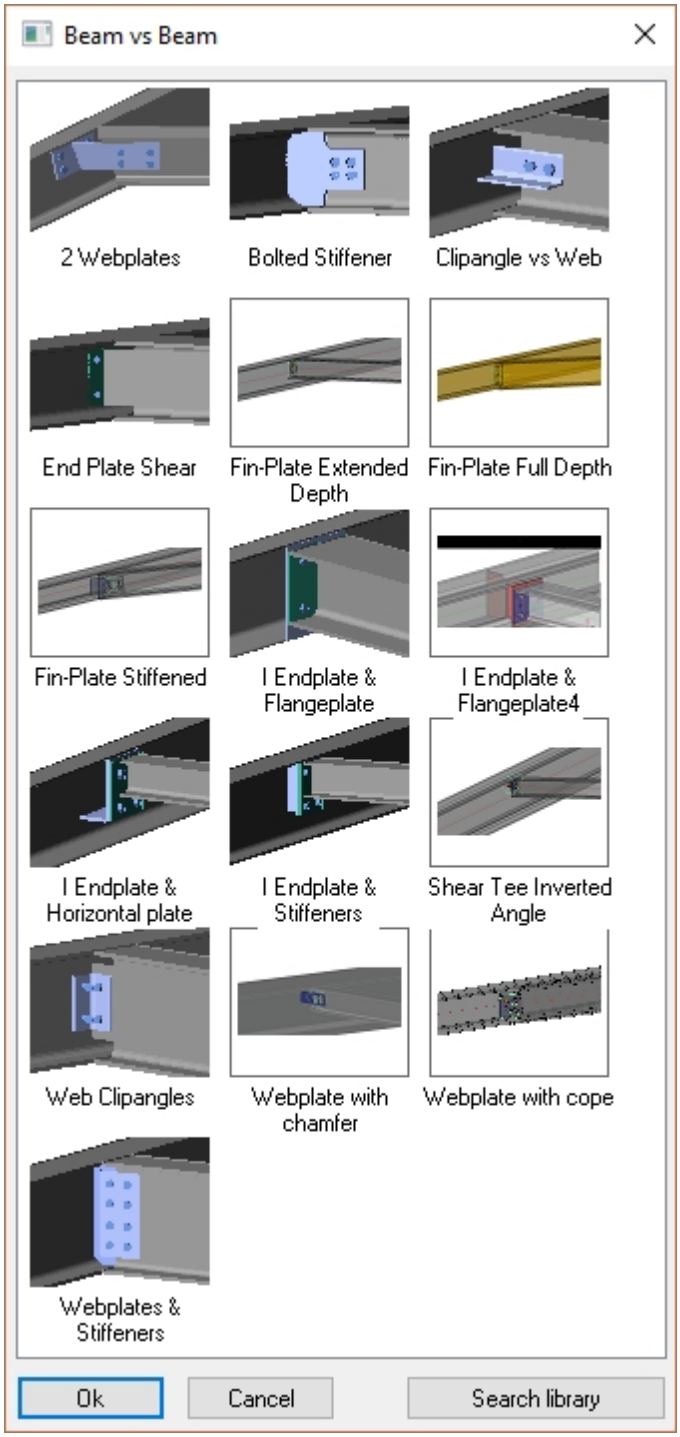
First select the supporting beam which is not to be lengthened, then select the incoming beam (which will be lengthened).

Note!

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

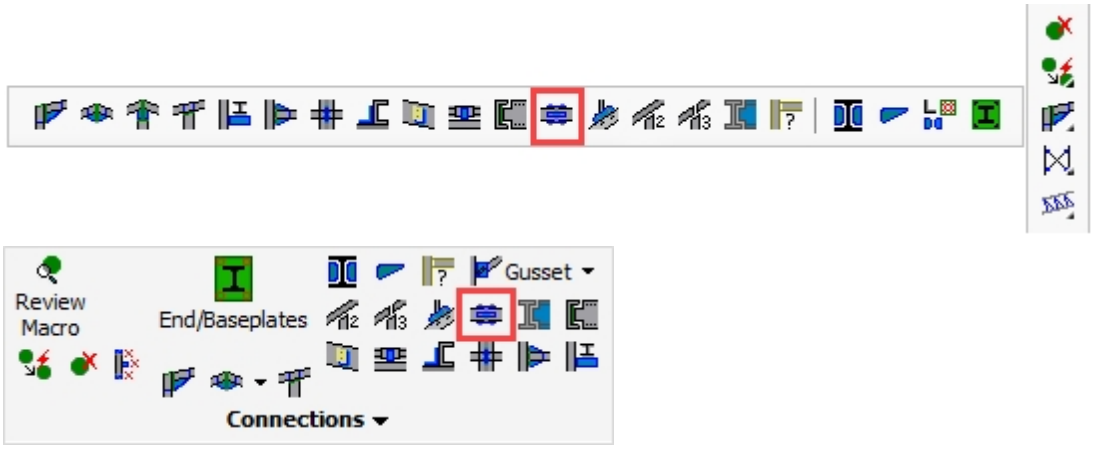
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Beam / Column Splice

Command -



The **Beam or column splice connection** requires that the profile be split at the point to be spliced.

Selection order

First select the base beam, then select the beam that will be lengthened.

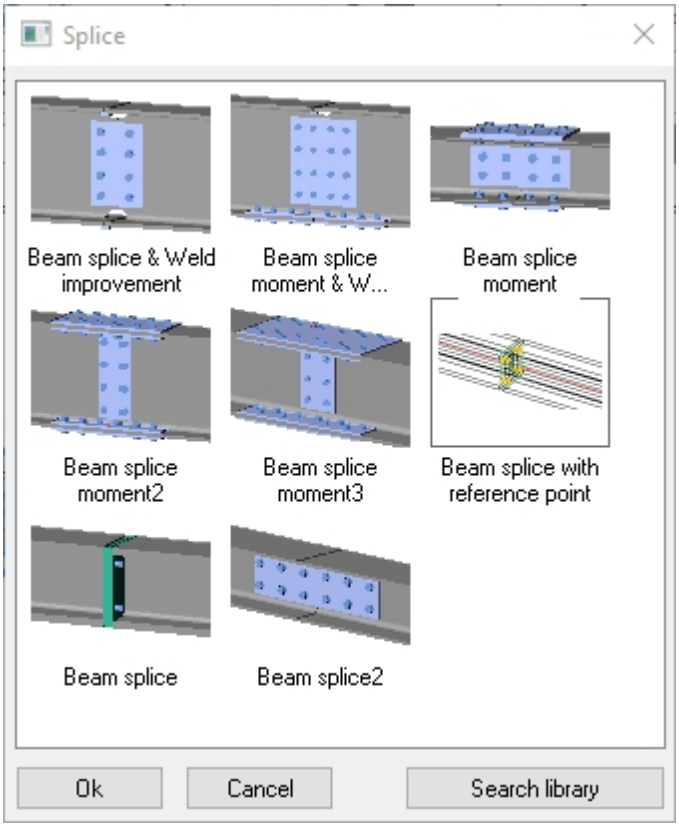
The ending of the first beam will determine the location of the splice and thus the location of the plates.

Note

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

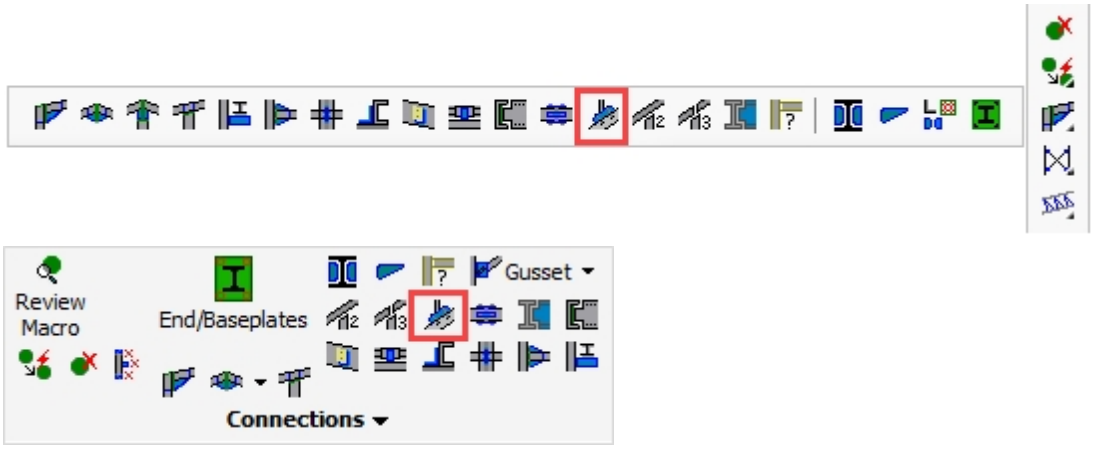
The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if a beam is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.



Connect Post

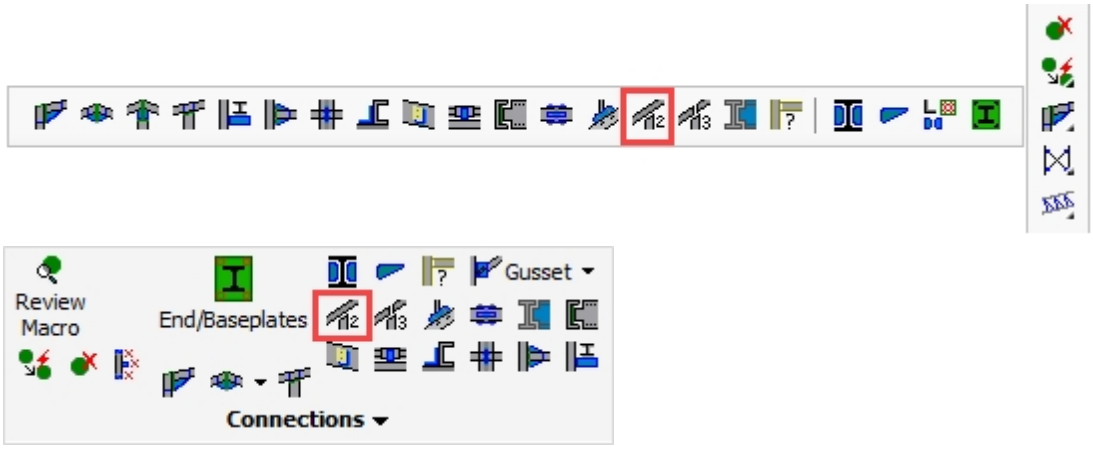
Command -



For more information about these connections, see the Connect Post topic in the stairs chapter.

Connect Post to Handrail - 2 Base

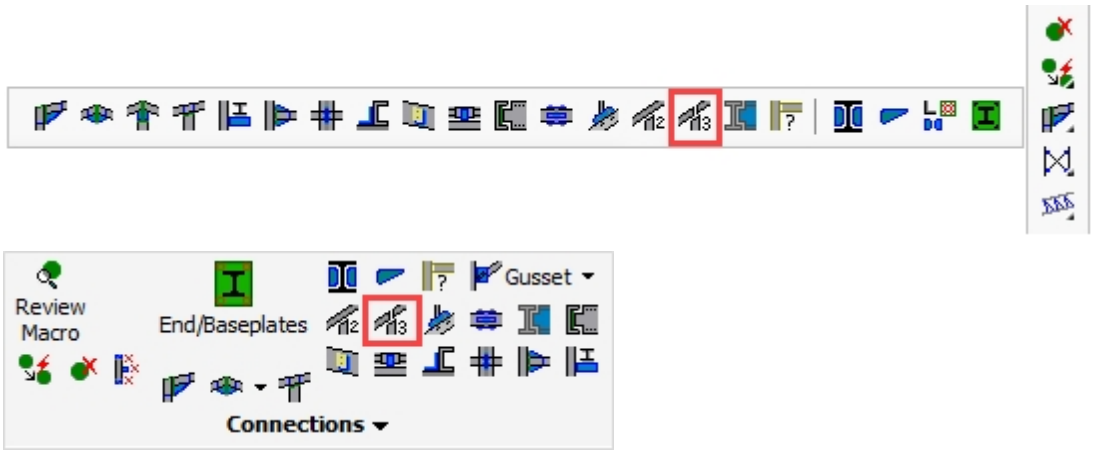
Command -



For more information about these connections, see the Connect Post to Handrail - 2 Base topic in the stairs chapter.

Connect Post to Handrail - 3 Base

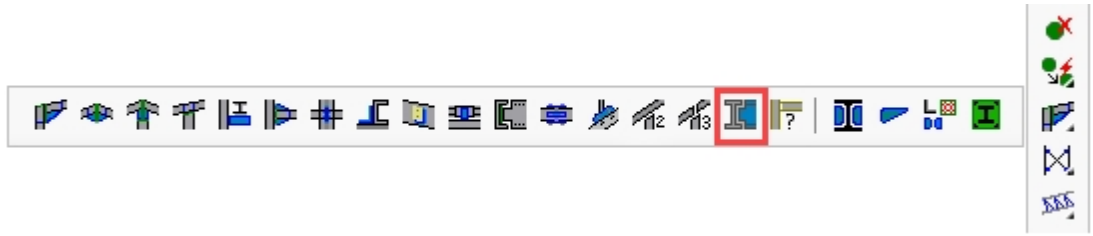
Command -

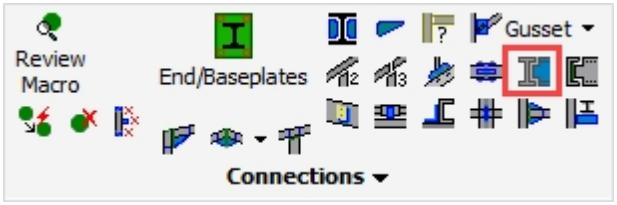


For more information about these connections, see the Connect Post to Handrail - 3 Base topic in the stairs chapter.

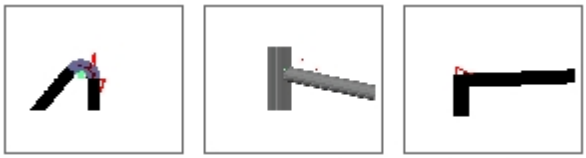
Weld Connections

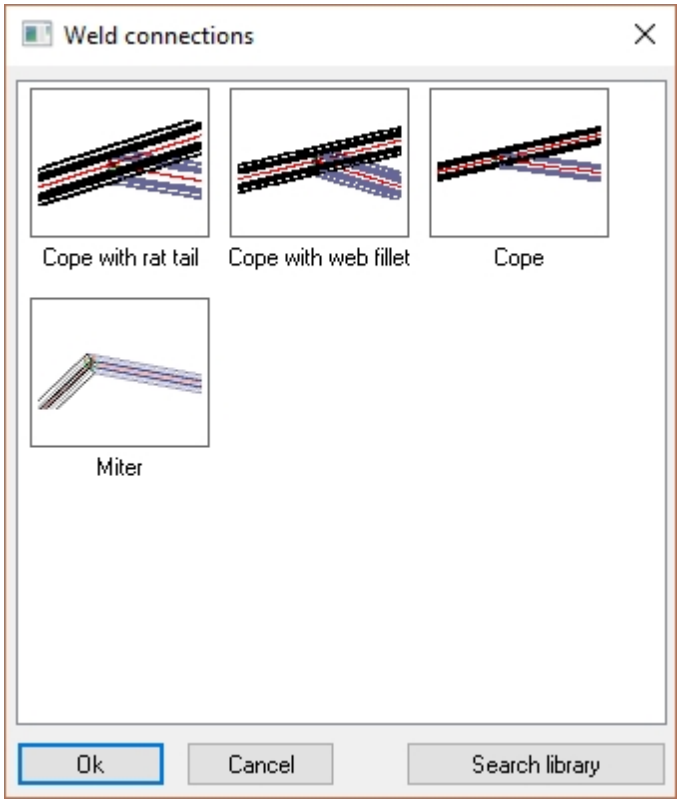
Command -





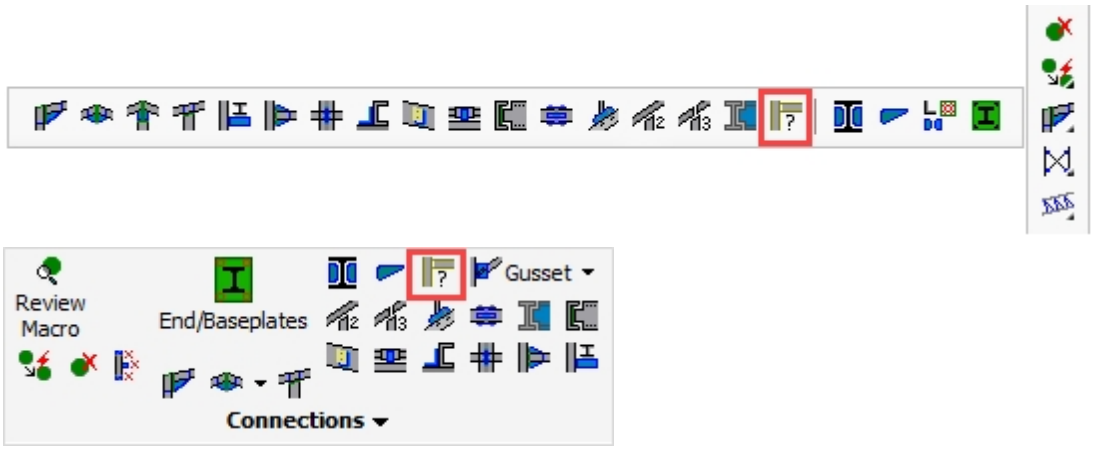
This command will draw a mix of weld connections such as rat tail copes and elbows :





New Connection from Group

Command -

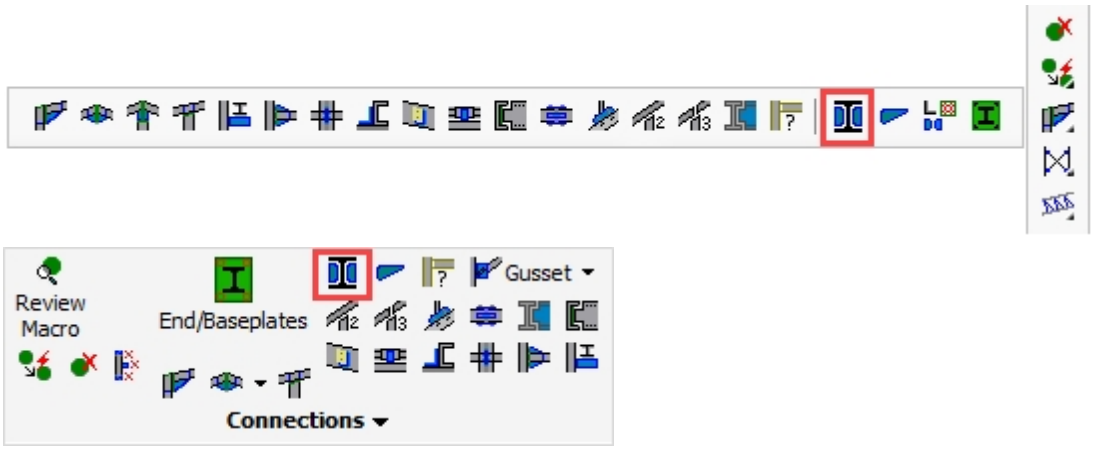


This command allows you to access all of the connection groups that are available in the library.

This can be useful in case someone has added a new group of connections to the library, but an icon for this group was not (yet) created.

Web Stiffeners

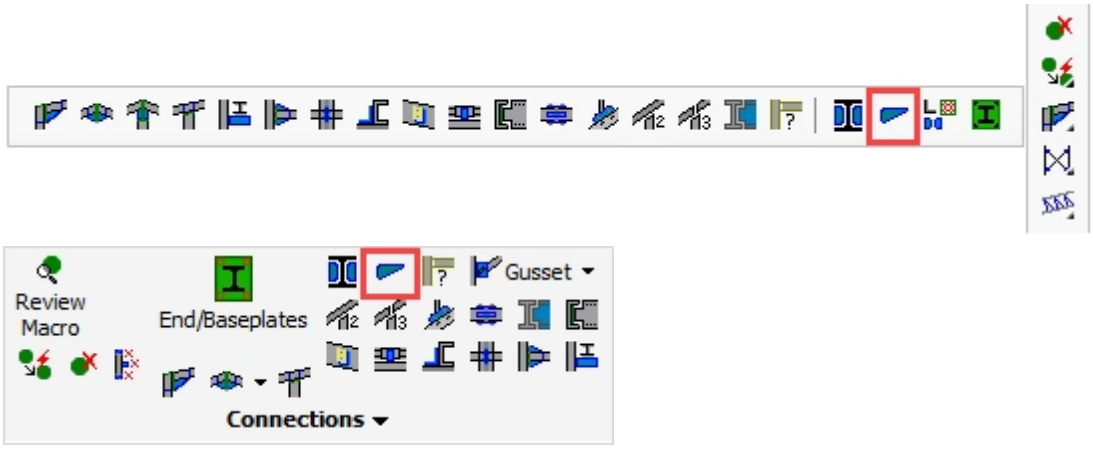
Command -



For more information about this, see the topic Web Stiffeners in the Tools chapter.

Flange Stiffeners

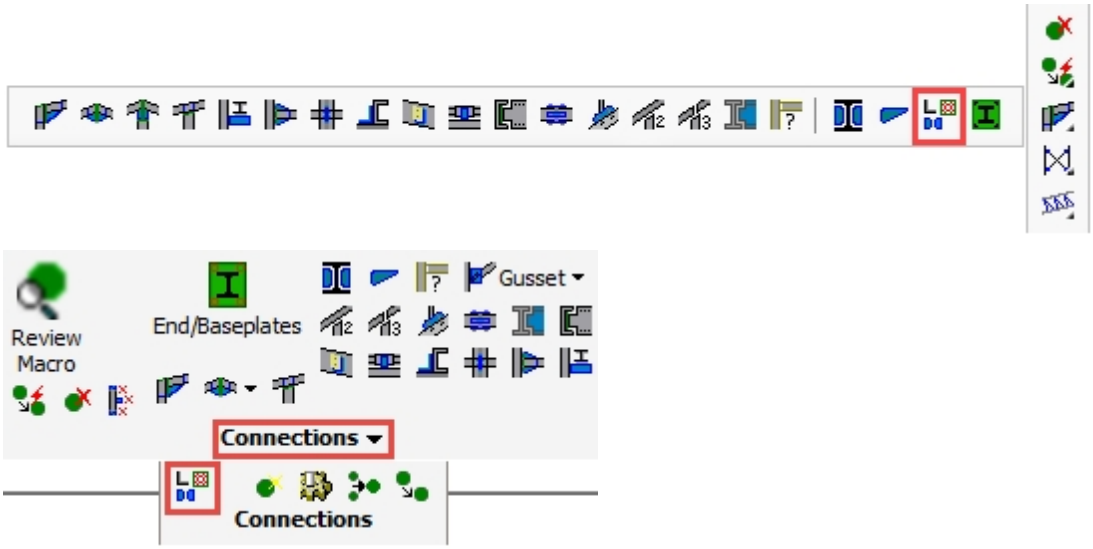
Command -



For more information about this, see the topic Flange Stiffeners in the Tools chapter.

Connection Components Library

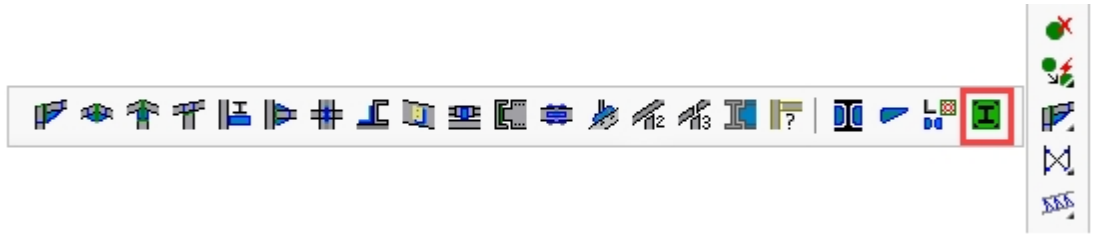
Command -



For more information about this, see the topic Connection Components Library in the Modify Macro chapter.

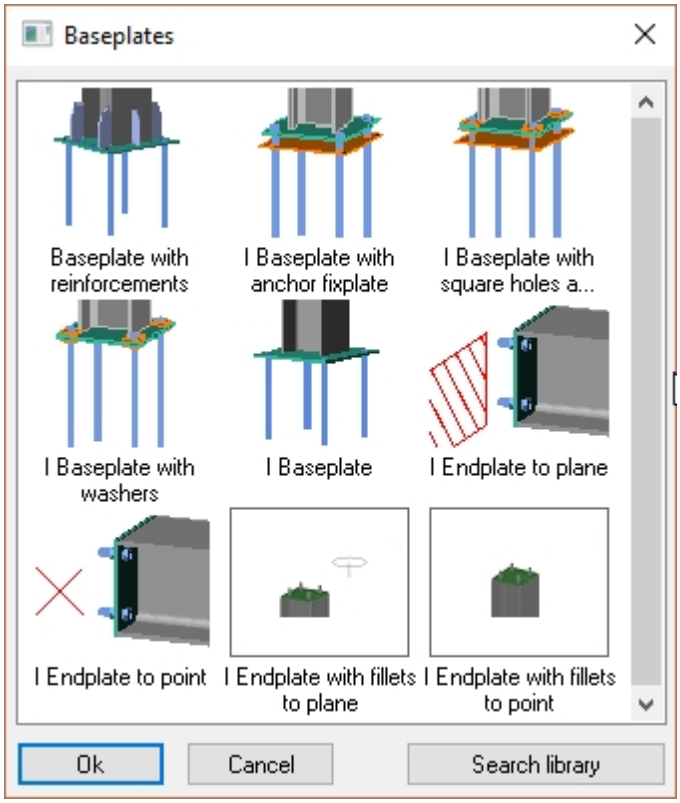
End / Base-Plates

Command -



The **End / Base-Plates** connection will add an end or cap plate to any profile shape.

Some of the available connections are :



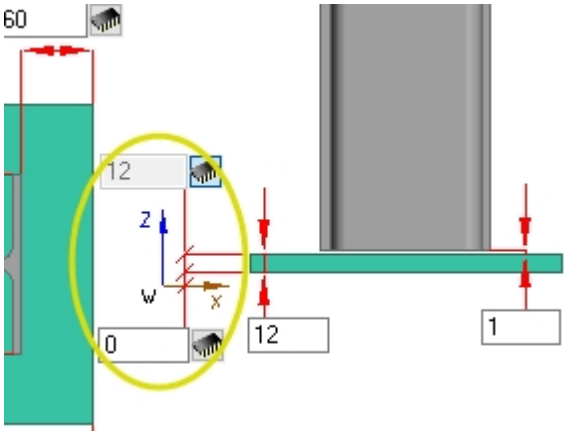
Note! When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections based on the profile type.

Therefore, the presented options may differ from those illustrated here.

Baseplates

By default, baseplates will always be drawn to the World Z plane of the drawing.

To change the height of the baseplate, one can use the offset in the dialog box indicated here :

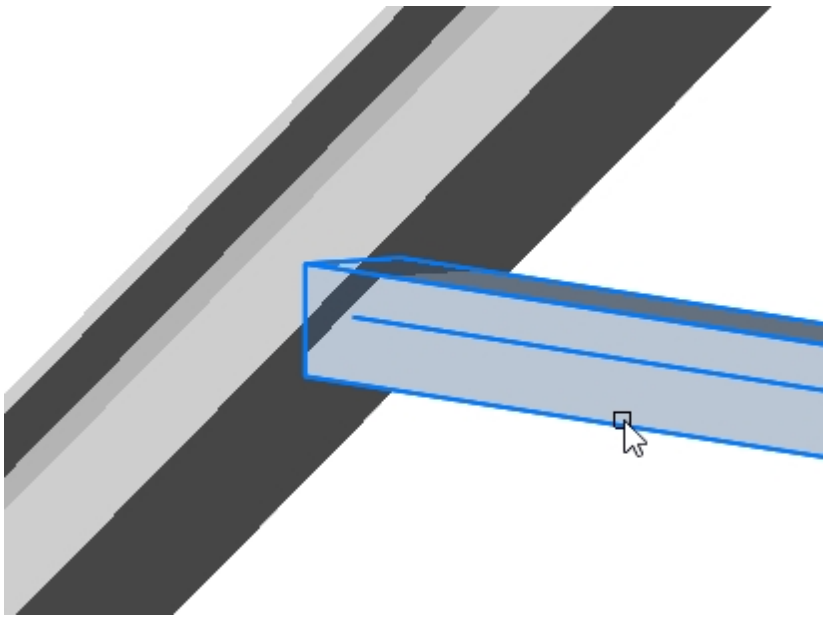


Endplate to plane

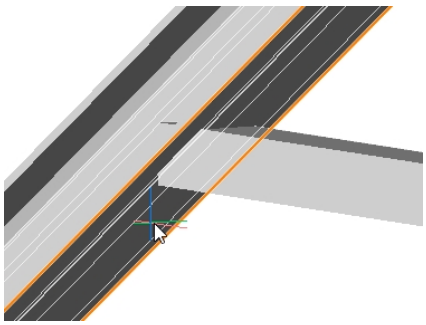
This is the preferred tool, as apposed to the *Endplate to point* tool.
The big advantage of this tool is that our new endplate will assume the inclination and the position of the selected planar surface.
Therefore, the endplate's position and rotation will immediately be correct.

The only condition is, of course, that we have a profile or plate in place to which the endplate should be positioned and bolted.

A possible example use of this tool :



Selecting the profile that needs an endplate attached to it



Selecting the underside of the flange (the circumference of the flange is highlighted here in orange).
We could select any planar surface of any profile, plate, structure, volume, or polyline.

Note It is not recommended to select the profile's own end face. Parabuild will not be able to determine the plate's position reliably. In this case the end cut of the profile changes the face that you selected, creating an unsolvable loop.

Endplate to point

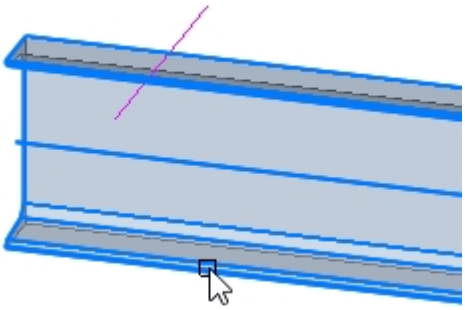
In some cases we have no desire to draw a profile or plate that determines the position of the

endplate.
Then we can use this tool to either select an point sub-geometry, or to use Object Snap to snap to any point to any object.

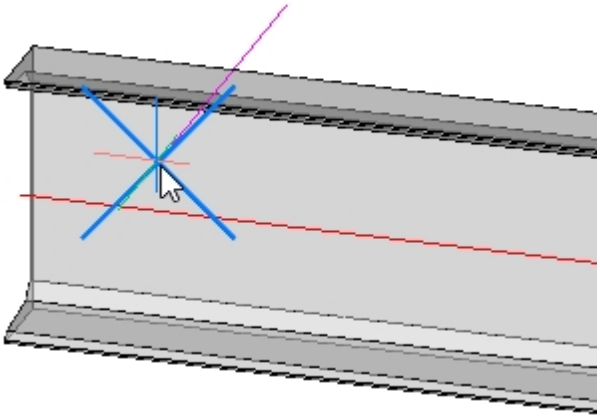
Selecting a point sub-geometry

This is the default behavior of the tool. We can use this to select any point on any object that Parabuild recognizes.
The endplate that is drawn will stay connected to the selected point by the macro sphere.

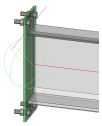
A possible example use of this tool is where a line determines the endplate's position :



Selecting the profile that needs an endplate attached to it



Selecting an endpoint of a line. We could select any point of any profile, plate, structure, volume, line, or polyline.



The e

The position of the endplate is determined by projecting the point to the axis of the profile.

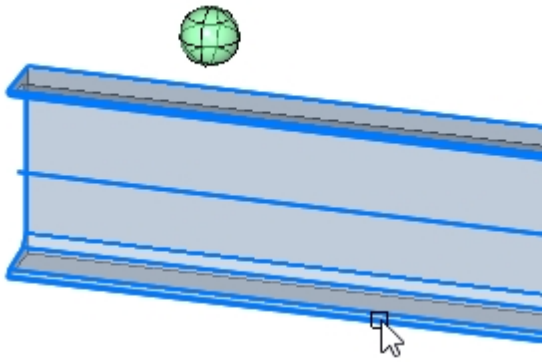
Note It is not recommended to select a point on the profile that needs to receive the endplate. Parabuild will not be able to determine the plate's position reliably. In this case the end cut of the profile changes the point position that you selected, creating an unsolvable loop.

Object Snapping

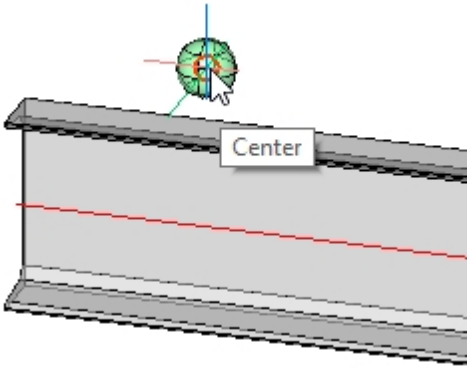
When the *Selecting a point sub-geometry* tool is too limiting, then we can use object snapping. With object snapping, we can select the midpoint of a line, center of a circle, intersection of lines, etc.
It is also object-independent so it will work on any object, even object types that are unknown to Parabuild.

To access this capability, follow the exact same procedure as for the point sub-geometry, but press <Enter> when prompted to select the point sub-geometry.

A possible example use of this tool :

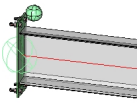


Selecting the profile that needs an endplate attached to it



After pressing <Enter> for not wanting to select a sub geometry : Selecting the center of the sphere with the help of Object Snapping. This is just an example.

We could snap to any point on any object. This can be a point on an object unknown to Parabuild such as 3D Solids or 3rd party software objects.



The end

This tool will draw a new point object at the location that you snapped.
The new endplate will stay connected to that point object by the macro sphere.

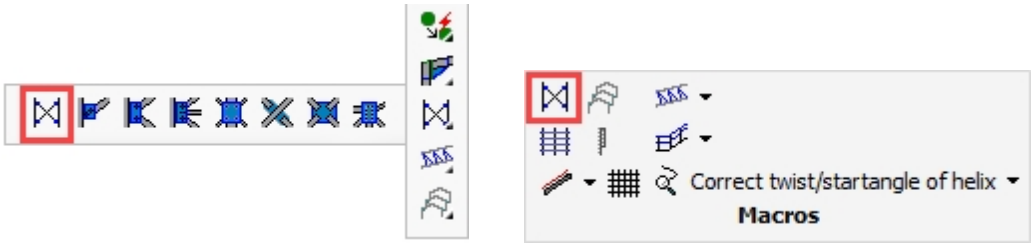
The position of the endplate is determined by projecting the point to the axis of the profile.

Bracing

- Bracing
- Gusset Plate - 2 Base
- Gusset Plate - 3 Base
- Gusset Plate - 4 Base
- Gusset Plate - 5 Base
- Gusset Plate - Middle 3 Base
- Gusset Plate - Middle 4 Base
- Transmission - 5 Base

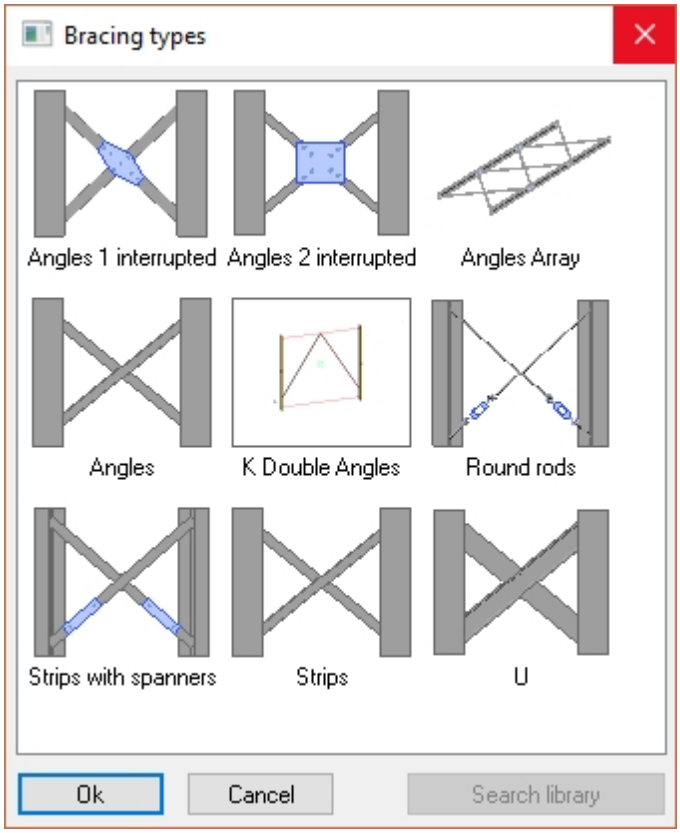
Bracing

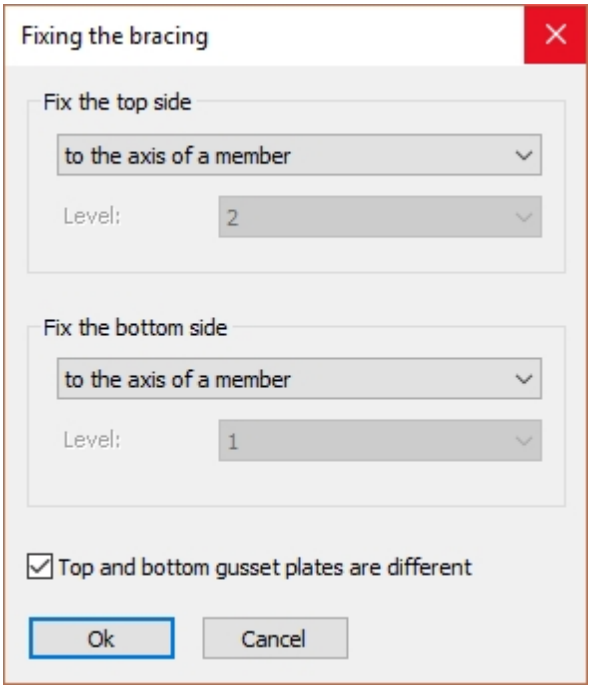
Command - **PrB_Bracing**



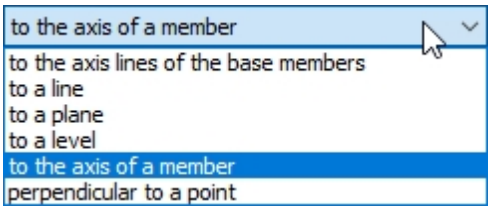
Activating the Bracing command will bring up the following bracing options dialog, from which you are prompted to select the Wind Bracing configuration.

After that, the **Fixing the bracing** dialog is shown where you are prompted to fix the top and bottom bracing elements:





The options for fixing the top and bottom bracing elements include:



All bracing will be drawn on the axes of the main members - which may be either columns or main lateral profiles.

The bracing fixing options only apply to transverse members.

- **To the axis lines of the base members** - This option will draw the selected bracing between the ends of the 2 main profiles, ignoring any transverse members.
- **To a Line** - The model line should be drawn to intersect with the axes of the base members.
  Note!
  - 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 main 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 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 (surface) 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 bracing in the roof. (You will be prompted to enter the applicable levels)

- **To the axis of a member** - The same procedure as with lines, but in this case the axis of the selected member will be used.
- **Perpendicular to a point** - You will be prompted to 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

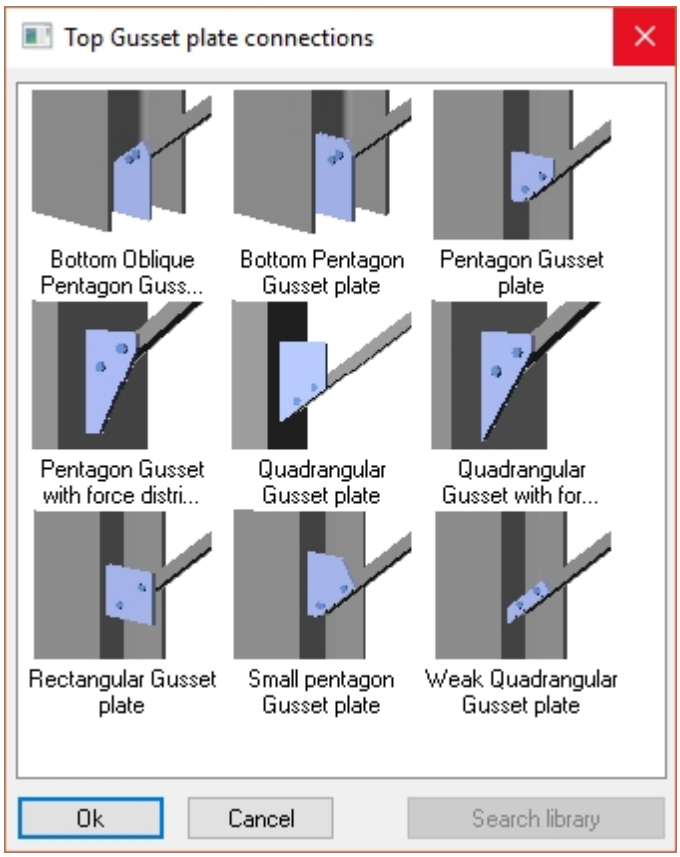
If this option is deactivated, the gusset plate dialog box will be shown just once and all four gusset plates will be the same.

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

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

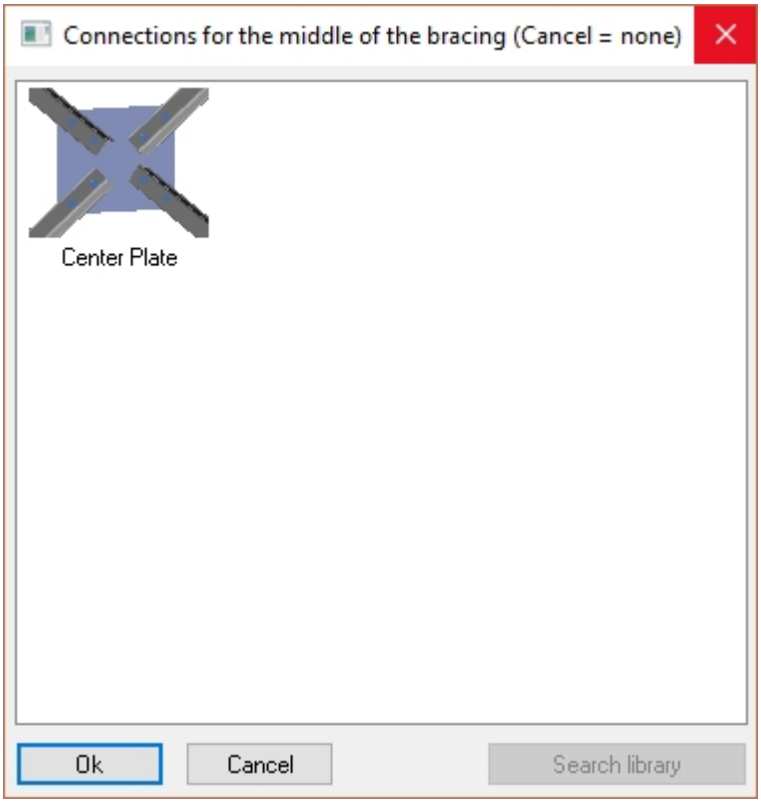


We choose the desired gusset plate and click on **Ok**.

It is acceptable to click the cancel button at this stage. In this case the bracing will still be drawn

but without gusset plates.

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

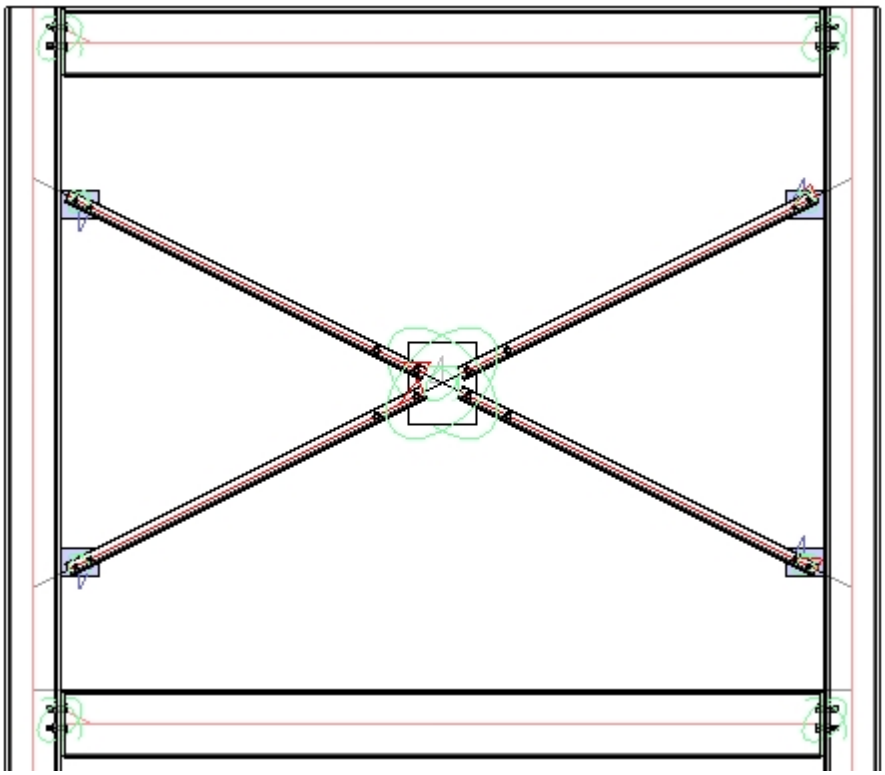


We choose an option and click on **Ok**.

It is acceptable to click the cancel button at this stage. In this case the bracing will still be drawn but without the center connection.

Now Parabuild draws the wind bracing.

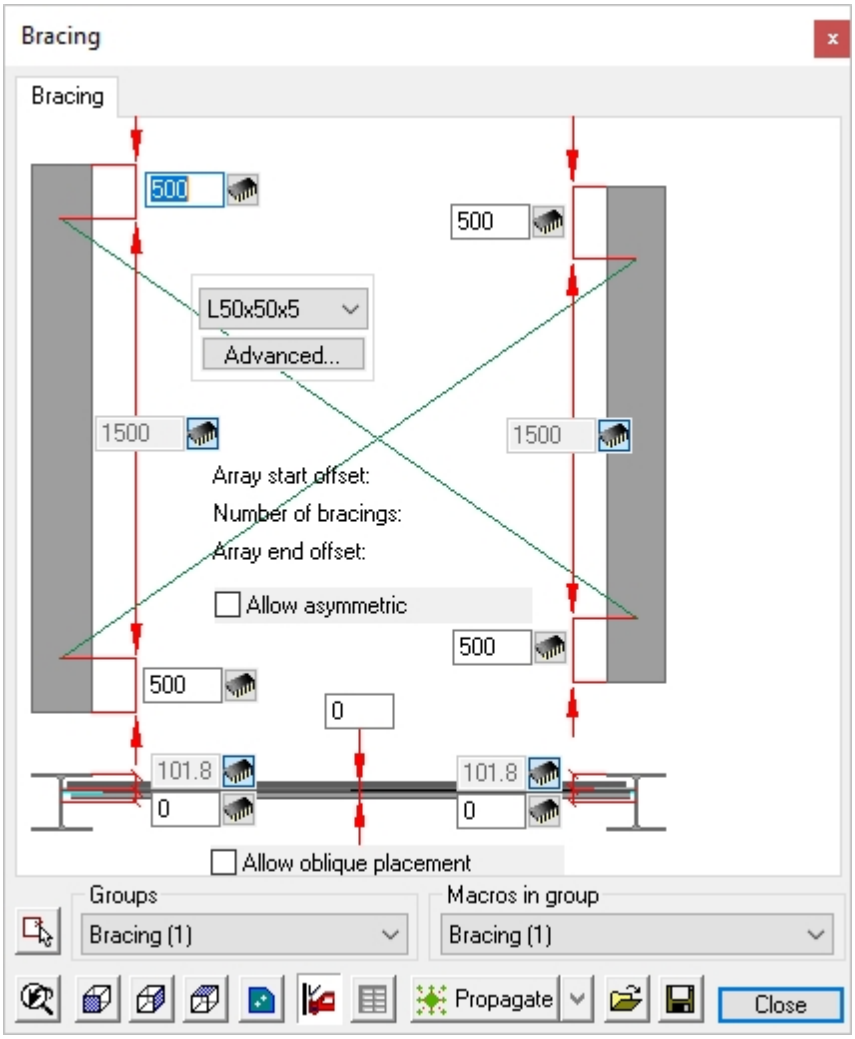
The previous prompts are repeated so that you can draw the same bracing on other locations.



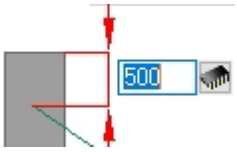
This example shows an angle cross brace with a center plate.

The bracing edit dialog

After completing the bracing, all of the dialog s of the newly drawn parts are displayed.



Bracing



- 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.

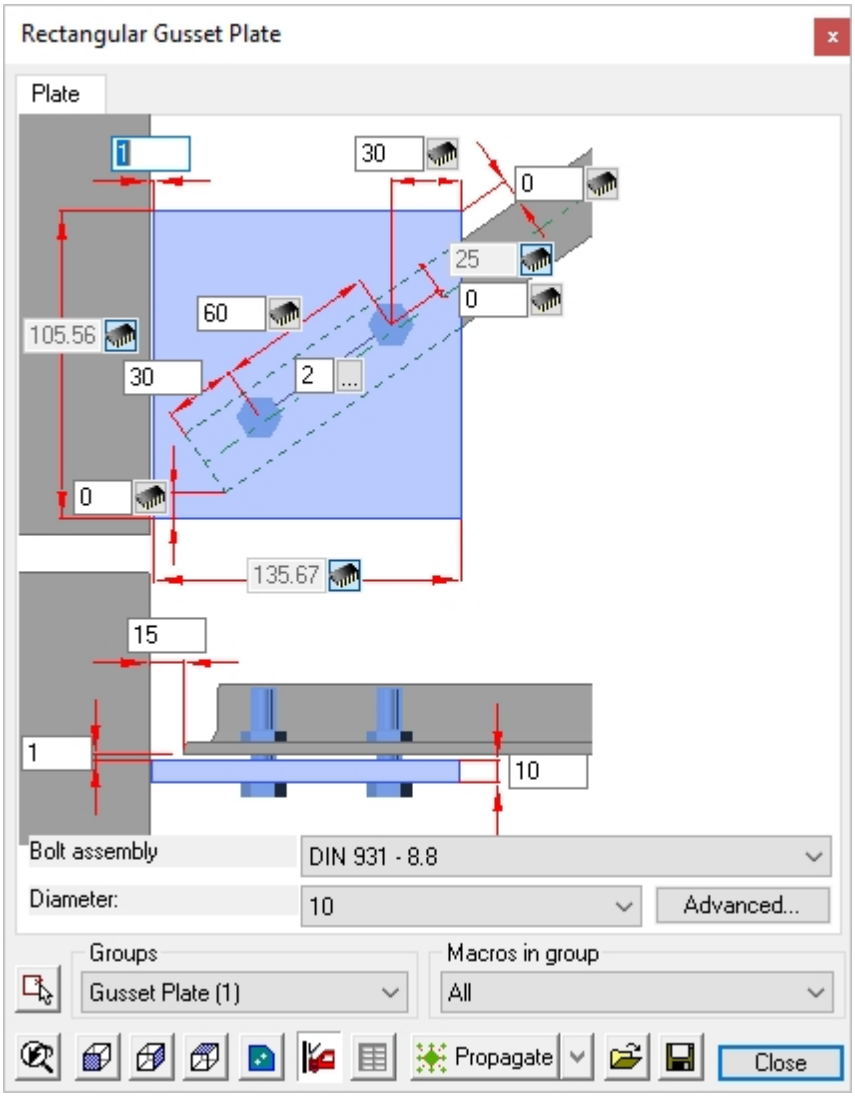
Note! if you change the profile type of the bracing using this tool, then the gusset plate connections will likely become corrupted. In this case you should redraw them.

- 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. This last setting gives you the possibility to draw a bracing between members whose axes are not aligned.
- **Note!** The gusset plates that Parabuild has drawn are suggestions and should be reviewed to make sure the dimension adhere to the load calculations.

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

For more information on these macro dialogs, see the topic Review macro dialog.

Gusset Plates

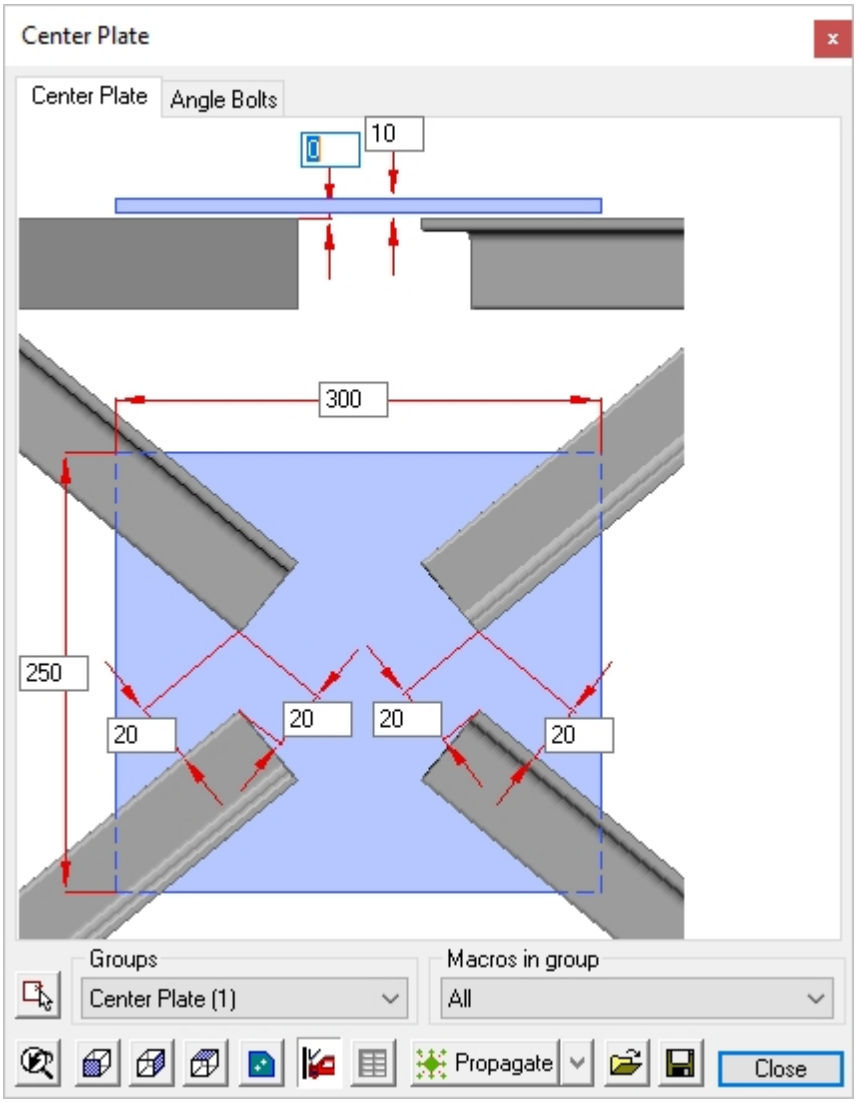


Here you may edit the gusset plates connecting to the main members.

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

For more information on these macro dialogs, see the topic Review macro dialog

Center Plate



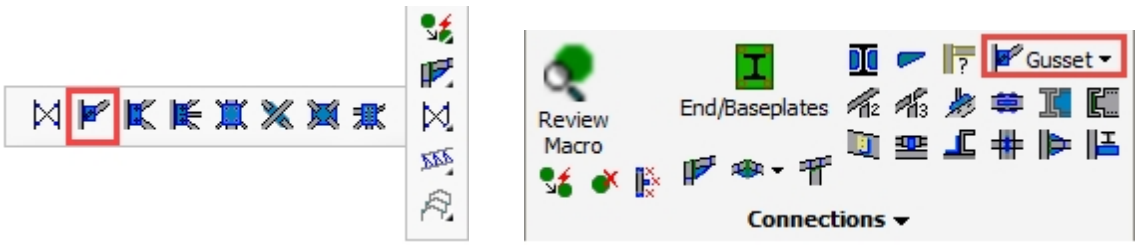
Here you may edit the center plates connecting the bracing members

The center plates can be modified individually with the Review macro command on the specific macro sphere.

For more information on these macro dialogs, see the topic Review macro dialog

Gusset Plate - 2 Base

Command -



The 2 base gusset plate group connects a single brace to either a beam or column.

On activating this command you will be prompted to:

1. Select the Column (or beam)
2. Select the bracing member

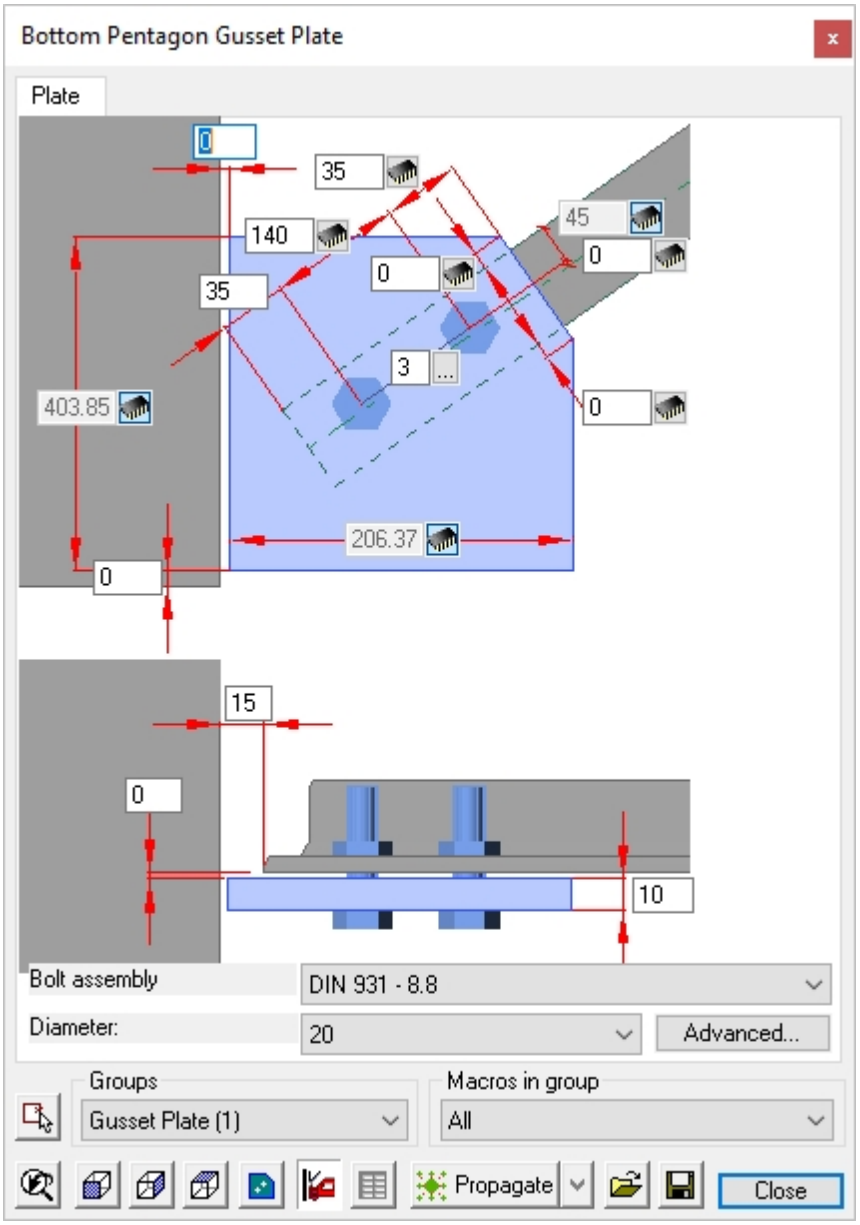
Note!

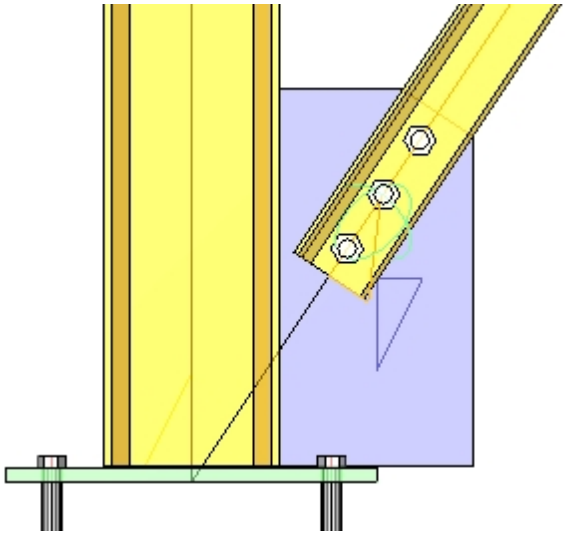
When selecting the profiles to be connected, Parabuild will automatically filter the proposed

connections.

The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if the gusset plate is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the selection dialog.

Therefore, the presented options may differ from those illustrated here.



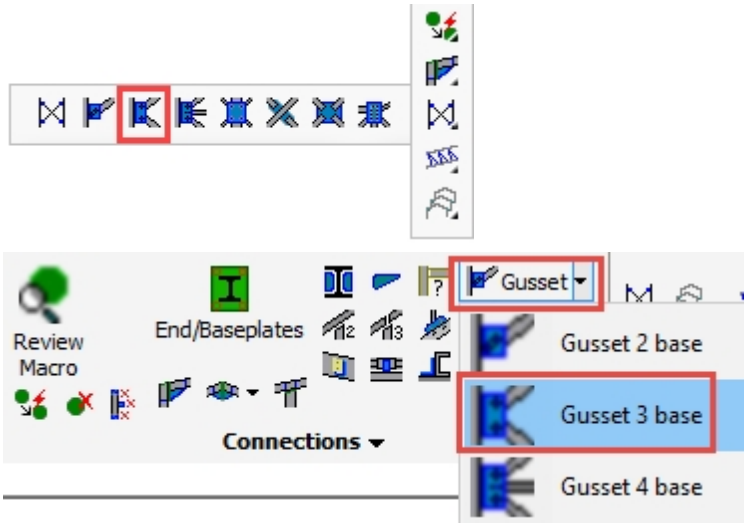


The above illustration shows the result of the edited customization dialog.

The shape of the gusset plate may be edited using the Modify Profiles or Plates or Modify Plate Vertices commands

Gusset Plate - 3 Base

Command -



The 3 base gusset plate group connects 2 bracing members to either a beam or column.

On activating this command you will be prompted to:

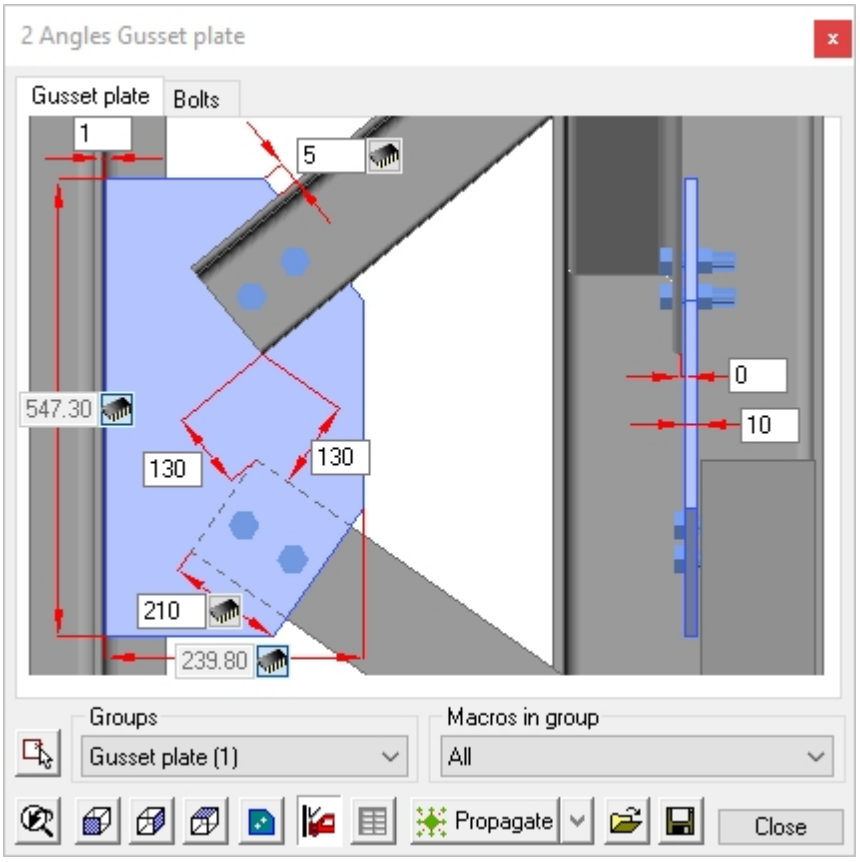
1. Select the Column (or beam)
2. Select the top (or right for beam) bracing member
3. Select the bottom (or left for beam) bracing member

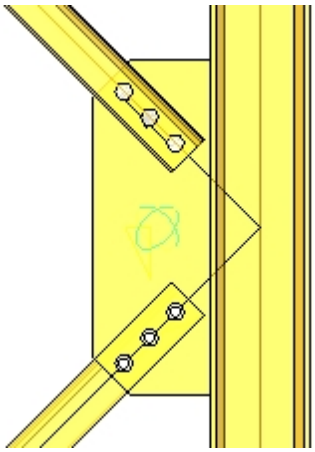
Note!

When selecting the profiles to be connected, Parabuild will automatically filter the proposed connections.

The filtering will not only be based on the profile types, but also their orientation with respect to each other. For example, Parabuild will recognize if the gusset plate is connecting to a column flange or web. This information is used by this command to filter out the incompatible connections in the connection selection dialog.

Therefore, the presented options may differ from those illustrated here.





The illustrations show the result of the edited customization dialog.

The shape of the gusset plate may be edited using the Modify Profiles or Plates or Modify Plate Vertices commands

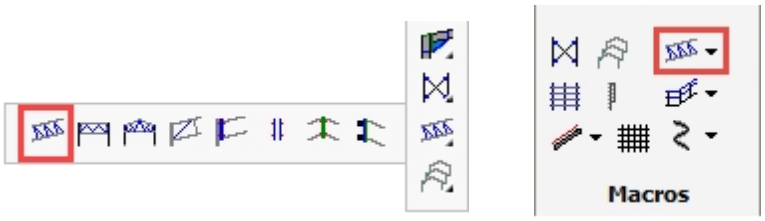
Trusses

- Truss
- Truss between Columns

- Truss for Portal Frame
- Truss Strut
- Truss haunch Connection
- 2 Trusses v Column
- Truss - Apex
- Truss to Truss

Truss

Command -



Activating this command will open the Truss dialog, from which you may choose a configuration.

The truss options include:

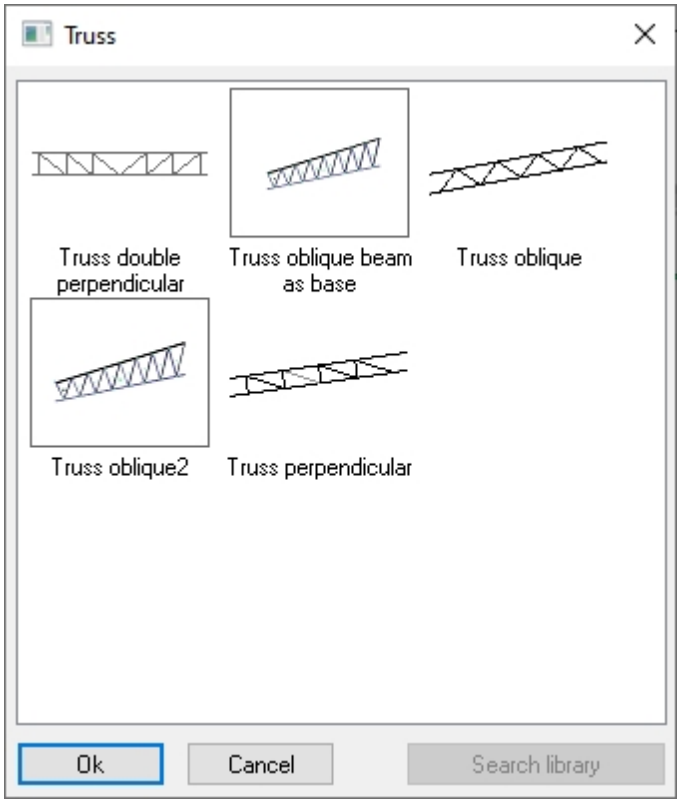
Truss double perpendicular

Truss oblique beam as base

Truss oblique

Truss oblique2

Truss perpendicular



Draw a Truss on a line

After clicking on **Ok**, just select the model line and the complete truss will be drawn on top of it.

Draw a truss by indicating 2 points

At the command line prompt you will be prompted to **Select the line that will determine the location of the parts, or <Enter> to draw a new line :**

If there is no preexisting line, press <Enter> and you will be prompted to indicate the first and second points of the line, after which the default truss will be drawn.

Some information about the truss

The line will determine the length and position of the top chord of the truss, it may be drawn between columns, beams, or independently in space, depending on its function and how it is to be supported.

The location of the top chord relative to the line is determined by the **Section displacement** which may be set from the Profile placement dialog. By default, all profiles forming the truss are set to **Middle**. This means that all the truss component members are set so that they intersect at their geometric centers eliminating eccentricities at the joints and is considered to be the optimum configuration for efficient load distribution.

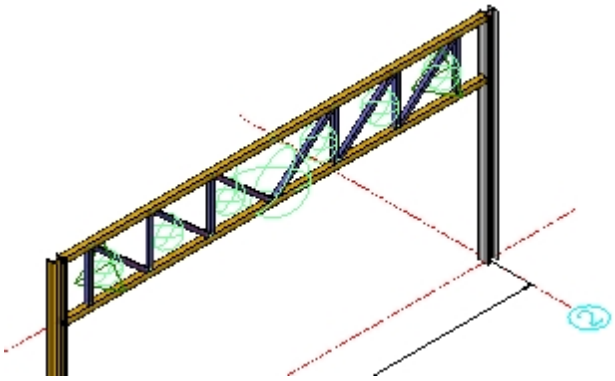
Altering the Profile placement will alter the truss geometry. If this is not desirable, then the line should (In the instance of connecting to columns) be placed below the column cap at a distance of 0.5 x the depth of the top chord ensuring the top of column = top of truss.

Likewise, the depth of the truss is determined by the same geometry.

Add truss end-connections

You can select bolted connections for connecting the truss chords to the columns from the Connections menus

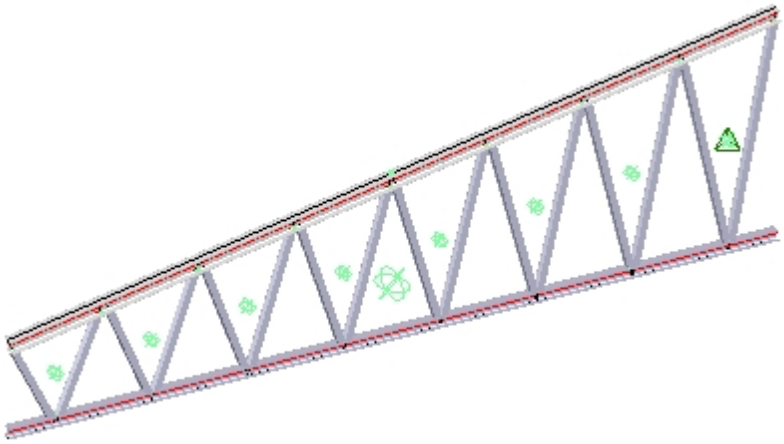
Truss double perpendicular



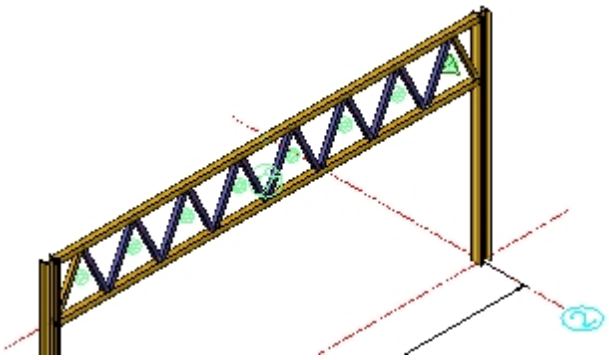
Truss oblique beam as base

This is the same as Truss oblique2, except the base of the entire truss will not be a line but it will be the top chord itself.

So for this truss the top chord will need to be adapted to determine the truss's size and location.

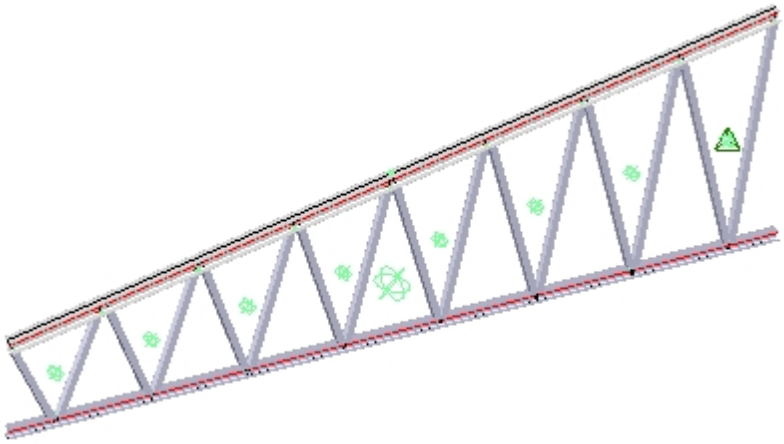


Truss oblique

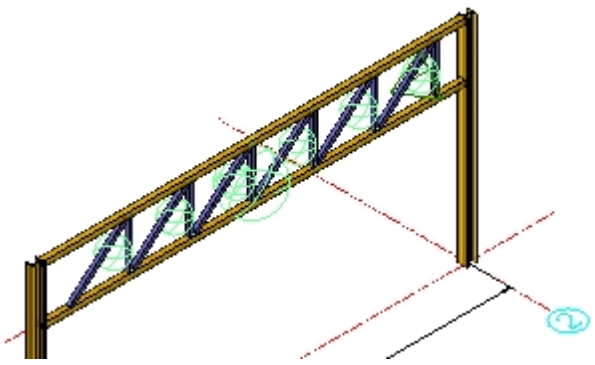


Truss oblique2

Unlike most other trusses, when the top chord of this truss is sloped, then the bottom chord will always remain horizontal.



Truss perpendicular



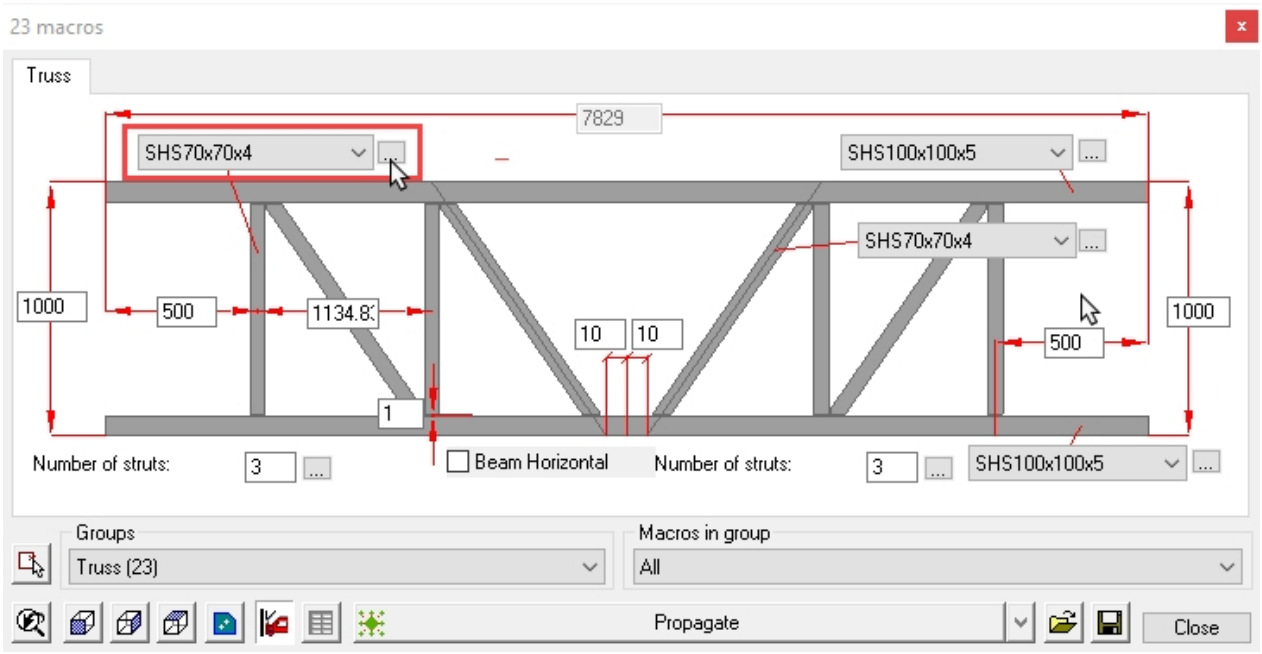
The macro edit dialog for Trusses

The edit dialog will vary according to the chosen truss, but the general principles will apply to all.

Here you are able to change the number of struts, truss depth, and profile types and sizes. The length of the truss is predetermined by the model line.

The truss is made up of a number of macros, each of which may be edited individually. Activate the Review macro command and indicate the green sphere of the macro that you want to modify individually.

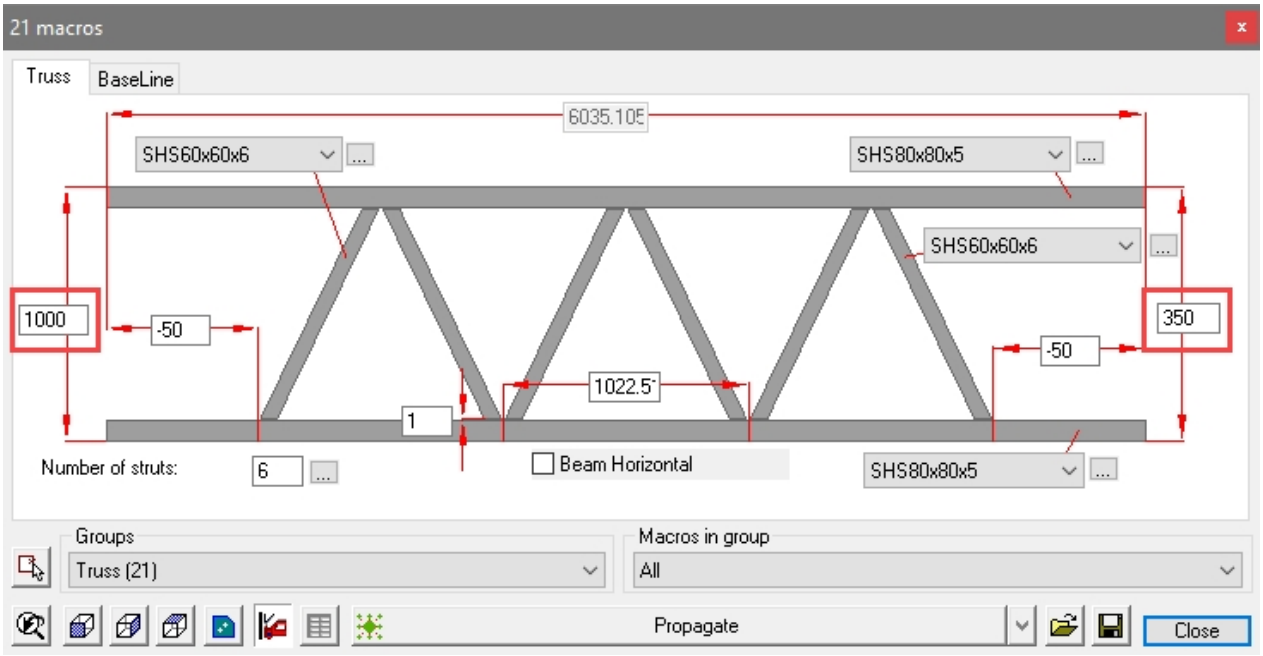
The geometry of the truss is determined by the Profile placement. This can be accessed by clicking on the button ... next to the profile size drop-down.



Note The top and bottom chords do not have to be parallel. The slenderness of both chords may be changed with the two depth options.
In the example below, the truss depth will be **1000** at one end - sloping to **350** at the other.

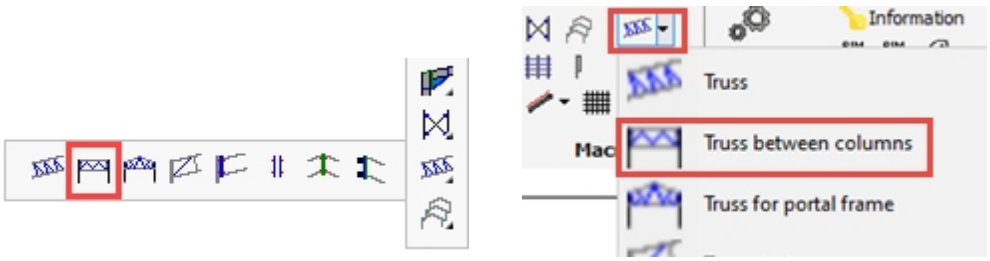
Making both ends the same will give a parallel truss.

Checking the **Beam Horizontal** checkbox will override the dimension differential.



Truss between Columns

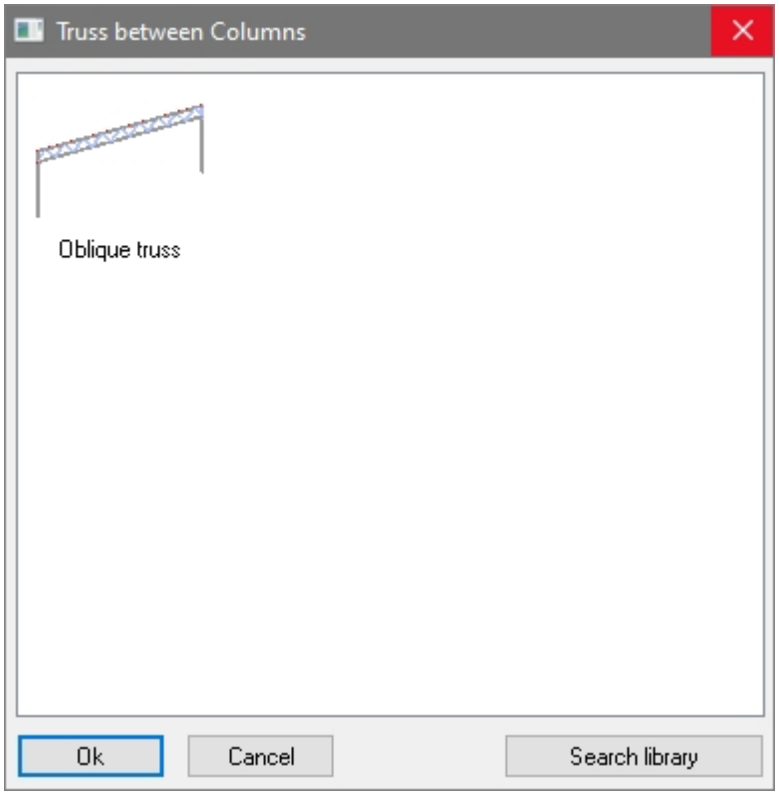
Command -

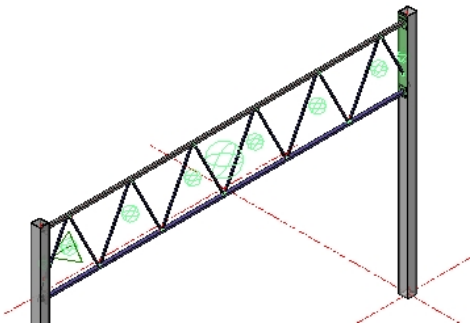


This command will draw a truss between square or rectangular hollow section columns.

This command will first prompt for the columns. When prompted to select the coordinate system, press <**Enter**>.

For now there is just 1 type of truss implemented in this tool :

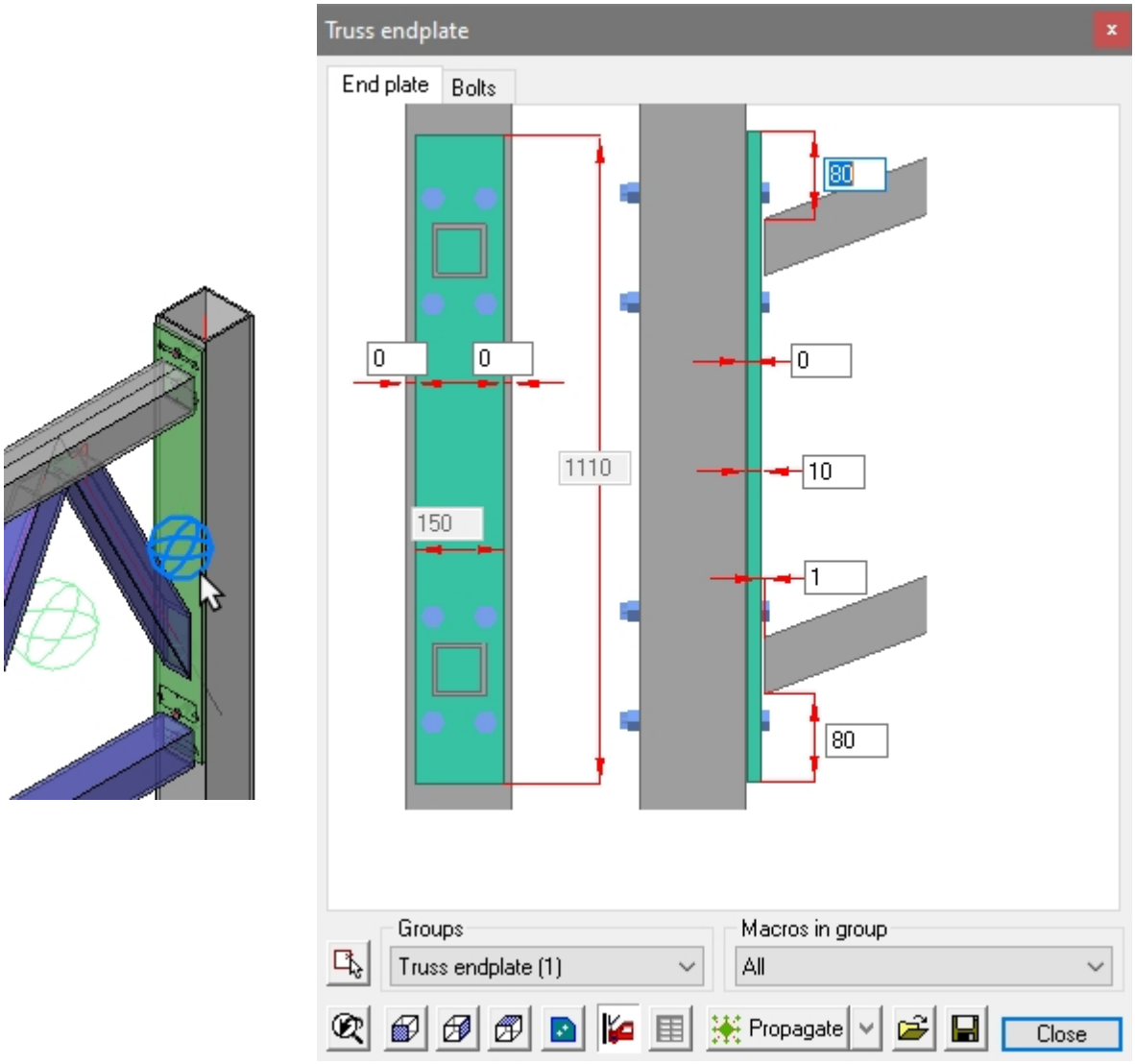




For more information about the truss dialog box's options, see the Truss topic.

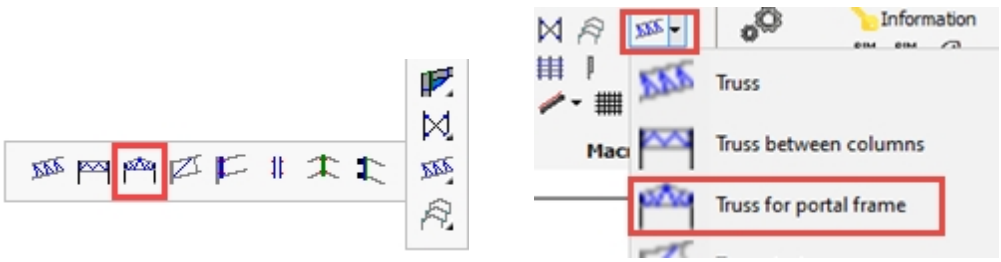
The truss is made up of a number of macros, each of which may be edited individually. Activate the Review macro command and indicate the green sphere of the chosen macro.

Example: To edit the end plate, select the macro sphere, press <**Enter**> or right-click the mouse button to open the macro edit dialog



Truss for Portal Frame

Command -



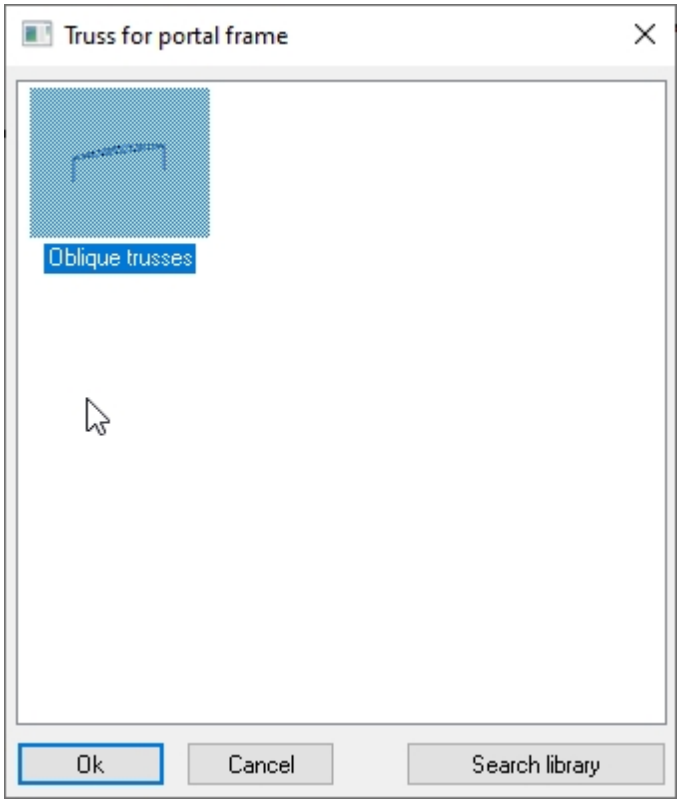
This tool can draw a double truss inside a portal frame.

It will ask you to select the following items in this order :

1. The column on the left hand side

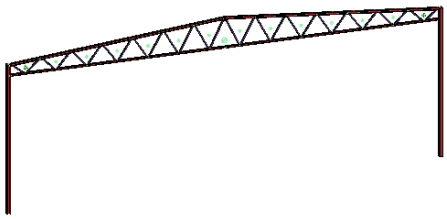
2. The left rafter on the left hand side
3. The rafter on the right hand side
4. The column on the right hand side

For now there is just 1 type of truss implemented in this tool :



When prompted to select the coordinate system, press <**Enter**>.

You will first need a a portal frame in order to use this tool

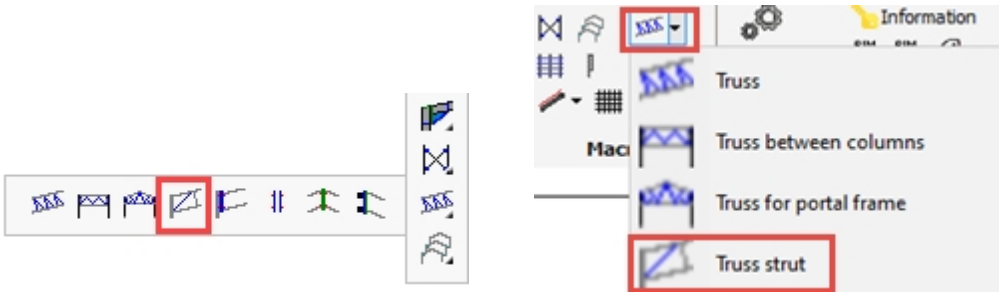


The end-result : The truss with a spliced bottom chord and connections to the columns

For more information about the truss dialog box's options, see the Truss topic.

Truss Strut

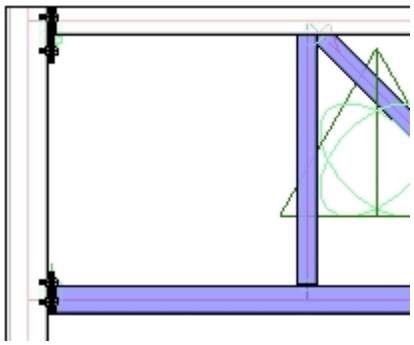
Command -



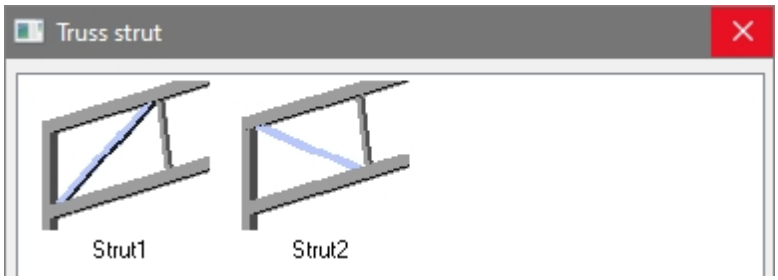
This command will add a strut to the end of a truss.

On activating the command you will be prompted to select in this order :

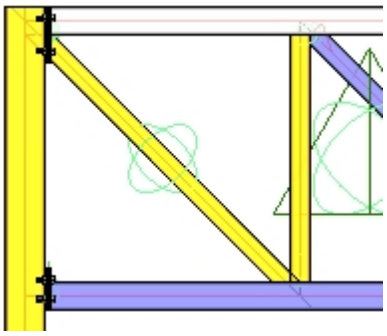
1. The column
2. The lower chord of the truss
3. The upper chord of the truss
4. The vertical strut

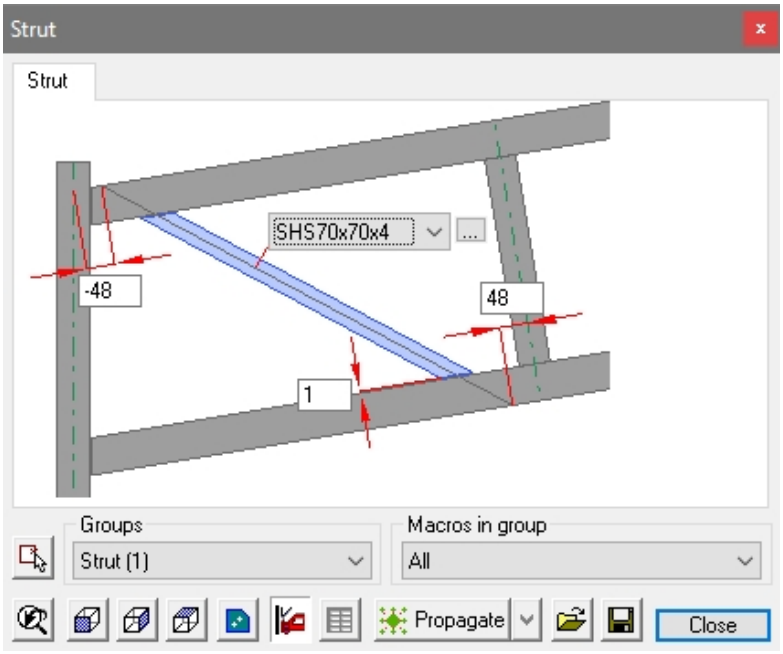


After selection, the options dialog will appear - select the required option:



The strut will be placed accompanied by the macro edit dialog

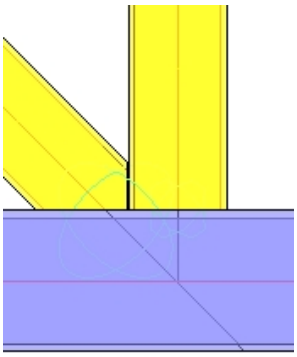




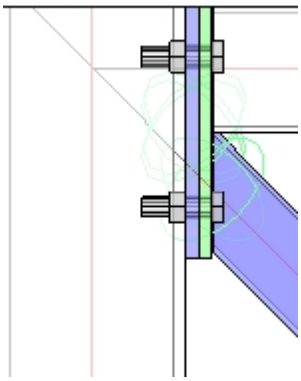
In this case, the reference dimensions have been edited to read 48, so that the center-line of the new strut intersects with the centers of the top and bottom chords of the truss.

Note that the color of the struts and the adjacent column indicate an interference. This is because the required cuts have not been included:

You can add these with the Add cut to macro tool that is available in the Cutting toolbar.



Cut 1

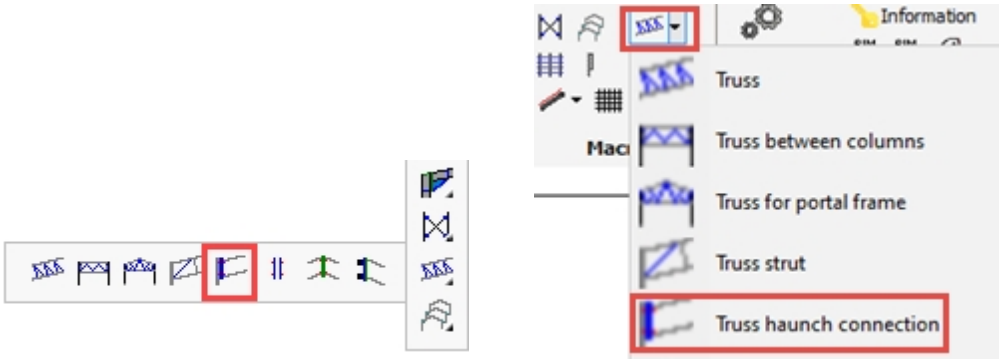


Cut 2

Running the Clash check command should indicate the interference has been resolved

Truss Haunch Connection

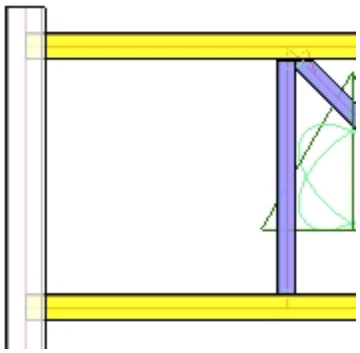
Command -



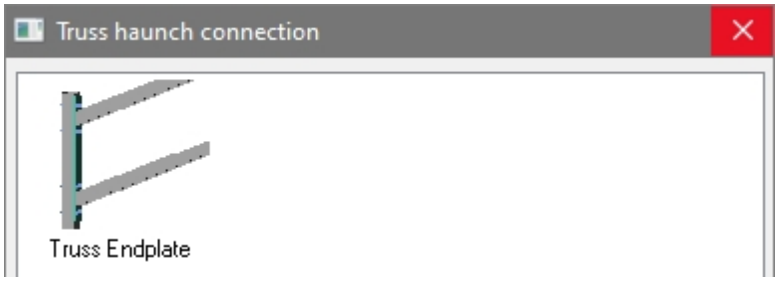
This command will add a full depth end-plate connection for a truss.

On activating the command you will be prompted to select in this order :

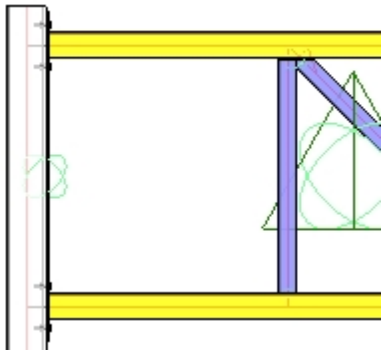
1. The column
2. The lower chord of the truss
3. The upper chord of the truss

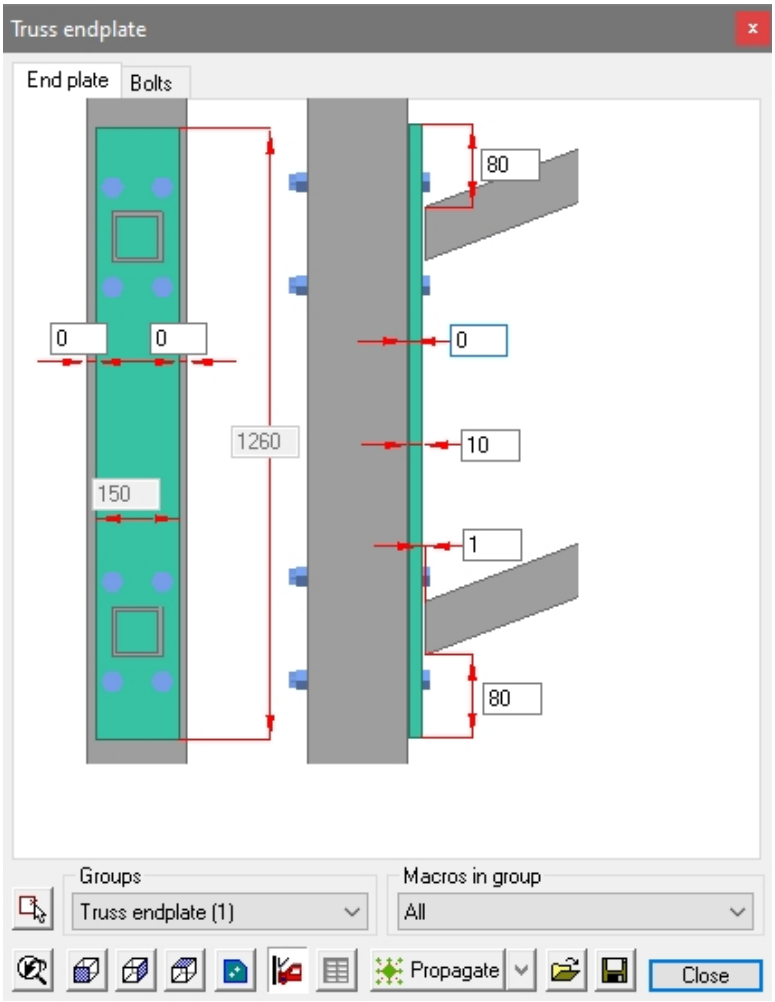


After selection, the options dialog will appear :



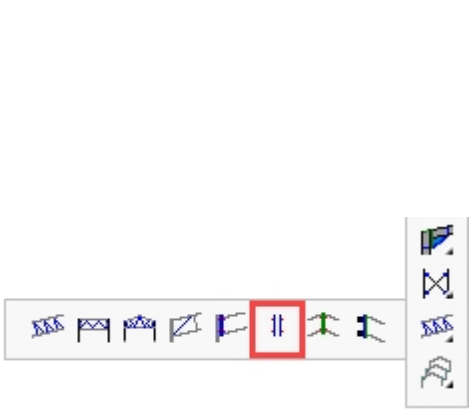
The end-plate will be drawn immediately :

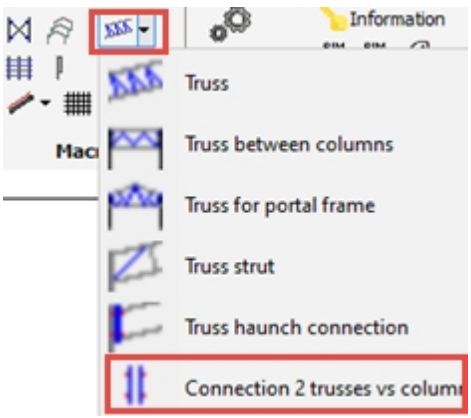




2 Trusses v Column

Command -

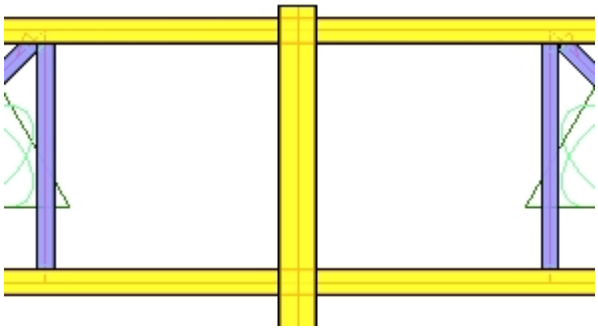




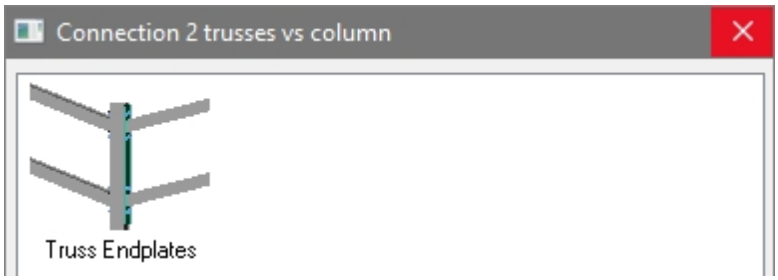
This command will add full depth end-plates truss connection where 2 trusses connect to one column.

On activating the command you will be prompted to select in this order :

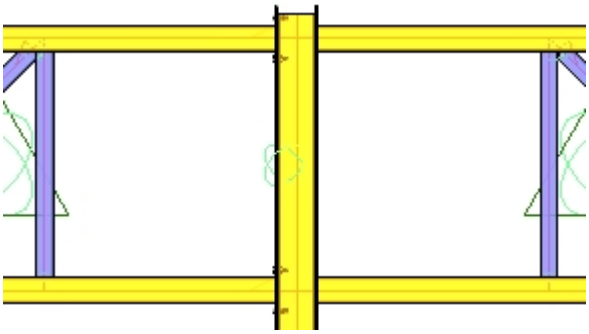
1. The column
2. The lower chord of the left truss
3. The upper chord of the left truss
4. The lower chord of the right truss
5. The upper chord of the right truss

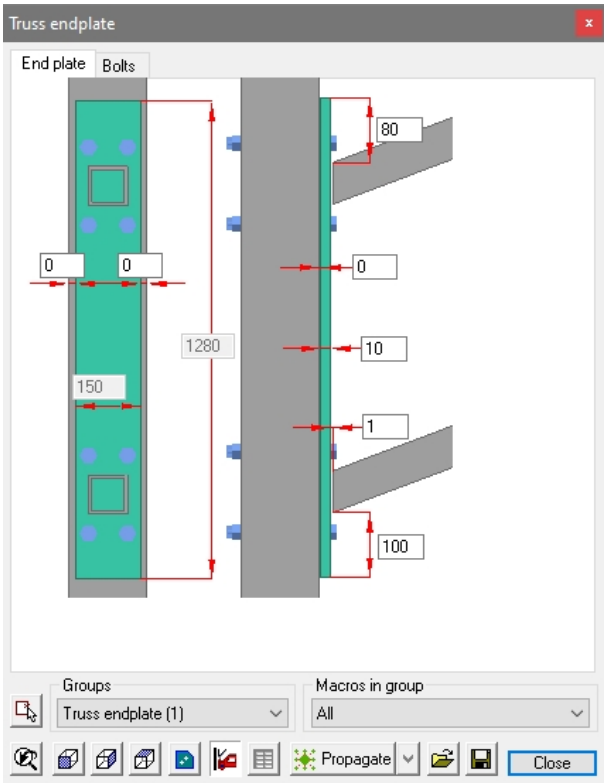


After selection, the options dialog will appear :



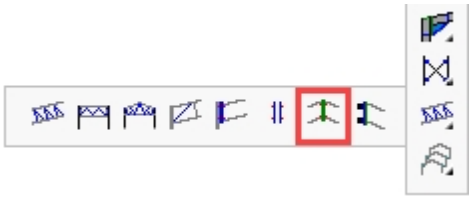
The end-plates will be drawn immediately :

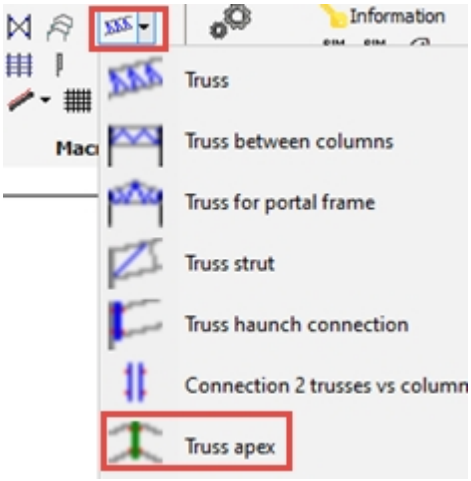




Truss Apex

Command -



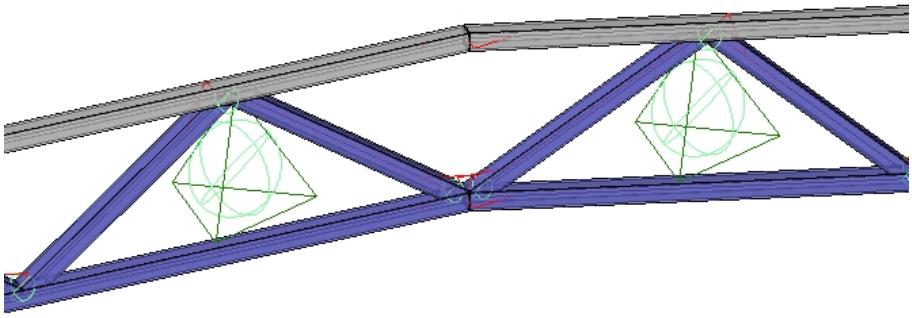


This command will add full depth apex end-plates for 2 trusses.

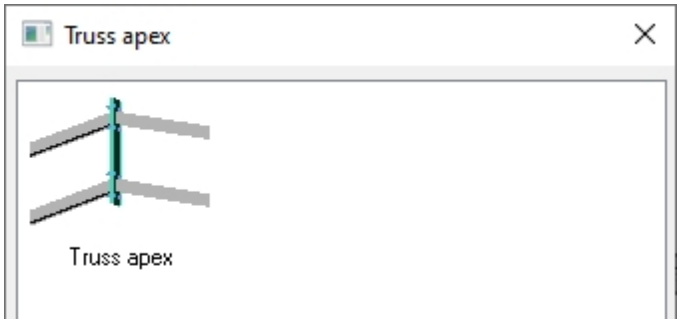
On activating the command you will be prompted to select in this order :

1. The lower chord of the left truss
2. The upper chord of the left truss
3. The lower chord of the right truss

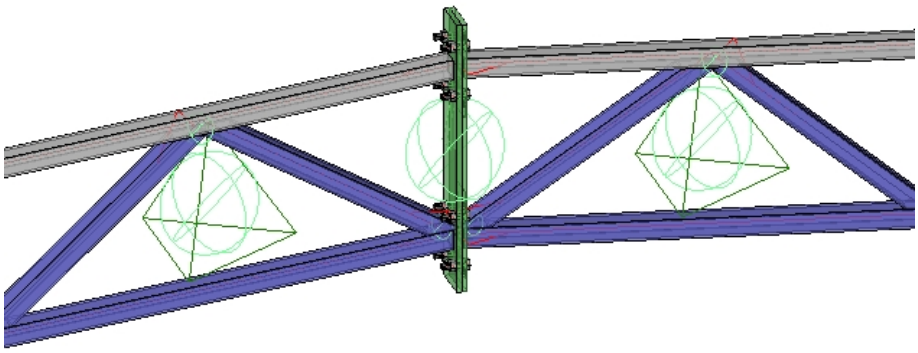
4. The upper chord of the right truss



After selection, the options dialog will appear :

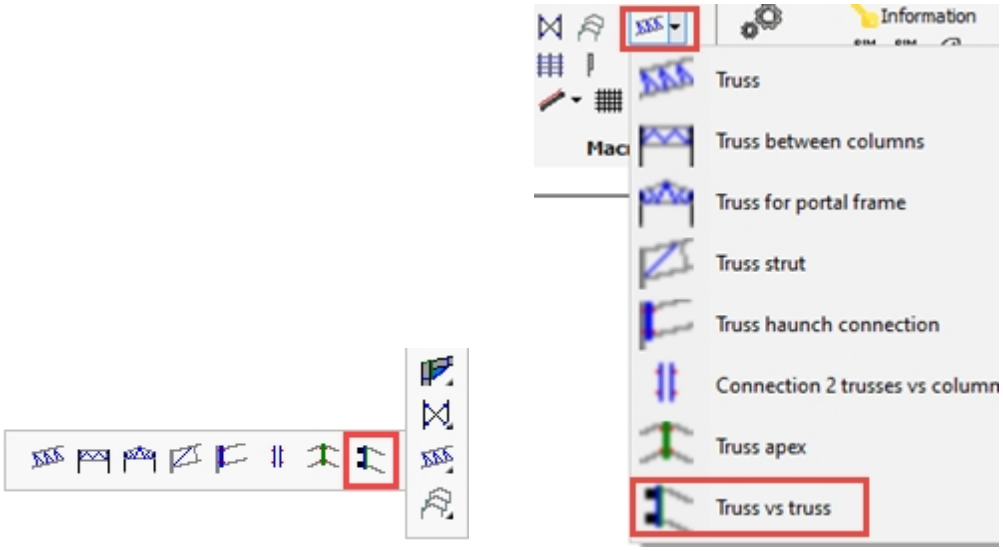


The end-plates will be drawn immediately :



Truss to Truss

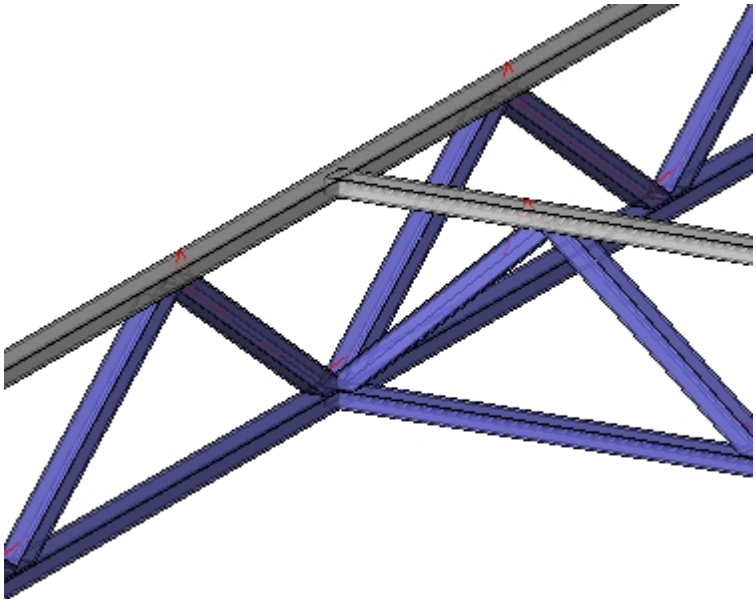
Command -



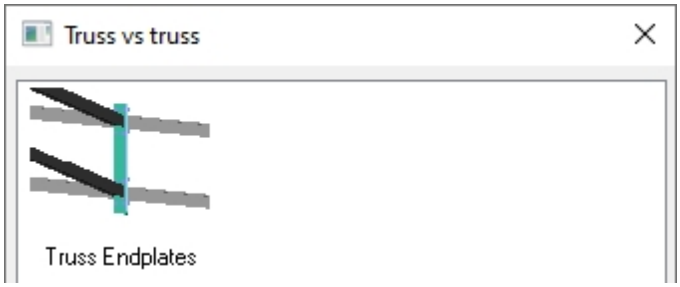
This command will add full depth end-plates for intersecting trusses in a T-connection.

On activating the command you will be prompted to select in this order :

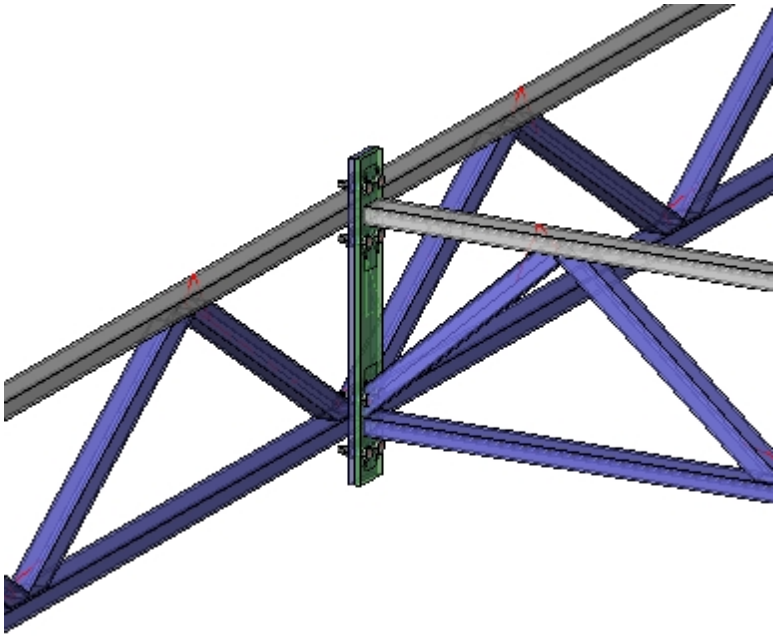
1. The lower chord of the truss that will be lengthened
2. The upper chord of the truss that will be lengthened
3. The lower chord of the truss that is continuous
4. The upper chord of the truss that is continuous



After selection, the options dialog will appear :



The end-plates will be drawn immediately :



Framework

- Frames /
    - Building Imperial
    - Building Imperial / No Connections
    - Building / No Connections
    - Building
    - Platform
    - Staircase / Cranked Stringers
    - Staircase / Floors only
    - Staircase / with UB Platforms
- Stair
- Railing
- Ladder
- Spiral stair
- Cladding

- Draw a Body
- Manipulate a Body

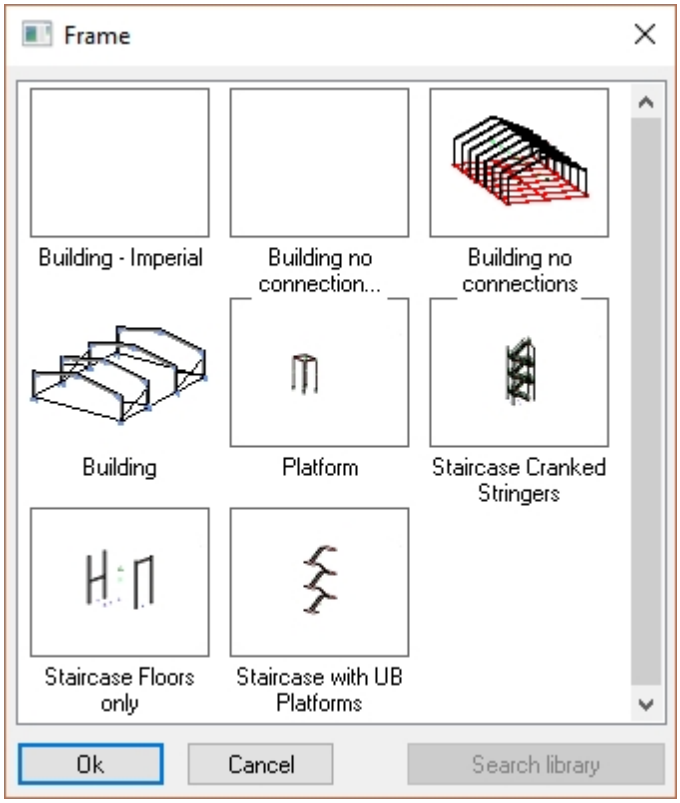
Frames

Command -

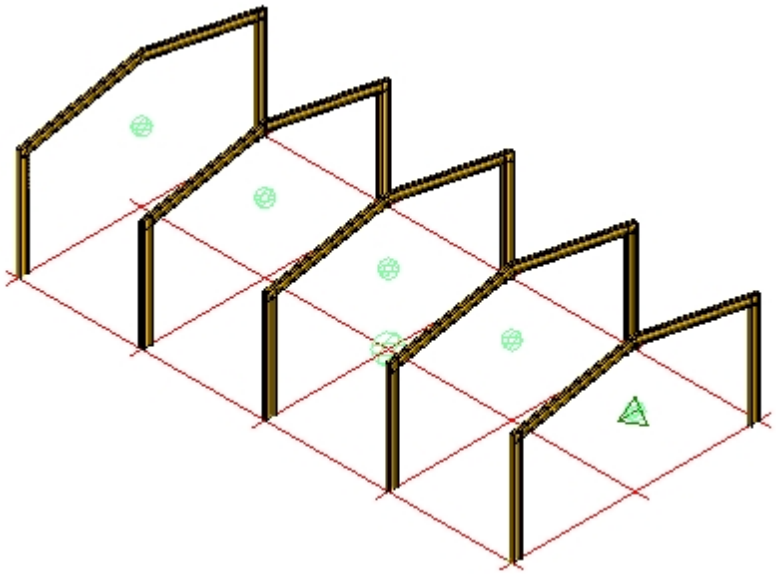
Activating this command will open the below dialog.

From here you can select one from the range of predefined building structures. Each one is explained in detail below:

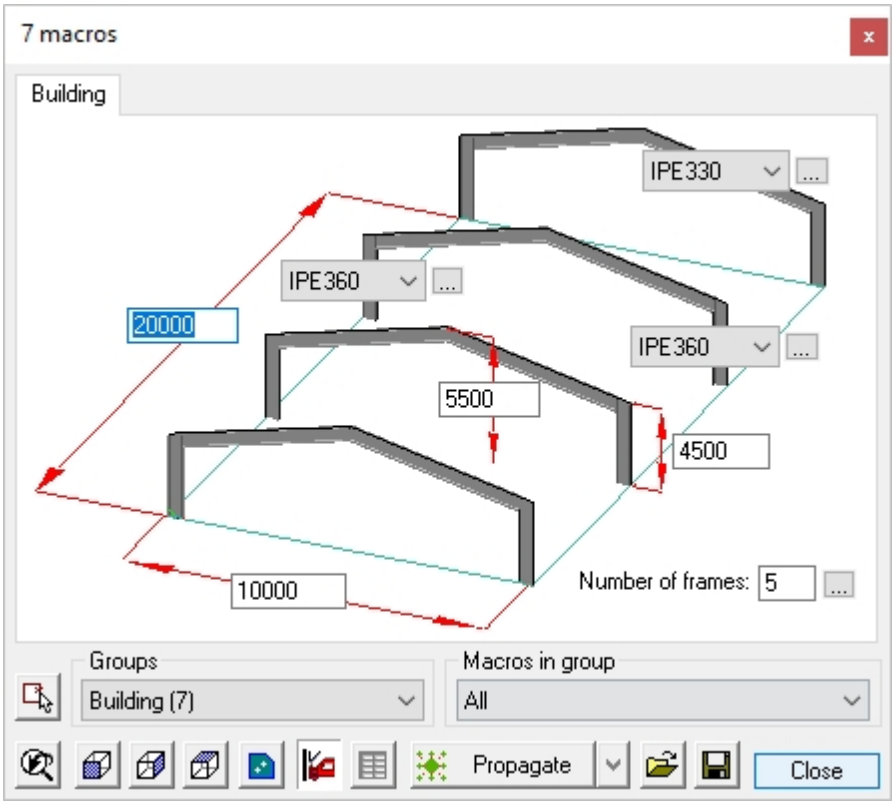
- Building - no connections
- Building - with defined connections and braced bays
- Platform
- Staircase Cranked Stringers - Stair tower
- Staircase - Floors only
- Staircase with UB platforms



Building - no connections



The customization dialog of this macro looks like this :

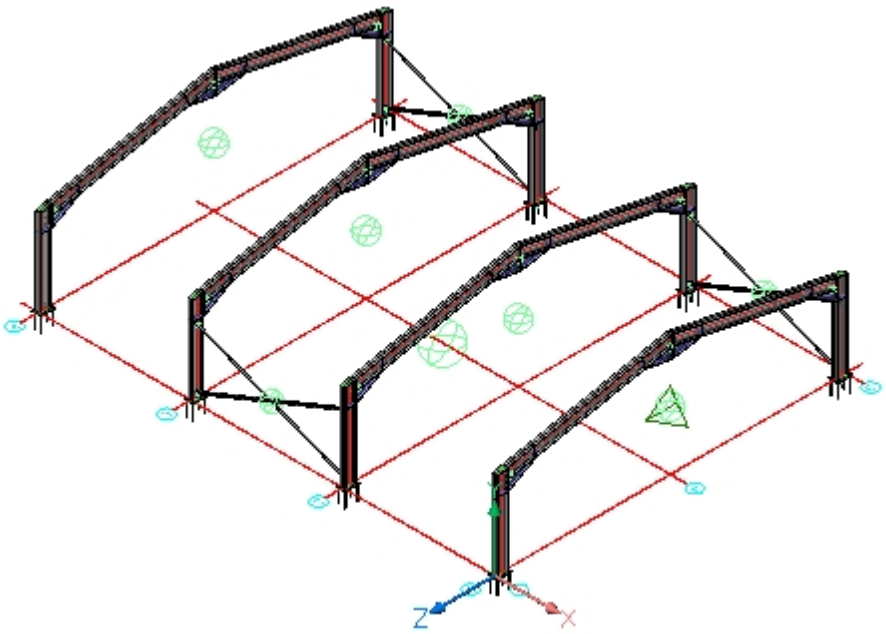


Here, you are able to fix the height, roof height, width, number of bays, and the steel section sizes.

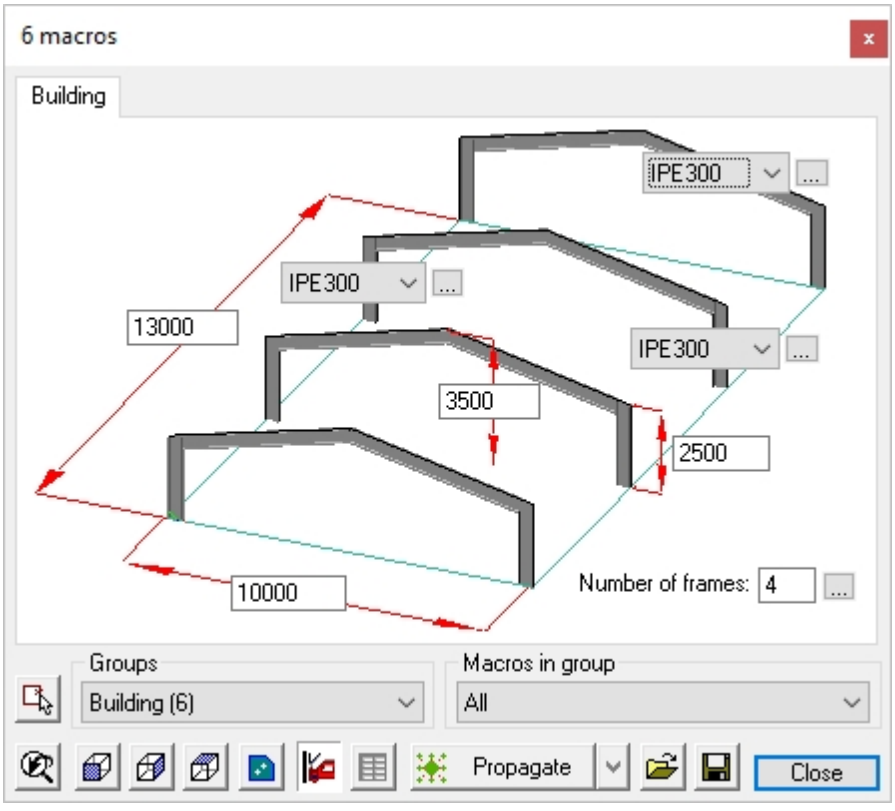
Note that by default the bays will always be equally spaced. This can be changed by pressing the ... button next to the number of bays value.

You can revisit any of the individual macros by using the Review Macro command

Building - with defined connections and braced bays



The customization dialog of this macro looks like this :



Here, you are able to fix the height, roof height, width, number of bays, and the steel section

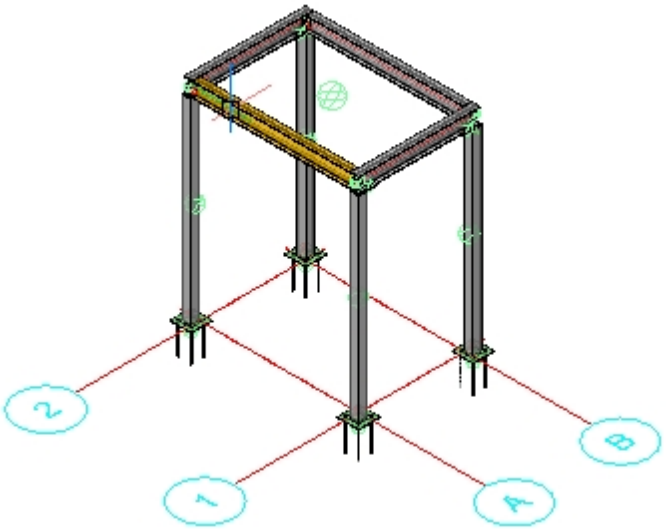
sizes.

Note that by default the bays will always be equally spaced. This can be changed by pressing the ... button next to the number of bays value.

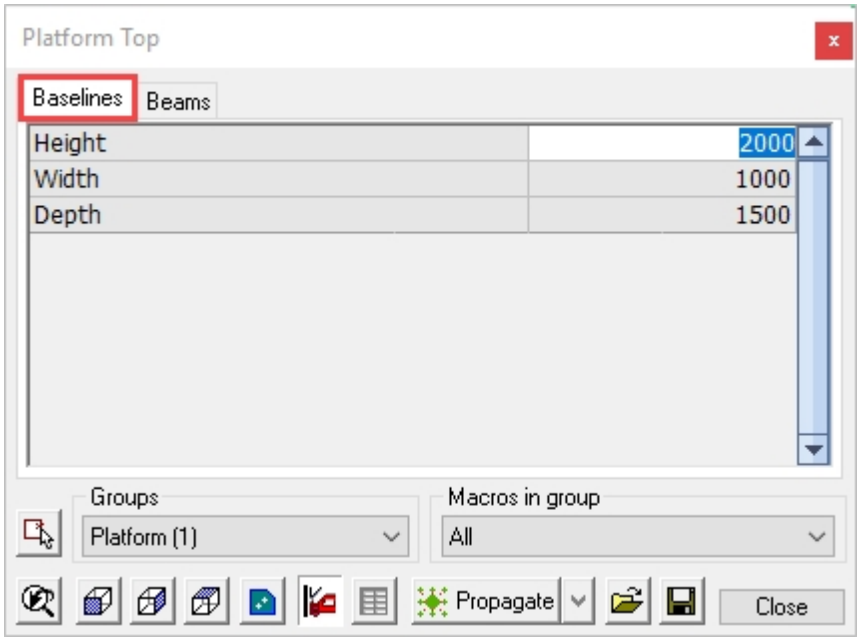
Sheeting rails and roof purlins may be added by activating the Cladding command

You can revisit any of the macros by activating the Review Macro command and selecting the individual macro spheres.

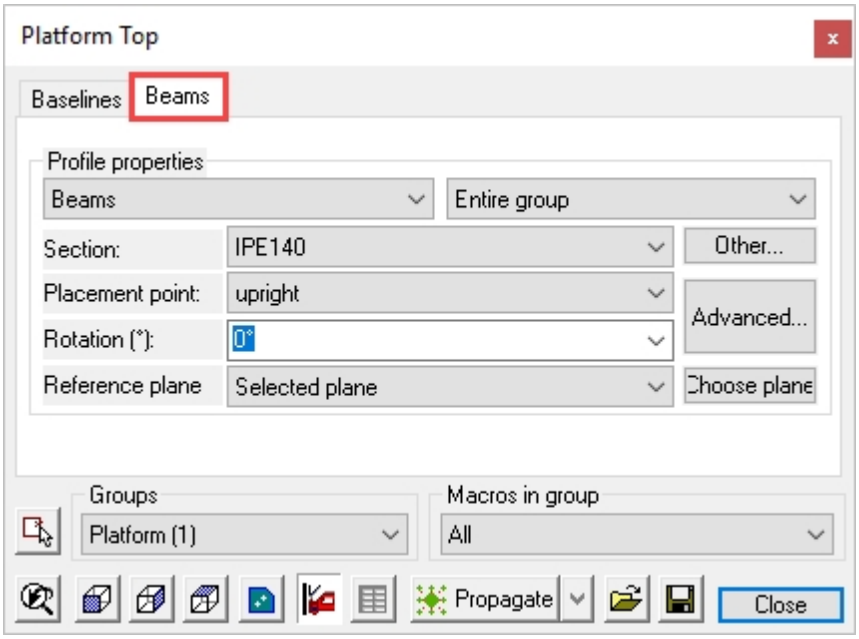
Platform



The customization dialog of this macro looks like this :



This tab enables you to edit the overall parameters of the platform, Height, Width, Depth



This tab enables you to edit the beam profiles and their placement

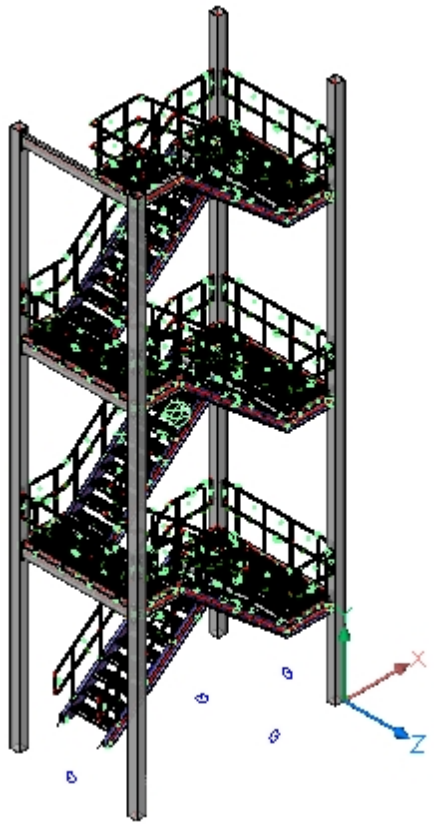
You can revisit any of the macros by activating the Review Macro command and selecting the individual macro spheres

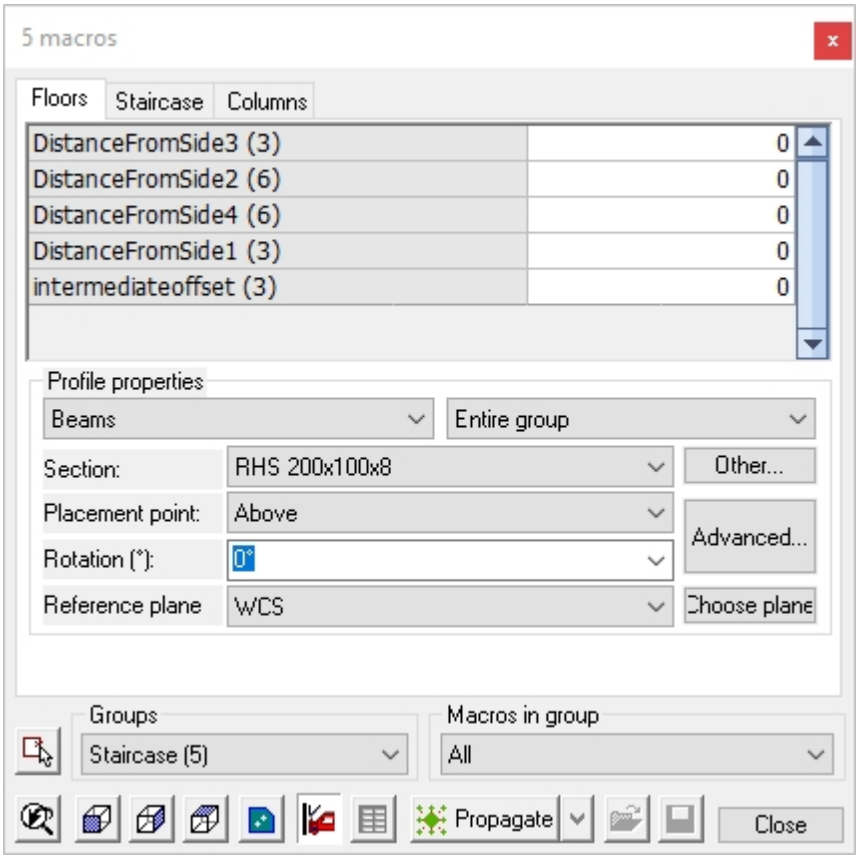
Staircase Cranked Stringers - Stair tower

This tool draws a freestanding stair tower.

There are 3 tabs to the dialog, enabling you to edit the **Floors**, **Staircase**, and **Columns**

Selecting the Review Macro command, and selecting any of the macro spheres, you are able to individually edit the connections, stair treads, and HandRailing to suit requirements





The starting height, X and Y starting position, width, and depth of the staircase are all determined by the blue planes a the bottom of the structure.

When any of these are moved, the staircase will adapt to this change.

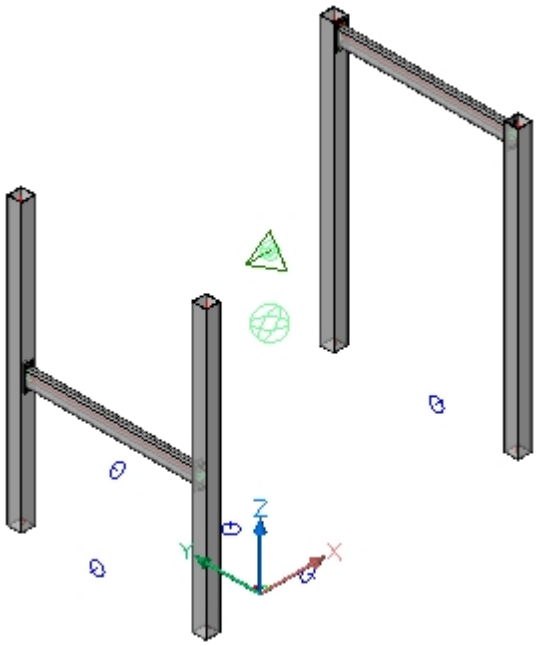
Staircase - Floors only

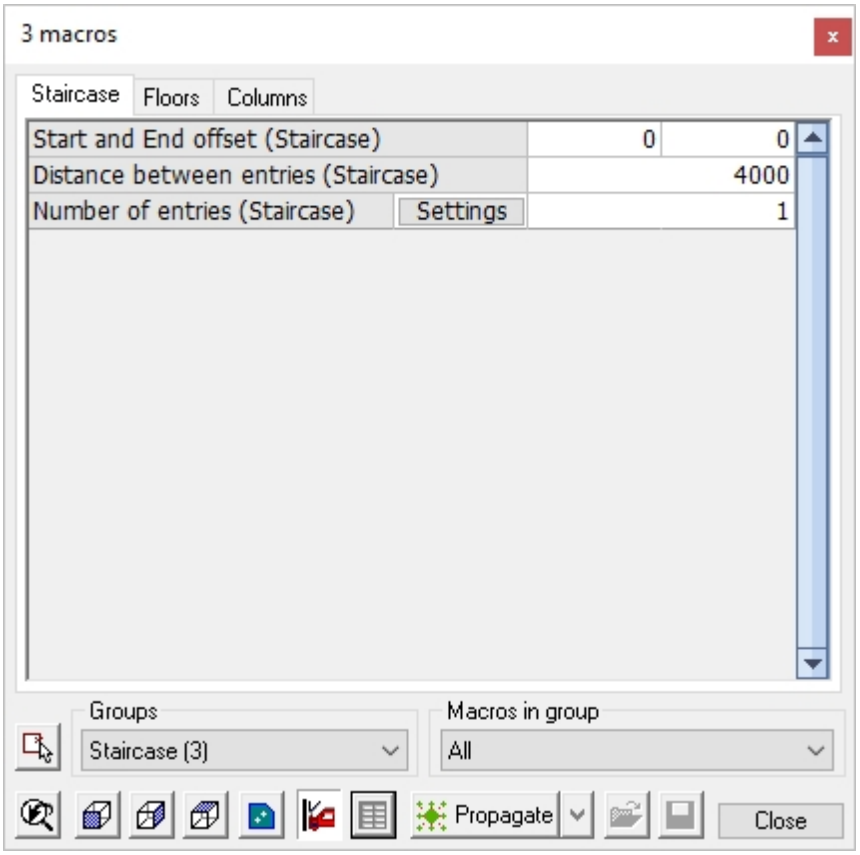
This tool draws a freestanding stair tower structure without the stair.

There are 3 tabs to the dialog, enabling you to edit the **Floors**, **Staircase**, and **Columns**

Selecting the Review Macro command, and selecting any of the macro spheres, you are able to individually edit the columns and beams to suit requirements

This command will allow the user to install a custom stair with parameters outside of the Staircase-Cranked Stringers





The starting height, X and Y starting position, width, and depth of the structure are all determined by the blue planes a the bottom of the structure.

When any of these are moved, the structure will adapt to this change.

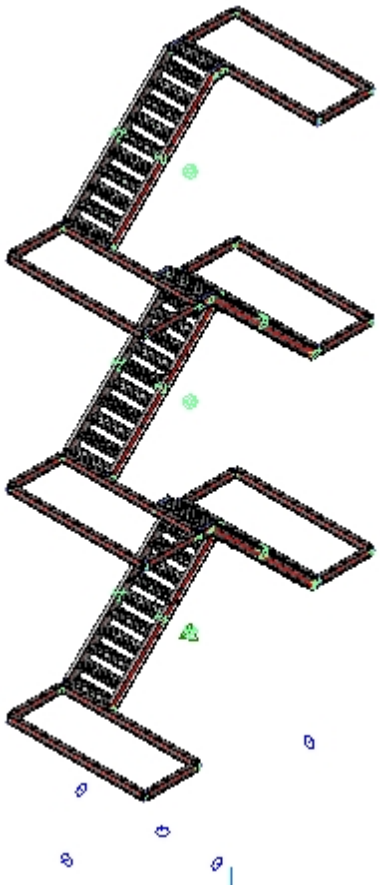
Staircase with UB platforms

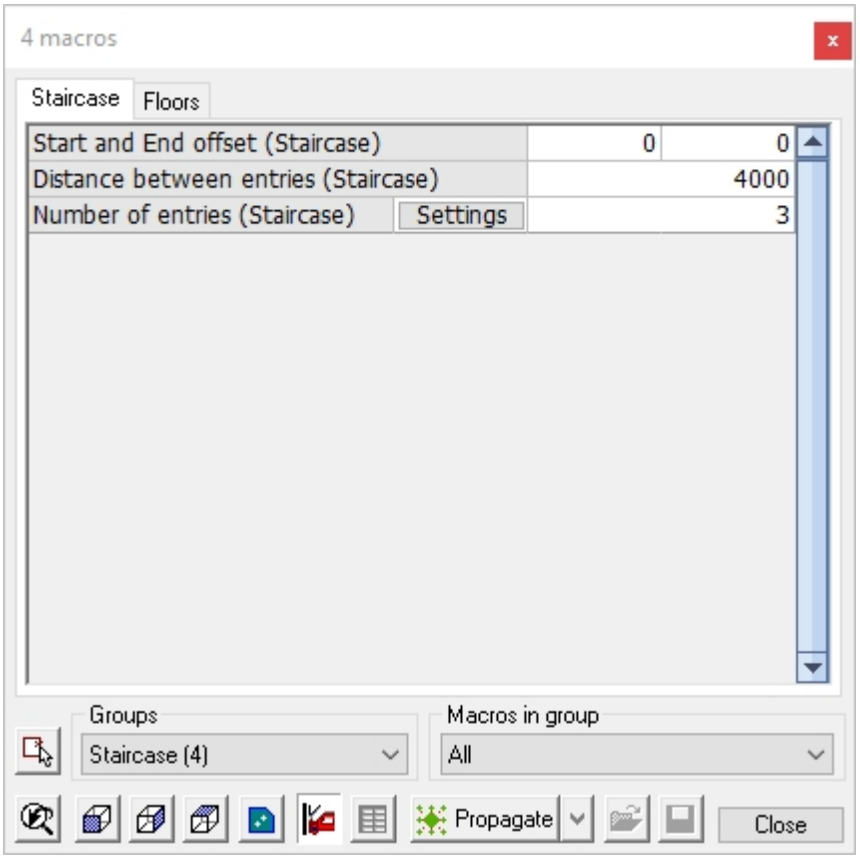
This tool draws a freestanding staircase without any supporting columns.

There are 3 tabs to the dialog, enabling you to edit the **Floors**, **Staircase**, and **Columns**

Selecting the Review Macro command, and selecting any of the macro spheres, you are able to individually edit the connections, stair treads, and HandRailing to suit requirements

This command will allow the user to install a custom stair with parameters outside of the Staircase-Cranked Stringers





The starting height, X and Y starting position, width, and depth of the staircase are all determined by the blue planes a the bottom of the structure.

When any of these are moved, the staircase will adapt to this change.

Stair

Command - **PrB_Stair**

For more information about this tool, see the Stairs chapter.

Railing

Command - **PrB_Railing**

For more information about this tool, see the Railings chapter.

Ladder

Command - **PrB_Ladder**

For more information about this tool, see the Ladder chapter.

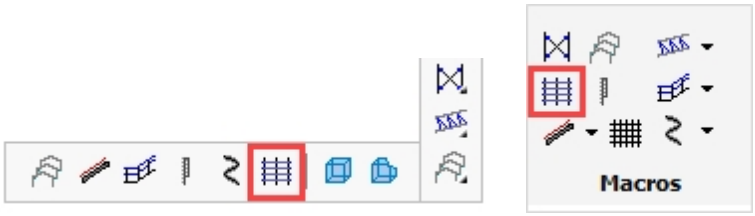
Spiral Stair

Command -

For more information about this tool, see the Spiral Stair chapter.

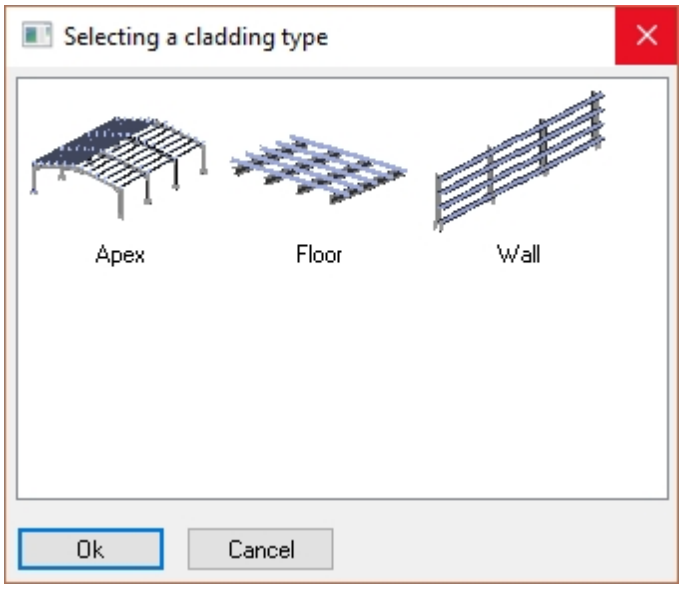
Cladding

Command - **PrB_Cladding**



Activating this command will open the **Select a cladding type** dialog - select one of the options and press **Ok**

In this manual topic, we will explore the 3 options: Apex, Floor, and Wall

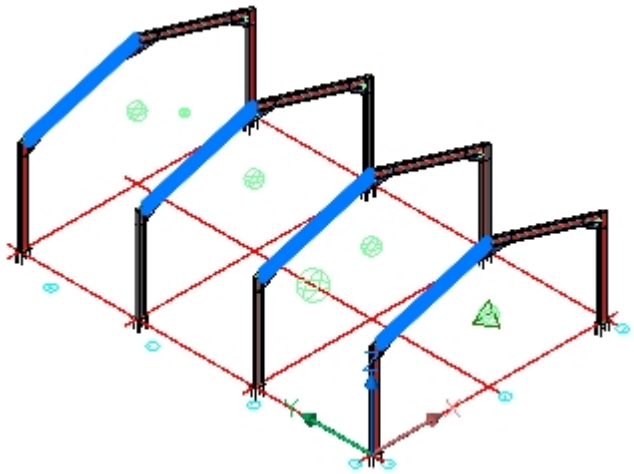


Apex

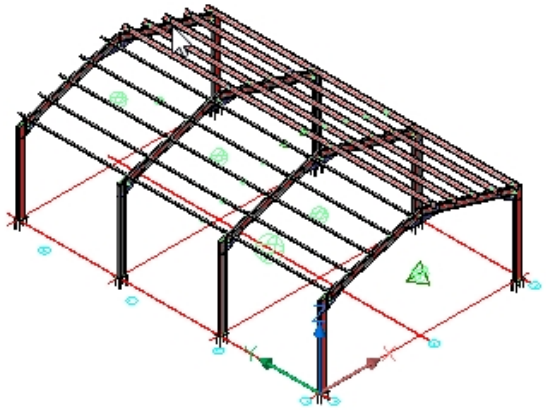
This tool will place roof purlins.

After confirming the **Apex** configuration, select the first row of roof beams on one side of the apex.

Then, select the next row of beams on the other side of the apex.



Select the roof beams on one side of the Apex

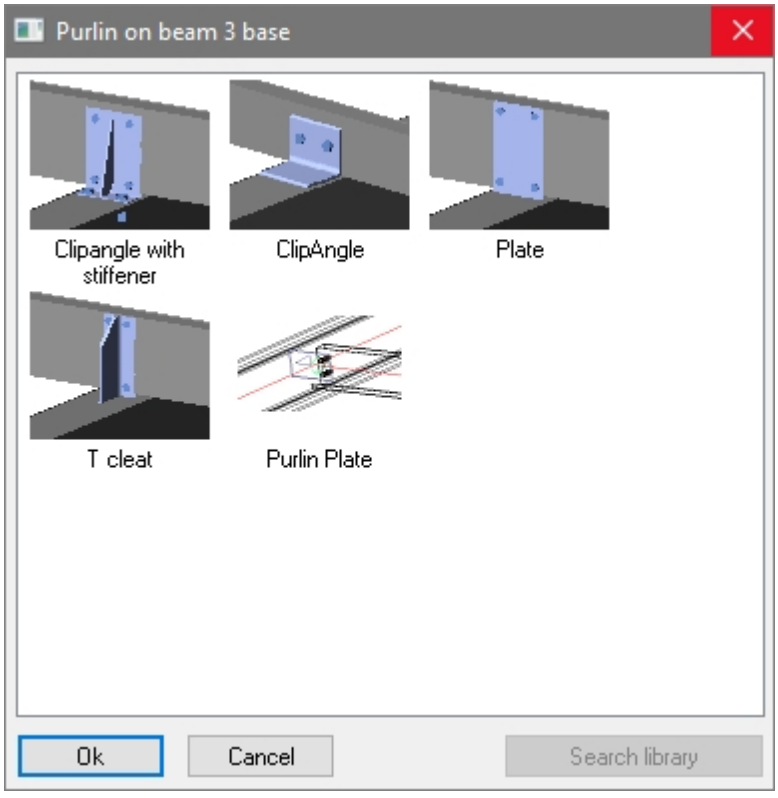


Purlins after selecting Connections

From the connection selection dialog, select the appropriate purlin to roof-beam connection.

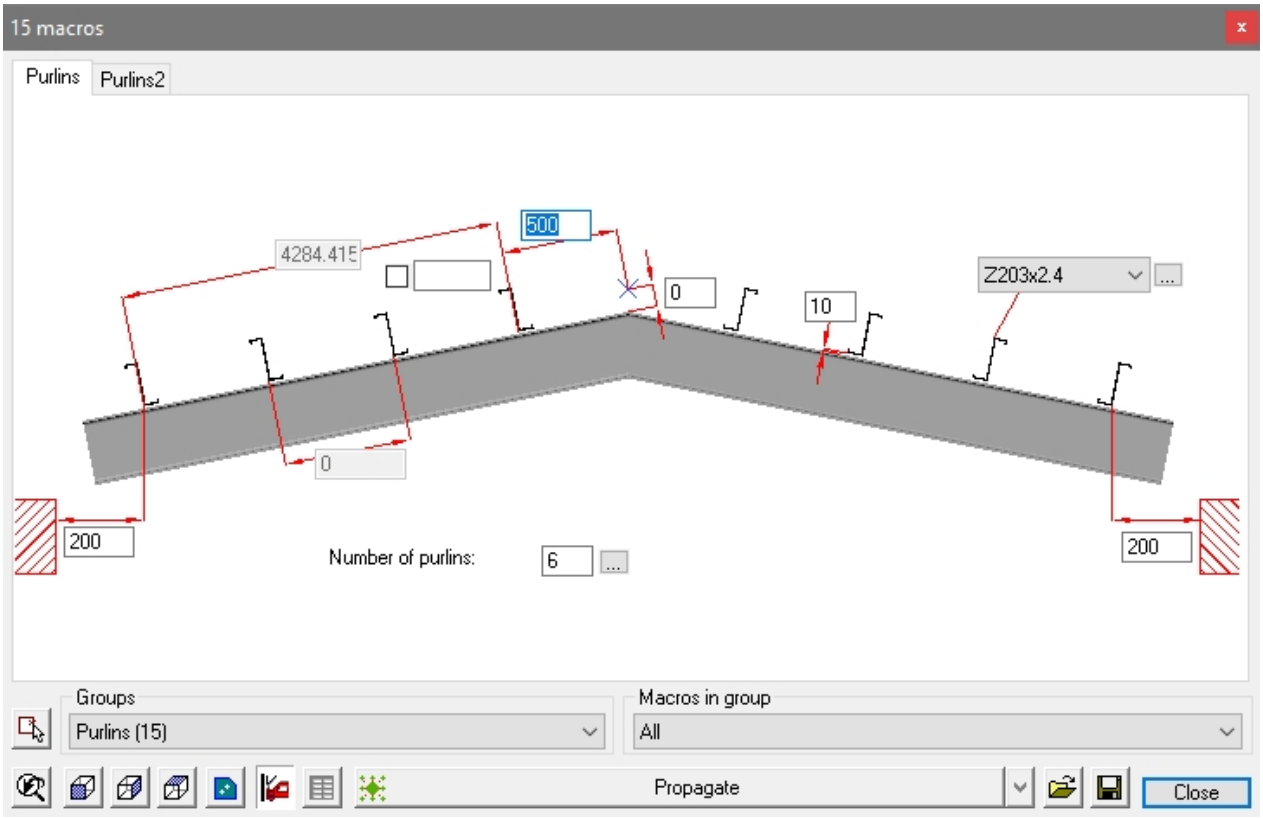
Where the purlins are interrupted, a second dialog will appear prompting you to select the bracket at the interrupted joint.

You may skip these connections by pressing the cancel button.



The purlins are drawn complete with the selected brackets accompanied by the edit dialog where you can change all the parameters of the purlins.

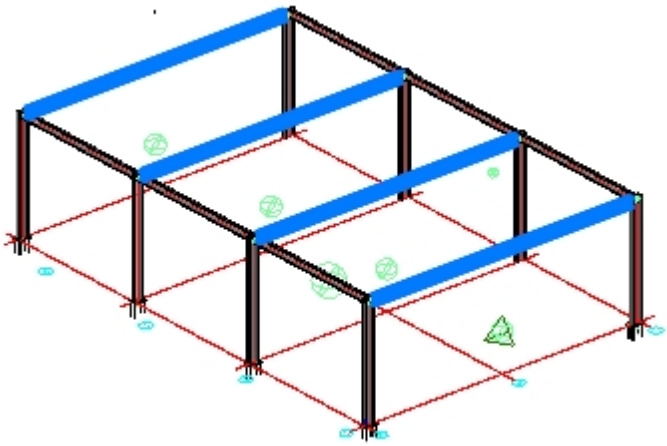
Read below to learn how to Edit the new purlin brackets



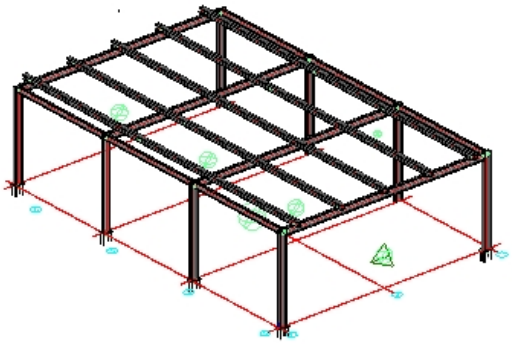
Floors

This command will place floor beams.

After choosing the **Floors** configuration, select the row of roof beams on which to support the floor.



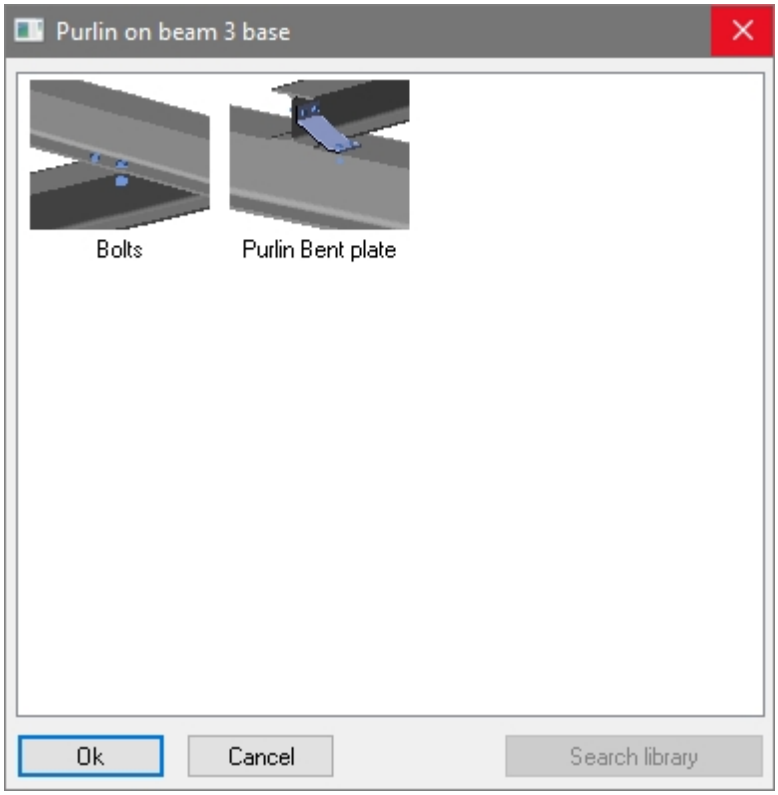
Select the supporting beams



Floor beams after selecting Connections

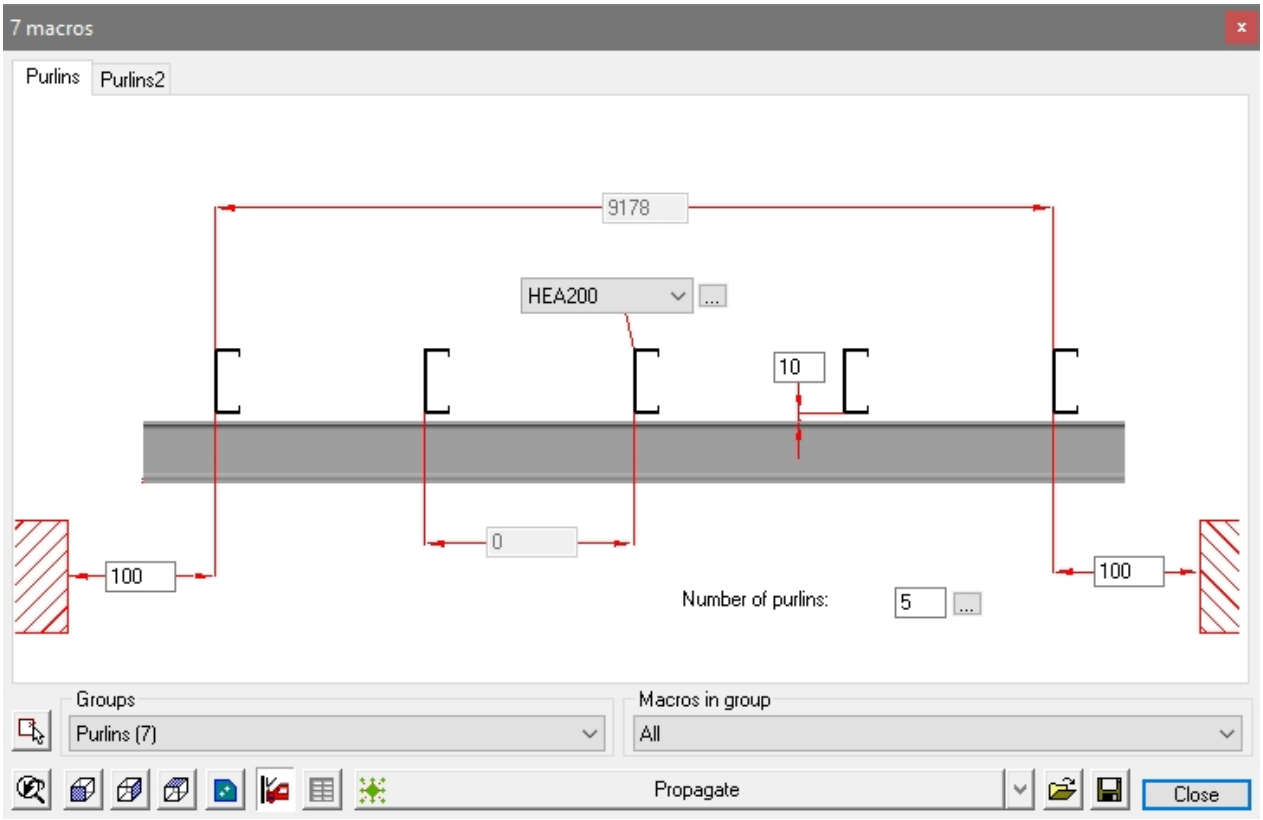
From the connection selection dialog, select the appropriate floor to beam connection.

You may skip these connections by pressing the cancel button.



The floor beams are drawn complete with the selected connections accompanied by the edit dialog where you can change all the parameters of the floor beams.
Any changes will be automatically reflected on the model.

Read below to learn how to Edit the new purlin brackets

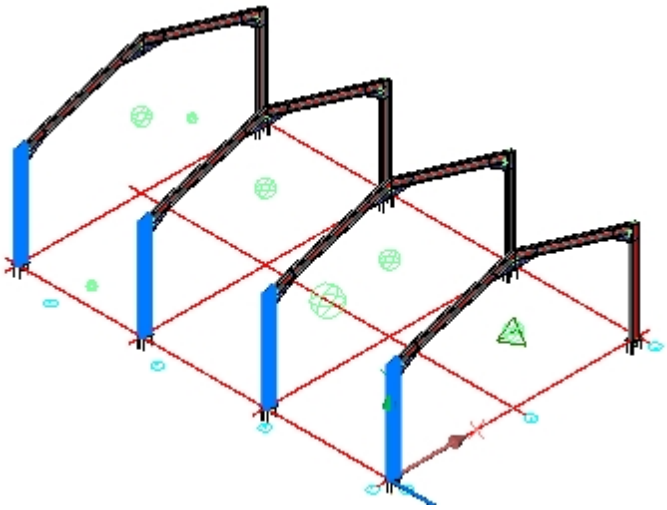


Wall

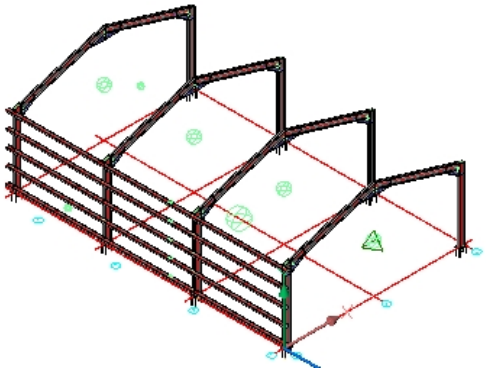
This command will place **Side Cladding** or **Girts.**

After selecting the **Wall** configuration, select the row of columns on which to place the side cladding girts.

At the prompt, select the side of the columns to which they are to be placed.



Select the Columns

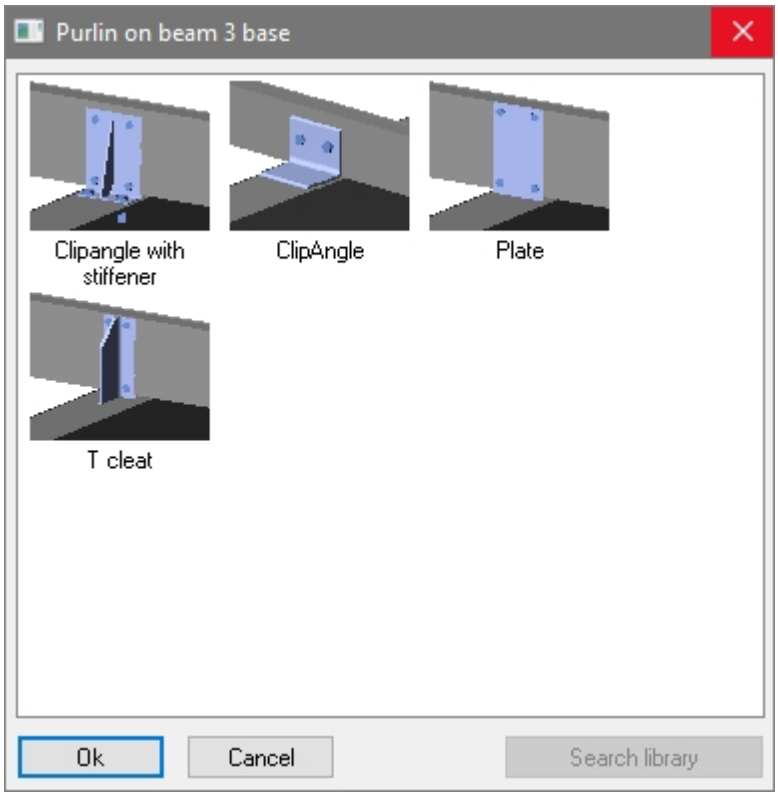


Side girts after selecting direction and Connections

From the connection selection dialog, select the appropriate side girt connection.

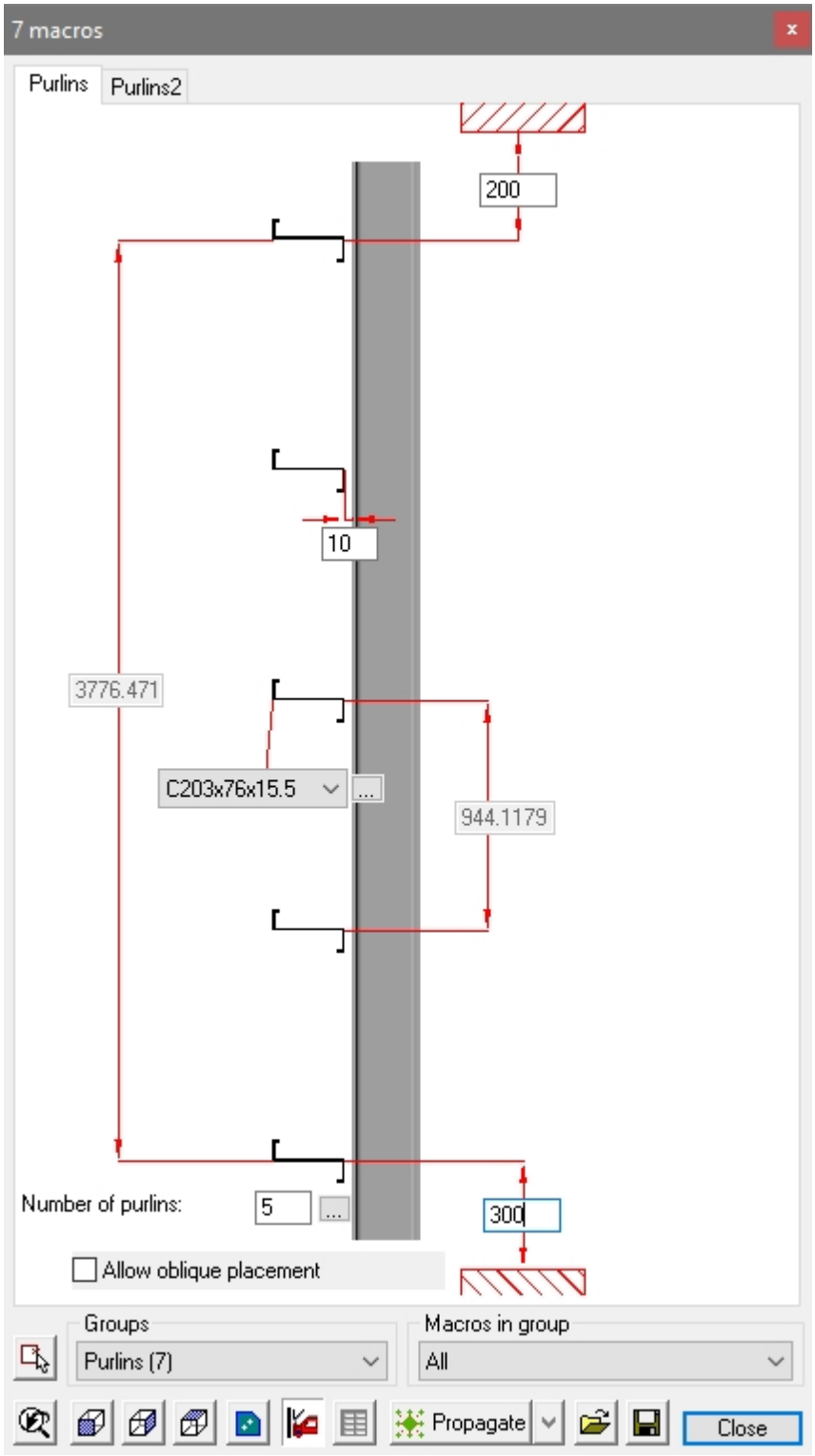
Where the girts are interrupted, a second dialog will appear prompting you to select the bracket at the interrupted joint

You may skip these connections by pressing the cancel button.

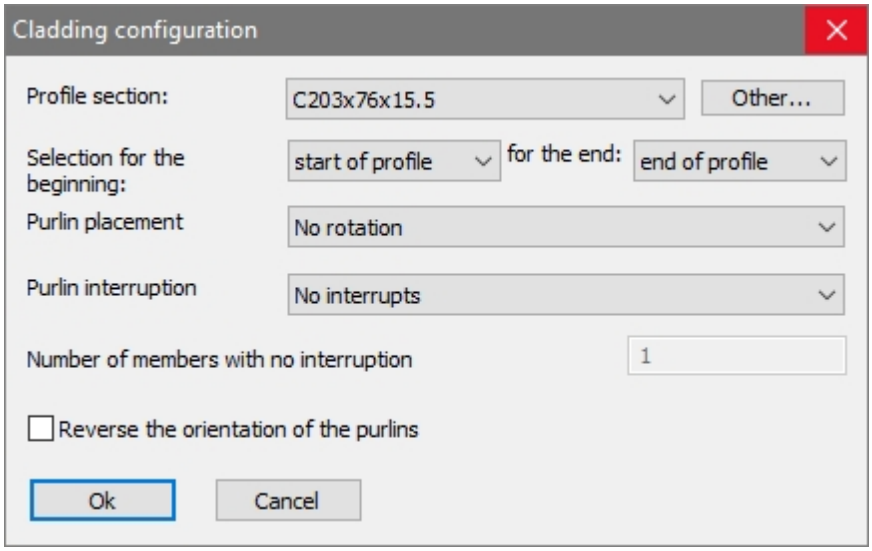


The side girts are drawn complete with the selected connections accompanied by the edit dialog where you can change all the parameters of the girts.
Any changes will be automatically reflected on the model.

Read below to learn how to Edit the new purlin brackets



The Cladding configuration dialog



All of the options in this dialog box are explained here :

Profile Section - The profile shape and size that needs to be used for the new purlins/girts

Selection for the Beginning / **End** - This will set the position of the first purlin. For the roof purlins this would be the eaves of the structure, for wall girts this could be the ground floor or a concrete slab.

From the drop-down menu, there are the following options:

> **To Plane** - Will allow you to select any column face of any profile or plate
>
> **To Grid line** - Will allow you to select any grid line.
>
> **To Beam** - Will allow you to select any beam.
>
> **To Column** - Will allow you to select any beam. Use this for floor or apex configurations only
>
> **Start of Profile** - This will measure from the start of the base profiles. This will be the selected columns in case of the wall cladding, or the selected beams in case of Floor or Apex configurations.
>
> **End of Profile** - This will measure from the end of the base profiles. This will be the selected columns in case of the wall cladding, or the selected beams in case of Floor or Apex configurations.

Purlin placement - This will set the orientation of the purlin relative to the base profiles. From the drop down menus, there are the following options:

> **No Rotation** - Accept the default rotation
>
> **Rotate follow-up row of purlins** - Alternate every second purlin rotation
>
> **Rotate for Zed purlins with overlap** - Zed purlins with an overlap will need a rotation for each instance

Purlin interruption - This will set the purlin span. From the drop down menus, there are the following options:

> **Interrupt at every location** - Span the purlins between each base profile
>
> **Draw the purlins between the base members** - This will draw the purlins flush : The top of the purlins will be coincident with the top of the base profiles. This also means that the purlins will be interrupted at each intersection with the base profiles.

No interrupts - This will span the purlins over the base profiles without any breaks

Skip interruption over some members - Do not break the purlins over a specified amount of base profiles.

Skip interruption over some members, Alternating - Same as above, but the breaks on the followup row of purlins will be alternated. If you use this then the interruption will not be placed on the same base profiles for all the purlins.

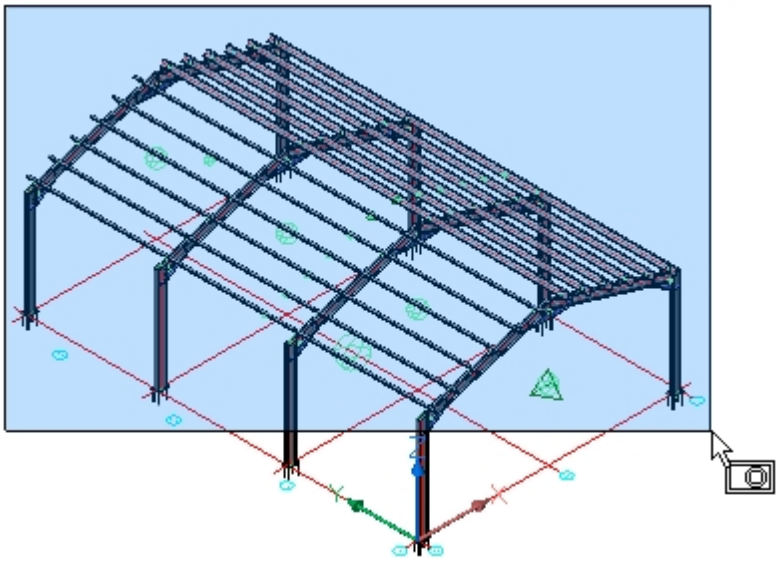
Number of members with no interruption - This option is to be used in combination with the above *Skip interruption over some members* options

Reverse the orientation of the purlins - This switches the start/end orientation of all the purlins

Edit the Purlin Brackets

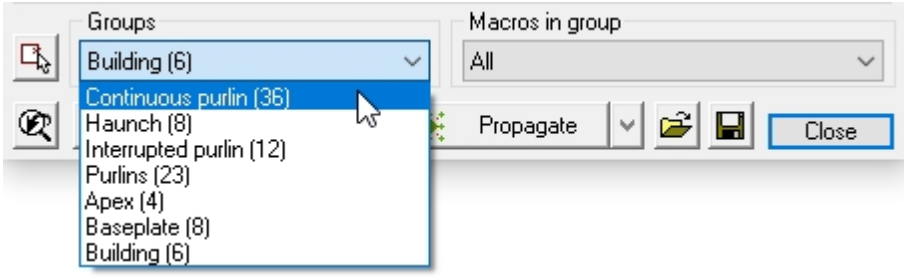
There are likely to be numerous brackets and to edit them one at a time would take too long. It is however possible to edit all the brackets simultaneously.

Click on the Review macro icon and block select the entire roof structure.



This will open the macro edit dialog. At the bottom of the dialog, open the **Groups** drop-down menu and select **Continuous purlin** (the number in brackets is the number of macros included in the selection).
This will automatically open the relevant edit dialog.

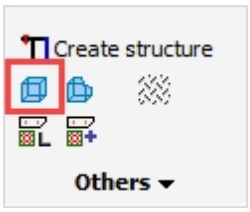
Note! If the option for **Interrupted purlin** was chosen, it will be necessary to edit the brackets separately.



This method will work for any connection type or group.

Draw a Body

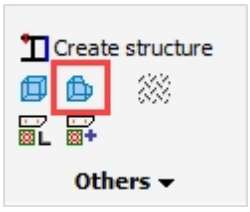
Command - **PrB_CreateBodyDlg**



For more information about this tool, see the Draw a Body chapter.

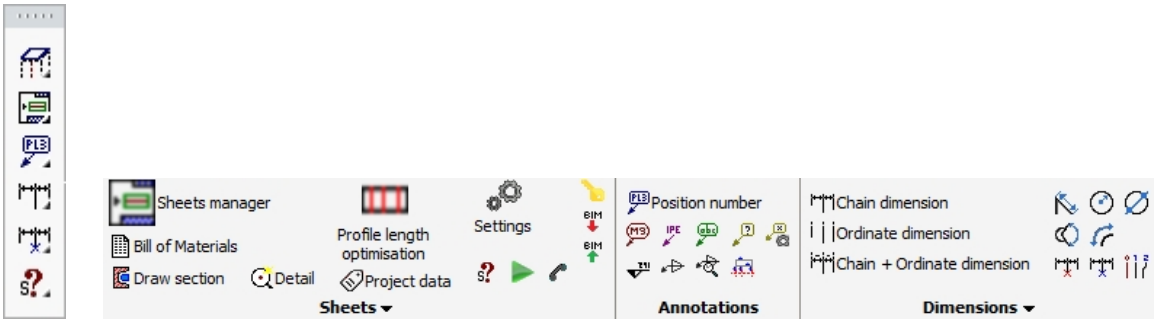
Manipulate a Body

Command - **PrB_ManipulateBody**



For more information about this tool, see the Manipulate a Body chapter.

Main - Output



This chapter is divided into two: The first topic covers welding as applied to the 3D model and includes a number of sub-categories listed separately under it's heading Welding / Numbering

The remaining topics apply to the preparation and editing of 2D drawings and their respective bills-of-material and schedules, including:

- Welding Numbering
- Sheets Manager
- Tools for 2D Drawings
- Dimensioning
- Modifying Dimensions
- Parabuild General

Welding / Numbering

- Assembly / Object selection
- Weld
- Disconnect weld
- Set as end-plate
- Switch profile triangle direction
- Restore group selections
- Numbering / Revisions
- Edit weld symbol

Assembly / Object Selection

Command - **PrB_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 group always stays intact, i.e. when you turn assembly selection back on all components of that assembly will 'stick together again'.

To make things clear: assembly selection only changes 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.

Weld

Command - **PrB_Attach**

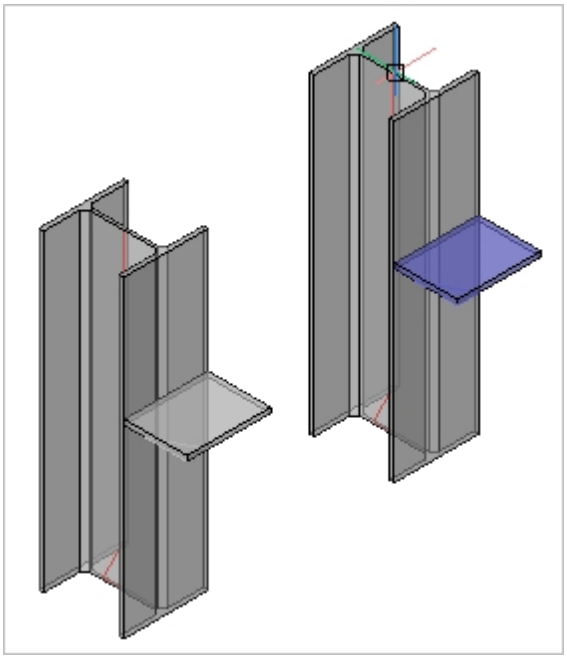
This tool allows you to weld one or more parts to a main part.

The welded sub parts will receive a blue or green color by default (if the visibility mode *Sub parts* is activated).
This allows you to see immediately which parts are welded.

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

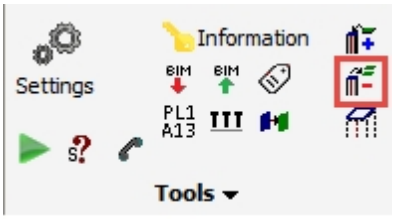
This means that if you select one part then all parts of that assembly will automatically be selected.
This behavior can be disabled (so you can switch between part selection and assembly selection).
To learn more about this see Assembly/Part selection switch

Note: A plate cannot be used as a main part to weld other parts to it. Only profiles can be main parts.
In cases where you really need to weld against a plate (for example the assembly is composed of plates only), then you can convert 1 plate to a strip and use the strip as main part.



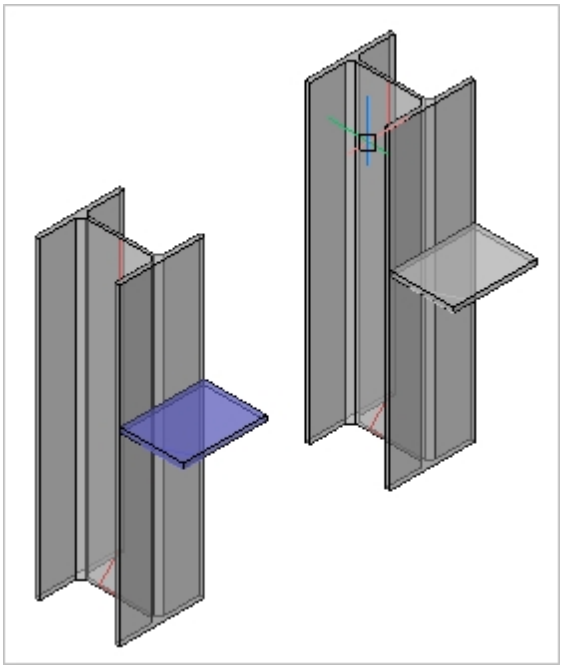
Disconnect Weld

Command - **PrB_Detach**



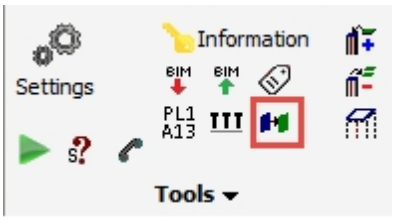
This tool allows you to disconnect a weld of one or more parts from a main part.

The disconnected sub parts will revert to their default color (if the visibility mode *Sub parts* is activated)..
This allows you to see immediately which parts are not welded.



Set as End-Plate

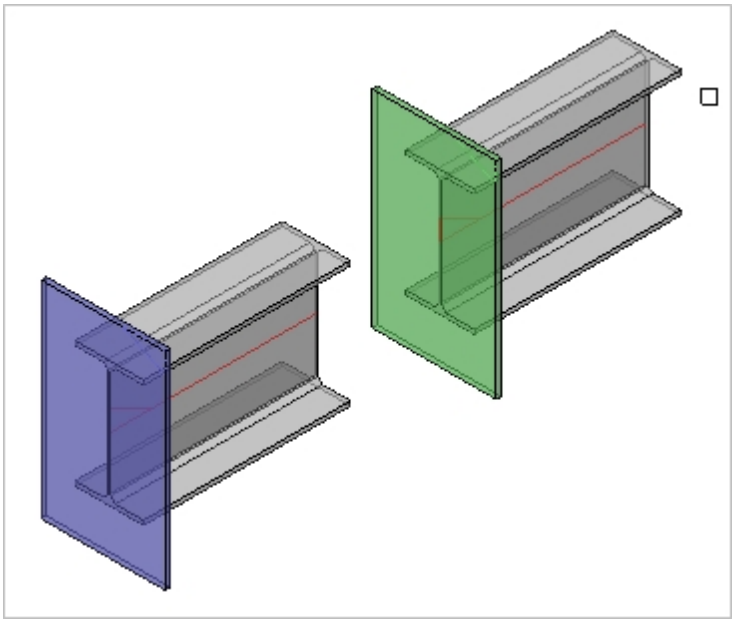
Command - **PrB_SetTracPlate**



With this command you can convert an ordinary welded (blue) plate to an End / Base-Plate (green).

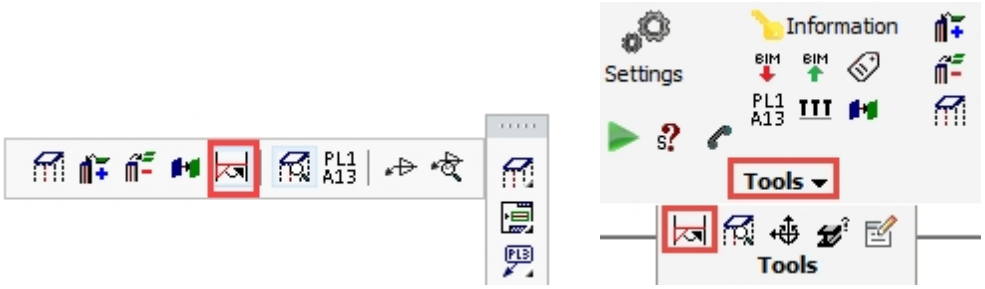
The only difference between the two is that the end (green) plate gets an additional front view in the assembly workshop drawing.

The properties panel of the plate has a property that does the exact same thing as this command.



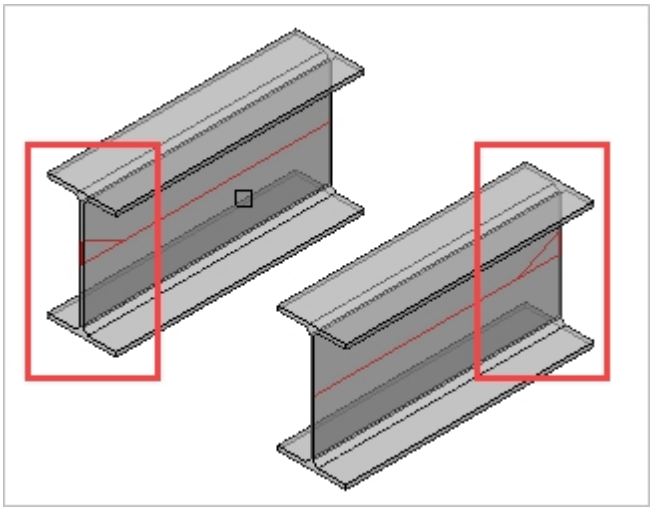
Switch Profile Triangle Direction

Command - **PrB_SwitchProfEcs**



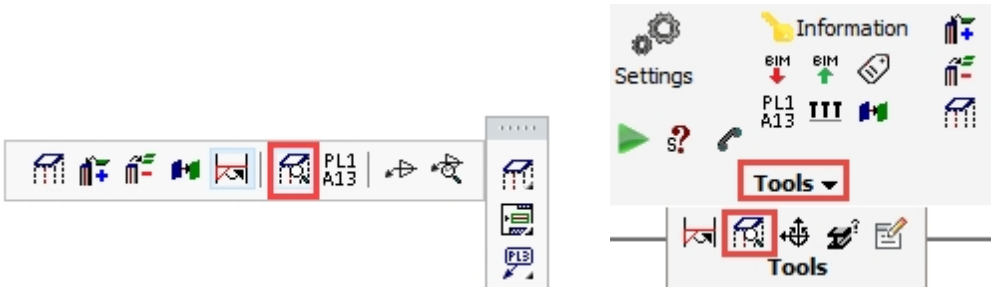
This tool allows you to switch the location of the triangle of a profile.

The triangle of a profile determines the profile's orientation on the workshop drawings, at least if the workshop drawings are set up to use the profile's direction for the views.



Restore Group Selections

Command - **PrB_ResetGroups**

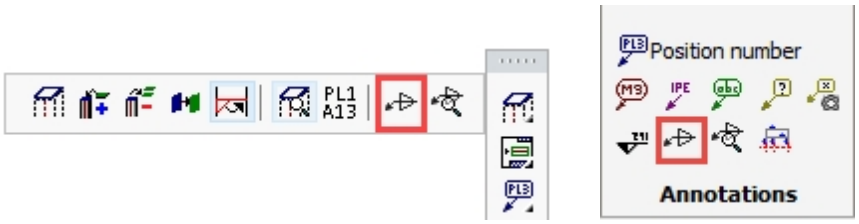


This tool rebuilds all the groups in the 3D drawing, based on the weld data that Parabuild has stored for all the parts.

This tool can be used if the selection groups are mismatching the actual welds.
Parabuild always manages the selection groups so that they perfectly match with the actual weld data.
But the selection groups could become mismatched after a corrupted 3D drawing was repaired.

Weld Symbol

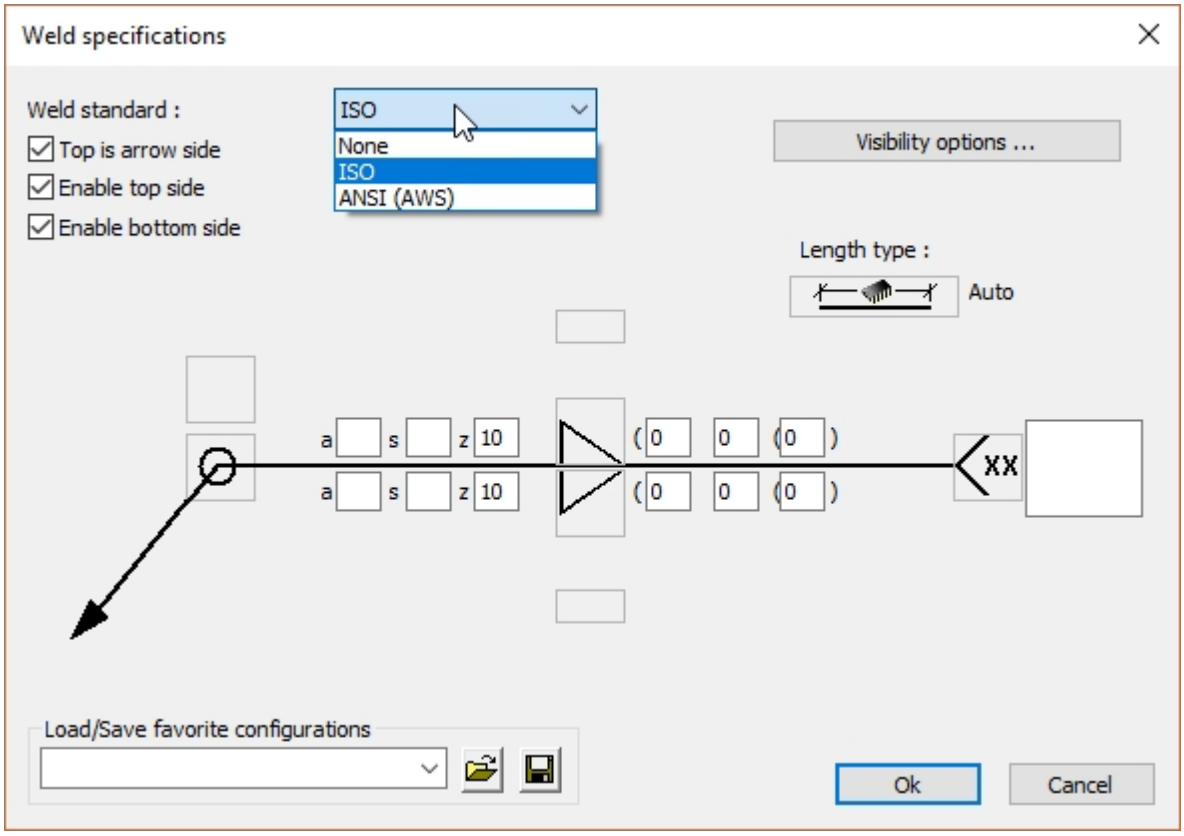
Command - **PrB_DrawWeldTag**



Activating the Weld Symbol command will open the weld symbol dialog.
There are 3 standards available:

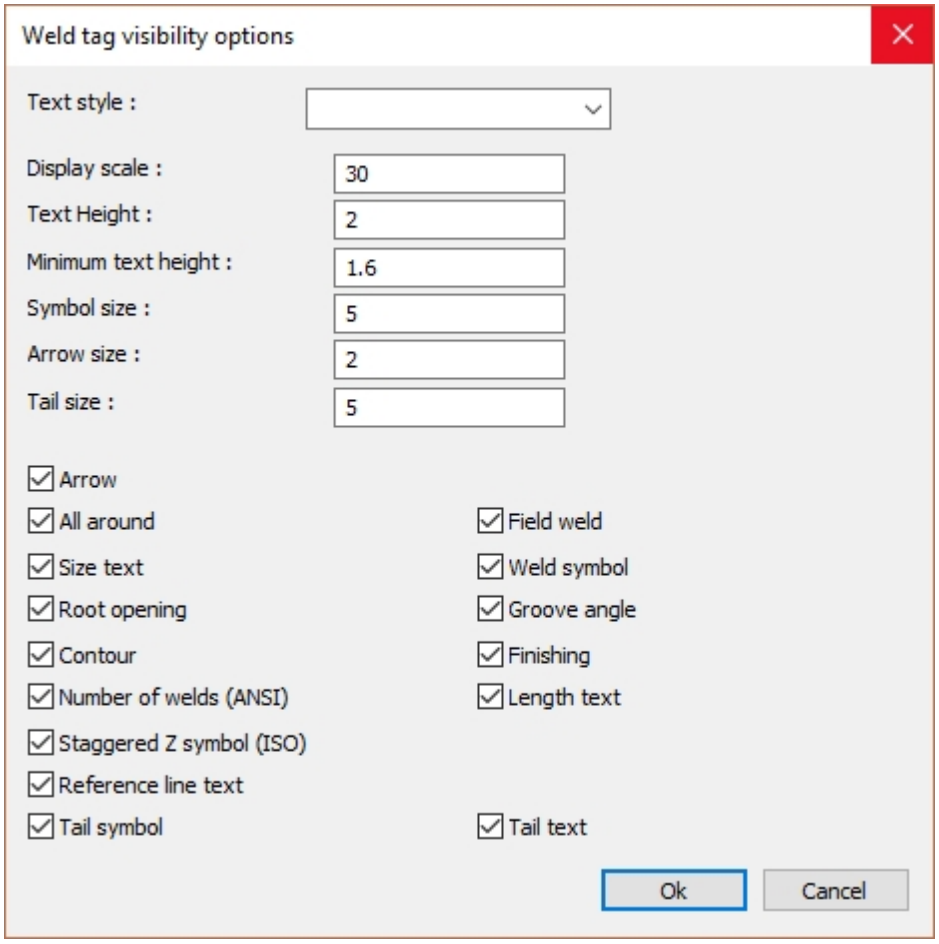
1. None
2. ISO (International Standards Organization)
3. ANSI AWS (American Welding Society)

Though the systems differ, there are many commonalities in the dialog, which will be explained here:



Visibility options

Selecting this button will allow you to customize how the weld symbol will look. The dialog is self-explanatory :

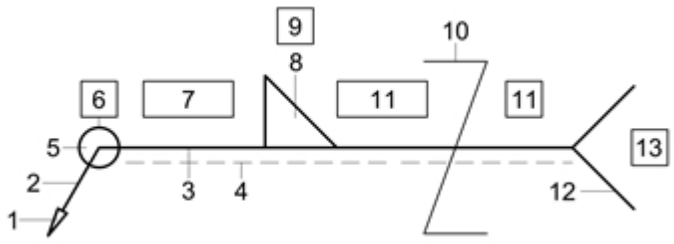


For more about the 2 weld standards, see ISO Weld Symbol and AWS Weld Symbol.

ISO Weld Symbol

The Basic ISO Weld Symbol

The diagram below illustrates the various elements making up the weld symbol



1. Arrow
2. Leader
3. Reference line
4. Dashed line
5. Weld type
6. Shop or Field weld
7. Weld size * a / s / z
8. Weld symbol
9. Weld face contour

10. Intermittent staggered weld symbol **
11. Number, Length and Spacing of weld elements **
12. Tail
13. Welding process reference and Welding class

Note!

* = Fillet welds only

** = Intermittent welds only

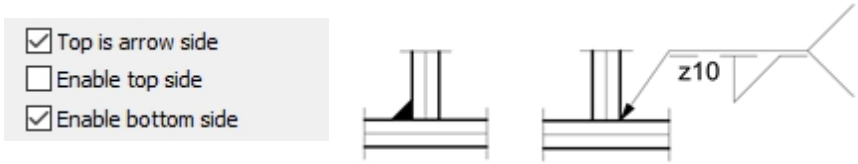
Note: Weld symbols on the full reference line relates to welds on the arrow side of the plate being welded. Weld symbols on the dashed line relates to weld on the far side of the plate. If the welds are symmetrical on both sides of the plate the dashed line is omitted. If the dashed line is below the full line then the symbol for the arrow weld is drawn above the reference line and the symbol for the far side weld is below the dashed line.

The following illustrations clarify this :

Continuous Fillet Weld - Arrow Side

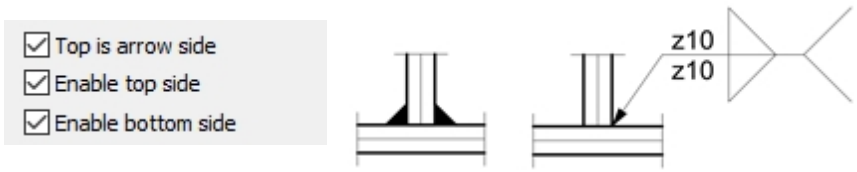
Dialog Setting *Weld* *Weld Symbol*

Continuous Fillet Weld - Far Side



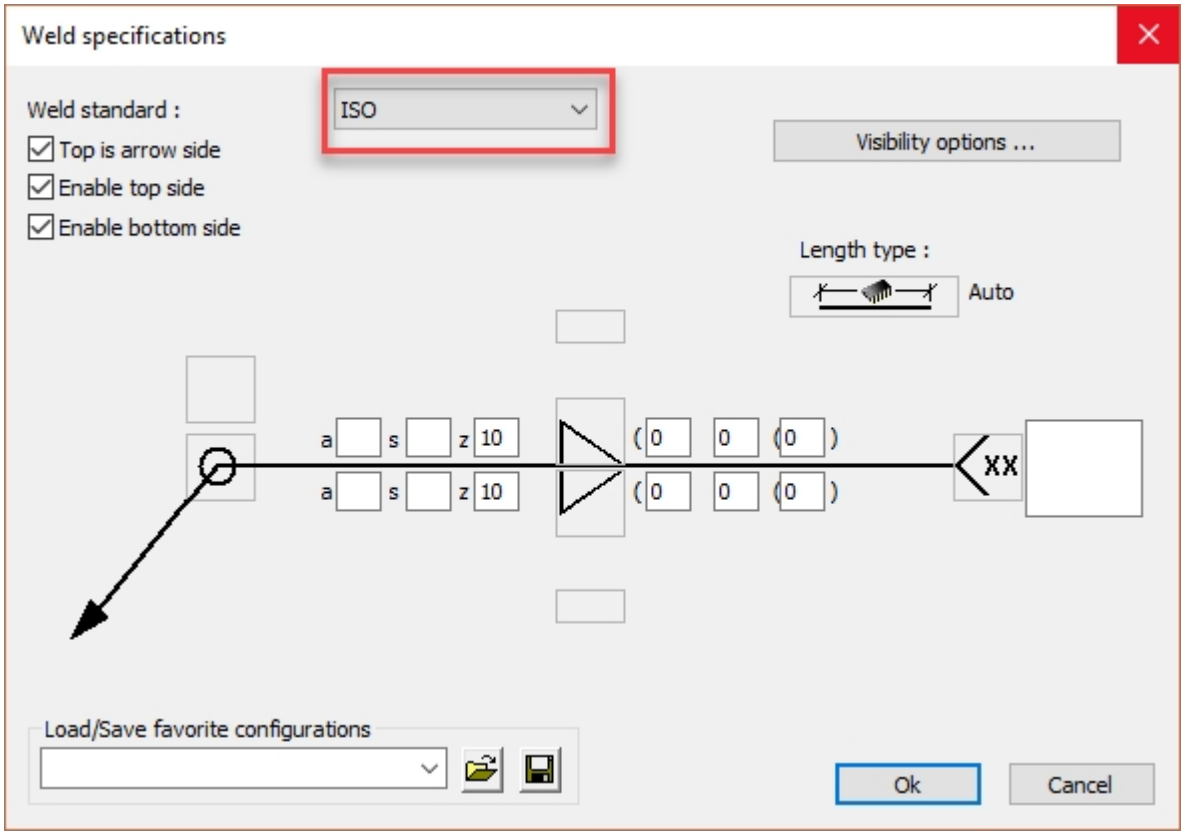
Dialog Setting *Weld* *Weld Symbol*

Continuous Fillet Weld - Both Sides

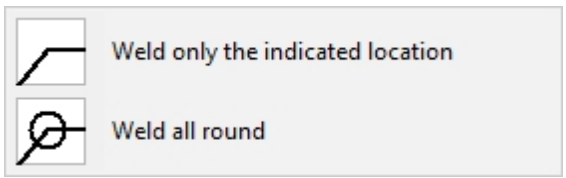


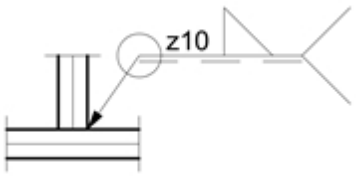
Dialog Setting *Weld* *Weld Symbol*

Editing the Parabuild ISO Weld Dialog

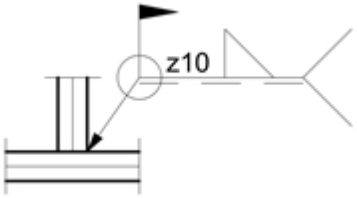


1. The **Arrow** is at the point of the mouse pointer and should be placed at the weld position
2. The **Leader** will stretch in compliance with the movement of the mouse pointer
3. The **Reference line** contains all the weld information for the weld on the near side of the plate being welded
4. The **Dashed line** contains all the weld information for the weld on the far side of the plate being welded
5. The **Weld type** is to indicate whether the weld is to be only at the indicated position or all-round - select either from the dialog



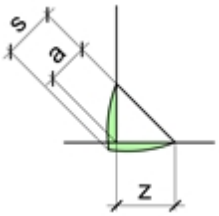


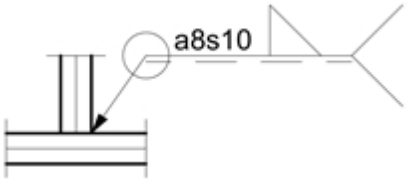
6. **Shop or Field weld** - select either from the dialog



7. **Weld size** - With the ISO system, the weld size is placed to the left of the weld symbol and is preceded by the letters: (**z**) / (**a**) / or (**z**) - depending on whether the leg length or throat thickness is to be specified. Unless otherwise instructed, it is usually the leg length that is specified, in which case only the (**z**) box needs to be added. If the throat thickness is to be specified, only the (**a**) box needs to be added.

(**s**) refers to the effective throat thickness for deep penetration welds and is placed in front of the throat thickness dimension (**a**) - therefore, if the effective throat thickness (**s**) is to be specified, both (**a**) and (**s**) boxes need to be added



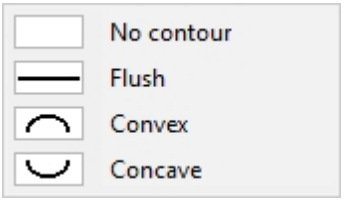


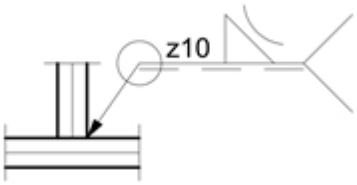
z = Leg Length

a = Throat thickness

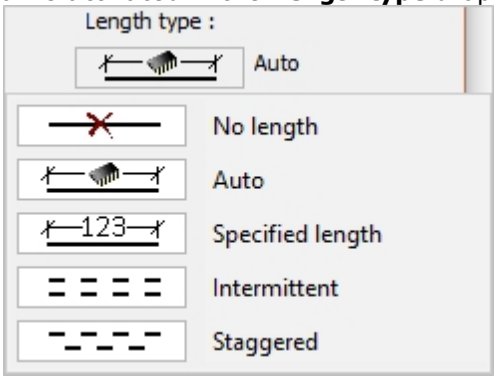
s = Effective throat thickness - for deep penetration welds

8. **Weld Symbol** - Available weld symbols may be chosen from the list
9. **Weld face contour** - select from the dialog





10. **Intermittent staggered weld symbol** - this will be added automatically when the **Staggered** option is activated in the **Length type** drop-down options menu

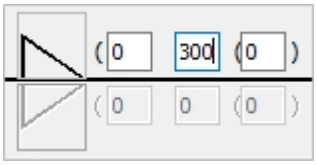


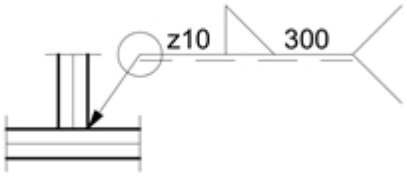
Notes on the length type drop-down

a. No length - No specified weld length

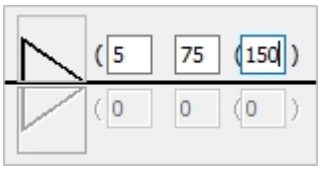
b. Auto - Weld length assumed to be full length

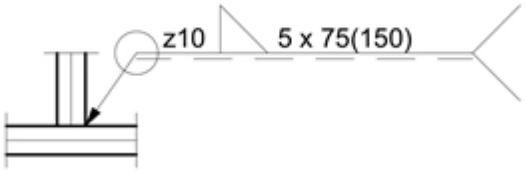
c. Specified length - enter the specified length in the weld length box (See **Number**, **Length**, and **Spacing** of weld elements)

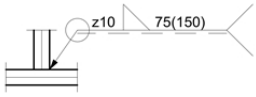




d. Intermittent - Enter the number of welds, weld length, and the spacing of the intermittent weld (See **Number**, **Length**, and **Spacing** of weld elements)
If the number of welds are either not known, or not necessary, simply leave blank, or enter a value of zero

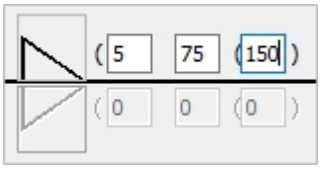


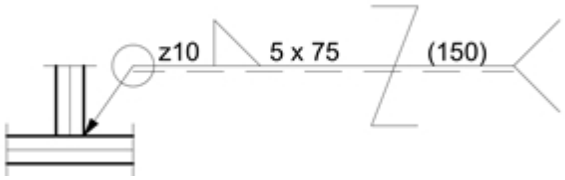


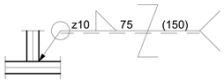


e. Staggered - As with Intermittent welds, enter the number of welds, weld length, and the spacing of the intermittent weld (See **Number**, **Length**, and **Spacing** of weld elements)

If the number of welds are either not known, or not necessary, simply leave blank, or enter a value of zero





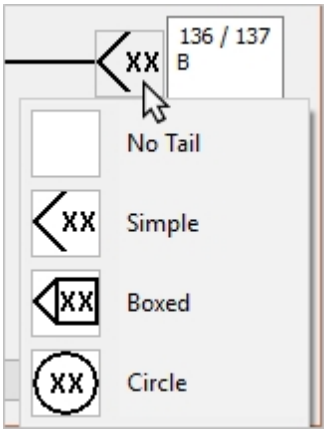


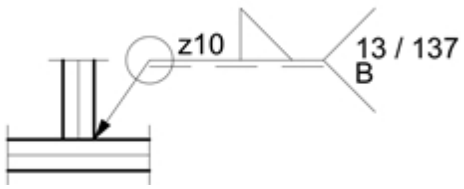
11. **Tail** - In the tail, ISO recommends placing the welding process reference and the weld class.

The welding process refers to the manner in which the weld, (e.g. Arc welding / ERW welding etc.) is to be placed and are in the form of a numeric code as specified by EN ISO 4063.

The Weld Class refers to the quality of the weld which are designated B / C / and D.

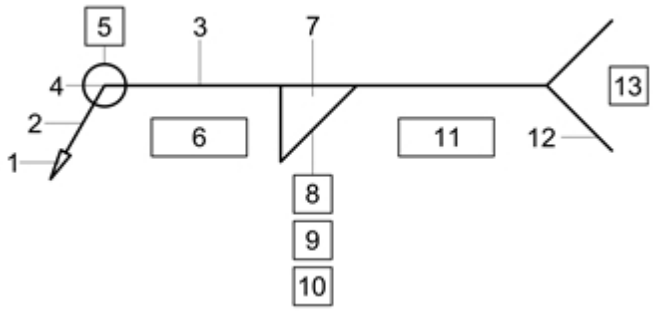
(For a list of codes and classes refer to the ISO website, or Parabuild's Practical Steel Construction web resource.





AWS Weld Symbol

The Basic AWS Weld Symbol



1. Arrow
2. Leader
3. Reference line
4. Weld type
5. Shop or Field weld
6. Weld size *
7. Weld symbol
8. Contour
9. Finish
10. Root Gap
11. Intermittent welds **
12. Tail
13. Welding process reference and Welding class

Note!

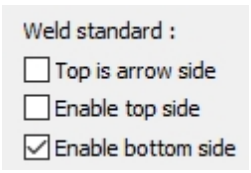
* = Fillet welds only

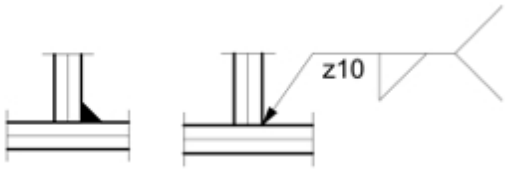
** = Intermittent welds only

Note: Weld symbols placed under the reference line relates to welds on the arrow side of the plate being welded. Weld symbols above the line relates to weld on the far side of the plate. Weld symbols placed above and below the reference line, indicate the weld is to be placed on both arrow-side and far-side of the plate.

The following illustrations clarify this :

Continuous Fillet Weld - Arrow Side



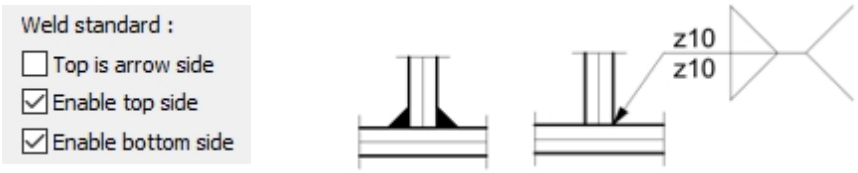


Dialog Setting *Weld* *Weld Symbol*

Continuous Fillet Weld - Far Side

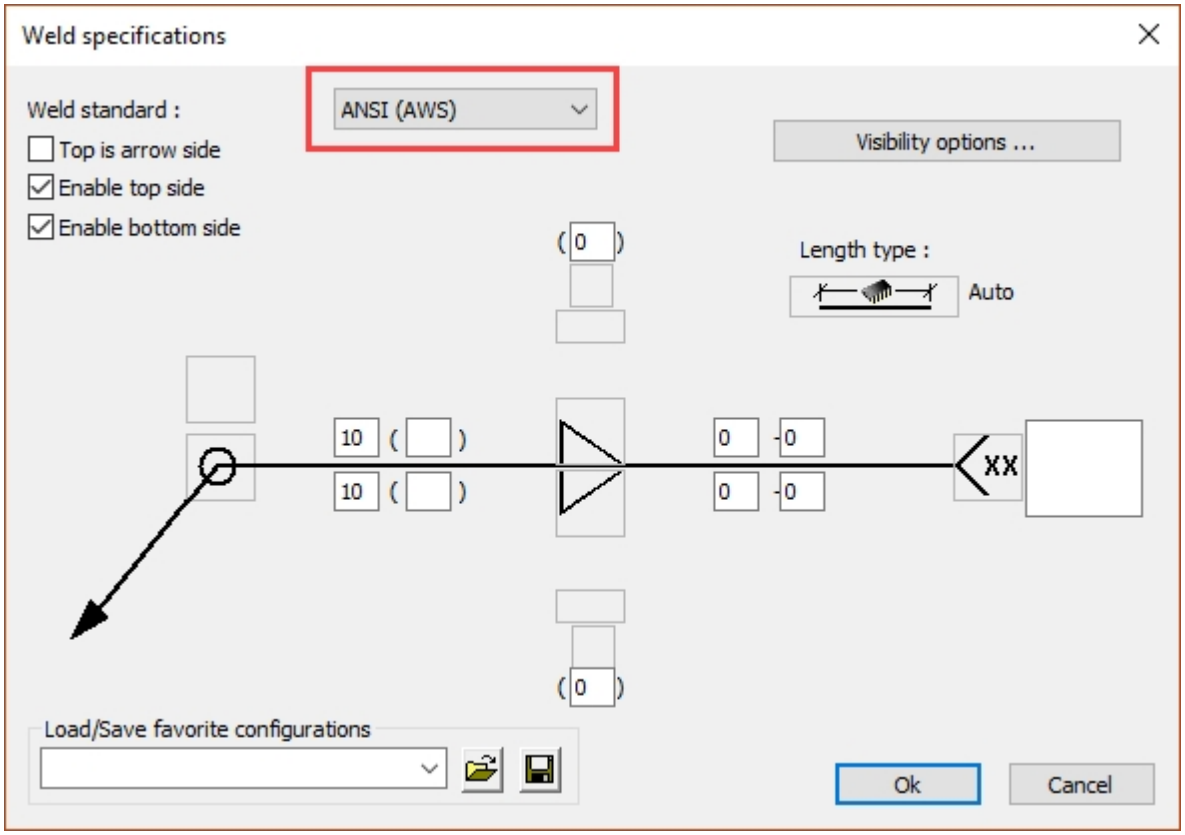
Dialog Setting *Weld* *Weld Symbol*

Continuous Fillet Weld - Both Sides

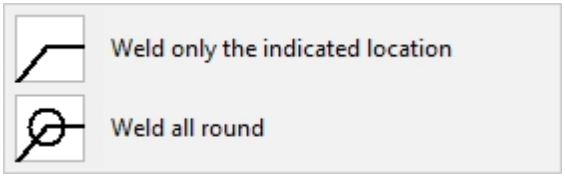


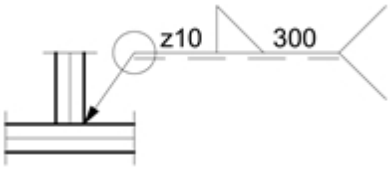
Dialog Setting *Weld* *Weld Symbol*

Editing the Parabuild ISO Weld Dialog

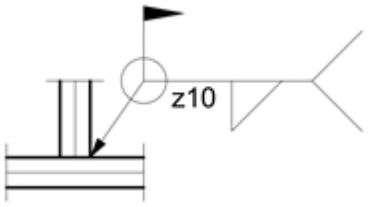


1. The **Arrow** is at the point of the mouse pointer and should be placed at the weld position
2. The **Leader** will stretch in compliance with the movement of the mouse pointer
3. The **Reference line** contains all the weld information for the weld on the near side of the plate being welded
4. The **Weld type** is to indicate whether the weld is to be only at the indicated position or all-round - select either from the dialog

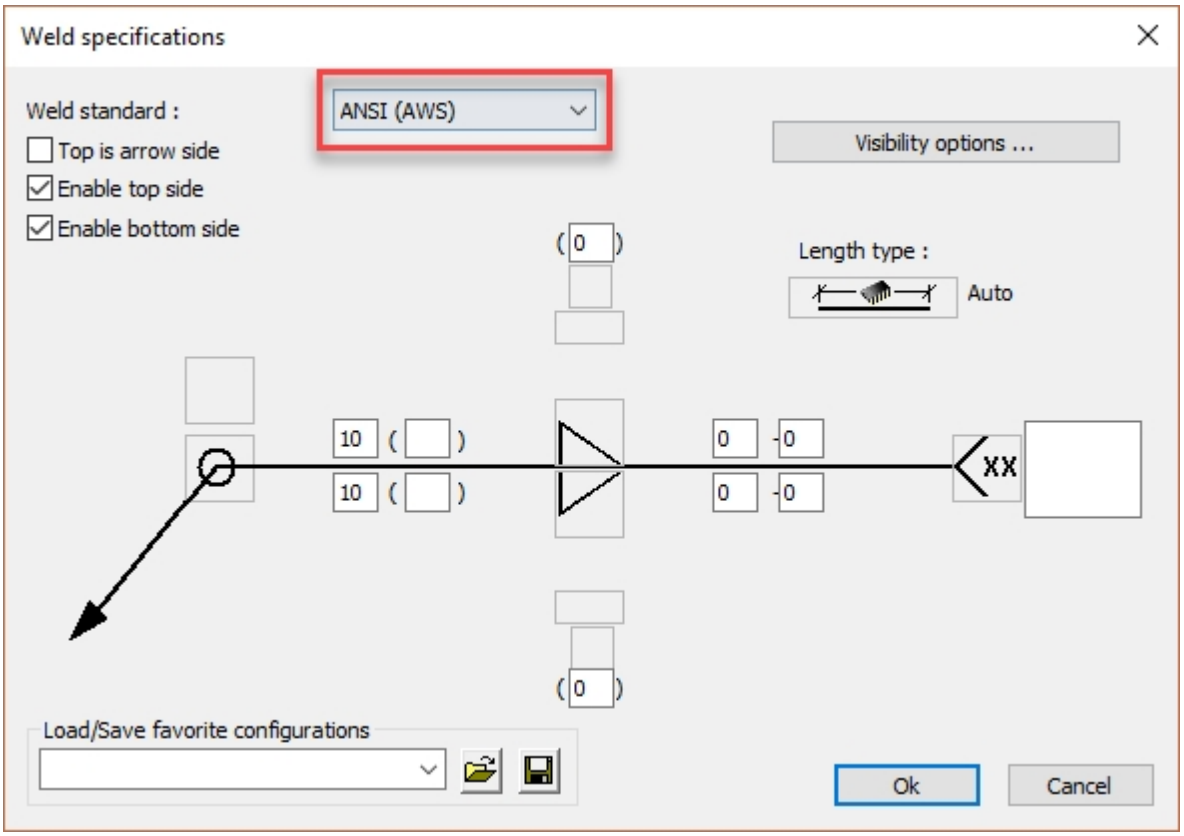




5. **Shop or Field weld** - select either from the dialog

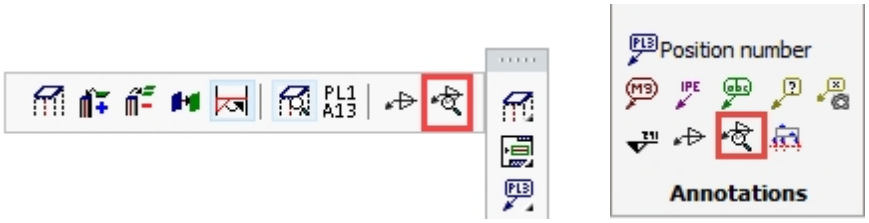


6. **Weld size** - With the ISO system, the weld size is placed to the left of the weld symbol and is preceded by the letters: (**z**) / (**a**) / or (**z**) - depending on whether the leg length or throat thickness is to be specified. Unless otherwise instructed, it is usually the leg length that is specified, in which case only the (**z**) box needs to be added. If the throat thickness is to be specified, only the (**a**) box needs to be entered.



Edit Weld Symbol

Command - **PrB_EditWeldTag**



This tool will open the configuration dialog so that you can change an existing weld symbol.

You can also double-click on a weld symbol which brings up the same dialog box.

Numbering / Revisions

Anything drawn - a bolt, a plate or a profile - 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.

Revision Lists

The purpose of the revision system is to enable the user to easily identify the differences between two revisions.

In the Bill of Materials dialog box, a part list can be made for two revisions which will clarify the differences between the two revisions.

Example:

revision 0: Position part PR5 total:5

revision 1: Position part 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 being deleted or modified.

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

It is allowed to create workshop drawings and part lists from a revision that is still “open” i.e. not yet locked.
The numbers are still kept stable (they do not change) if the revision is never locked.

Numbering

The composition of the part number

The Parabuild numbering system allows for a Prefix / Number / and an optional Suffix i.e. (PL 1 A)

The prefix can be used to identify the part as a Plate, Member, Structure, or an Assembly.

The default Prefixes are:

PL – for Plates

PR – for Members

St – for Structures

A – for Assemblies

These values may be changed from within the Global Settings. Additionally you may change the **Start number** form it's default value of 1 to any number you choose - the numbers will be assigned sequentially beginning with the start number i.e. 100, 101, 102 ...

How Parabuild assigns Part / Assembly numbers

This is the order (or hierarchy) that Parabuild uses for assigning numbers to parts :

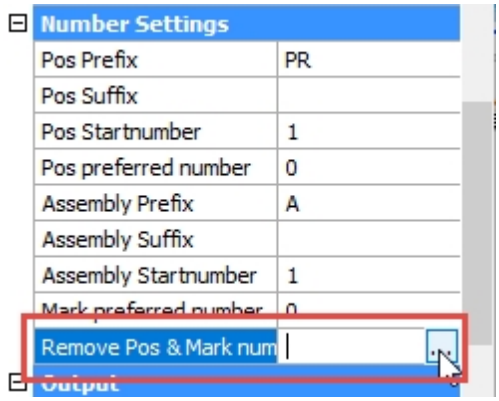
1. **Standard part number**
   If the geometry of the part matches with one of the standard parts (off the shelf parts), then this part will get the standard part's prefix, suffix and number.
   For more information about this mechanism, see the Parabuild Global settings dialog.
2. **Previous number** (not visible in properties)
   If there was no match with a standard part, and the part had a previous number, then it will reassign this number to the part if it is not already in use by another part. The previous number is stored in a 'Previous number' property, which is not visible in the Properties dialog.
3. **Preferred number** (see properties)
   If the previous number is already occupied, and the part has a Preferred number assigned to it, then Parabuild will try to assign that number to the part, provided it is not already in use.
4. **Start number** (see properties)
   If the preferred number was not set or is already occupied, then Parabuild will try to assign the *Start number* to the part, provided it is not already in use.
5. **Next available number** (see properties)
   Finally, if the *Start number* is already occupied, then Parabuild will assign the part the next available number, counting upwards from the start number.

Identical parts will however be assigned the same number, the quantities of which will be reflected in the pos / assembly sheets, and the bills of material.

This number assignment will be done the next time you generate a bill, workshop drawing, or when you press the **Assign part numbers** button in the Sheets manager, and will be immediately reflected in the assembly list.

Removing the Previous number

If at some point you want to force Parabuild to use the preferred number, then use this button in the properties dialog :



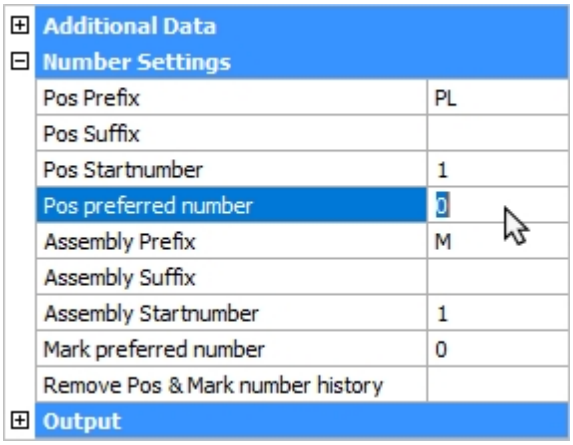
This action button will remove the previous number property that may have been stored in the part(s).

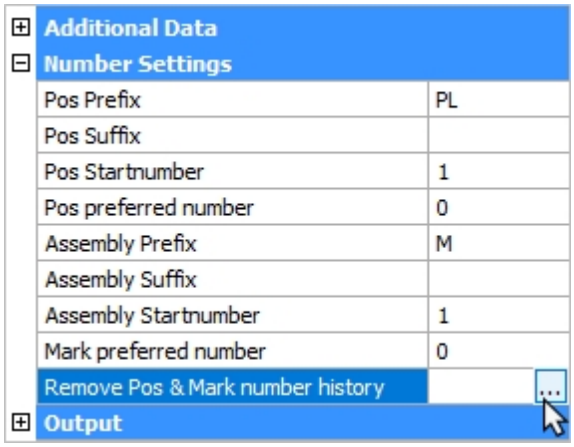
Preferred numbers

Preferred numbers may be assigned to parts or assemblies.

You can change the automatically assigned part number by assigning a preferred number. The default value is 0, indicating that a preferred number has not been assigned – here you may enter the preferred number provided it's not already in use. If it is, Parabuild will automatically assign the next available (free) number. Note! only numeric characters will be recognized.

If, at some point, you want to force Parabuild to use the preferred number, then use the Remove **Pos & Mark number history** button illustrated below. Parabuild will then remove the part's previous number property so that the preferred number property will be used, provided it's not already in use.





Add a Suffix

Another option available is to add a suffix to the Part / Assembly number. This suffix may be any combination of alpha/numeric characters. i.e. **A1**, **Rev-A** …….

This works best when you wish to identify the part as a variant, i.e. If there are a number of identical parts, but one has an additional hole or some other minor difference – working with the automatic numbering system, that small variation will cause Parabuild to recognize it as a different part, consequently, it will be allocated a different number, and likewise, the same will apply to the assembly.

by adding a suffix to part (PL 1) creating (PL 1 **A**) will identify it as a variant of (PL 1). This is acceptable to Parabuild as it will not cause a conflict with (PL 1) – this same principle will apply to the assembly.

Conclusion

The above mechanisms were put into place to give the user the perfect mix of these capabilities :

- Assigning numbers fully automatically (do nothing, Parabuild does the numbering)
- Assigning numbers semi automatically (prefix, suffix and start number)
- Assigning numbers fully manually (preferred numbers)
- To store and find previous numbers for parts (Store preferred numbers in Revision manager + Properties panel)
- *Never* being able to make a mistake with assigning wrong numbers to parts. Parabuild will never allow you to make a mistake by assigning 2 different parts the same part number if you assign part numbers manually.

Parabuild always has the last say about the part number, so that is how we ensure that no mistakes can be made.

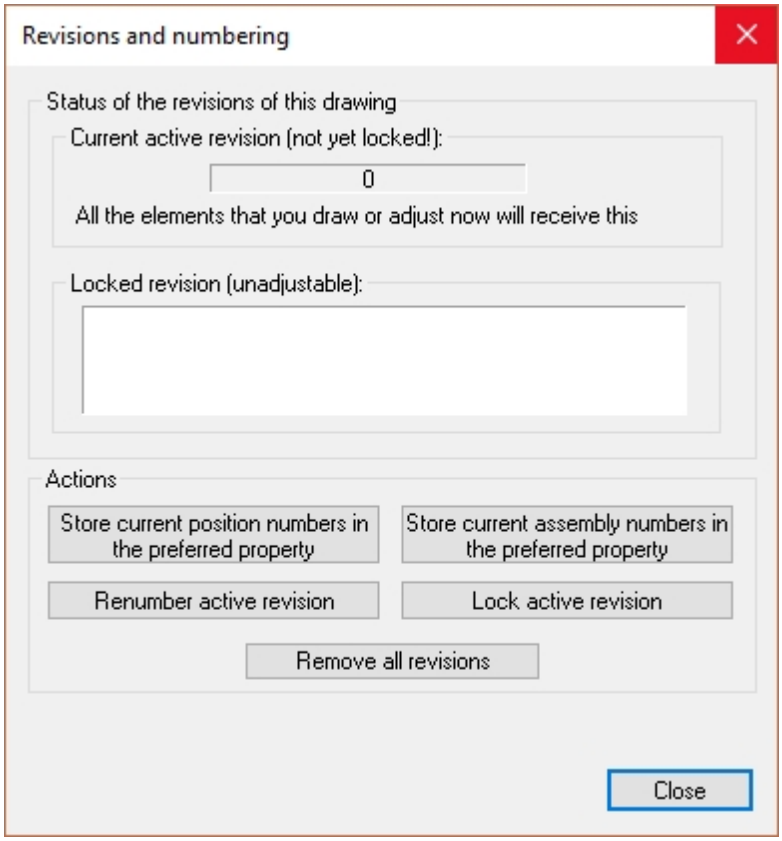
Revisions manager

Command - **PrB_RevisionManager**

Activating this command will open the Revisions manager dialog.

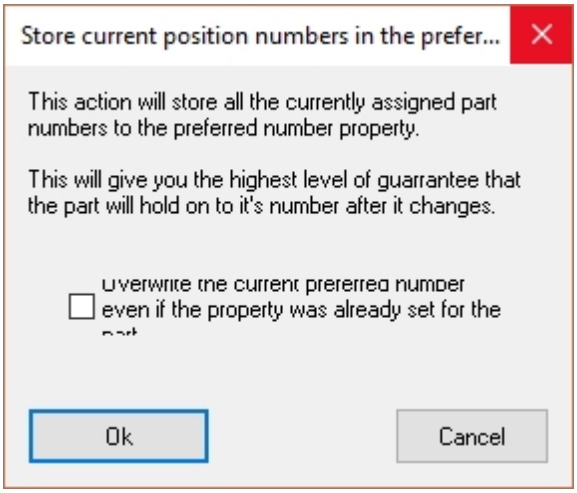
A few points to note:

- Working within a revision is compulsory.
- Anything drawn - a bolt, a plate or a profile - 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 modification 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.

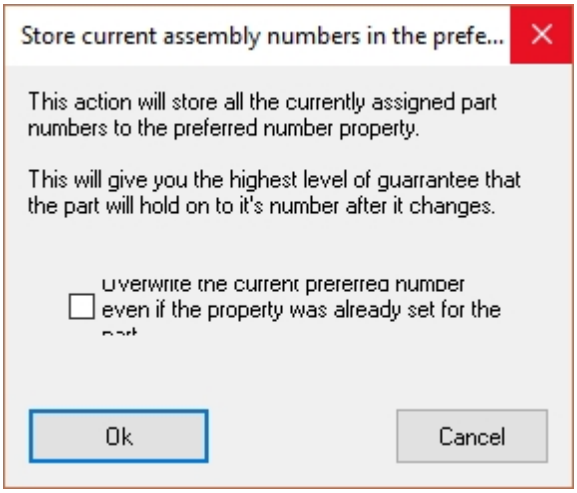


The actions in this dialog are explored below :

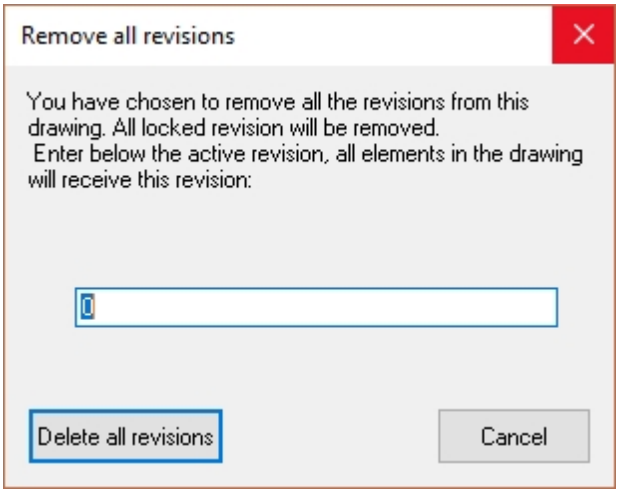
- **Current active revision** (**not yet locked**) - will display the current revision
- **Locked revision** (**Nonadjustable**) - will display all the currently locked revisions
- **Store current position numbers in the preferred property** - this will copy all current Position numbers to the preferred numbers for all parts.
  This is a handy tool when someone needs to do changes to the model, but GA drawings or shop drawings are already done or printed.
  When you apply a change to assembly A7, it might loose it's number A7 (when there were 2 assemblies with number A7 and only 1 of them has changed).
  But you will still see the "7" number in the preferred number property thanks to this tool.
  A user could change the suffix of that assembly to "B" which would result in an assembly number "A7B".
  Parabuild is happy because it does not create a conflict with A7. And the user has integrated the old number in the new number for easy recognition.



- **Store current assembly numbers in the preferred property** - this will copy all current Assembly numbers to the preferred numbers for all parts. This is basically the same as the position number storing tool.



- **Renumber active revision** - Use this to fill in unused part numbers. This may corrupt shop drawings because it may change the numbers of all parts. It will not corrupt GA drawings.
- **Lock active revision** - Will lock the active revision - and automatically allocate the next sequential revision number.
- **Remove all revisions** - This action will remove all revisions from the drawing and all locked revisions will be removed.
  This may corrupt shop drawings because it may change the numbers of all parts. It will not corrupt GA drawings.



Though the use of locking revisions is advisable, it is possible to work in revision 0 without ever manually locking.

For example, if revision "0" is locked and it contains parts (PL1) and (PL2), and after locking (PL1) is modified - It will then automatically get Revision "1" assigned to it.

An important point to note with Revisions is that part/assembly numbers will never be overwritten. So in the example of (PL1), it will not be assigned (PL1) again after the change, it will get PL3 or whatever number that is not yet in use.

The preferred number will also not enforce it to (PL1) because it can't override the locked revisions. The revisions "lock-in" their numbers and they have a greater priority than the preferred numbers.

Another tool for locked revisions is the revision bill, which can be generated with the Bill of Materials dialog. It will show an overview of the pos and assembly numbers in each revision.

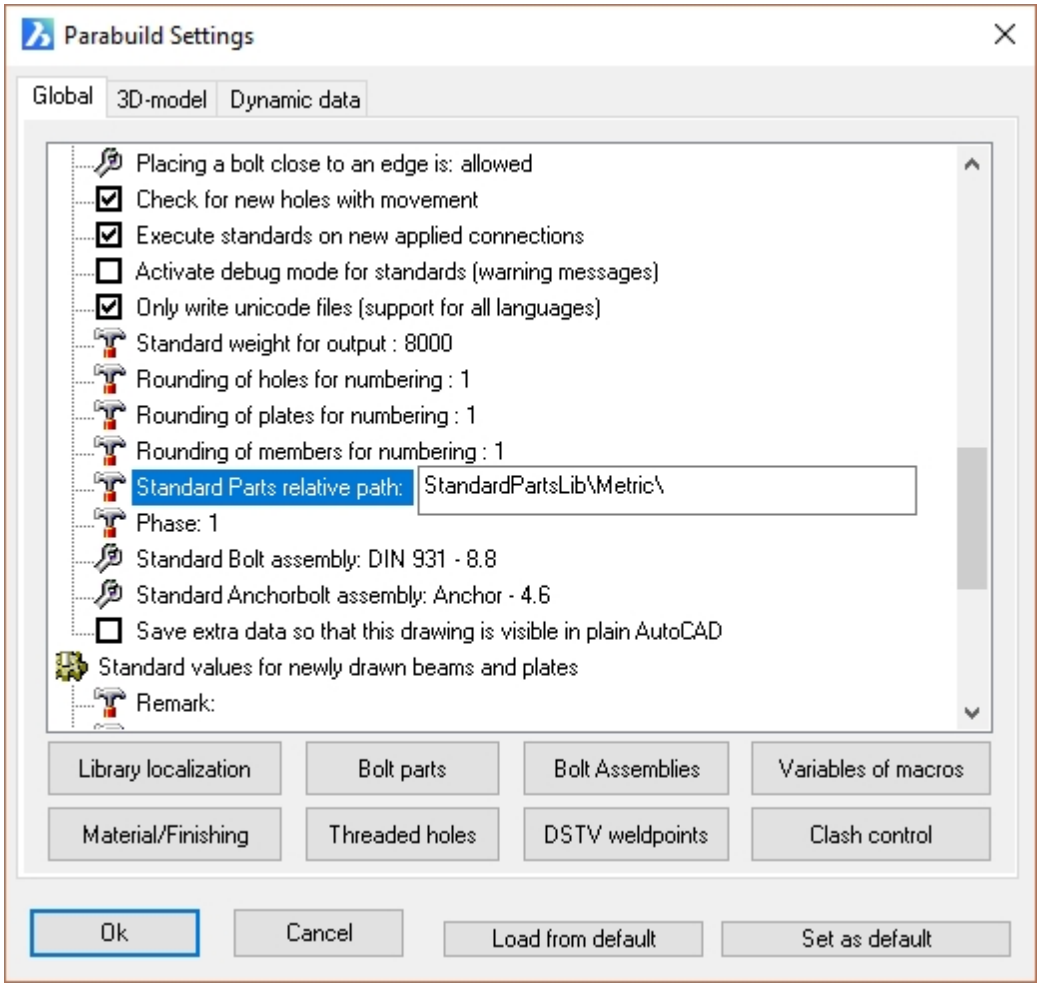
This bill will only work when there are at least 2 locked revisions.

Standard Parts Library

The numbering system in Parabuild will also look at standard parts in the library, and will use the numbers found in those drawings to automatically assign numbers in the current 3D drawing.

Standard parts can be seen as "Off the shelf" parts that do not need shop drawings, and should get a pre-defined position number so that the persons in the shop know which shelf-part to use by the position number only.

For more information about this, see the Parabuild global settings dialog topic.



Output : Sheets, BOM, CNC generation

This topic covers the control and output of all 2D sheets including: Position and assembly sheets, Bills, Schedules, General arrangement and detail drawings.

Additionally you can :

Generate Position and assembly sheets

Generate DXF files

Generate DSTV NC files

Generate WrapAround for pipes

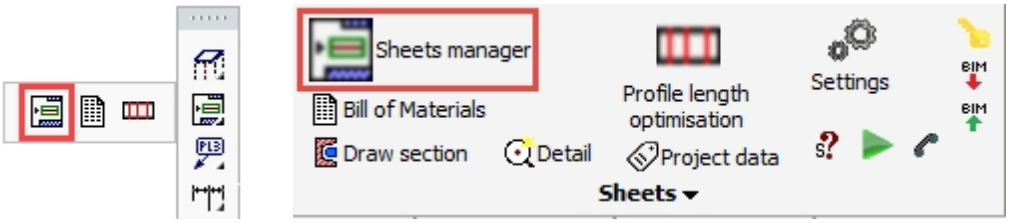
Generate step files

Generate bills of material

Add plan views and elevations to general arrangement drawings

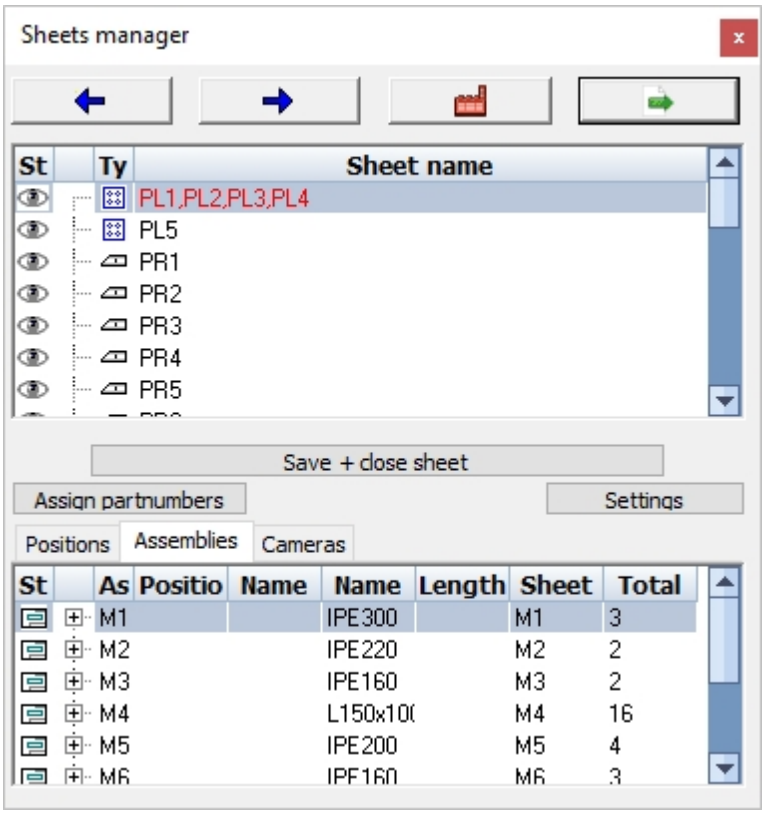
Sheets Manager

Command - **Prb_Sheets**



The Sheets Manager enables you to manage all the output that results from the 3D model. This includes: Position and assembly sheets, Bills, Schedules, and General arrangement & detail drawings.

The Sheets manager dialog can be accessed from the main ToolBar - which, when opened, will remain open until it's closed, or may be docked to the edge of the work area.

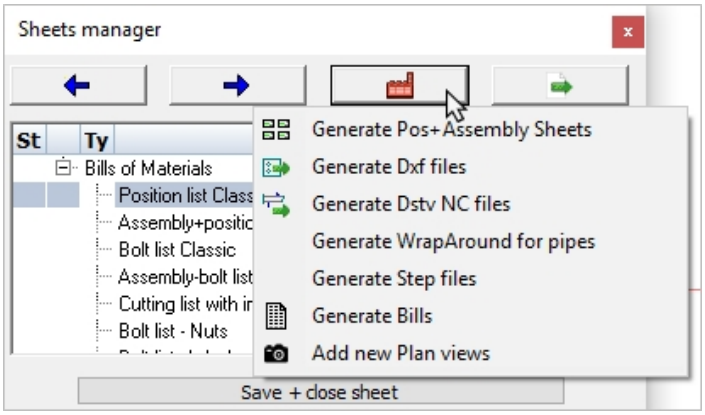


This dialog contains the following main elements:

Top buttons

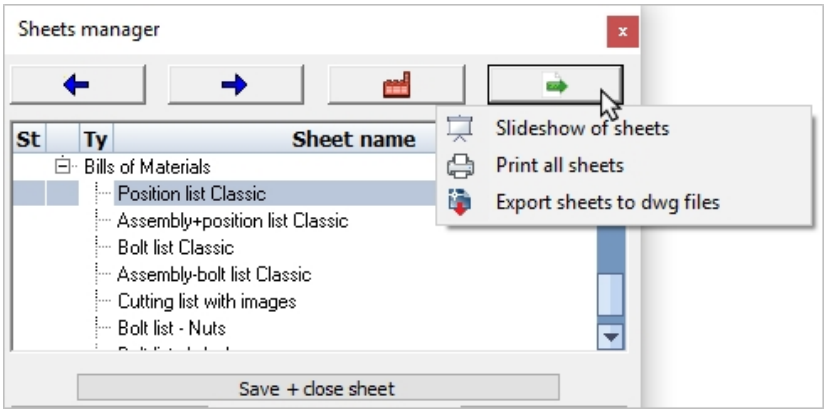
The Top buttons allow you to view, generate, export and print drawings and CNC files.

Activating the left and right arrows enable you to view and scroll through all the 2D drawing sheets that have been prepared



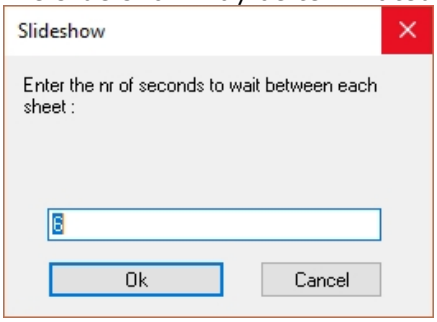
Activating the Generate button allows you to:

- Generate Position and assembly sheets
- Generate DXF files
- Generate DSTV NC files
- Generate WrapAround for pipes
- Generate step files
- Generate bills of materials
- Add plan views and elevations to general arrangement drawings



Activating the Export button allows you to:

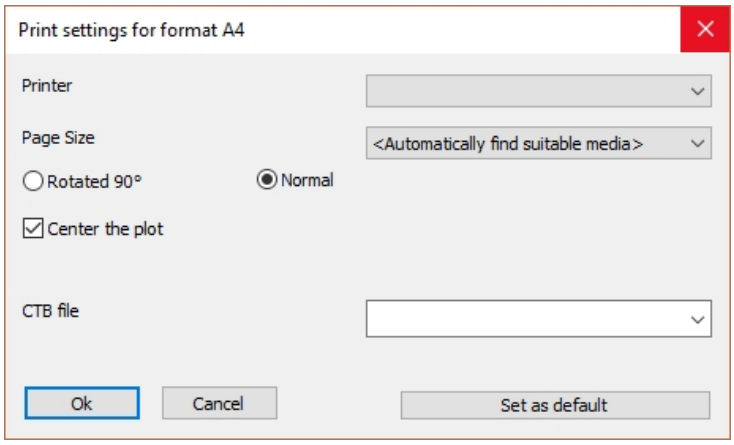
- **Create a slide-show of drawing sheets**
  You can enter the delay in seconds between sheet display.
  The slide show may be terminated at any time by selecting the **Stop** button



- **Print all sheets**

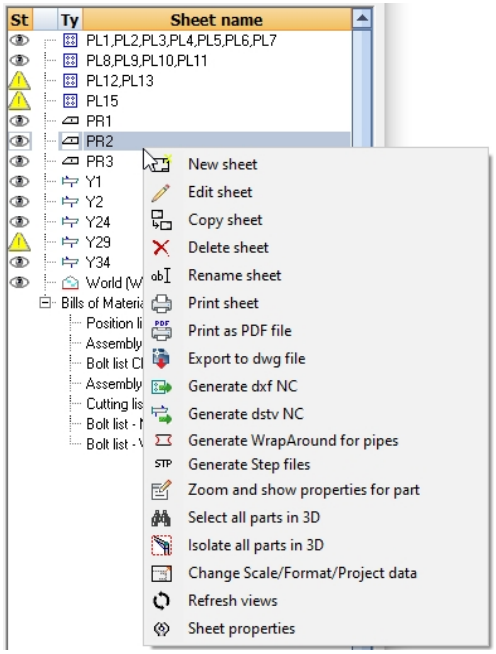
Activating this command will print all 2D sheets listed in the Sheets manager - the following dialog will prompt you to:

- Select the printer
- Select page size
- Select the desired CTB plot file
- You may then save these settings as default values for the current format



- **Export sheets to DWG files**
  Activating this command will export all drawing sheets listed in the Sheets manager in DWG format. The file will be located in the same folder as the location of the 3D model.

Top List

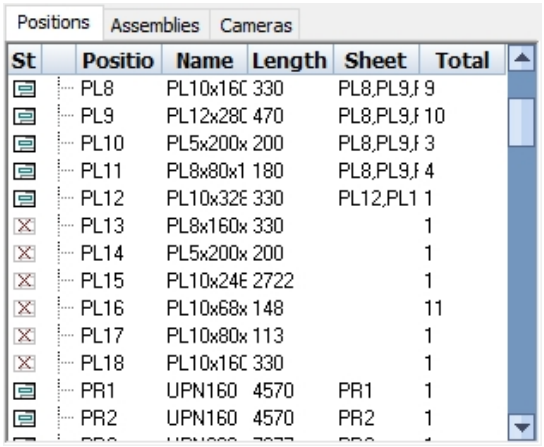


The **Top list** contains all the 2D sheets created, and also all available bill of materials. Right clicking a 2D sheet in this list will bring up a series of available tools.

Bill of Materials

Scrolling to the bottom of the top list, there are a number of pre-defined materials lists. Double-clicking a bill here will generate just that bill and will also open it in the default editing program as it was set in Windows.
Right clicking on any of these BOM items will bring up the available BOM tools.

Bottom List



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 on an assembly in this list will bring up a series of options.

At the head of the bottom list, there are a number of options:

Save and close sheet - will save and close the current sheet

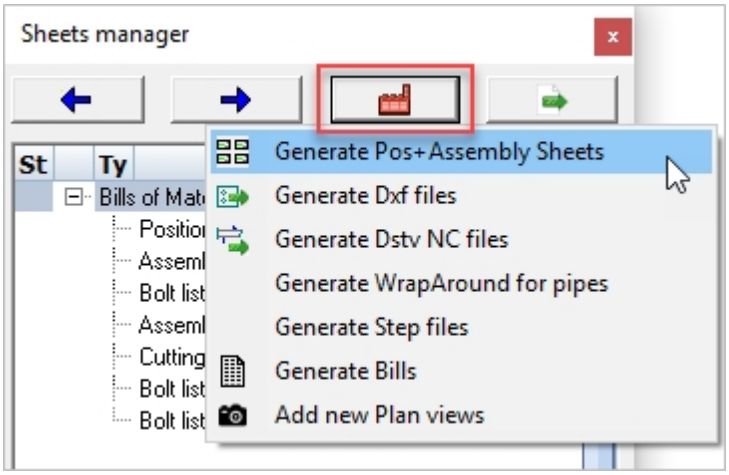
Assign part numbers - this tool will assign part numbers to all the parts in the drawing. To learn more of how this is done, see the Numbering chapter.

Settings - For adjusting the settings for BOMs, CNC output, workshop drawings and GA drawings (For more about this see Settings for workshop drawings)

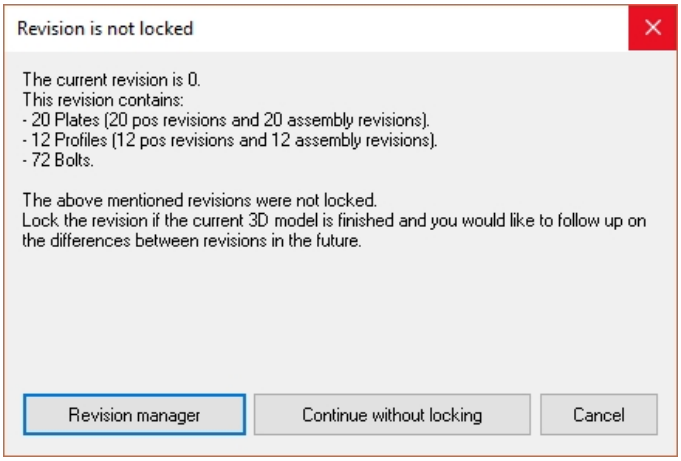
The Tabs: **Positions** / **Assembles** / **Cameras** - these tabs provide a way to filter the bottom list on the part type

Generate Pos. and Assembly Sheets

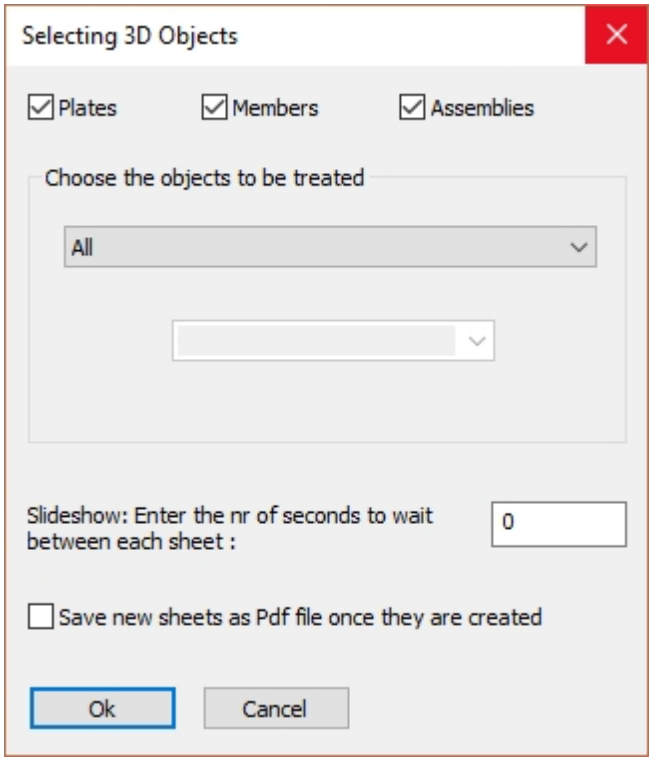
From within the sheets manager, click on  to start the function **Generate Pos+Assembly drawings**



A dialog will appear to warn you that the revision is not locked - here you have the option to lock the revision using the Revision manager or to continue without locking. It is safe to continue without locking.



Before generation of the 2D sheets you will be prompted to select which 3D objects are to be processed.



By opening the **All** drop-down menu, you have the choice to process a part of the 3D drawing, 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 parts that will be shown on the sheet is only the number of parts counted in the phase/revision selection.

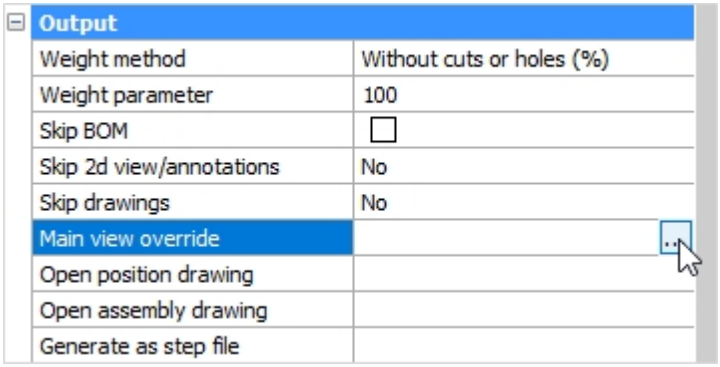
If you've changed the 3D drawing which would cause the number of parts on the sheet to change, then you can start the function Refresh views on the sheet(s) to update the bill on the sheet(s).
The refresh view function is done automatically on all sheets when you print or export the 2D sheets.

Assembly shown upside-down

Sometimes, the assembly will be drawn upside-down if you use Parabuild's default shop drawing options.

The cause of this is the main profile's **Y** axis, which is flat (in combination with the **No cardinal view direction** in the default option)

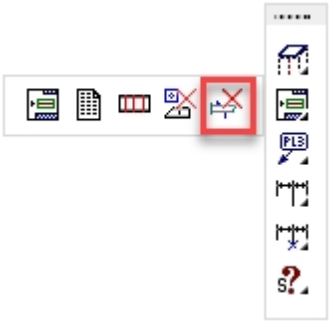
To fix this, we can use the view override of the main part in Properties



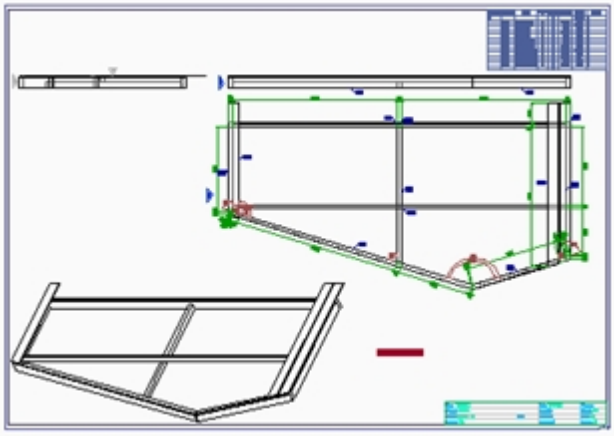
Select the part and open the **Properties** panel - scroll down to Output and click on the button under **Main view override**.

Following the command line prompt, *Select a plane that determines the main view's direction, or <ESC> to clear the current plane selection*

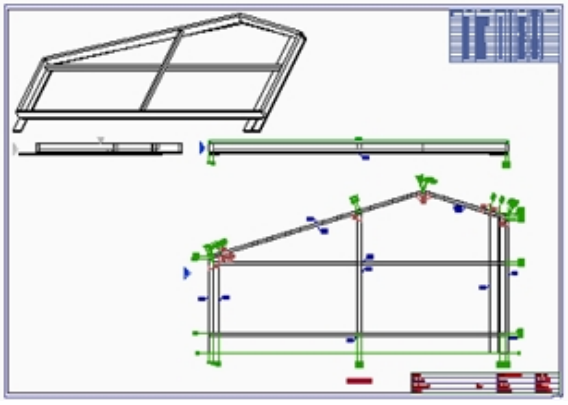
Select the front face of one of the members making up the assembly.



Then it's necessary to erase the history of the assembly. Otherwise, the existing history will be used which doesn't have the override.



Before



After

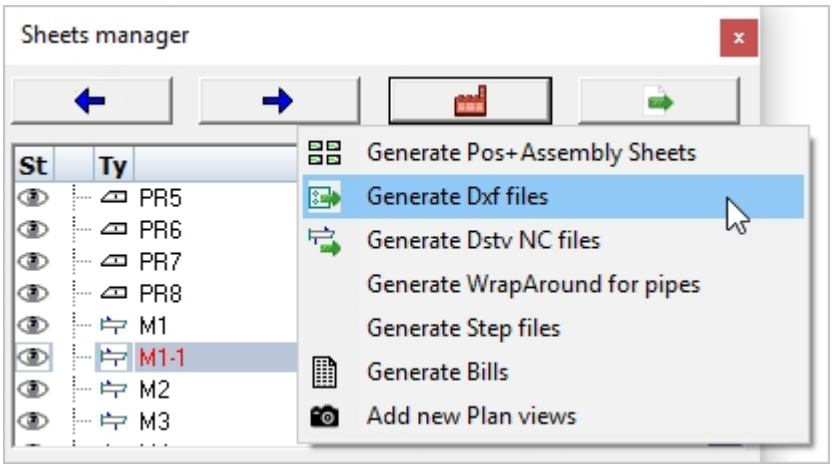
Generate DXF Files

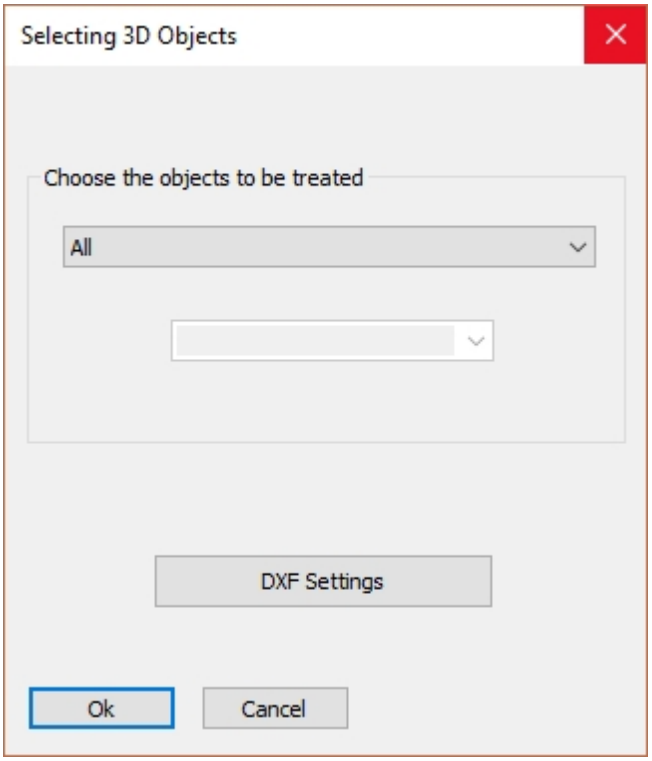
You can start this function by clicking on the button  in the sheets manager 

The dropdown allows you to filter the selection on : All / Selection / Phase / Revision.

Parabuild will only create DXF files for plates. These DXF files are only meant for plate table machines (laser, plasma, jet) that only have 2D capacity.

The DXF files will be stored in the same folder as the location of the 3D DWG file.



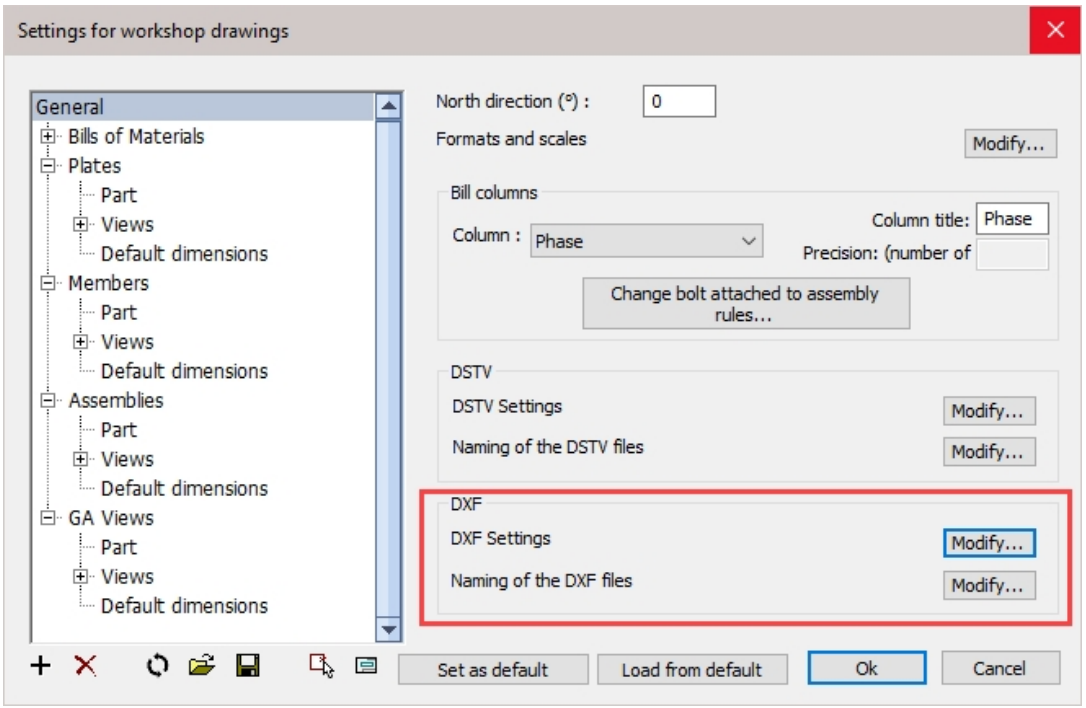


Pronest compatibility

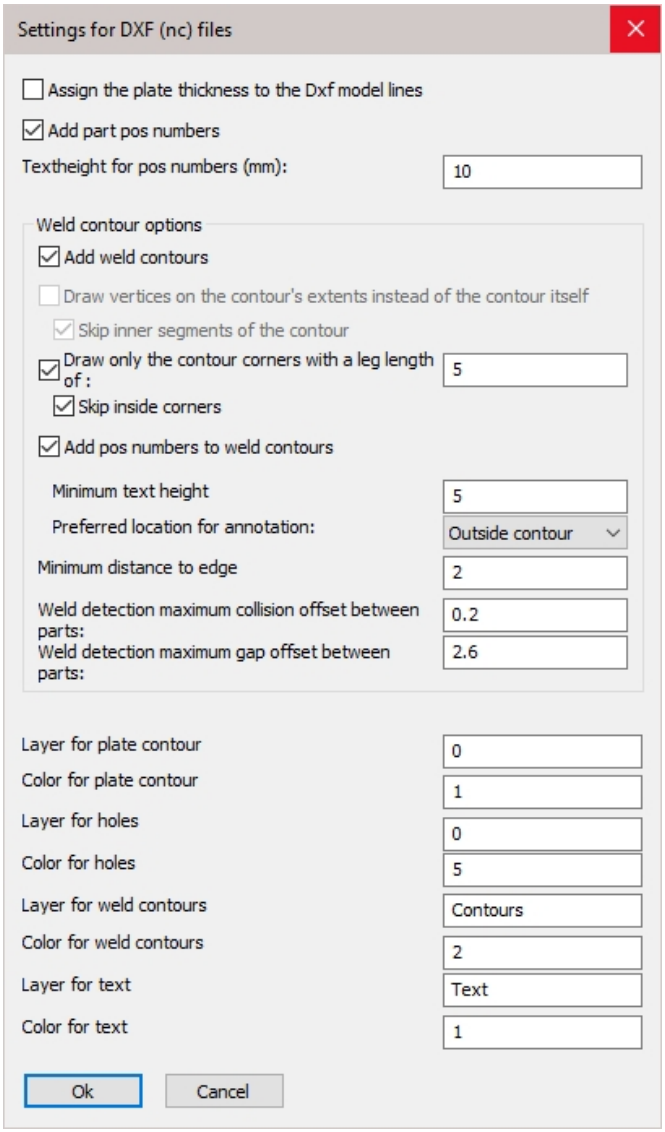
A ProNest PNL Job file is always generated automatically whenever DXF files are generated

DXF Settings

Settings for DXF files may be changed from the Sheets manager / Settings / General



Clicking the DXF Settings / **Modify** button will open the following dialog where you can change the following settings :



The options in the dialog box are explored below.

Assign the plate thickness to the Dxf model lines - When active, the thickness of the plate will be assigned to the model lines in the dxf files. This creates a semi 3D model and the machine can know the required thickness of the plate this way. Some machines can't read the dxf file when this option is active.

Add part position numbers - 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.

Text height for position numbers - Set the text height for position numbers on the part.

Add weld contours - When active, the weld contour options become visible.

Draw only the contour corners with a leg length of : When this is active, only the corners of the contour are scribed to save machine -time with scribing work.

Skip inside corners : Inside corners will occur when an I-shaped profile is welded with the I shape to the profile. This case would result in 4 inside corners. You can skip these corners with this option.

Add position numbers to weld contours - Wen active, position numbers will be added to weld contours.

Minimum text height - Set the minimum text height for the position numbers of the contour. Parabuild will deviate from the standard text height in order to fit the text in places without much

room, but will not use a text height below this value.

Preferred location for annotation : Normal behavior is to place this annotation outside of the contour. You can choose inside of the contour, but if the text is too large to fit in the contour then it will be placed outside of the contour.

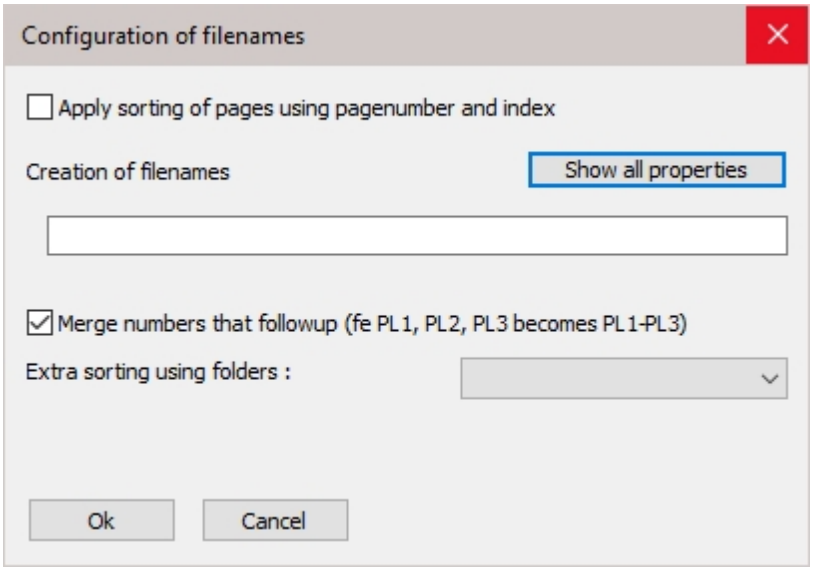
Minimum distance to edge - Set the minimum distance that should be kept between the text and the edge of the plate

Weld detection, maximum collision offset between parts - If the welded parts in 3D collide with each other too much then the weld contour won't be added to the DSTV file.

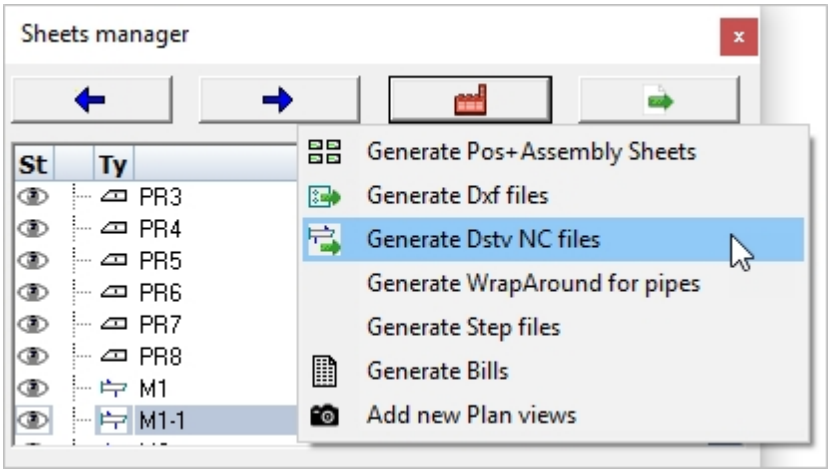
Weld detection, maximum gap offset between parts - If the gap between the welded parts in 3D is too much then the weld contour won't be added to the DSTV file.

***** The remaining options allow you to set the layers and colors of the different line types and texts.

Naming of the DXF files button will open the following dialog where you can change the following settings :



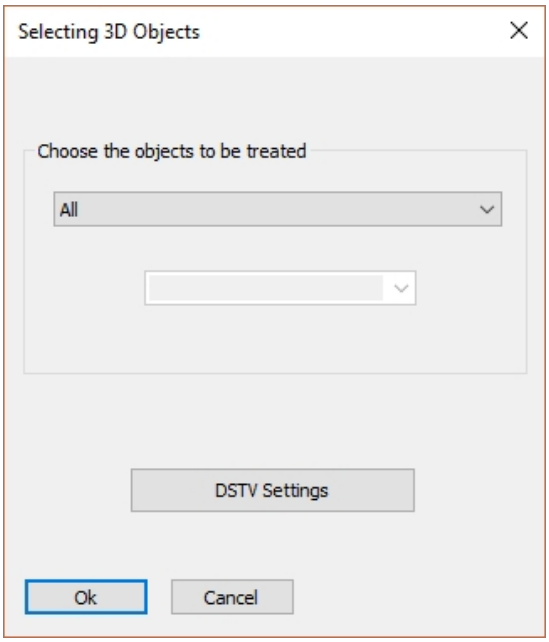
Generate DSTV NC Files



DSTV NC files can be read by most cutting- and drilling CNC machines. Also some plate machines (laser, plasma) can read DSTV files.

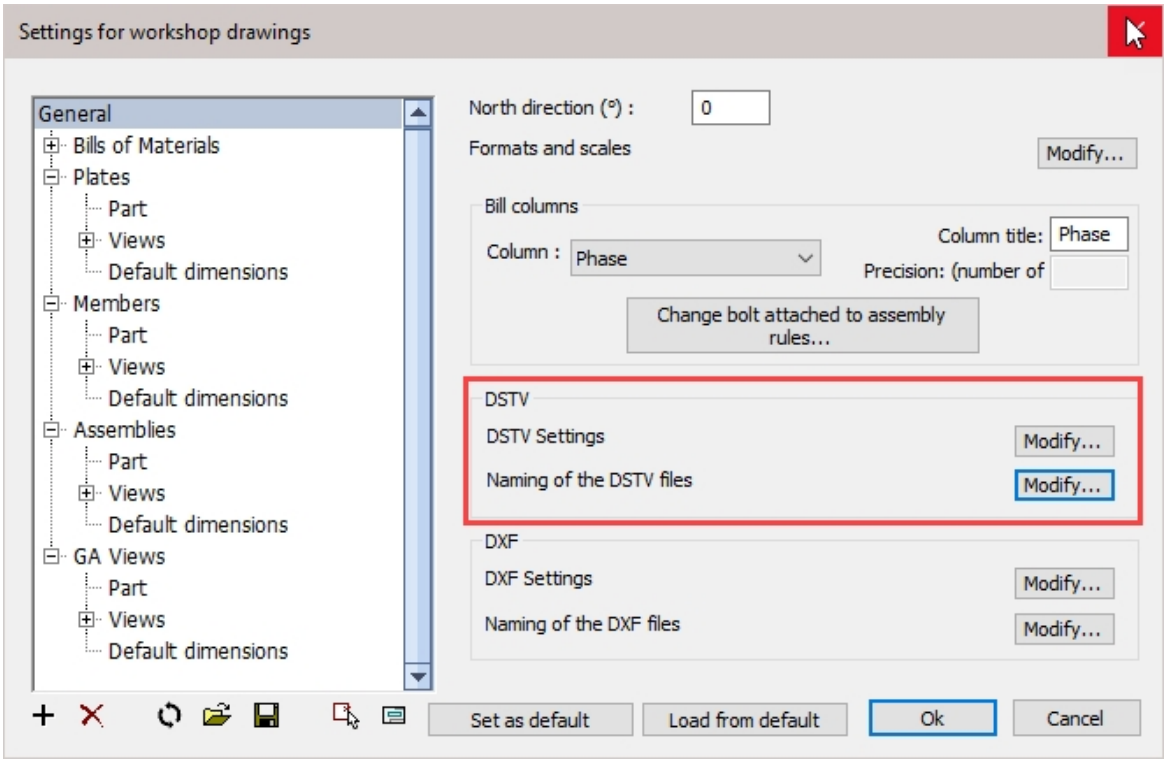
The DSTV NC files will be stored in the same folder as the location of the 3D DWG file.

You may also change the DSTV NC Settingslocation from within this dialog :

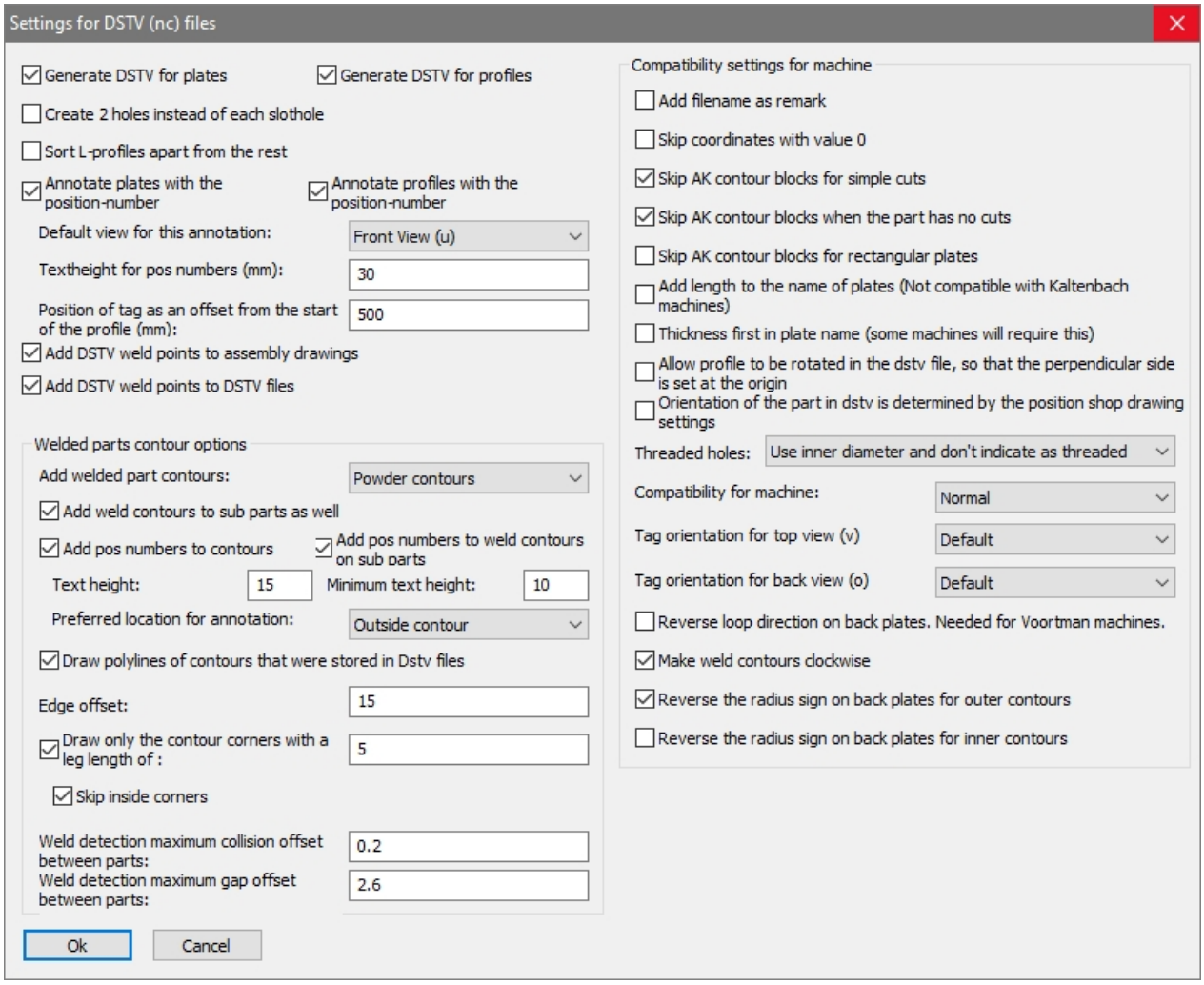


DSTV NC Settings

Settings for DSTV files may be changed from the Sheets manager / Settings / General



Clicking the DXF Settings / **Modify** button will open the following dialog where you can change the following settings :



The options in the dialog box are explored below.

Generate DSTV for plates - When activated, DSTV files will be generated for plates.

Generate DSTV for profiles - When activated, DSTV files will be generated for profiles.

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.

Default view for this annotation - Here, you can choose on which side of the part the position number should be scribed.

Text height for position numbers - Set the text height for position numbers scribed on the part.

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 this in the DSTV weld points topic.

Weld Contour types - Here you may choose between either, Punch / Powder / or Both. If you choose 'Both' then all the contours are added double to the DSTV file.

Add weld contours to sub parts as well - When you enable this, then weld contours will also be added to sub parts that are touching other sub parts. This will occur in assemblies where 2 sub parts are touching each other.
If this option is disabled, then only the subparts that are touching the main part will get a contour on the main part.

Add position numbers to contours - When you enable this, then weld contours that are placed on a main part will get a position number text.

Add position numbers to weld contours on sub parts - When you enable this, then weld contours that are placed on sub parts will also get a position number text. This will occur in assemblies where 2 sub parts are touching each other.

Text Height and **Minimum text height** : Here you may select the text height and minimum text height. If the part number text is located (especially) inside the contour, then a reduced height may be used. With this option, you can choose the minimum text height that the function is allowed to use for this purpose.

The **Preferred location of the annotation** - The default behavior is to place the annotation outside of the contour. If you choose to place it inside the contour, but the text is too large, then Parabuild will automatically place it outside anyway.

Selecting 'No Preference' will allow Parabuild to select the most suitable position.

Draw PolyLines of contours that were stored in the DSTV files - When this is enabled, Parabuild will draw a contour in the 3D model for each contour that it has written in the Dstv file. This is useful for checking the contour results, but this can clog your 3D model with lines.

Edge offset - Parabuild will trim the contour so that it does not come any closer to any edges than this offset value.

Draw only the contour corners with a leg length of : When this is active, only the corners of the contour are scribed to save machine -time with scribing work.

Skip inside corners : Inside corners will occur when an I-shaped profile is welded with the I shape to the profile. This case would have 4 inside corners. You can skip these corners with this option.

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.

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 in the DSTV file.

Skip AK contour blocks for simple cuts - Some machines always need the AK block, others don't support the AK block at all.

Skip AK contour blocks when the part has no cuts - See above

Skip AK contour blocks for rectangular plates - See above

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.

Thickness first in plate name - Some machines will require this

Allow profile to be rotated in the DSTV file so that the perpendicular side is set at the origin

Orientation of the part in DSTV is determined by the position shop drawing settings

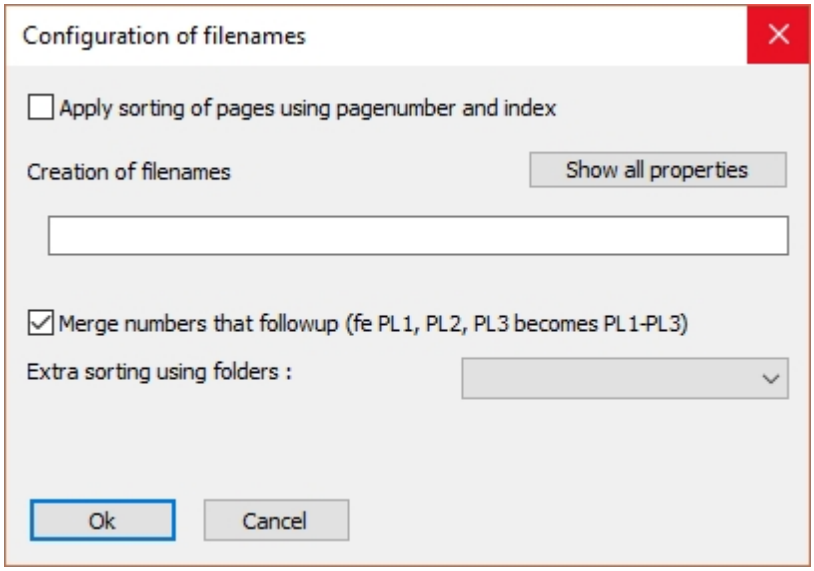
Reverse loop direction on back plates - This is needed for Voortman machines

Make weld contours clockwise

Threaded holes

Compatibility for machine - These are a collection of changes to the standard that some old machines need

Clicking the DSTV Settings / **Naming of the DSTV files** button will open the following dialog where you can change the following settings :



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 gaining time because one has to measure less when welding.

The points will normally be made by the machine with the tip of the (currently mounted) 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.

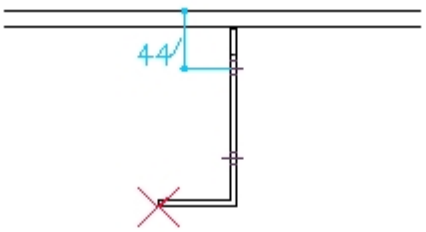
At the top of the dialog you see all points in the group. You can create new points or remove points with the buttons next to the list.

If you select a point from the list then the options for that point will be visible beneath it and you can modify these.

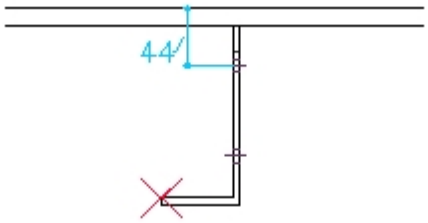
Properties of a point

Name: The name for the point. We will later use this name if we want to use this point for a welded element.

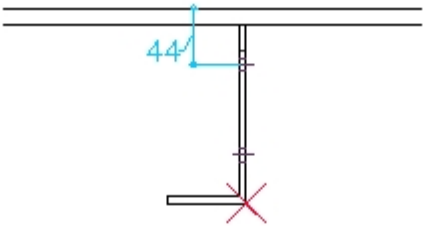
Choose corner: You choose the corner where the weld point should be positioned on the welded element. The following illustrate all possible choices with an example:



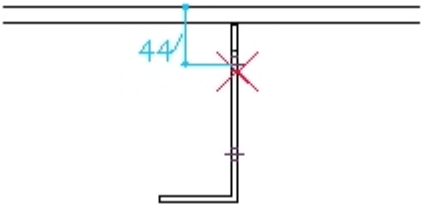
Corner left under



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.



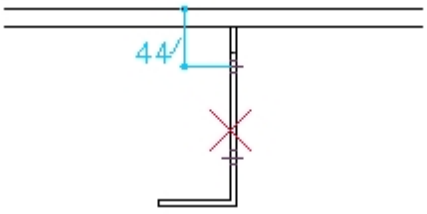
Corner right under



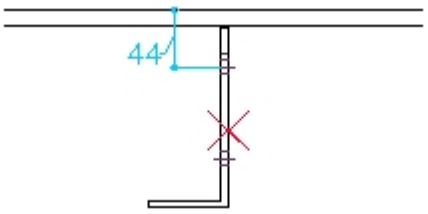
Corner right above - The point was not placed entirely at the top because the drill head cannot come this close to the flange. This clearance space is adjustable, see further in the manual.

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.

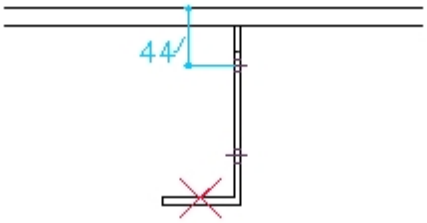
The following illustrate all possible choices with an example:



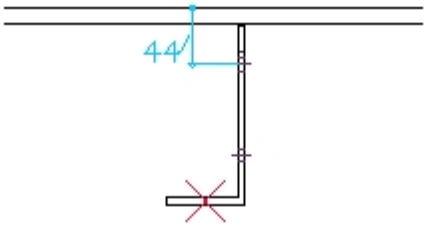
Intersection left



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.

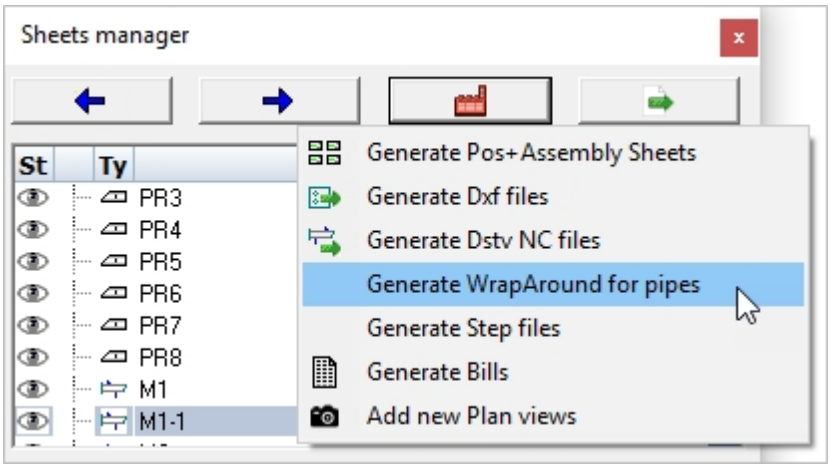
This property exists both for plates and profiles.

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.

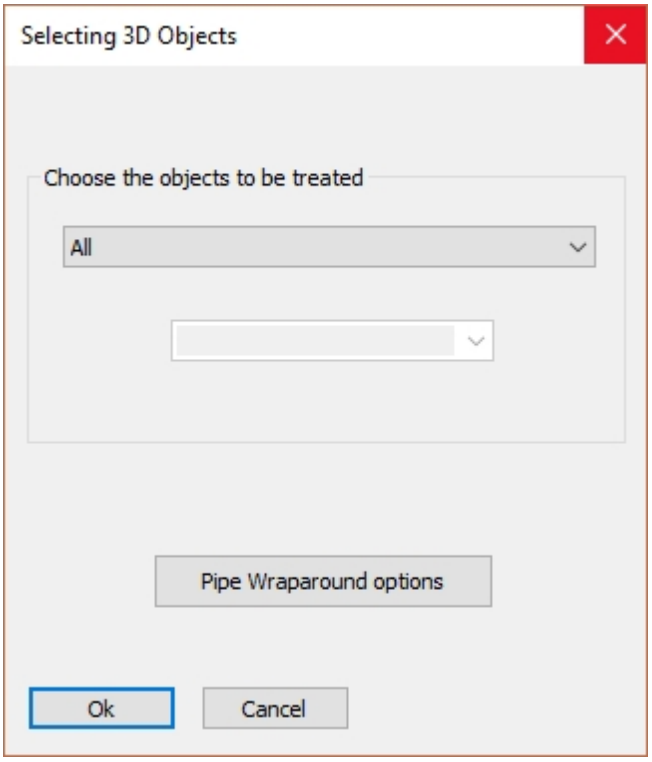
Assembly drawings and DSTV files will also get different names. Normally you have one DSTV file for each position number, but that is no longer possible when that profile has weld points. In that case the file name of the assembly drawing and the DSTV file will contain both the mark number and the position number. The result is more files and drawings, but it is an inevitable disadvantage.

Generate WrapAround for Pipes



This tool allows you to generate WrapAround details in DXF format for pipes and circular hollow sections.

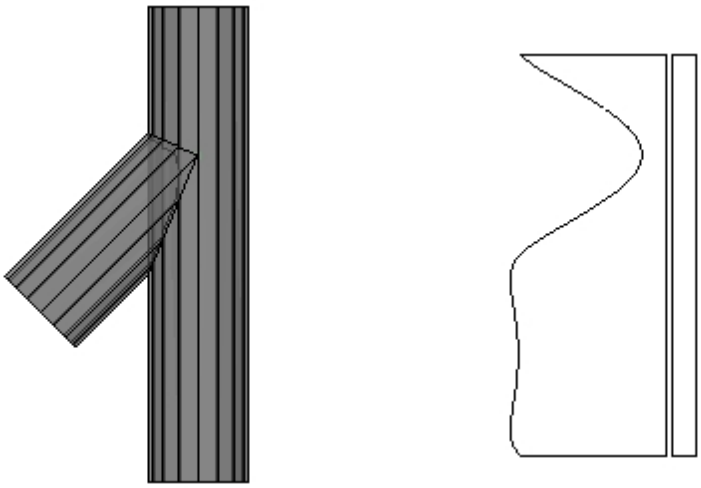
On activating the command the following dialog will appear :



It allows us to filter the objects to be treated on :

- **All** - will generate WrapAround DXF files for all elements in the 3D model
- **Selection** - will prompt you to select the required objects
- **Phase** - where you will need to enter the project phase
- **Revision** - where you will need to enter the required revision

In the same dialog it is also possible to directly change the Pipe WrapAround options

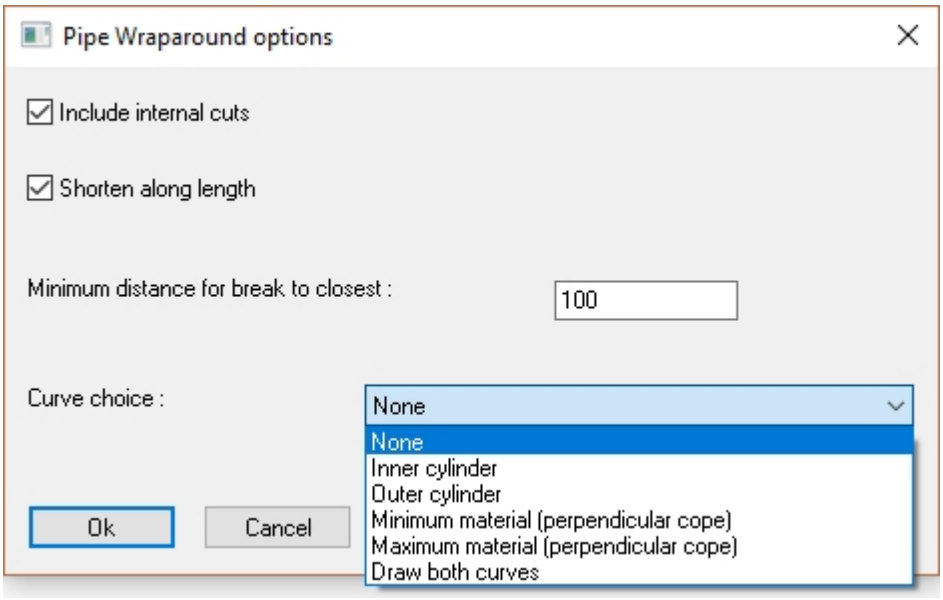


Example of a 3D model

The resulting wraparound generated by this tool

Pipe Wraparound Options

This tool will generate a wraparound of circular hollow sections. A wraparound is a foldout of the pipe on a flat 2D surface.



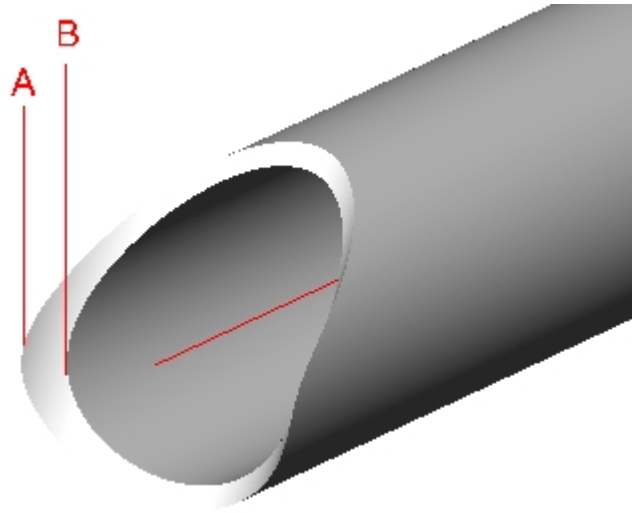
The different options in the dialog are explored below :

Include internal cuts - If disabled, only the cuts at the endings of the pipe would be processed.

Shorten along length - If enabled, the wraparound will be shortened on the straight lines, saving the space.

Minimum distance for break to closest - This relates to the shortening : It is the minimum distance that should be kept between the shortening line and the closest curves that can't be shortened.

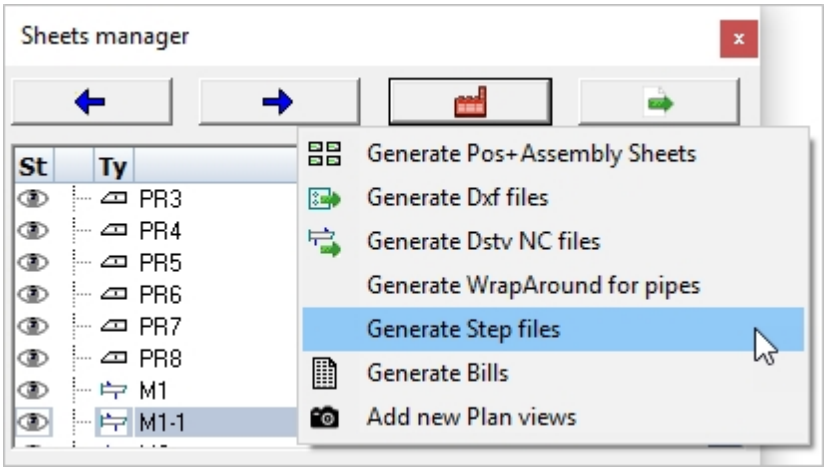
Curve choice - Generating the wraparound of the 3D model will cause a loss of information. We are after all exporting just 1 layer of the pipe, whereas the actual pipe in 3D consists of many layers that could be exported.

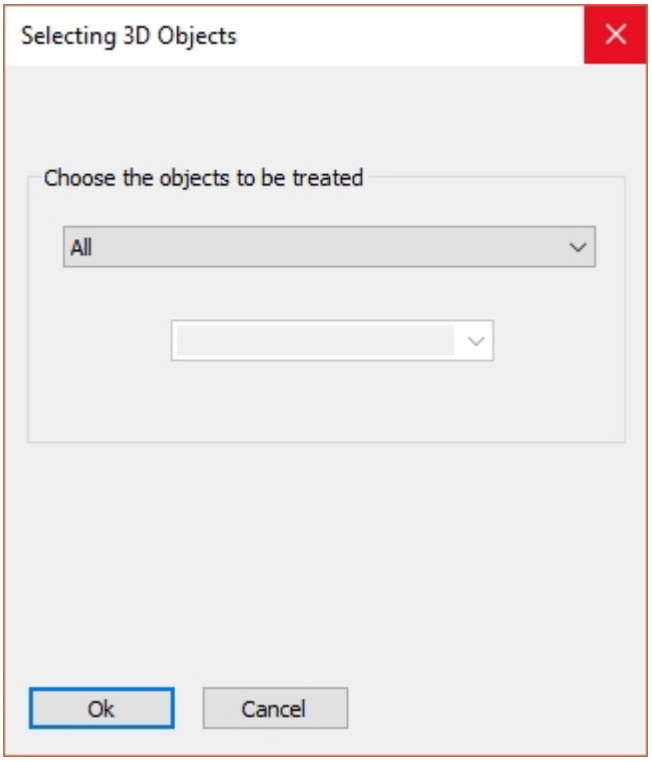


These are the available options for the curve choice :

- **Inner cylinder** - This is cylinder B in the illustration
- **Outer cylinder** -This is cylinder A in the illustration
- **Minimum material** - Parabuild will choose the inner or outer automatically depending on which would cut the most amount of material away. Perpendicular cutting is assumed.
- **Maximum material** - Parabuild will choose the inner or outer automatically depending on which would cut the least amount of material away. Perpendicular cutting is assumed.
- **Draw both curves** - This option will draw both the inner and the outer curves.

Generate STEP Files





This tool will generate a step file for each position number in the drawing.

STEP file is a CAD file format, usually used to share 3D models between users with different CAD systems.

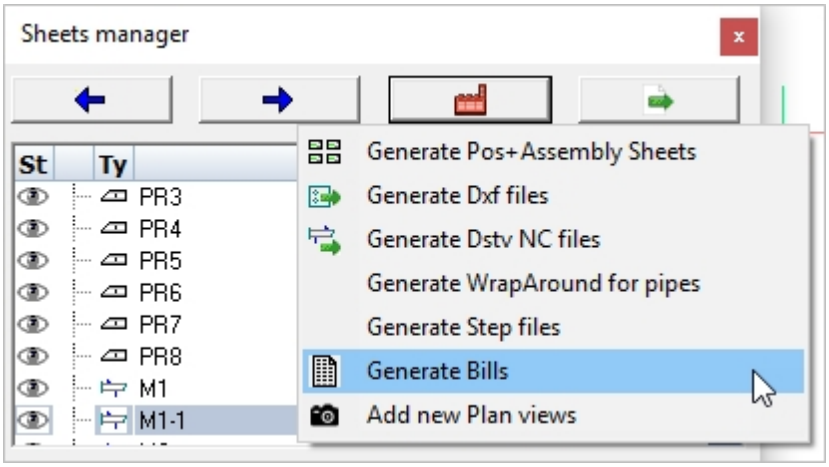
The step file is often needed for CNC machines that can automatically cope (plasma/laser) hollow profiles.

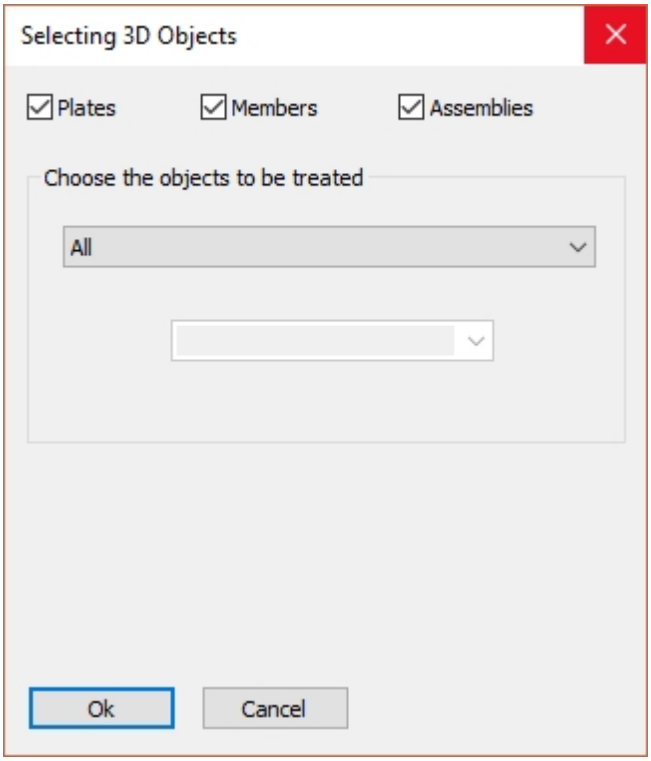
The number of parts to be produced is added to the name of the file : "PR5 x 6.stp"

Note

At the time of writing this command will only work in BricsCAD and when the Communicator module is installed and licensed.

Generate Bills



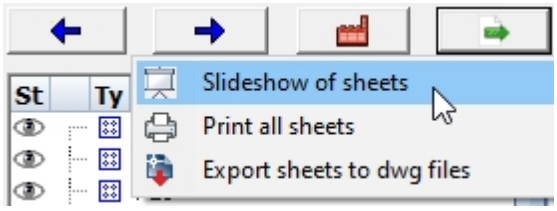


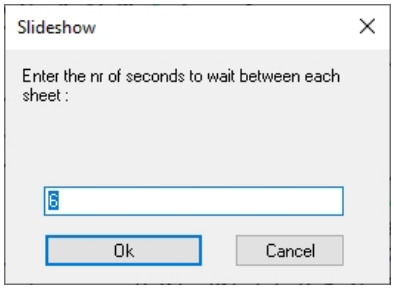
When using this tool, all bills of materials that are listed in the sheets manager will be generated.

The files will be stored in the same folder as the folder of the 3D DWG file.

If you want more or other bills in this list, then see the topic Bills of Materials

Slideshow of sheets

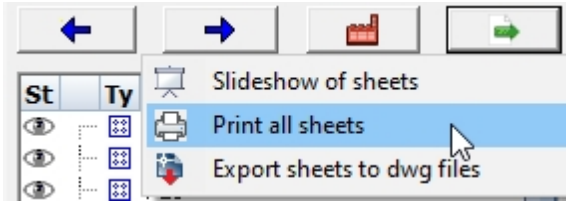




This tool allows you to create a slide-show of all the sheets.

You can enter the delay in seconds between sheet display.
The slide show may be terminated at any time by selecting the **Stop** button.

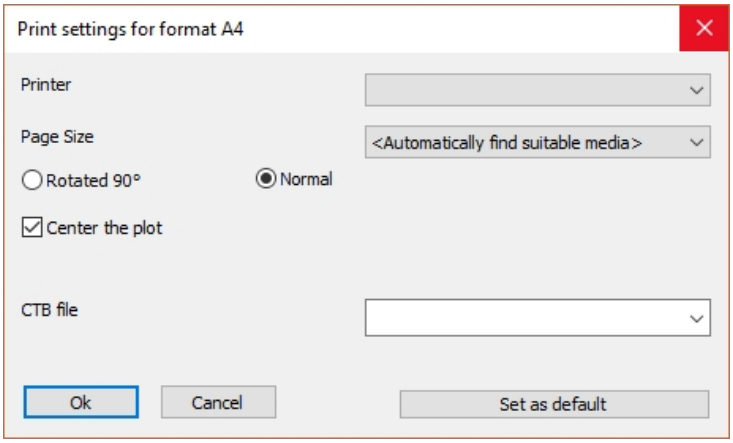
Print all sheets



This tool allows you to print all of the sheets.

Before the actual printing is executed, Parabuild will first refresh all of the shop drawings.
This is done to ensure that the number of parts in the BOM of each corresponds with the actual parts in the 3D model.
During this process, it will warn you in case you're about to print an expired drawing.

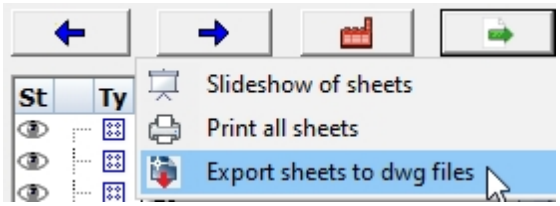
The print dialog will be prompted once per page size.
This allows you to assign a different printer per size.



The dialog has the following options :

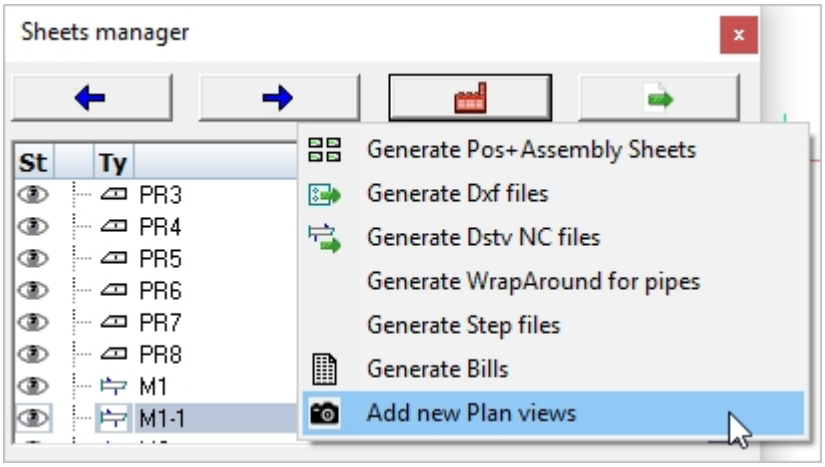
- The printer
- Page size
- The desired **CTB plot file** - this is used to match the color of entities to the desired line thicknesses on the printed sheet
- **Set as Default** - Press this button to save the current configuration as the default for all future print sessions. The default is only saved for the current paper size.

Export sheets as DWG files



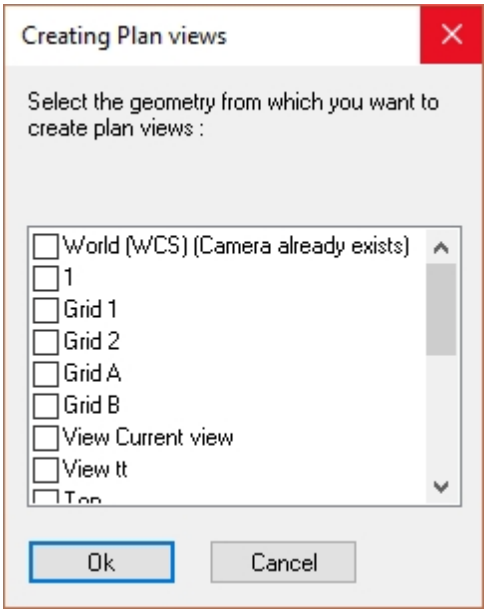
This tool allows you to export all of the sheets to DWG files.
The files will be located in the same folder as the location of the 3D model.

Add New Plan Views



With this tool you can create a new general arrangement view based on a grid, a level, an ISO view or the current 3D view.
The new view will be drawn on a new 2D sheet if no sheet is currently active.

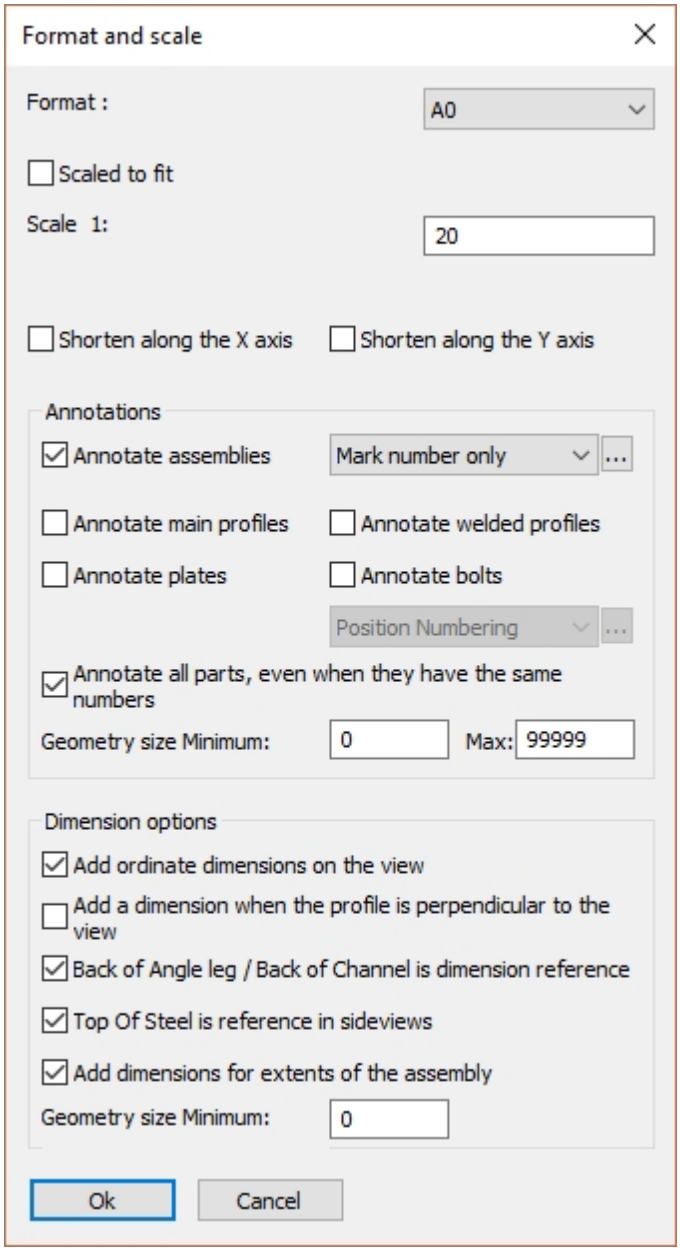
Activating this command will open the **Creating Plan views** dialog. Here you may select which view(s) to generate by checking one or more items in the list.



By selecting multiple views you will be prompted to select the position where each view is to be drawn. Unless you've planned exactly where each view is to be positioned within the sheet, this method can lead to some editing of the view placement afterwards.

Placing one view at a time may sometimes be easier. To place one at a time means repeating the **Add new plan views** process for each view.

After that the **Format and scale** dialog will appear, enabling you to set the drawing parameters:



The different options in the dialog are explored below :

Format - Set the size for the new sheet

Scaled to fit - Selecting this option will set the drawing scale to fit the new view.

Scale - Manually sets the drawing scale. The scale will be set as a ratio to 1.

Shorten along the X or Y axis - When enabled, the view will be shortened in X and/or Y directions. This is usually only used on floor plans for laying out the anchors.

Annotate assemblies - When you activate this, each assembly will automatically receive an annotation. Choose the annotation style for those annotations in the drop-down next to it. More information on the annotation styles can be found at Tools for 2D drawings

Annotate main profiles - When you activate this, each main profile will automatically receive an annotation

Annotate welded profiles - When you activate this, each welded profile (non-mains) will

automatically receive an annotation

Annotate plates - When you activate this, each plate will automatically receive an annotation

Annotate bolts - When you activate this, each bolt will automatically receive an annotation

Annotate all parts - When activated, parts that have the same part number will still each be annotated

Geometry size - This options works as a filter for the annotations. Parts smaller than the minimum or larger than the maximum will not receive an annotation. The length of the axis of the parts are used for this filter.

Add ordinate dimensions to the view - Checking this checkbox will add ordinate dimensions to the view.

Add dimension when the profile is perpendicular to the view - Checking this checkbox will add dimensions when the profile is perpendicular to the view - it will not add dimensions when the profile is at an angle to the view.

Back of Angle Leg / Back of Channel is dimension reference - Checking this checkbox will place dimensions from the back (Heel) of the channel or angle.

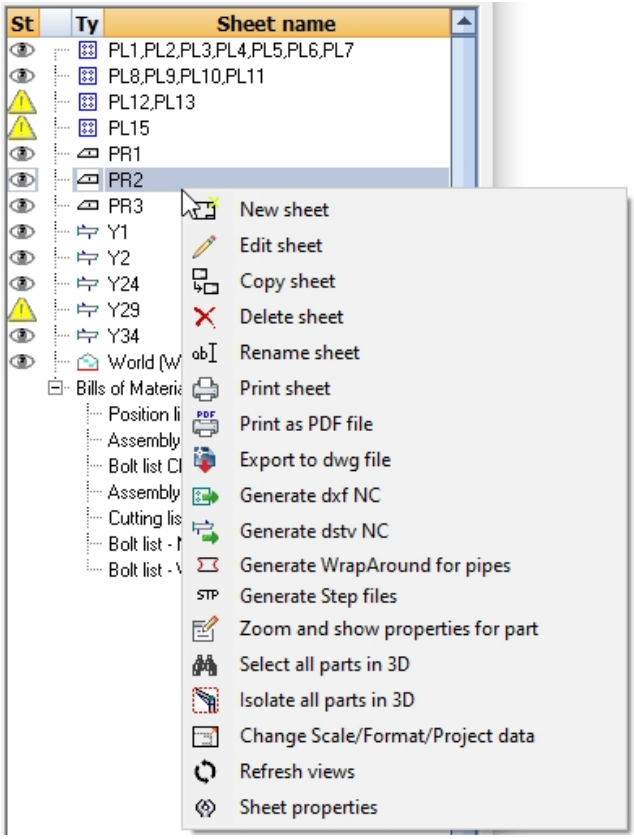
Top of Steel is reference in side views - Checking this checkbox will make the top-of-steel the reference for placing of dimensions.

Add dimensions for extents of assembly - Checking this checkbox will add O/All dimensions to the view.

Geometry size Minimum - This options works as a filter for the dimensions. Parts smaller than the minimum will not be dimension-ed. The length of the axis of the parts are used for this filter.

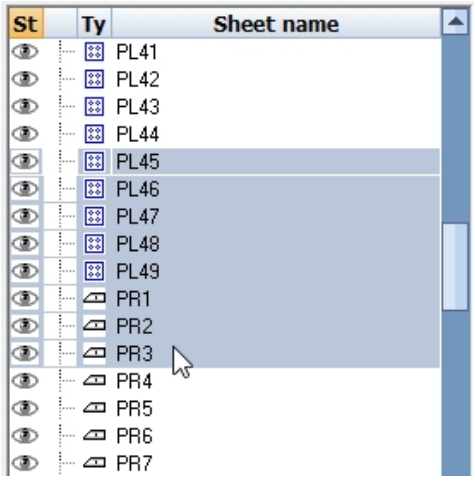
Right Click on a 2D sheet

When you right-click on a 2D sheet from within the Sheets manager  you can perform a range of actions on the sheet.



Most of these functions can also be executed on multiple sheets simultaneously.

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.



We will explain in more detail the available actions :

New sheet - A new empty sheet will be created - you will be prompted to:

- Choose the type of sheet from the drop-down options
- Choose the sheet format (Sheet size)
- Choose a settings group from the drop-down options
- Select a unique sheet name

 **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 make a copy of the entire sheet.

 **Delete sheet -** The 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 format, per different 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.

 **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.

 Generate WrapAround for Pipes - Generates a Dxf file that contains a wraparound of CHS/Pipe profiles.

 Generate Step Files - Generate a step file for each position number in the drawing. At the time of writing this command will only work in BricsCAD and when the Communicator module is installed and licensed.

 **Zoom and show properties for part** - will zoom in and highlight just 1 occurrence of the 3D model - together with the Properties window

 **Select all parts in 3D** - will zoom in and highlight all the 3D models that are drawn on the sheet

 **Isolate all parts in 3D** - This tool will hide all the 3D models that are not drawn on the sheet

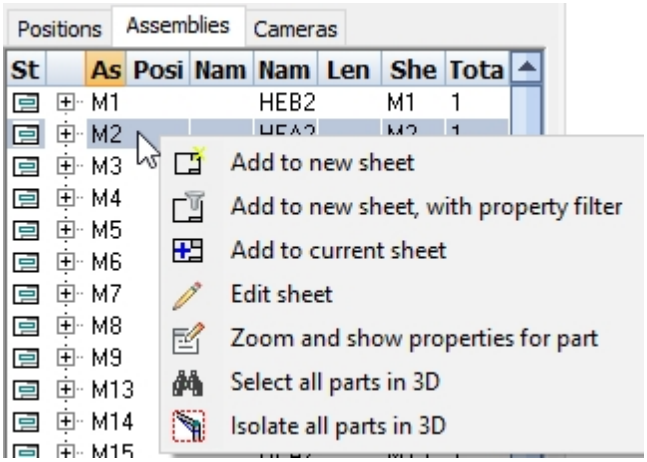
 **Change Scale / Format / Project data** - will allow you to change the sheet size, scale and project data of the selected sheet. The project data refers to the project data used in the title block. Enabling this checkbox will refresh the title block with the latest Project data as set in the 3D model.

Refresh views - When you perform this function on a General Arrangement view, all of the views on the sheet will be refreshed. All the changes you made to the 3D model up to this point will be adopted by the view(s).
If you run this tool on a non-expired shop drawing, then the BOM on the sheet will be renewed so that it matches the current amount of parts in the 3D model.
If you run this tool on an expired shop drawing, then you will be prompted to 'match up' the sheet to a new 3D model.

Sheet properties - This will enable you to edit the properties of the currently active sheet, but only work with the currently active sheet.

Right Click on a Part number

When you right-click on a position or assembly number from within the Sheets manager  you can perform a range of actions on that part number.



We will explain in more detail the available actions :

Add to new sheet - The selected part number will be added to a New Sheet, with automatic dimensioning. This will function will do the same as the Generate Pos. and Assembly sheets, but it will do so only for the selected part number.

Add to new sheet with property filter - This will do the same as above, but it allows you to filter the number of parts on the shop drawing on : All / Phase / Revision

Add to current sheet - This will add the selected part to the sheet that is currently active. At the time of writing, the views will not be dimensionned or annotated.

Edit sheet - This allows you to edit the sheet on which this part is currently drawn (if any)

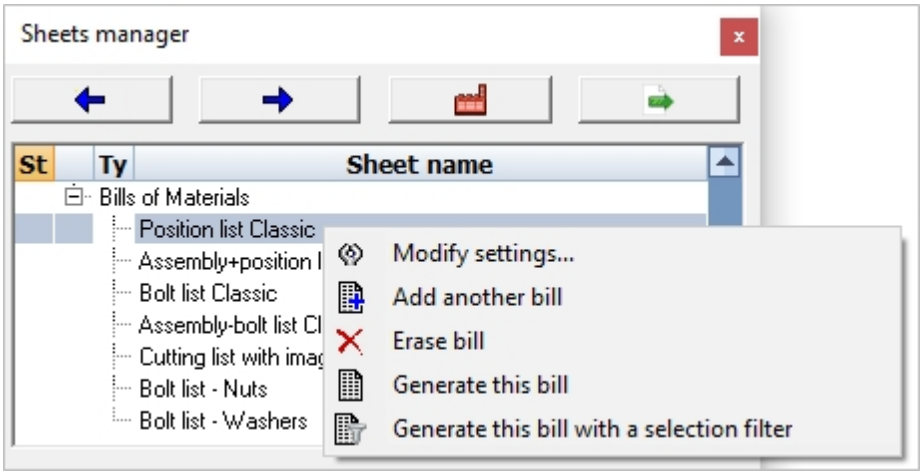
Zoom and show properties for part - will zoom in and highlight the selected part on the model - together with the Properties window

Select all parts in 3D - will zoom in and highlight all parts with this part number on the 3D model

Isolate all parts in 3D - This tool will hide all the models in 3D that are not drawn on the sheet

Right Click on a BOM Item

When you right-click on a BOM item from within the Sheets manager you can perform a range of actions on the bill.



We will explain in more detail the available actions :

Modify settings - This allows you to edit the formatting options for the bill

Add another bill - This allows you to add an additional bill to the list in the sheet manager

Erase bill - This allows you to erase a bill from the list

Generate this bill - This will generate the selected bill. The bill will be stored in the same folder as the current 3D model drawing file.

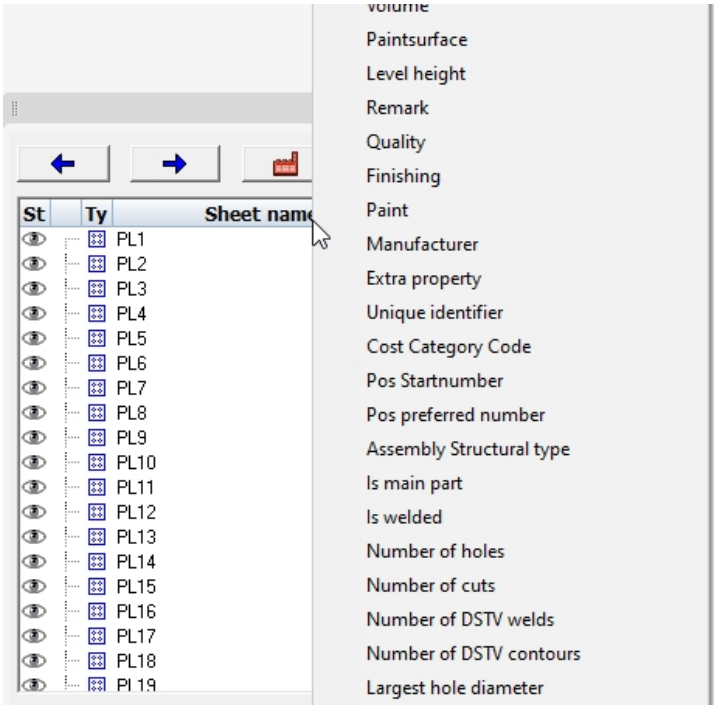
Generate this bill with a selection filter - This will do the same as the above, but it allows you to filter the number of parts in the bill on : All / Phase / Revision.

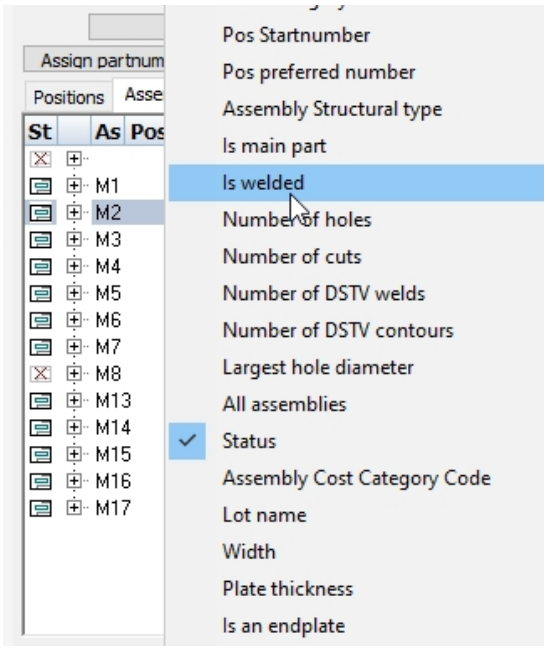
Left or Right Click on the list headers

The Sheets manager has a sheet list and a parts list.

Right-click

On both lists, we can right-click the header :



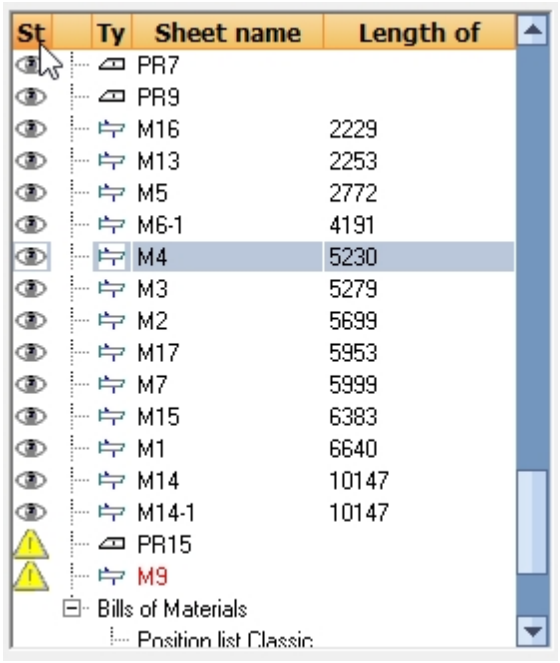


By doing so, we can activate additional columns in the lists.

This will give you more information about the part or about the parts on the sheet.
All the properties that Parabuild stores for the part can be shown this way.

Left-click

On both lists, we can also left-click the header :



This will influence the sorting of the entries in the list.

It would allow you to see all of the expired drawings together if you click on the *Status* header, like shown in the image above.

Parabuild often needs multiple columns to complete the sorting. In the above example, the first sorting columns is *Status*, the second is *Length of main member* and the third is *Sheet Name*.

To specify the priority list, Parabuild sets the last selected column first, and the second last column as second, etc..

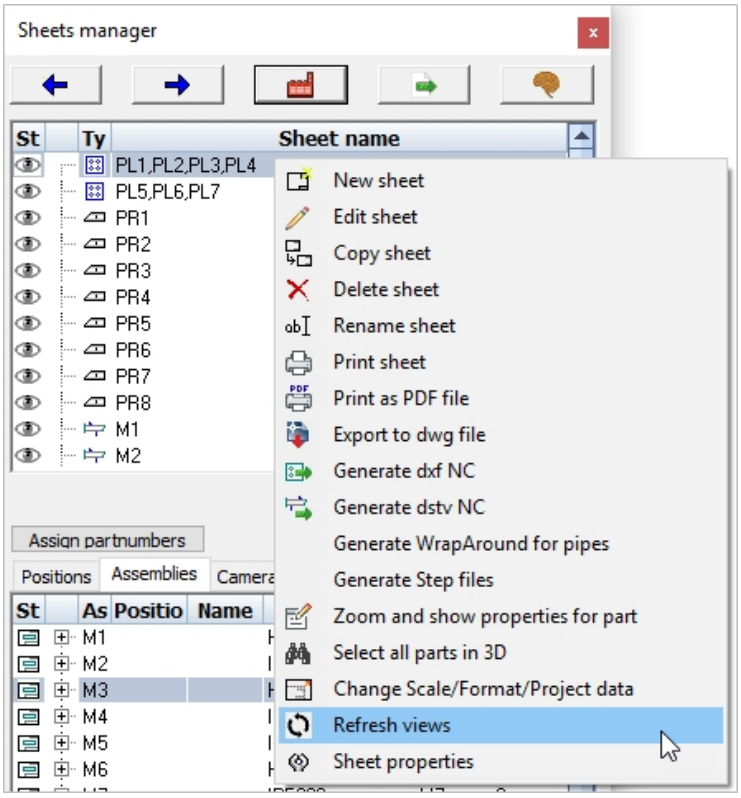
So to achieve the above sorting you would have to select the columns in this order :

1. Sheet name
2. Length of main member
3. Lastly, the Status

Refresh Views

There are 4 methods to refresh the view(s) of a 2D sheet.

- By right-clicking on a sheet in sheets manager
- By right-clicking on a view (if the BricsCAD/AutoCAD context menu is enabled)
- In the properties panel of the view
- By pressing the <F5> function key while 1 or more views are currently visible



On a GA drawing

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).

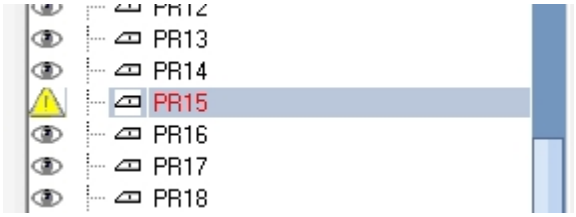
While updating the view, Parabuild will keep the dimensions and annotations that are drawn on the view and if necessary they will be moved or stretched.

It is possible that the connection between 2D view and 3D model is lost. This happens when a 3D model was deleted. Or it might happen when a 3D model falls out of scope of the camera. In this case the dimension remains untouched and the value of the dimension will contain '?' because it becomes immeasurable.

On a shop drawing

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 in the sheet manager.
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 views of the position or assembly, and nothing further is changed or added.

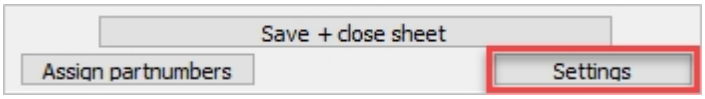
You will still need to modify the dimensions for the elements of the assembly that are added or modified.

This tool is especially useful for large assemblies in which you have invested time adapting them.

Settings for Workshop Drawings

Command - **Prb_Sheets**

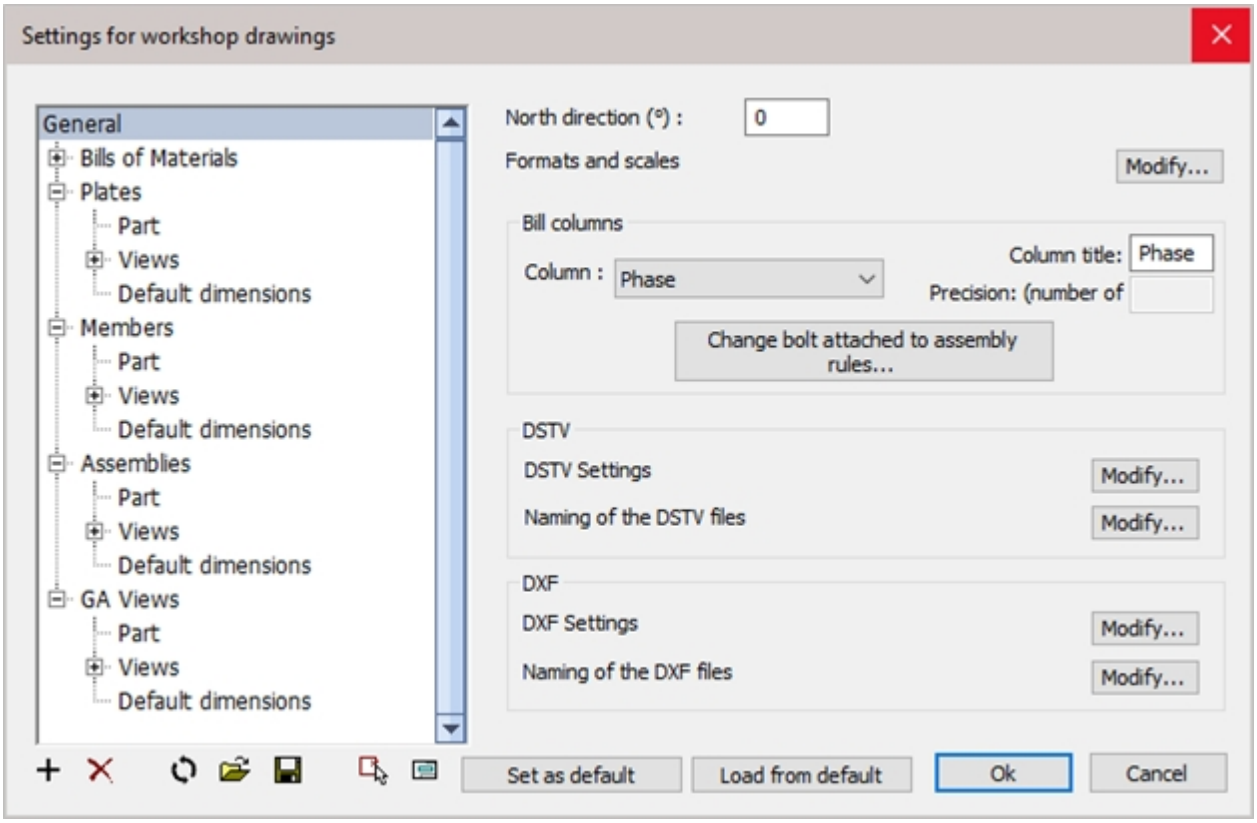
The settings dialog can be accessed form the Sheets Manager / Settings button



In the dialog box, the *list on the left* is divided into 6 main headings:

- General - Where you can change the sheet format and scale, organize the columns for the bills of material, and edit the settings for DXF and DSTV files
- Bills of material - Where you can create or erase a new bill, determine a folder where the bill will be stored and retrieved, and change the name and settings.
- The remaining headings: Plates / Members / Assemblies / and GA Views share the same settings format and dialogs and are dealt with collectively. Each heading has 3 items: Part / Views / Default dimensions

Selecting one of the 6 headings from the list will display different options in the right of the dialog



At the bottom of the dialog, we have a few controls that need some explanation :

 This will create a new group. A group could be for example a truss or a stair. A new group should only be created if you need to be able to change all of the object type's output settings.

 This will erase a group. The four default groups can't be erased.

 This will remove all of the changes done in the current dialog, and revert all of the options to the default 'out of the box' settings. This will not erase settings that have been stored to file.

 With this button you can load a settings file that has been stored previously. All of the settings on this dialog box and their sub-dialogs are stored in this file.
These files are stored in the folder : \Parabuild\Pb_Lib\Workshop Drawings\Settings\

 With this button you can save all of the settings on this dialog box and their sub-dialogs into a file.
These files are stored in the folder : \Parabuild\Pb_Lib\Workshop Drawings\Settings\

 This feature has been temporarily disabled because of stability issues

 This feature has been temporarily disabled because of stability issues

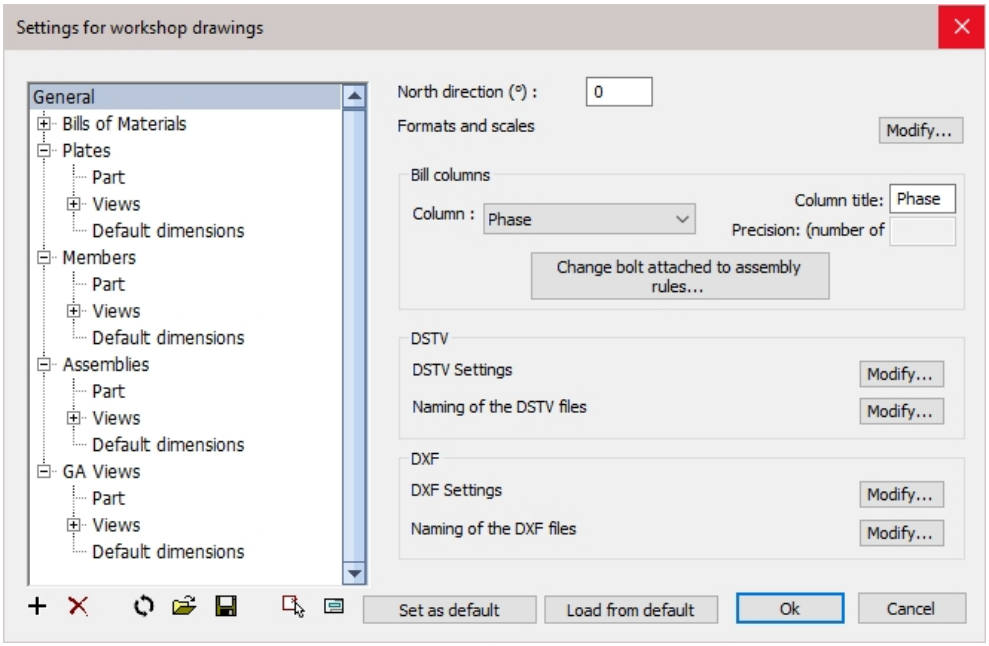
Set as default Use this button to store all of the current settings into the standards file.
The next time that Parabuild starts, it will load this file automatically.
This standards file is located here : \Parabuild\DefaultWorkshopSettings-English.dat

Load from default Use this button to load all of the settings from the standards file.

This standards file is located here : \Parabuild\DefaultWorkshopSettings-English.dat

General

Selecting **General** will open the following dialog:



All of the options on this dialog are explored below :

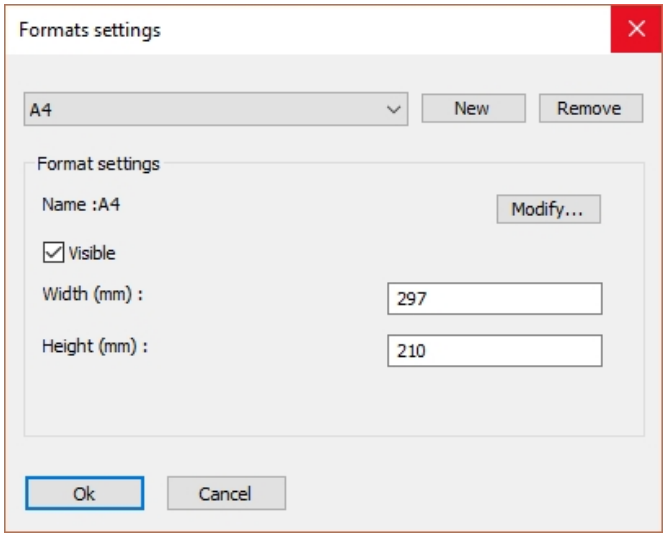
North direction (°)

The shop drawings can optionally use the cardinal directions for views. Parabuild uses the current drawing's NORTHDIRECTION variable to know this direction.

Formats and scales

This dialog box allows you to add more formats that can then later be used by Parabuild for the generation of sheets.

Do note that after adding a new format, you might also have to create a new title block that should be used in combination with the new format. More about title blocks can be found in the Settings for page and layout.



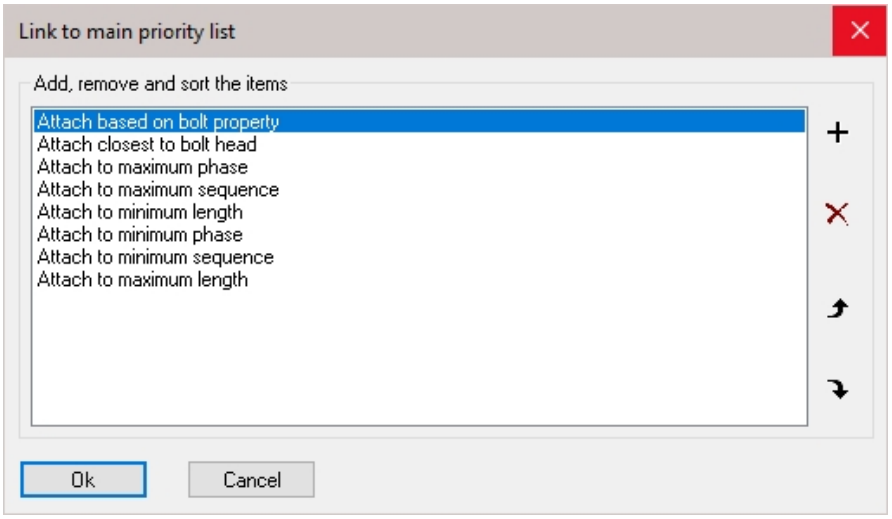
Bill Columns

This allows you to configure the bill column of the bill of materials.

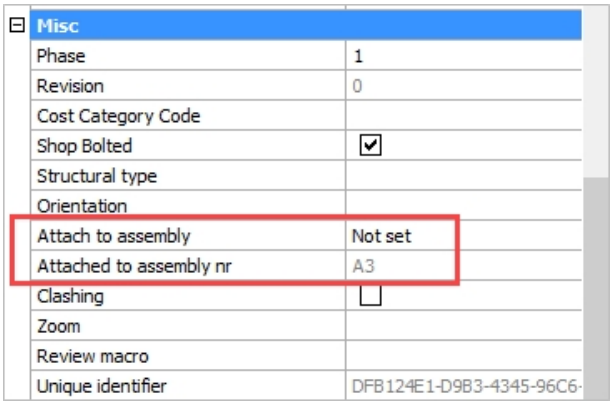
You may also change the column title to suit your preference. For columns that contain numerical values, you may add the precision value - i.e. 0.01 / 1 etc...

Change bolt attached to assembly rules

With this priority list you can influence the automatic assignment of reference assembly of all bolts. The list is ordered according to priority. You can add, delete, and sort all items in the list.



In exceptional cases where the automatic choice taken by Parabuild is not as desired, it is still possible to manually set the reference assembly in the properties of the bolt:



The property **Attach to assembly** allows you to override the main priority list for the bolt by choosing **Closest to bolt head** or **Closest to Bolt end.**

The property **Attached to assembly nr** can't be changed. This will indicate which assembly the bolt is currently attached to. It is the assembly that was effectively chosen by the either the main priority list or by the *Attach to assembly* property if it has been set.

Dstv Settings

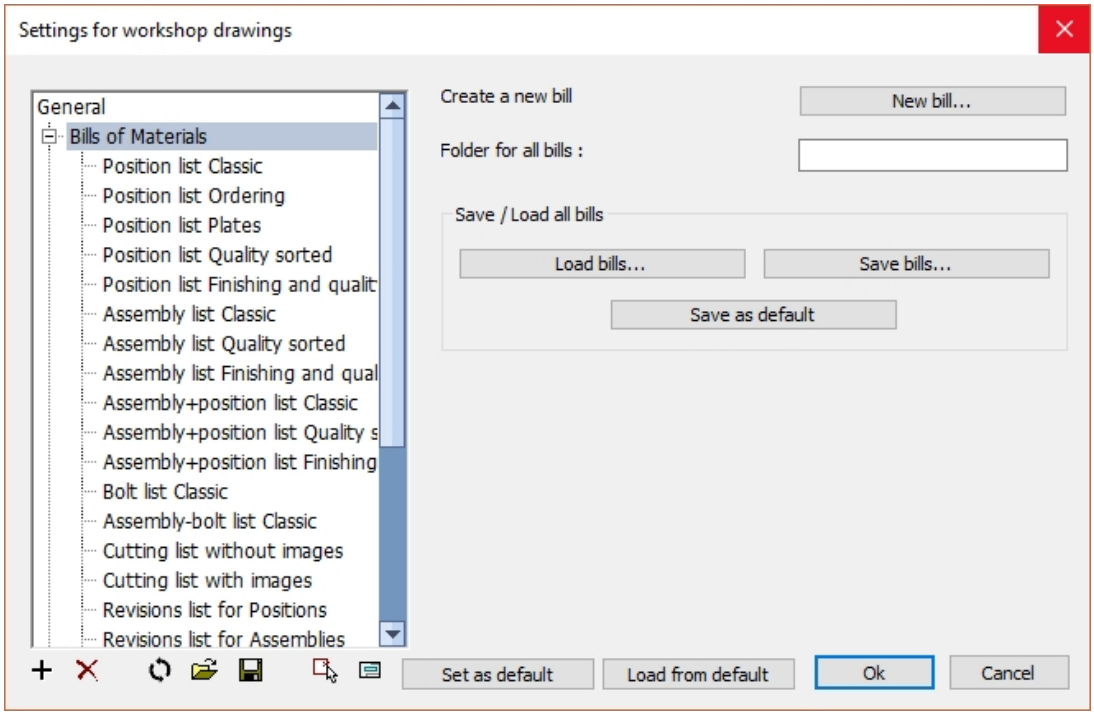
For more information about the DSTV options, see the DSTV NC Settings topic.

Dxf Settings

For more information about the DXF options, see the DXF Settings topic.

Bills of Material

Here you can activate, add or remove bill of materials.



All of the options on this dialog are explored below :

New bill - Create a new bill of materials from scratch.

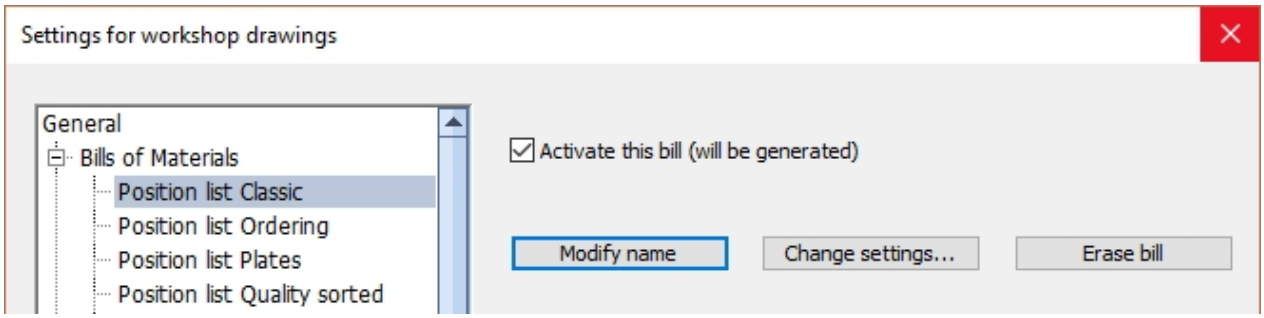
Folder for all bills - Here you may create add a sub folder name, and all of the bills will be written in this sub folder in the same folder as the 3D drawing

Load bills - Load a file that contains all your favorite bills.

Save bills - The current bills will be stored 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 is selected from the list on the left, you can perform the following actions:



All of the options on this dialog are explored below :

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

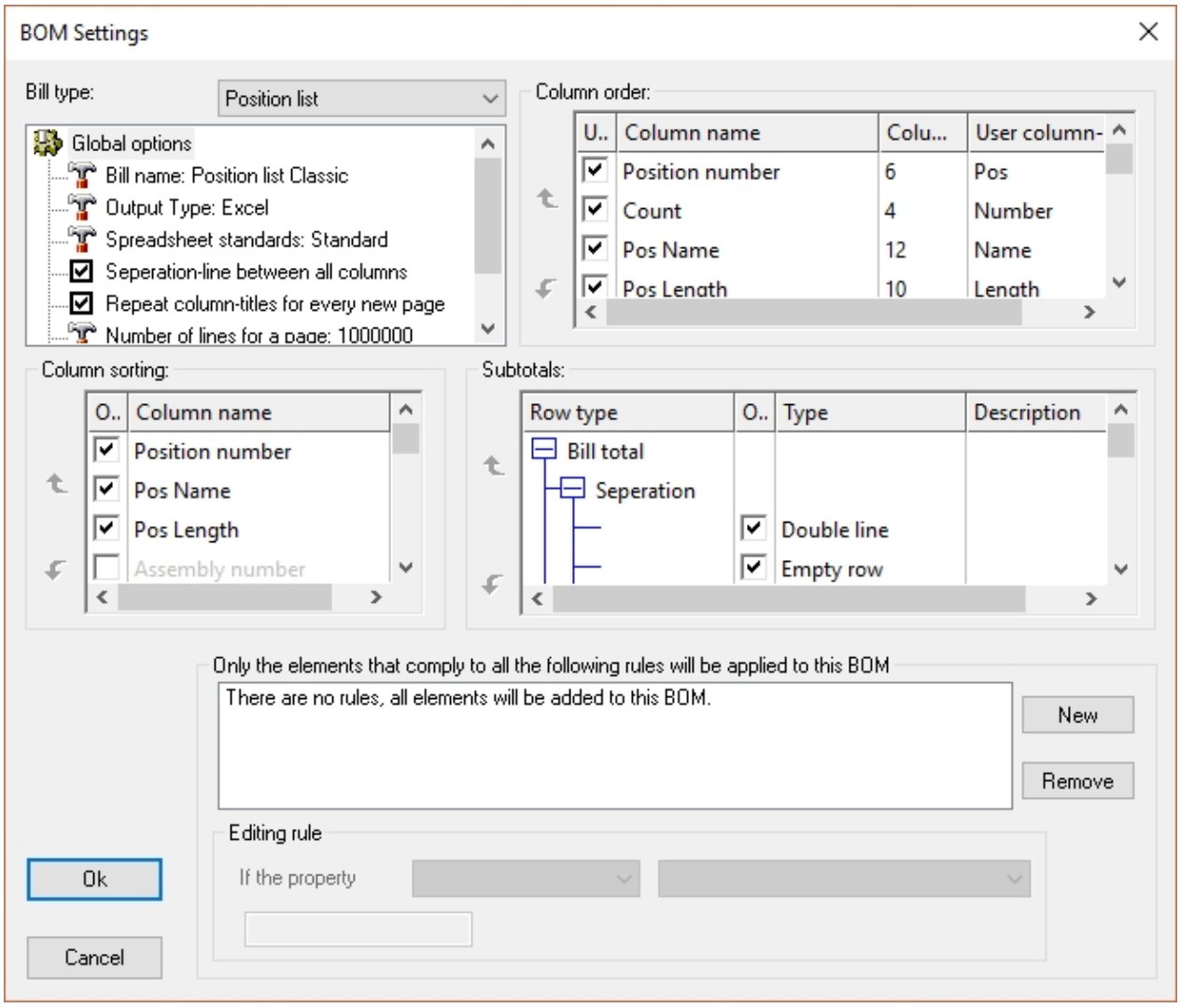
manager

Change settings - Here you can customize the Formatting options which includes: **Bill type**, **Columns**, **Columns order**, **Column sorting** options, and **Sub-totals**

Erase bill - Permanently erases this bill

Bill Formatting Options

Here you can customize the **Bill type** , **Column order**, **Column sorting** options, and **Sub-totals**



All of the options on this dialog are explored below :

Bill type:

Bill name - here you can enter a new bill name

Output type - here you can select from the drop-down either: Excel / Regular text / Parabuild spreadsheet

Spreadsheet standards

Separation line between all columns

Repeat column title for every new page

Number of lines for a page

Add totals to the end of the bill

Column order:

Each column may be activated or deactivated

Column name - this is the descriptive name of the column

Column width - this may be edited as required

User column - these are the headings as they appear on the bill

Column Sorting:

These columns will determine the sorting of the parts on the bill.

Sub-totals:

This table is slightly more complicated, because every row no longer works independently. This table has a tree structure.

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.

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:

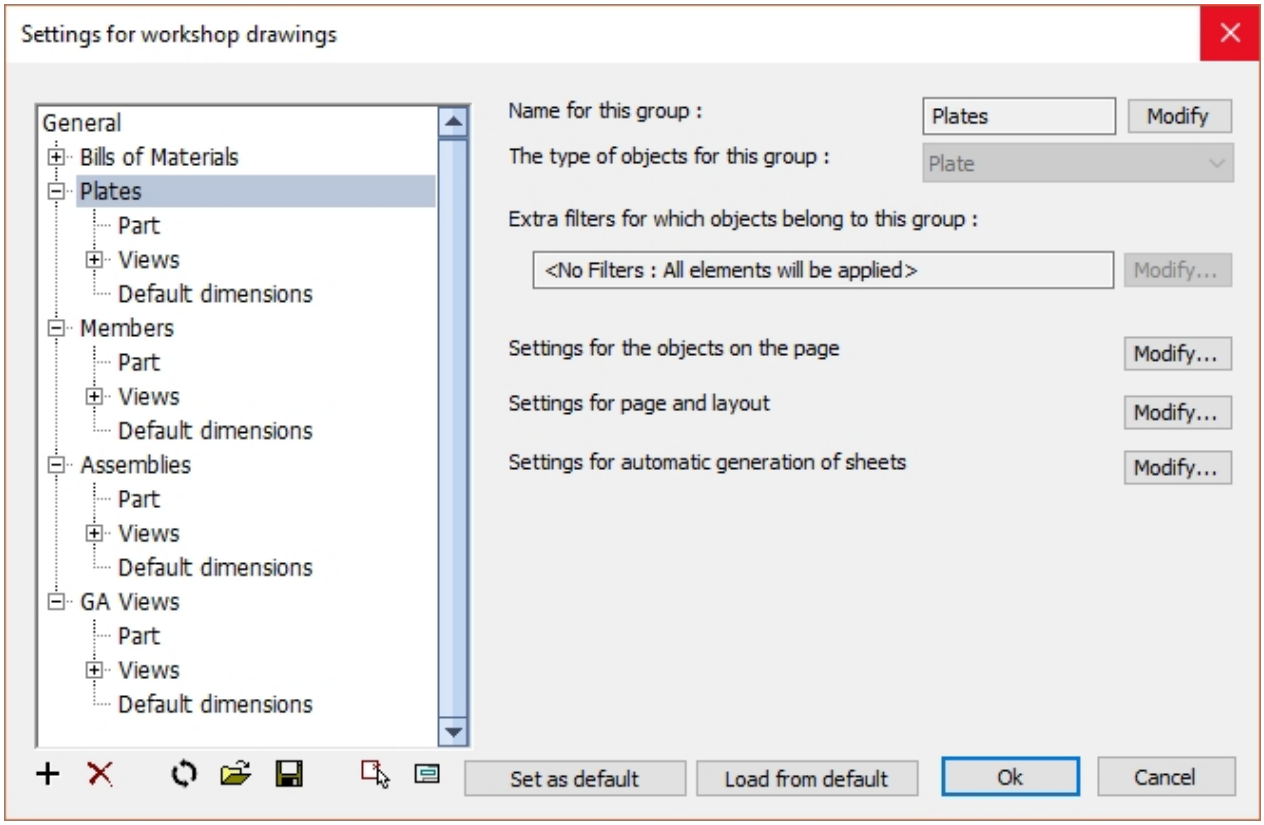
If the property **Position Quality starts with SS**

In another part list, but then for concrete the following rule would be used:.

If the property **Position Quality starts with concrete**

Plates / Members / Assemblies / GA Views

Here we look at the main groups (**Plates, Members, Assemblies** and **GA Views**) collectively as they share the same settings and accompanying dialogs.



The sections and sub dialog boxes are explored here :

Name for this group - This will reflect the name from the left column.

Settings for objects on the page

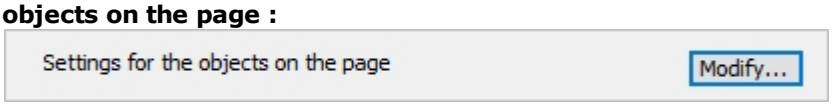
Settings for page and layout

Settings for automatic generation of sheets

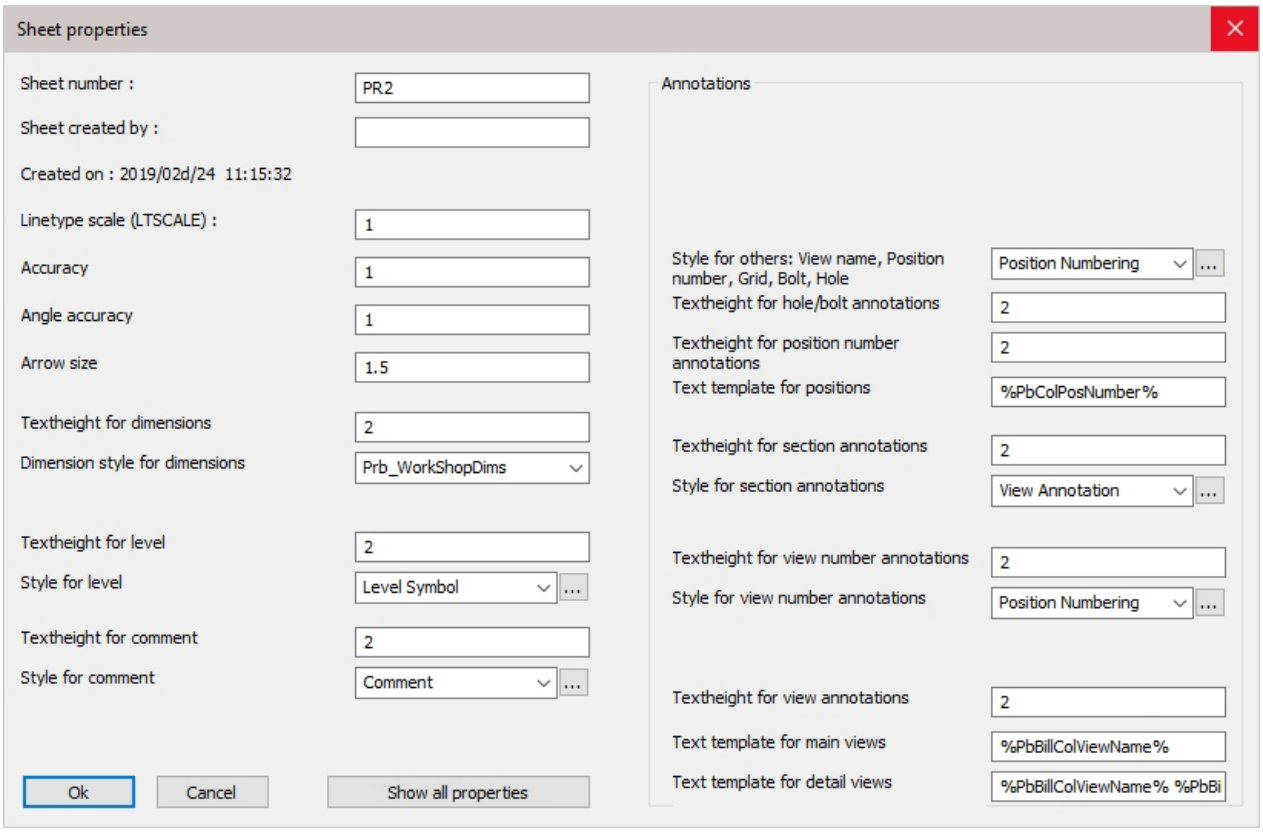
Sheet properties

This dialog may be opened in one of two ways:

1. By Right clicking on a 2D sheet in the Sheets manager. This will enables you to edit the properties of the currently active sheet, editing these properties will only work with the currently active sheet.
2. It can also be opened by clicking **Settings** in the Sheets manager and selecting either the **Plates / Members / Assemblies / Views headings** - and selecting the **Settings for the objects on the page :**

   Editing the settings by this route will effectively establish them as a default for all future sheets of this type.



Non applicable settings will be enabled or disabled for certain sheet types. This is shown in the descriptions below.

In this dialog box you can change the following settings :

Sheet number - By default, the sheet number is the same as the sheet name, here, you may change the number manually

Sheet created by - For tracking purposes

Created on - The created time and date are recorded automatically

(*These first 3 properties are written to KISS files - See* Export BIM to File *for more information on KISS file settings*)

Line type scale - The scaling of dashed lines and center lines

Accuracy - The measurement accuracy for linear dimensions

Angle accuracy - The measurement accuracy for angle dimensions

Arrow size - The size of the arrows of dimensions.

Text height for dimensions - The text height of all dimensions on the sheet

Dimension style for dimensions - The dimension style (AutoCAD) for all dimensions on the sheet

Text height for level - The text height for all level annotations on the sheet

Style for level - The text style for all level annotations on the sheet

Text height for comment - The text height for all comment annotations on the sheet

Style for comment - The text style for all comment annotations on the sheet

Text height for assembly annotations - The text height of all assembly number annotations on the sheet. (*Not applicable for Plates and Members*)

Text style for assembly annotations - The text style of all assembly number annotations on the sheet. (*Not applicable for Plates and Members*)

Text template for assemblies - This is a text override for assembly number annotations. Enter the text that should be shown in the frame of the annotation. Use variables that can be entered between % symbols. Look at all the available variable names by clicking the button *Show all properties* button at the bottom of the dialog.
If you leave this value empty then the override is not in effect and the text of the annotation style will be used.

Style for others: View name, Position number, Grid, Bolt, Hole - The style for all other annotations such as members, holes, bolts, etc...

Text height for hole/bolt/part annotations - The text height for annotations on holes and bolts.

Text height for position number annotations - The text height for position number annotations.

Text template for positions - This is a text override for position number annotations. Enter the text that should be shown in the frame of the annotation. Use variables that can be entered between % symbols. Look at all the available variable names by clicking the button *Show all properties* button at the bottom of the dialog.
If you leave this value empty then the override is not in effect and the text of the annotation style will be used.

Text height for section annotations - The text height of the annotation of end plate views and section views on assembly workshop drawings. (*Not applicable for Plates, Members and GA views*)

Text style for section annotations - The text style of the annotation of the end plate views and section views on assembly workshop drawings. (*Not applicable for GA views*)

Text height for view number annotations - The text height of view numbers 1,2,3,4 that are shown on workshop drawings. (*Not applicable for GA views*)

Text style for view number annotations - The text style of view numbers 1,2,3,4 that are shown on workshop drawings. (*Not applicable for GA views*)

Text height for grid annotations - The text height for the grid balloon text.

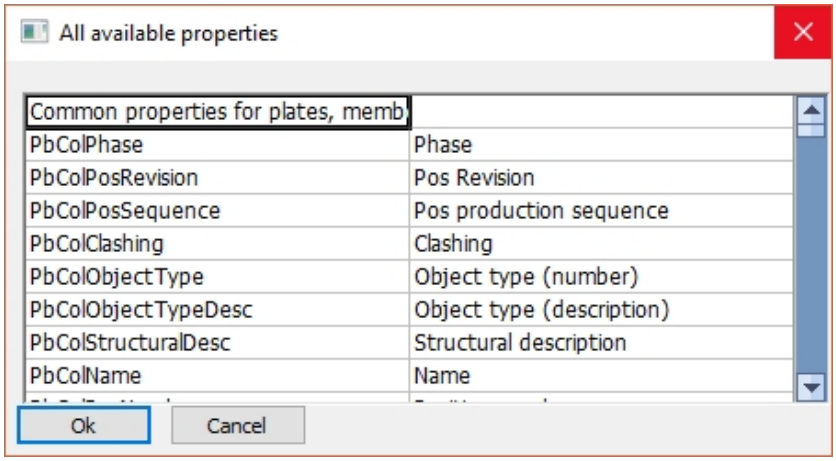
Text height for view annotations - The text height will generally be larger than other annotations.

Text template for main views - This is a text override for main GA view annotations. Enter the text that should be shown in the frame of the annotation. Use variables that can be entered between % symbols. Look at all the available variable names by clicking the button *Show all properties* button at the bottom of the dialog.
If you leave this value empty then the override is not in effect and the text of the annotation style will be used.

Text template for detail view - This is a text override for detail view annotations. Enter the text that should be shown in the frame of the annotation. Use variables that can be entered between % symbols. Look at all the available variable names by clicking the button *Show all properties* button at the bottom of the dialog.
If you leave this value empty then the override is not in effect and the text of the annotation style will be used.

Show all properties

With this button you can review all variables that can be used in the text template of annotations.



Empty values in the fields

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.

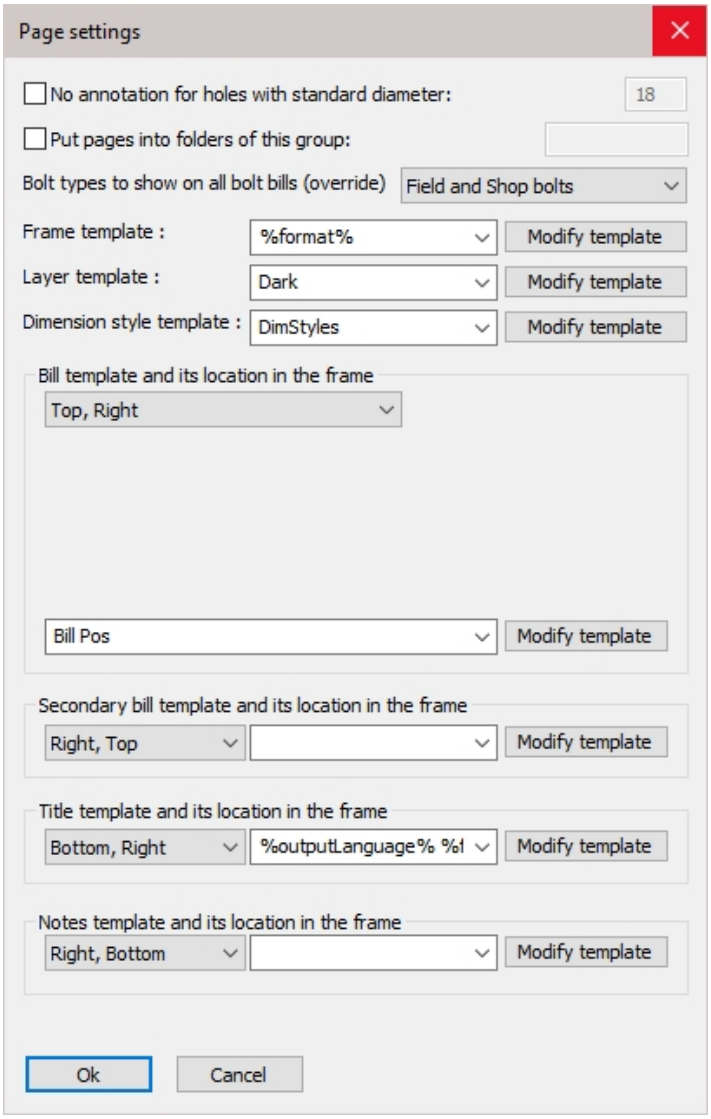
Also if the text template of an annotations is empty then the text template in the annotation style will be used. The reason why these settings still exist here is so that we can still choose different text heights on a per-sheet basis.

Settings for page and layout

In this dialog we can choose the standard drawing sheet templates.

It is possible to personalize the templates to reflect your company's or clients identity.

To do this, you can open and modify these template .DWG files in the Parabuild directory: Parabuild\Pb_Lib\Workshop drawings\



In this dialog box you can change the following settings :

No annotation for holes with standard diameter - Checking the checkbox will result in holes with the given diameter not getting any annotations on the 2D drawings. Not implemented yet

Put pages into folders of this group - Not implemented yet

Bolt types to show on all bolt bills (Override) - With this option you can set the type of bolts that will be fed into all the bolt bills that are activated in the lower settings

Frame template - By default, this is set to %format%. It is important to keep this variable, because the frame template to be used is dependent on the format. As the format is chosen automatically by Parabuild, we can't enter an actual format here but we use the variable to automate the frame template selection.

Layer template - Here you may switch the layer template from light to dark. Both templates were created to accommodate users working with a light background while others prefer to use a dark background for sheets. Do note that the layers are usually only inserted the first time per 3D drawing. After the first time, the layers exist already and they are not overwritten.

Dimension style template - The dimension styles in this template drawing will be inserted in the current 3D drawing, so that the styles can be used for dimensions on the 2D drawing.

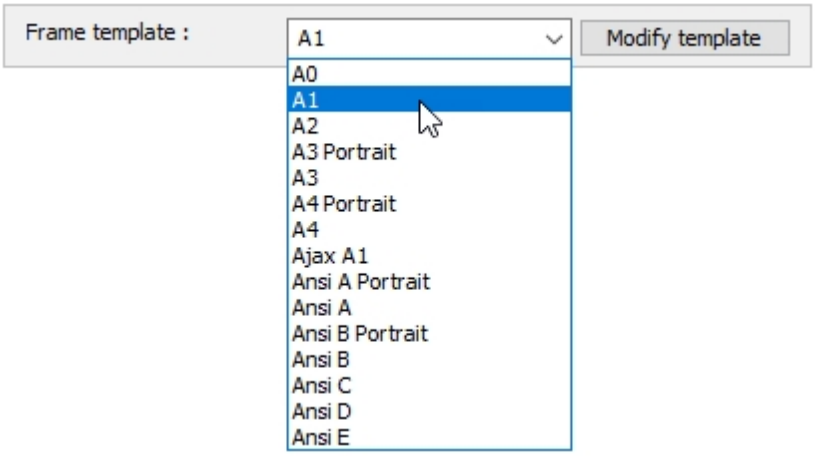
Bill template and its location in the frame - This is the main bill that contains the list of all parts on the 2D drawing

Secondary bill template and its location in the frame - The secondary bill is usually used to list all the site and/or shop bolts that are linked to the parts on the 2D drawing.

Title template and its location in the frame - The title blocks of the sheet can be personalized by opening the drawings in the folder : \Parabuild\Pb_Lib\Workshop drawings\Title Templates\
The default value is "%outputLanguage% %format%", which could result in for example "English A4". So in that case the file "English A4.dwg" will be used in the Title templates folder.
Both strings can variate, so therefore the variables are necessary.
If the user works for several contractors that each want their own logo on the title block, then this can be accommodated by using the variable %Contractor_Name%.
This variable can be entered in the Project data of the 3D drawing.
For this to work, the user will also have to create the title blocks for each contractor.

Notes template and its location in the frame - The notes is an extra template that can be used to freely add more information on the sheet.

More about frame templates

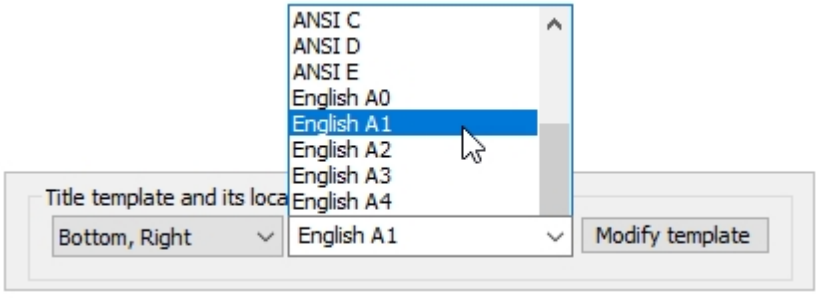


These template files are located in the folder \Parabuild\Pb_Lib\Workshop Drawings\Sheet Templates\.

The overall size of the templates will match with the size of the standardized ISO or ANSI sheets.

Should you wish to create a non-standard template, then you can add a new frame template here, but don't forget to also add the accompanying format in the Format scales dialog.

More about Title Templates



These template files are located in the folder \Parabuild\Pb_Lib\Workshop Drawings\Title Templates\.

You can add or delete text fields, rename the fields, rearrange them, and add your own or the companies identity in the form of a logo.

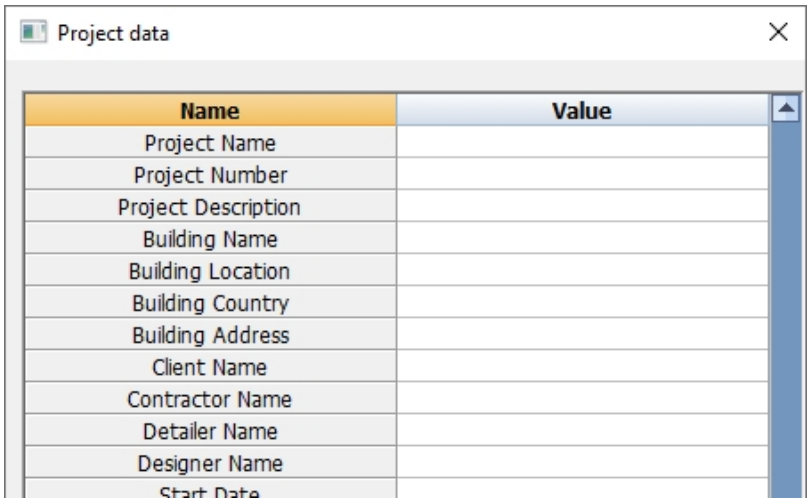
Name:Project_Name		Date:Prb_Date	Pr nr:Project_Number
Project:Project_Description		Drawn:Detailer_Name	Scale:Prb_Scale
Contractor:Contractor_Name	Rev:Revision	Checked:Checked	Units:Prb_Unit
File:PrB_FullFileName		Approved:Approved	Format:Prb_Format

The default template is shown above.

Each row first contains the title header and then the variable - e.g. Name: Name

The variables will be resolved through the Parabuild General / Project data dialog.

By entering the appropriate values adjacent to the Name will substitute the variables in the title block template.



Engineering Project Somewhere		PRB Associates PRB House - Somewhere	
Name:Whatever		Date:18/4/2019	Dwg nr: 11001
Project:Whatever		Drawn:Michael Brown	Scale:1/20
Assignment:Whatever	Rev: 0	Checked:David Jones	Units:mm
File:PrB_FullFileWhatever		Approved:Fred Smith	Format:A1

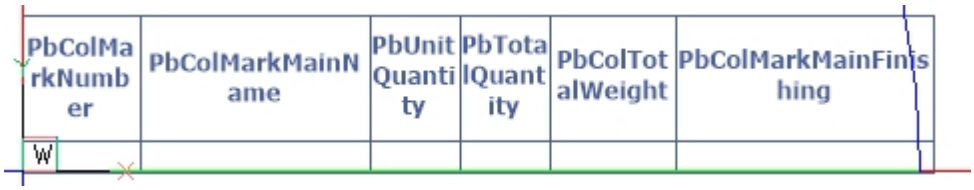
A typical edited title block with fields added, together with the company logo

More about Bill templates

These template files are located in the folder \Parabuild\Pb_Lib\Workshop Drawings\Bill Templates\.

The column titles are normal texts inside a table object. Just double-click a text field to edit them.
You can add or delete columns or rows, rename rows, and rearrange them all with the BricsCAD/AutoCAD table tools.

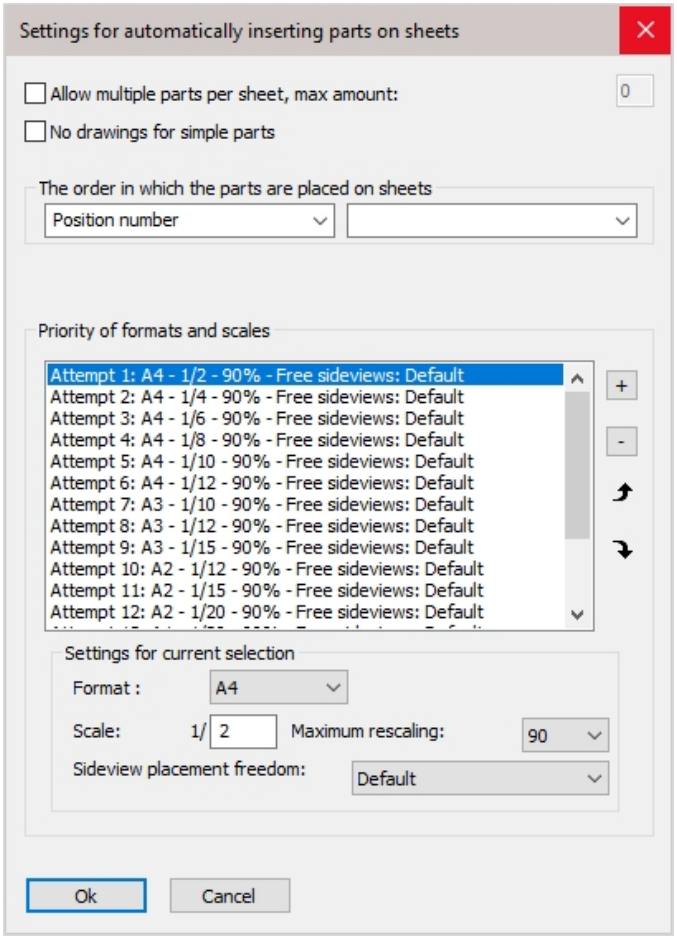
The column headings are variables and will be replaced with their values in the bill. For a list of available variables go to Settings / Settings for objects on the page / Show all properties



Settings for automatic generation of sheets

This dialog allows us to influence the automatic generation of shop drawing sheets.

More specifically, we can choose when and which sheet formats and scales Parabuild may use.



In this dialog box you can change the following settings :

Allow multiple parts per sheet, max amount - If disabled, Parabuild will always produce 1 sheet per part

No drawings 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. Not implemented yet

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.

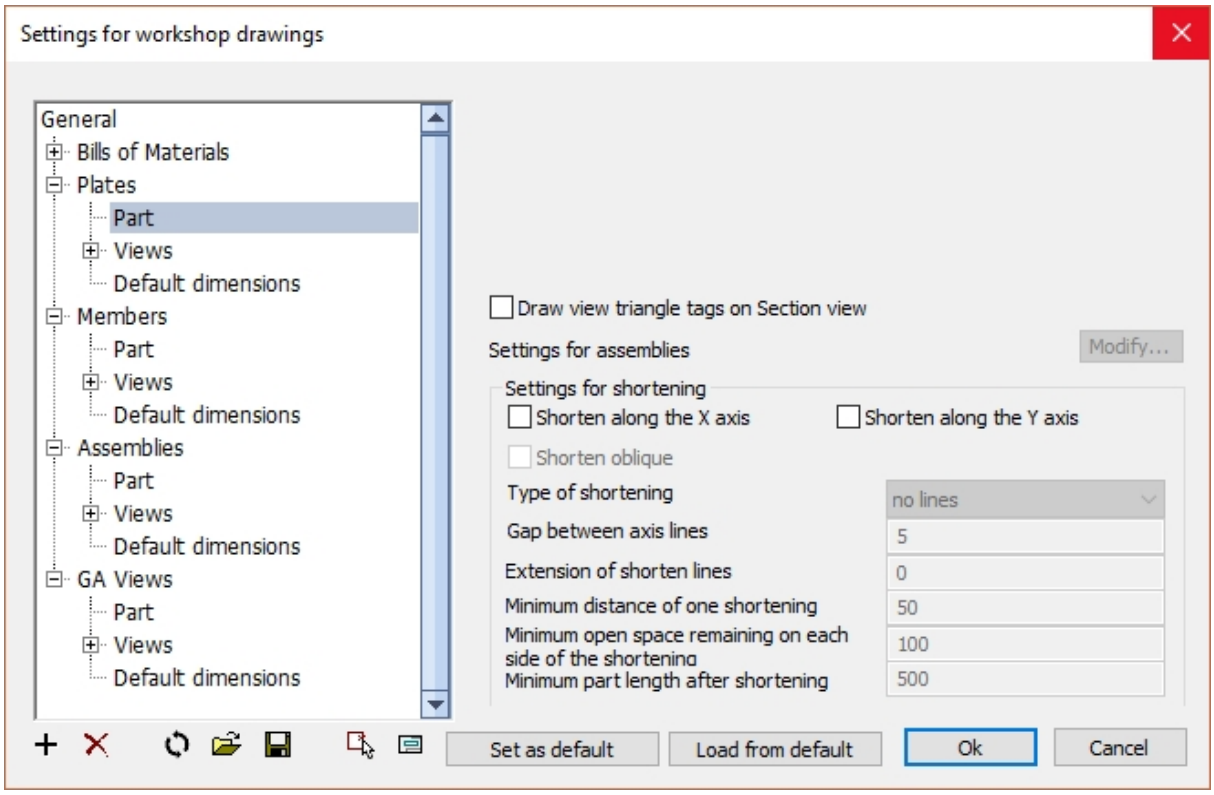
Maximum re-scaling - With which percentage can Parabuild re-scale the views even further in order to let the part fit on the sheet? This re-scaling will apply for the most part to 3D views on the sheet. Other view types are not re-scaled automatically.

Side view placement freedom - Enter the freedom that Parabuild has in rearrangement of the side views:

- **Always -** The side views can be drawn freely on the sheet.
- **Default -** The side views will be outlined most of the time, but in some cases where the part is big and not linear Parabuild will not align the side views in order to achieve more efficient space usage on the sheet (for example for a handrail).
- **Minimal -** The side views always need to be aligned with each other.

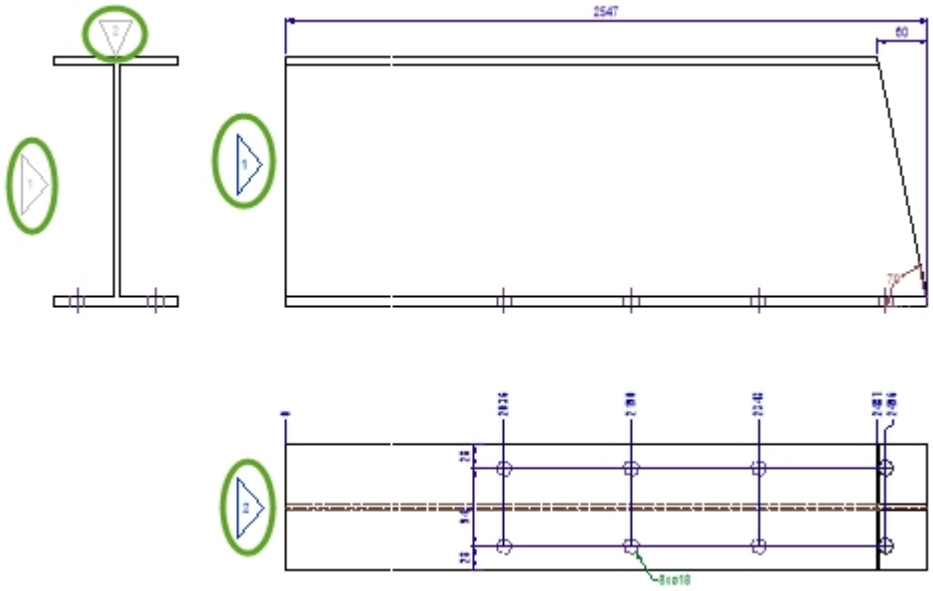
Part

In this dialog you can change the shortening options of the side-views as well as configure the extra section views and end plate views.



In this dialog box you can change the following options :

Draw view triangle tags on section view - When enabled, this will draw triangular annotations with the view numbers next to each side view and also on the section view:



Settings for assemblies - For more about these options, see the Assembly-specific settings topic.

Shorten along the X or Y axis - When active, the side views will be shortened linearly to save space on the sheet. The side views are always shortened equally.
Features such as holes or cuts are not shortened.
Use the options below to configure the 'aggressiveness' of the shortening.

Shorten oblique - When active, also oblique lines will be shortened.

Type of shortening - Choose the type of lines that should be drawn at each break location that was created by the shortening.

Gap between axis lines - This is the small gap between two shortening lines. If no shortening lines were chosen this will create a small opening of the shortened lines.

Extension of shorten lines - How far the shortening lines should extend beyond the model lines of the part.

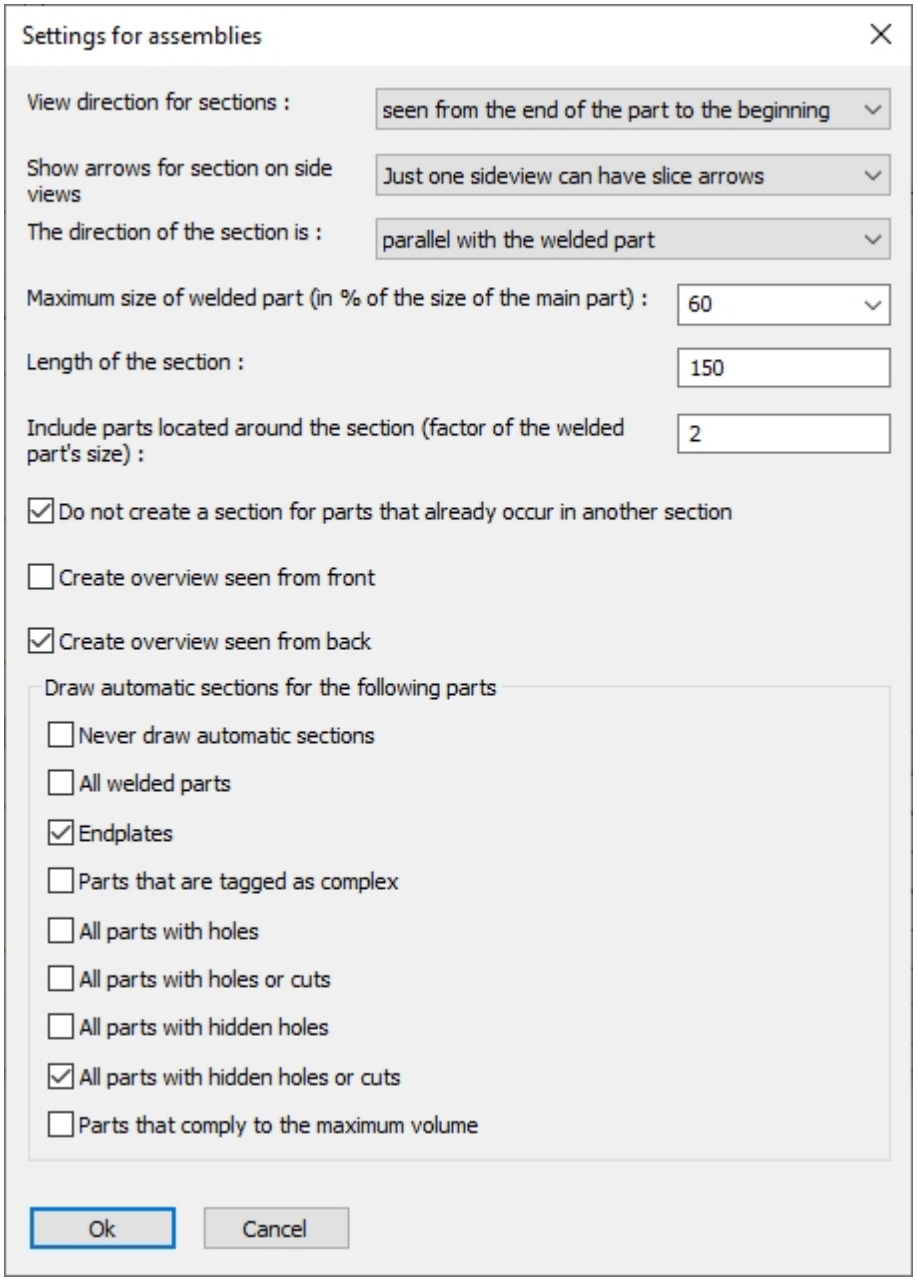
Minimum distance of one shortening - The minimum distance that is required between 2 features. If the distance is smaller, the shortening will not be done. This value will prevent shortenings being created between the holes of a hole pattern.

Minimum open space remaining on each side of the shortening - The open space that needs to be kept between the shortening line and the feature.

Minimum part length after shortening - The minimum length of the part after all the shortening have been applied. This setting will avoid a 8000mm long profile being shortened into a 50mm part, which would look strange on the sheet.

Assembly-specific settings

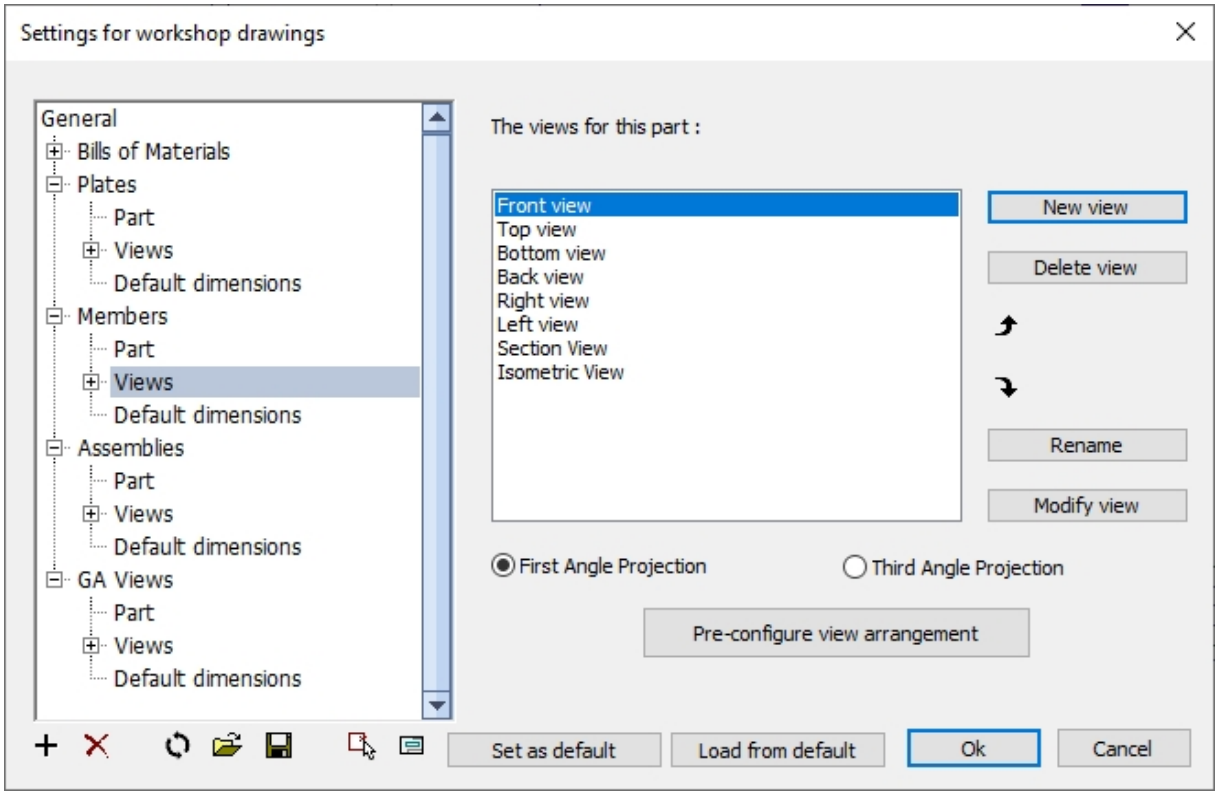
In this dialog you can influence which additional section views Parabuild should automatically add to the assembly drawings.



This dialog is due for review - most options are replaced by newer ones.

Views

In this dialog you can configure all of the views that should be drawn on the sheet.



In this dialog box you can change the following options :

New view - Create a new view by entering a unique name

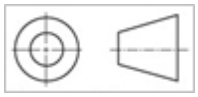
Delete view - Enables you to delete the selected view

Rename - Enables you to rename the selected view

Modify view - For more information about this, see the Modify view settings topic

First angle projection - First angle orthographic projection is used often in the EU

Third angle projection - Third angle orthographic projection is used widely in the USA



Pre-configure view arrangement - Allows you to change and re-arrange the views of the profile or assembly settings.

Note

Using the *Pre-Configure* dialog box as well as the First/Third angle radio buttons can cause a reset of all of your views.

It is therefore best to first use the Pre-configure dialog to configure the views.

If needed after that you can tweak the arrangement some more with the View Settings and Configure view directions dialogs view.

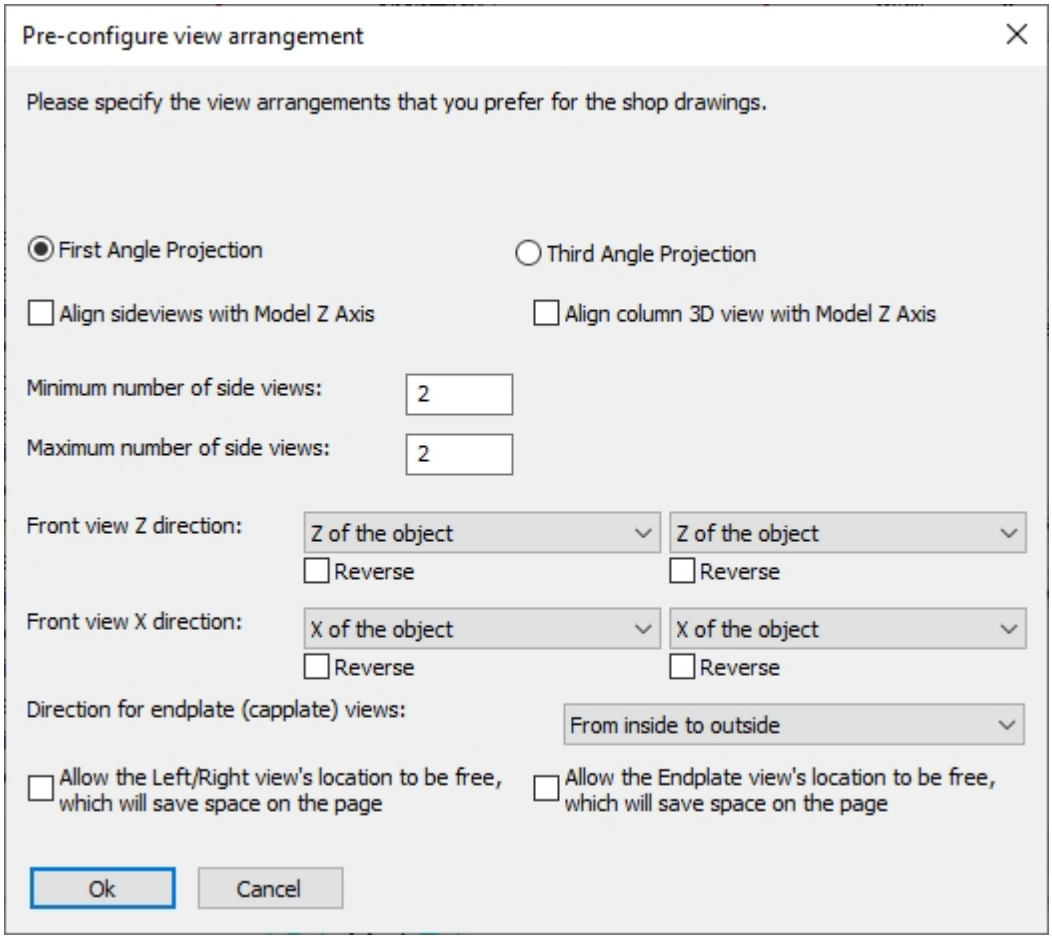
Pre-configure View Arrgt.

The Pre-configure dialog box will change and re-arrange the views of the profile or assembly settings.

Using this dialog box can cause a reset of all of your views.

It is therefore best to first use this dialog to configure the views.

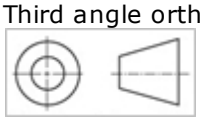
If needed after that you can tweak the arrangement some more with the View Settings and Configure view directions dialogs view.



In this dialog box you can change the following options :

First angle projection - This option will arrange the side views and the left/right views so that they are arranged according to the First angle orthographic projection.
First angle orthographic projection is used often in the EU.

Third angle projection - This option will arrange the side views and the left/right views so that they are arranged according to the Third angle orthographic projection.
Third angle orthographic projection is used widely in the USA.



Align side views with model Z axis - When activated, columns will be drawn upright, beams

horizontal, rafters sloped, etc...

Each part will have the same slope in 2D as it does in 3D.

When deactivated, the main member of the assembly will be drawn horizontal on the page.

Align column 3D view with Model Z axis - When activated, the 3D view of columns will be drawn upright.

The 3D view of all other assemblies will be drawn horizontal, even rafters and bracings.

Minimum/Maximum number of side views - The difference between the minimum and maximum number of side-views determines how many side-views will be activated dynamically.

A dynamically chosen side-view is a view that will only be drawn when there are parts on the view that would be invisible on the minimum side views.

If you want a fixed number of views, set minimum and maximum to the same number.

Front view Z direction - The Front view is the main view, on which most other views are based. The **Z** direction is the viewing direction.
With this option you can choose the viewing direction to follow :

- The object itself, so that we're looking at the part locally (this is the default)
- The world directions, so that we can look at it from the 3D model's direction
- The cardinal directions. The AutoCAD/BricsCAD variable NORTHDIRECTION will be used to determine the cardinal directions in 3D. The front view will then point to the North/South/East/West direction in the 3D model.

This option works in combination with the *Align side views with model Z axis* option.
The *Align side views* option decides most of the placement of the side views. The *Front view Z direction* option only has an influence on the viewing direction of the side views

Front view X direction - This is the same as *Front view Z direction*, but this relates to the X direction of the view instead of the viewing direction.
The Y direction of the view is derived from the Y and X directions.

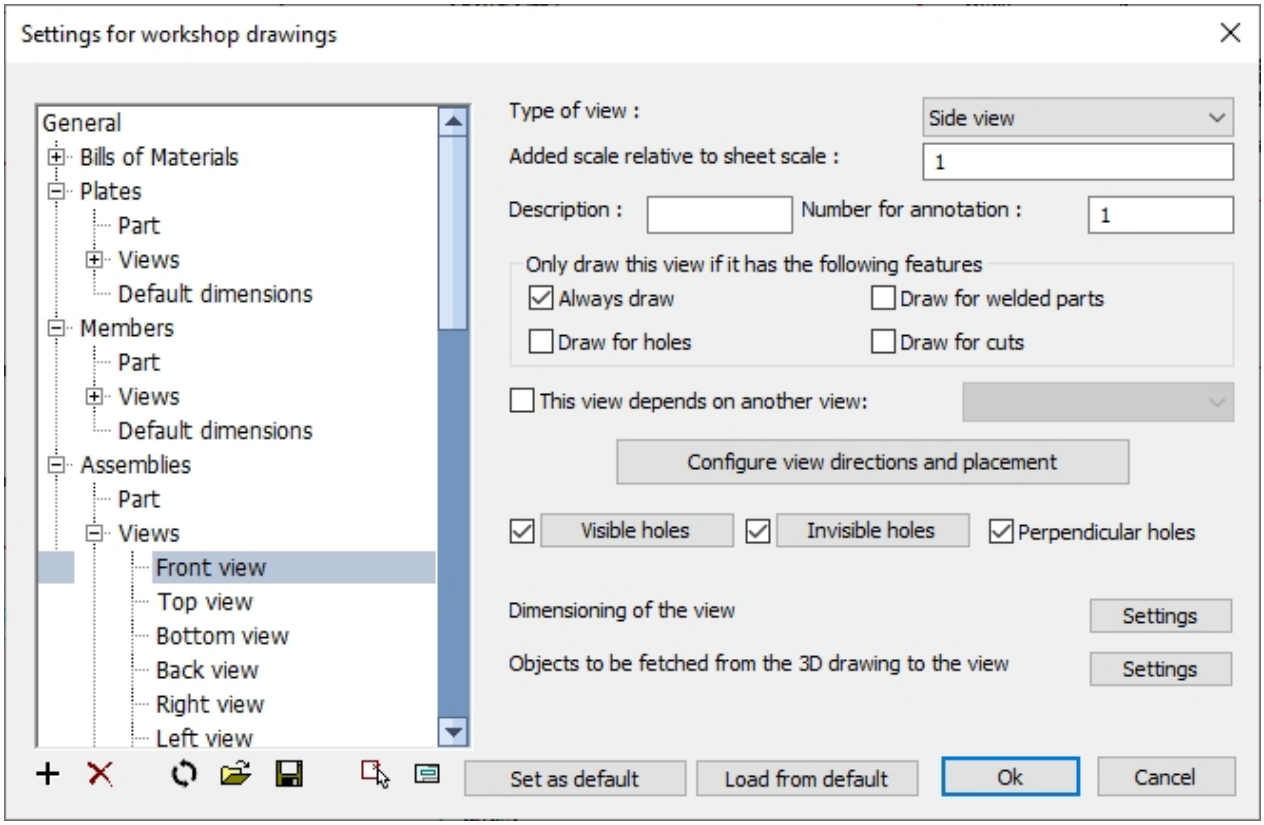
Direction of End plate views - Use one of the options to change the viewing directions of all of the end plate views

Allow Left/Right view's location to be free - When this is enabled, the Left/Right views will not be located next to the correct side views as instructed by the angle projection rules.
One would enable this to allow Parabuild to fit the part on a smaller format and a better scale because Parabuild will be able to move the view to a free space.

Allow the endplate view's location to be free - Same as above but this applies to all the endplate views.
The endplate views are not restricted by the angle projection rules.

Modify View Settings

This dialog allows you to change the settings of all the views that will be generated in the future.



In this dialog box you can change the following options :

Type of view - The main purpose of this type if for Parabuild so that it know whether this is a side view, an endplate view or something else

Added scale - This currently has no effect as the scale is determined dynamically by Parabuild.

Description - This description is just a data field stored in the view. It could be used as extra information in an annotation.

Number for annotation - This is the actual view number that will be shown in the triangular annotation of side views.

Only draw this view if it has the following features - When none of the checkboxes are enabled, then the view will never be drawn. When *Always draw* is activated it will always be drawn. When one of the other 3 checkboxes is enabled, the view will only be drawn when there are parts on the view that would be invisible on the other side views.

This view depends on another view - Enabling this will make this view's directions dependent on another view. This makes it easier to do changes to the view direction options, as there are less options that have an influence.

Configure view directions and placement - For more about this see the Configure view directions topic

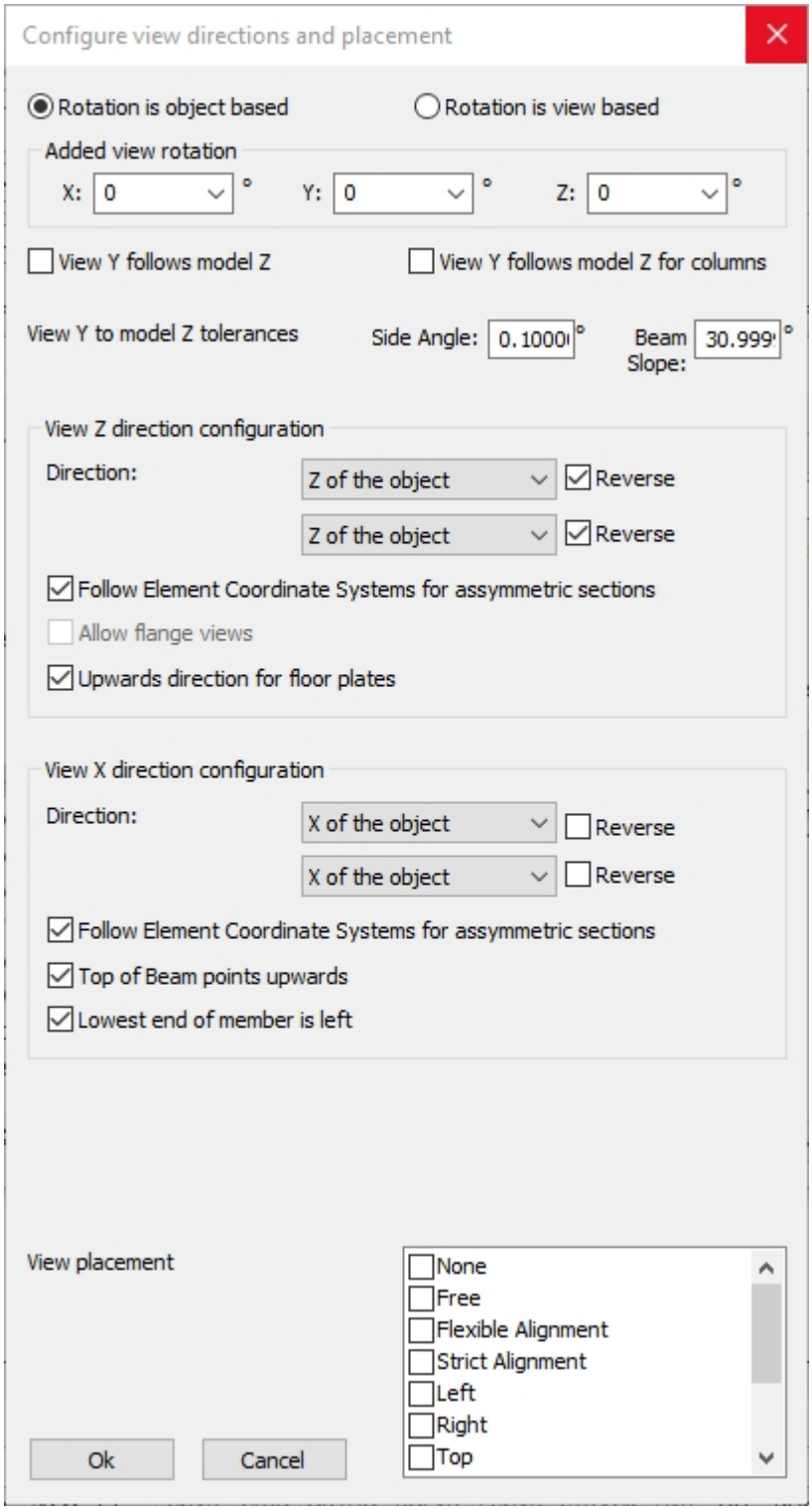
Visible/Invisible/Perpendicular holes - These options allow you to change the appearance of holes, or completely disable them. Perpendicular holes are holes seen from the side.

Dimensioning of the view - This allows you to activate or deactivate the automated dimensions and the different annotation types for the view.

Objects to be fetched from the 3D drawing to the view - In this dialog you can choose whether the axis lines, grid lines, 3D Solids, ... should be copied over to the 2D view. For more about this, see the Objects to be copied from 3D to 2D views topic.

Configure View Directions

This dialog box determines the viewing direction and the placement rotation of the view on the sheet.



In this dialog box you can change the following options :

View Y follows model Z - When activated, columns will be drawn upright, beams horizontal, rafters sloped, etc...
Each part will have the same slope in 2D as it does in 3D.
This has no influence on columns.

View Y follows model Z for columns - Same as the above option, but this option applies to columns only.

View Z direction configuration

- **View Z direction** - The Z direction is the viewing direction.
  With this option you can choose the viewing direction to follow :
  - The object itself, so that we're looking at the part locally (this is the default)
  - The world directions, so that we can look at it from the 3D model's direction
  - The cardinal directions. The AutoCAD/BricsCAD variable NORTHDIRECTION will be used to determine the cardinal directions in 3D. The front view will then point to the North/South/East/West direction in the 3D model.

- **Follow Element Coordinate Systems for asymmetric sections** - This is an override of the above *View Z direction* option.
  When enabled, it forces Parabuild to use the X Y and Z directions of the object if the object has an asymmetric section shape such as a channel or an angle. This will ensure the for example that the "back plate" of a channel is always drawn visible or always drawn invisible, and not mixed visible/invisible.

- **Upwards direction for floor plates** - This is an override of the above *View Z direction* option.
  When enabled, all floor plates will always point upwards no matter what the other direction options are set to in this dialog.

View X direction configuration

- **View X direction** - This is the same as *View Z direction*, but this relates to the X direction of the view instead of the viewing direction.

- **Follow Element Coordinate Systems for asymmetric sections** - This is an override of the above View *X direction* option.
  When enabled, it forces Parabuild to use the X Y and Z directions of the object if the object has an asymmetric section shape such as a channel or an angle. This will ensure the for example that the "back plate" of a channel is always drawn visible or always drawn invisible, and not mixed visible/invisible.

- **Top of Beam points upwards** - This is an override of the above View *X direction* option.
  When enabled, all of the 'Top of Steel' faces of beams in 3D will always be drawn upward on the view no matter what the other direction options are set to in this dialog.

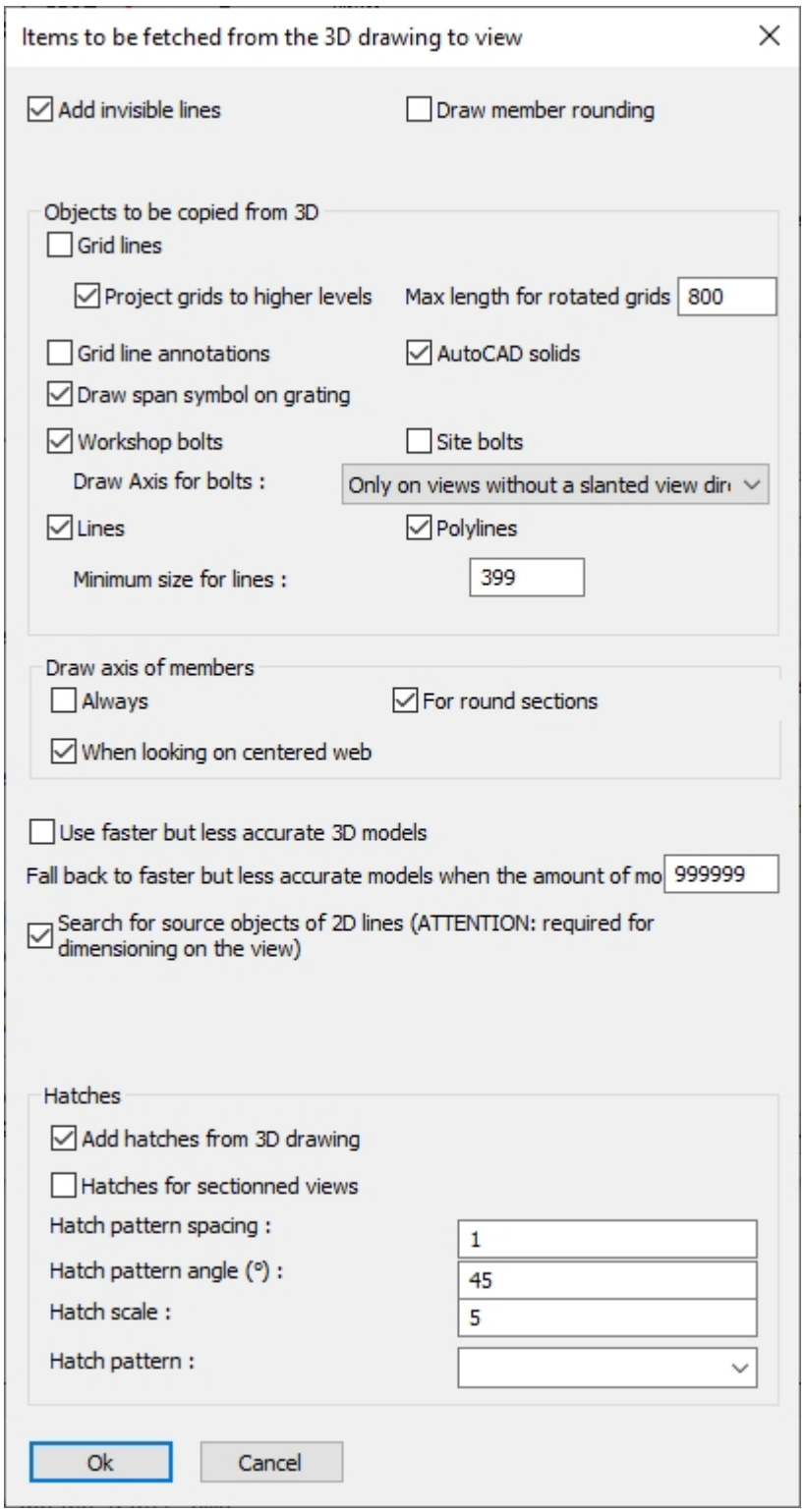
- **Lowest end of member is left** - This is an override of the above View *X direction* option.
  When enabled, all of the members that have a slope such as rafters and bracings will always be drawn with their "lowest" end in 3D on the left hand side on the view. No matter what the other direction options are set to in this dialog.

View placement - With this option you can influence how free Parabuild can move the view around on the sheet.

The *Strict Alignment* option refers to the *First/Third Angle projection* rules : When enabled, the location of the side views and Left/Right views will be strictly done according to the *First/Third Angle projections*.
The Flexible Alignment does also follow the *First/Third Angle projection* rules at first, but the views will be displaced when the view alignment takes up a lot of space on the sheet. This can happen for railings, trusses and skids.
Depending on the type of view, Parabuild does not always adhere to all of the options that are enabled here.

Objects to be copied from 3D to 2D views

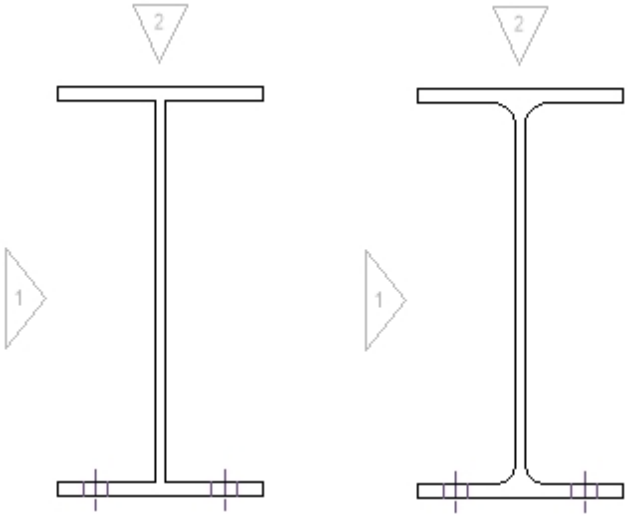
This dialog box allows us to decide which 3D objects are added to the 2D view.
We can also configure how the translation from 3D to 2D should be done.



In this dialog box you can change the following options :

Add invisible lines - On workshop drawings, these lines should always be drawn. But on 3D views or GA views, the invisible lines will quickly overcrowd a view.

Draw member rounding -



Example 2D view with member rounding disabled

Example 2D view with member rounding enabled

Grid lines - Enable this option to copy grid lines to the 2D view

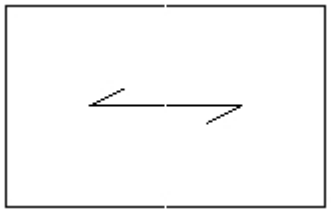
Project grids to higher levels - When enabled, the grid lines on the 'floor' will be copied to higher levels, even if the floor falls out of scope of the view.

Max length for rotated grids - Thanks to this option, elevation views will get grid lines. Use this option to set the size of these 'rotated' grid lines.
Set this value to 0 to disable grid lines on elevation views all together.

Grid line annotations - This option causes the grid lines to automatically get grid line annotations on the view.

AutoCAD solids - When enabled, AutoCAD 3D Solids will be shown in the view

Draw span symbol on gratings - When enabled, grating panels will get a span symbol on 2D views. The span symbol indicates the direction of the load bars :



Workshop bolts - Typically, workshop bolts are shown on both the shop drawings and on the GA views

Site bolts - Typically, site bolts are not shown on the shop drawings

Draw Axis for bolts - When enabled, a red axis will be drawn for all bolts

Lines/Polylines - When enabled, lines/polylines that are drawn in 3D will also be copied to the view.
This option has no effect on workshop drawings because the workshop drawings are not based on a camera. Instead they work based on a part number filter.

Minimum size for lines - If a line is smaller than the size entered here, then the line will not be copied to the view. This option was created to overcome performance problems that happened with large drawings that contain many small lines. 2D drawings that act as an underlay will often contain many of these lines.

Draw axis of members - This option will draw the red axis line for members.

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.

Hatches - When enabled, all of the parts that are cut off by the view limitation of the camera will receive a hatch.
Use the additional options to configure the hatch's appearance.

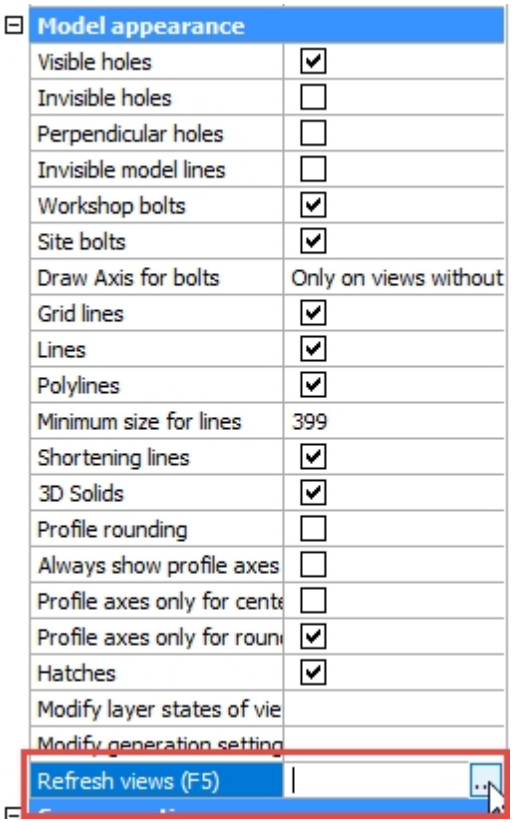
Change Existing View Settings

To change the settings of an existing view you can use the Properties panel.

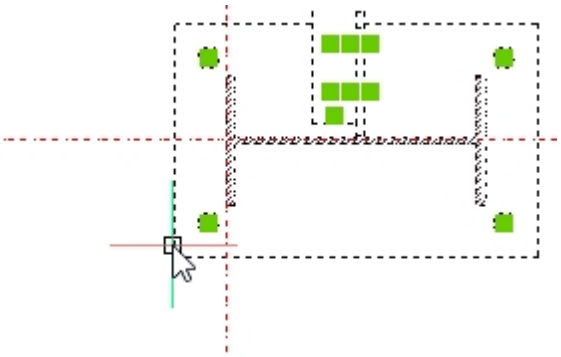
Properties changed here will only apply to the chosen view, and will not affect the general settings.

Note that none of the *Model Appearance* options will be reflected on the view after you change them.
These appearance settings have an impact on the 2D lines in the view, and in order to see the changes you have to refresh the view with the button in the view.

Alternatively, you can also push the <F5> function key. This will refresh all the views that are currently visible on your screen.



Double-clicking an existing view will open the Modify view Settings dialog directly.
This dialog will then change the options of the selected view only.

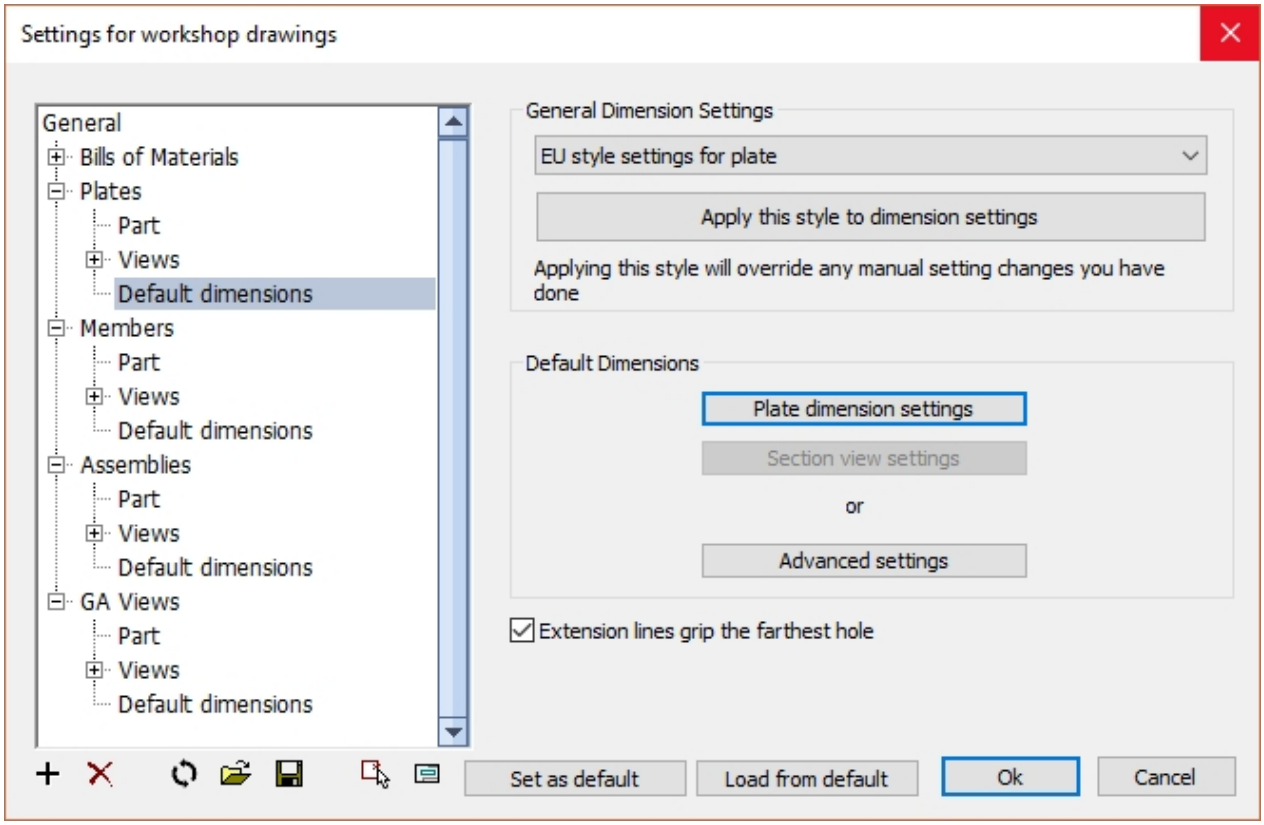


Default Dimensions

Default dimensions are dimensions that Parabuild draws automatically for you on the workshop drawings of plates, members and assemblies.

These are programmed dimensions based on the settings in this dialog box.

You can access this dialog from the Sheets Manager / Settings



In this dialog box you can change the following options :

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 this option after you've changed other settings, otherwise your other settings will be lost.

Plate dimension settings - The settings for the dimensions on all plates. This dialog box contains instant help for each settings. (*This button is only available for Plates)*

Side view settings - The settings for the dimensions on all side views. This dialog box contains instant help for each settings. (*This button is only available for Members and Assemblies*)

Section view settings - The settings for sections and end plate views. (*This button is only available 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.

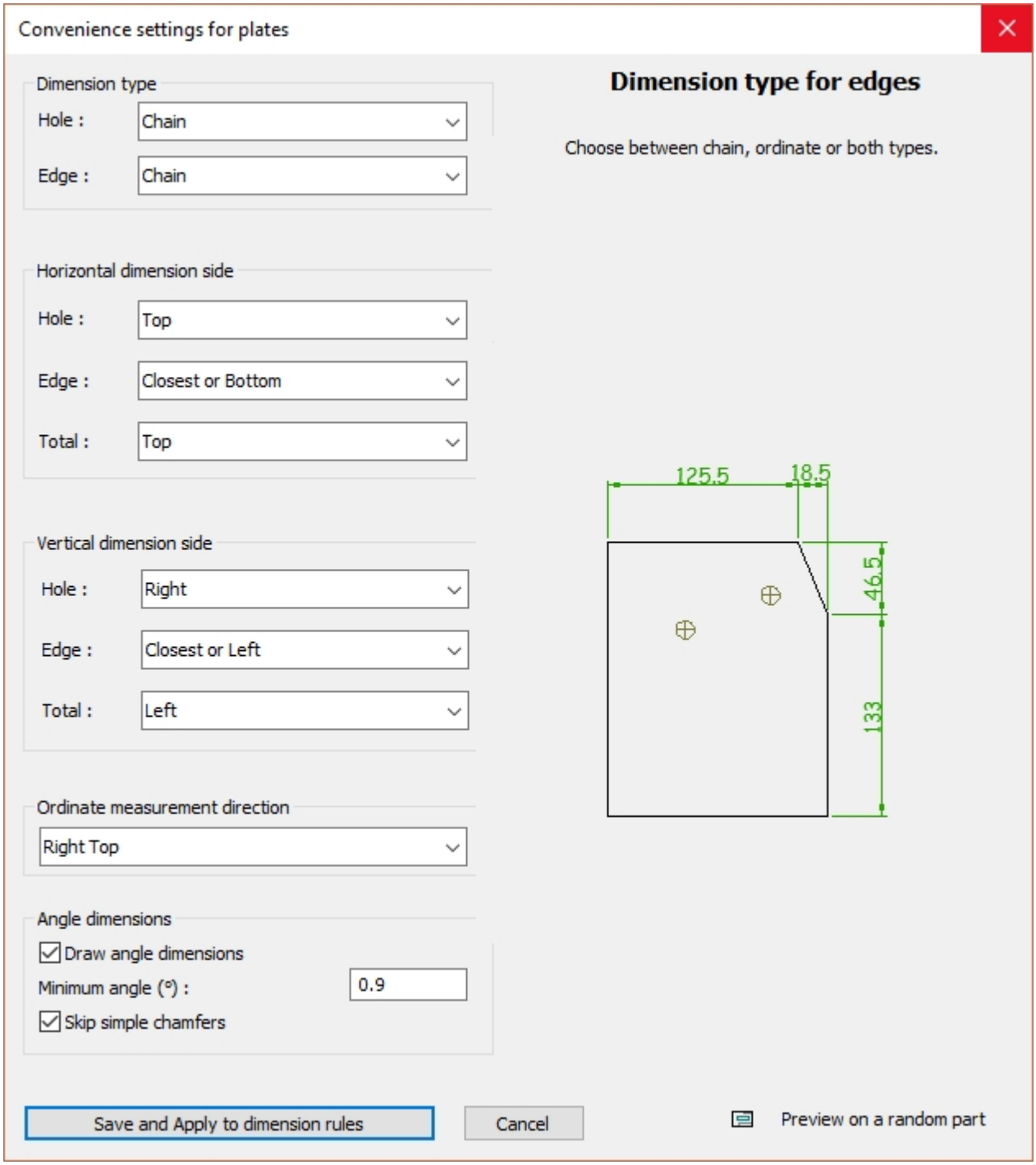
Plate Dimension Settings

This is an interactive dialog - placing the mouse over an option will open an example on the right side of the dialog.

Click on 

to preview your selected options on a test drawing before

applying.

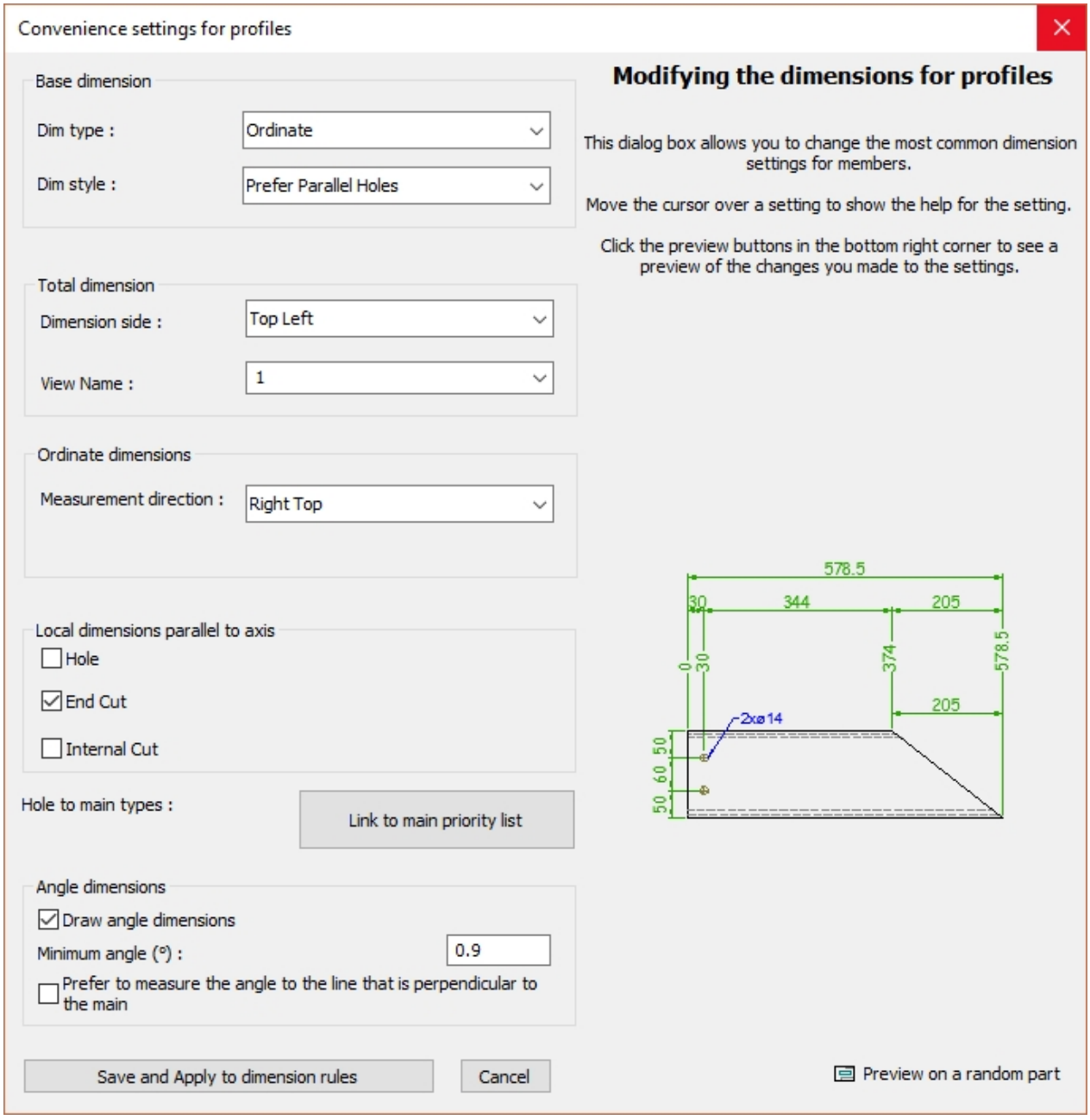


Side View Dimension Settings

This is an interactive dialog - placing the mouse over an option will open an example on the right side of the dialog.

Click on 

to preview your selected options on a test drawing before applying.

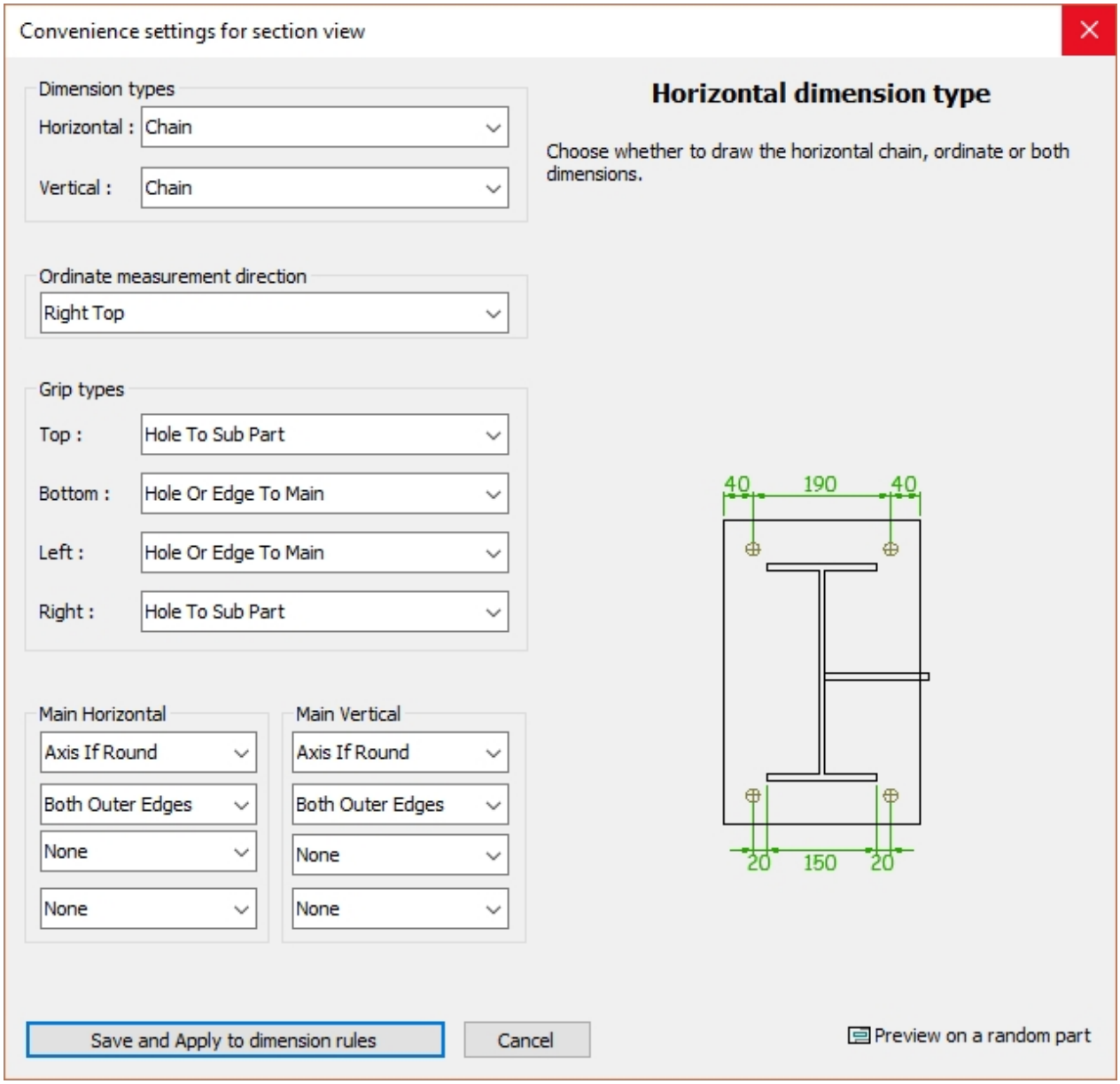


Section View Dimension Settings

This is an interactive dialog - placing the mouse over an option will open an example on the right side of the dialog.

Click on 

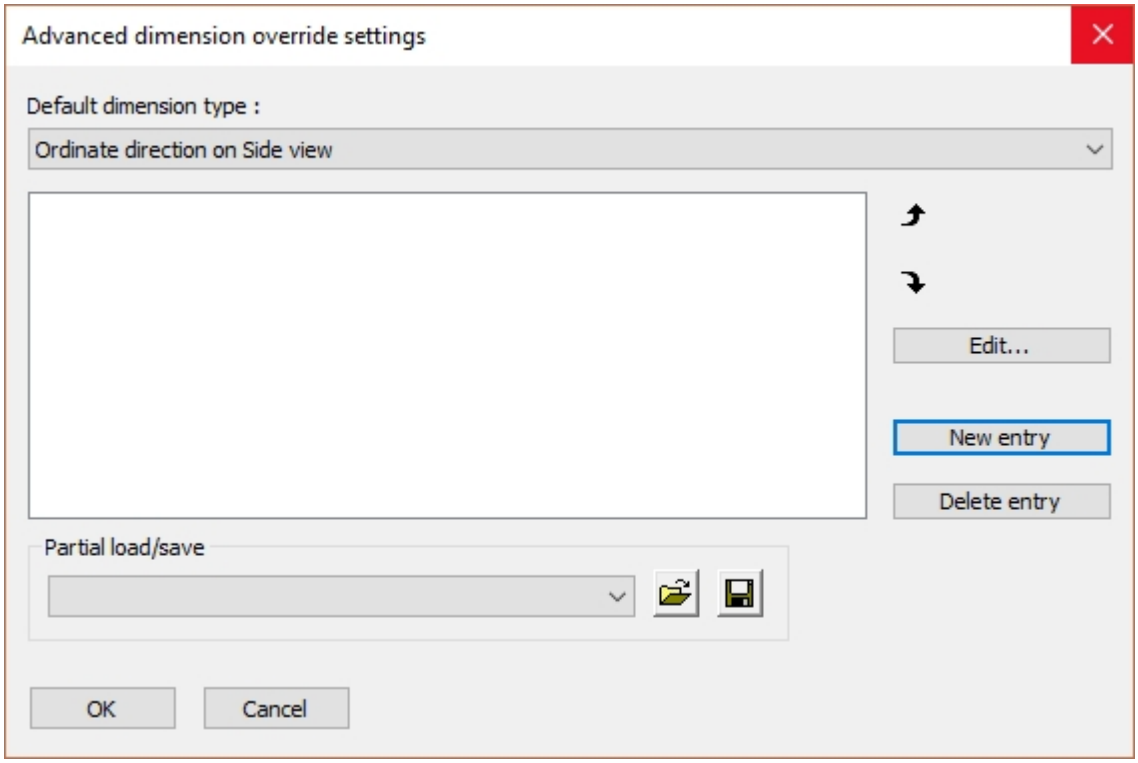
to preview your selected options on a test drawing before applying.



Advanced Dimension Settings

With difficult but advanced settings you can change the dimensions even more to your needs. The settings you change here may get lost when you change something in one of the other settings.

Activating this button will open an interactive dialog where you can edit **Base dimensions** / **Local dimensions** / **Others**

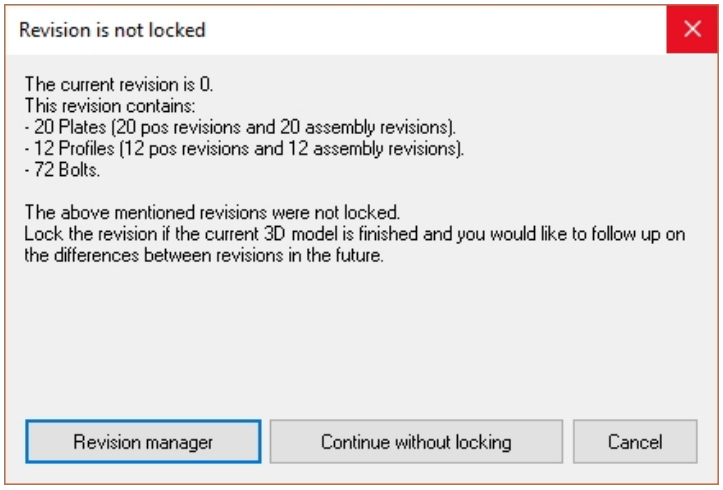


Bill of materials

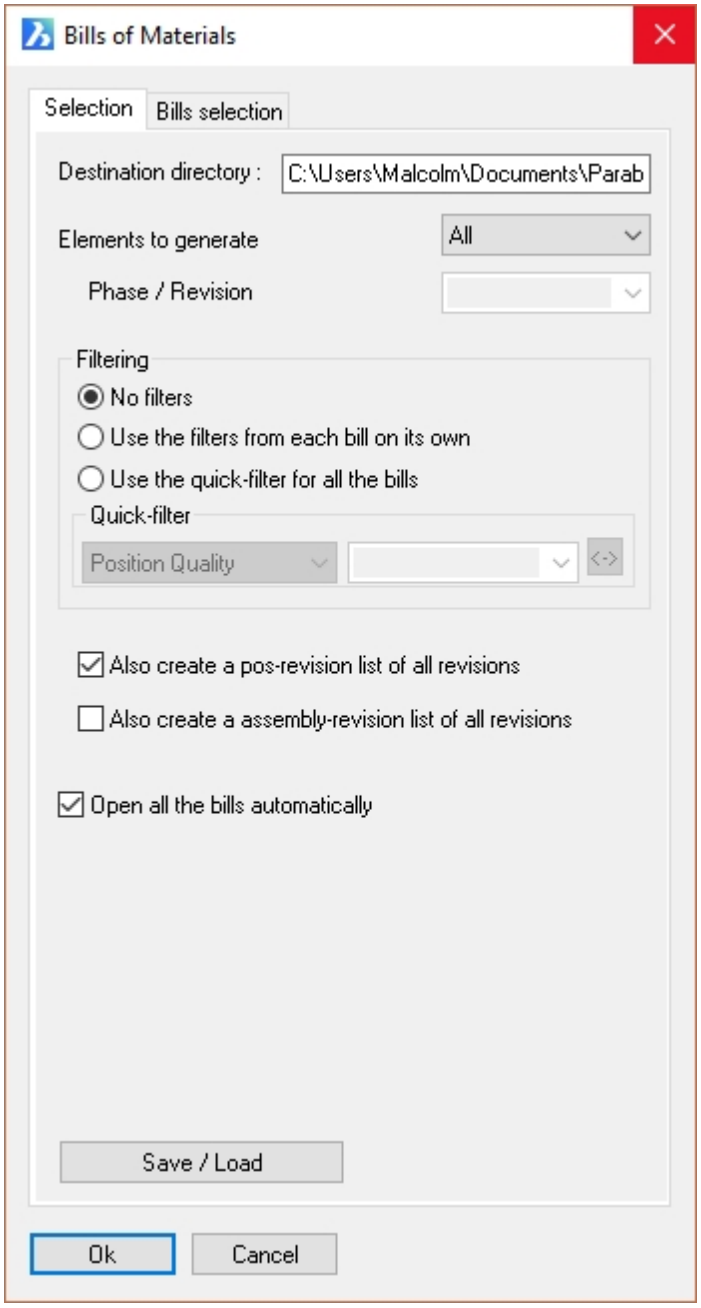
Command - **Prb_Bill**

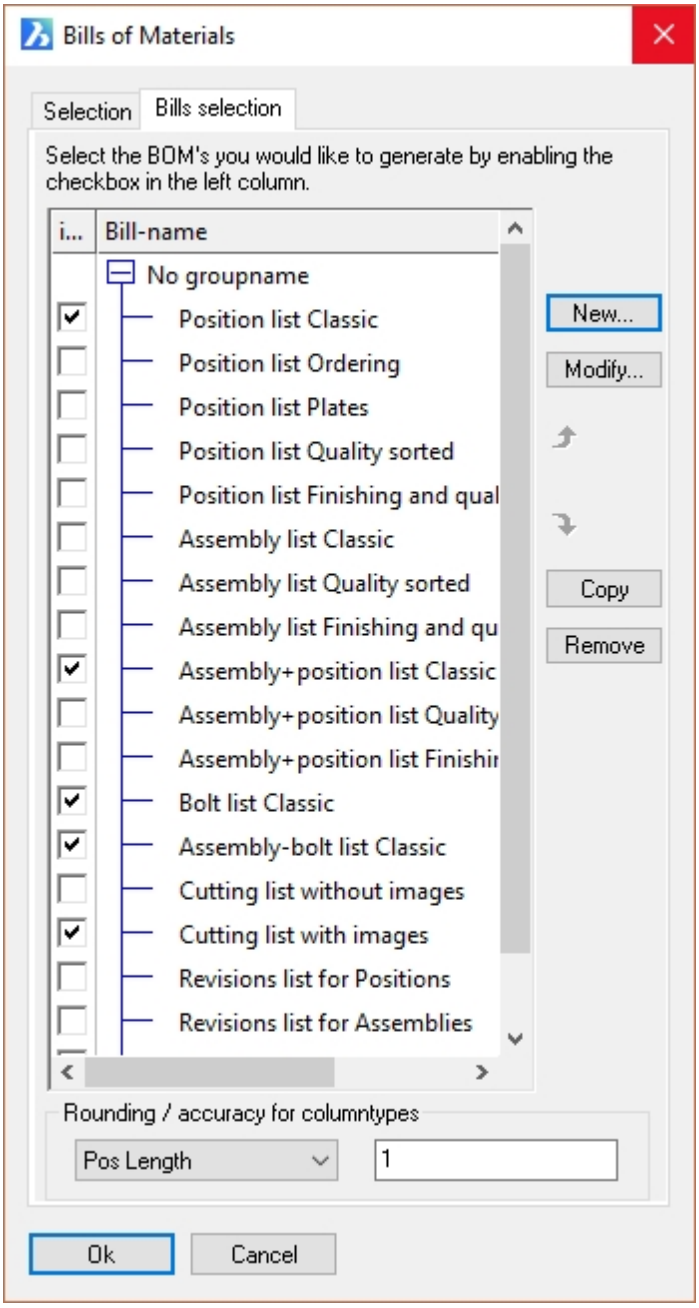
Activating this command will open the dialog to notify you that the revision is not locked. More information about revisions can be found in the Revision manager topic.

There is no harm in not/never locking the revision.



Next the Bills of material dialog is shown. This dialog has 2 tabs, **Selection** and **Bills selection**:


Bills of Materials
Selection
Bills selection
Destination directory :
C:\Users\Malcolm\Documents\Parab
Elements to generate
All
Phase / Revision
Filtering
No filters
Use the filters from each bill on its own
Use the quick-filter for all the bills
Quick-filter
Position Quality
Also create a pos-revision list of all revisions
Also create a assembly-revision list of all revisions
Open all the bills automatically
Save / Load
Ok
Cancel



The Selection Tab - has the following options:

- **Destination directory** - here you may choose a directory to store the bills - the default is the directory where the 3D model is stored
- **Elements to Generate** - here you may choose, **Selection** / **All** / **Phase** / **Revision**. If you select Phase or Revision you will be prompted to enter the name of the phase or revision in the **Phase/Revision** field
- **Filtering** - here the options are:
    - **No filters** - where all bills will be produced
    - **Use the filters from each bill on its own** - For information on this go to Element Filters

- **Use the quick-filter for all bills** - Selecting this radio button will activate the Position Quality drop-down options list from which you may select an item.

- You may also create a **Position list for all revisions**, **create an Assembly list for all revisions**, and **Open all the bills automatically** as they are written by activating the checkboxes.

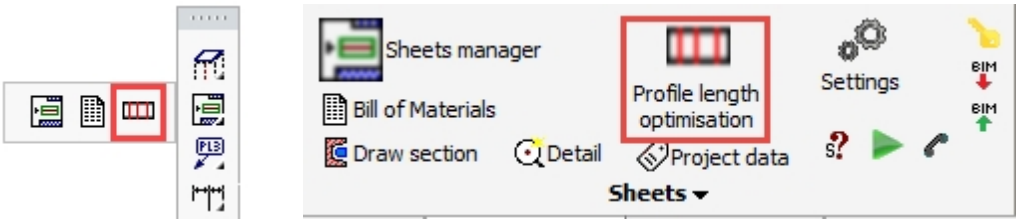
The Bills selection Tab

Here you may select the bills of material you would like to generate by enabling the various checkboxes

- **New -** will enable you to create a new Bill from the Formatting options dialog
- **Modify** - will enable you to modify an existing Bill from the Formatting options dialog
- **Copy** - will enable you to make a copy of an existing bill
- **Remove** - will enable you to remove a bill from the list
- **Rounding accuracy for column types** - allows you to change the rounding accuracy per column

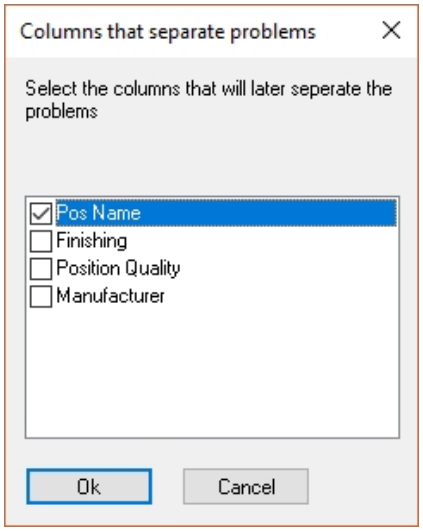
Profile Length Optimization

Command - **PrB_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 first 2 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.

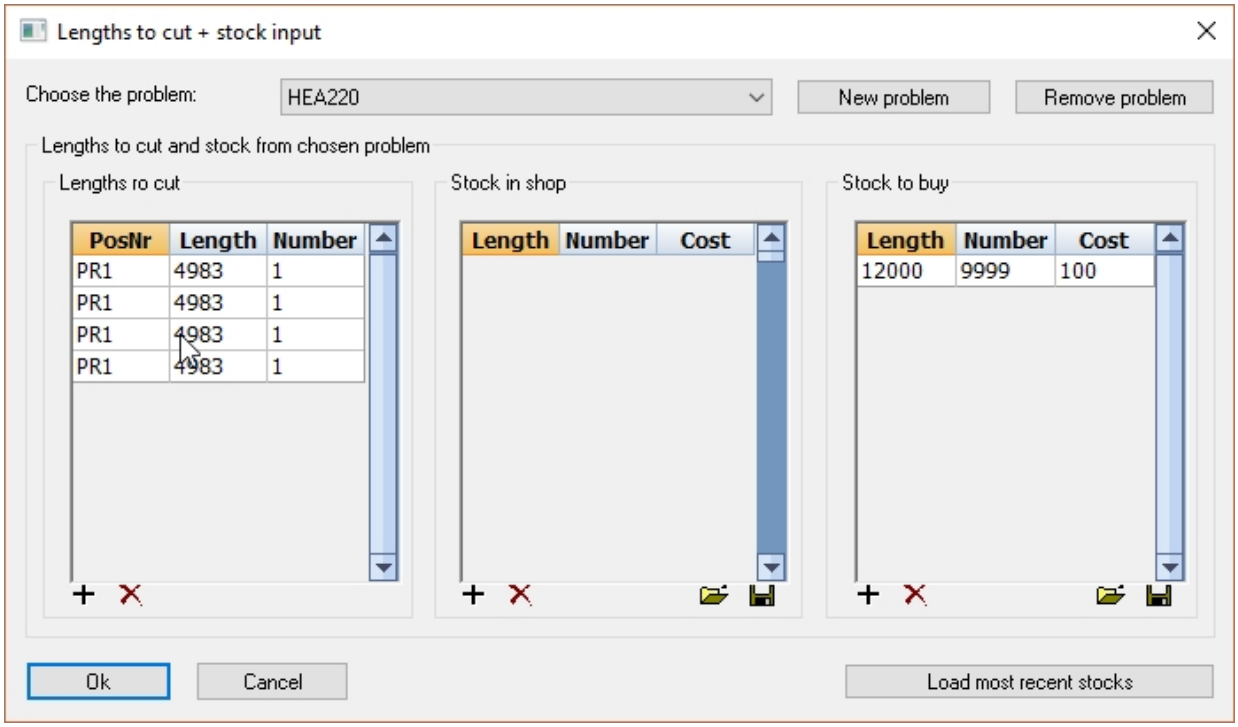
There are two types of stocks: 'Workshop stock' and 'To buy stock'. They are split apart so we can later easily renew the workshop stock: Removing used lengths and adding new remainder lengths. You can automatically load the most recently saved stocks with the button

at the 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:

"x:\Parabuild\Pb_Lib\Cutting Stock\HEA200\... "

x:\Parabuild\Pb_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 the price ratio of the length. Later while calculating efficient combinations these prices 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 higher cost ratio than the shop lengths. The cost ratio difference depends on how fast you want to get rid of the stock.

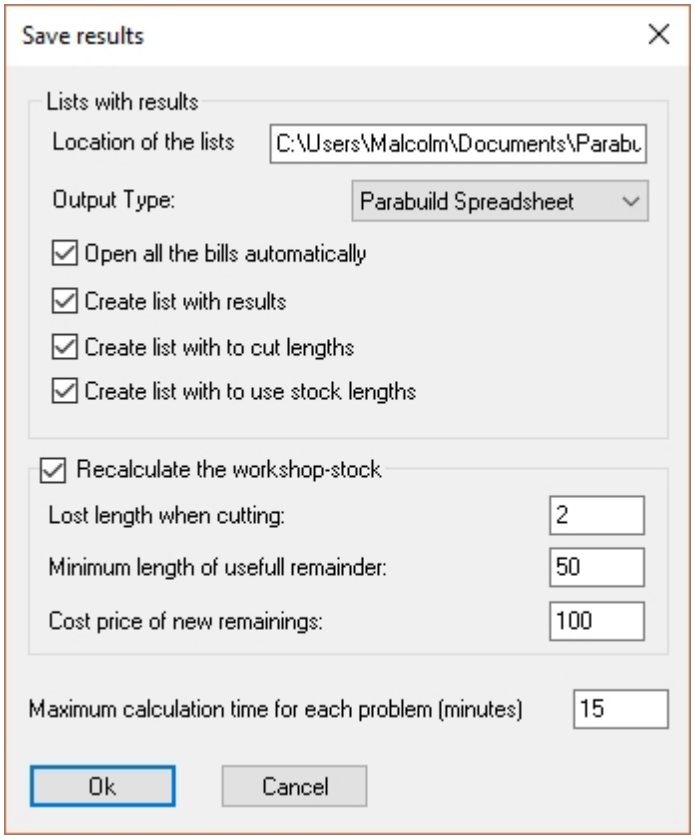


The third dialog box is meant for saving the end results. You have to enter the location where the lists with results have to be saved, and also which type of lists you want.

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:

x:\Parabuild\Pb_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.



Tools for 2D Drawings

- Position number tag
- Assembly number tag
- Part name
- Comment
- Other annotations
- Level
- Weld symbol
- Edit weld symbol
- Auto tags on views
- Show annotation settings
- Draw Section
- Detail in G.A. drawing

Position Number Tag

Command - **(Prb_TagGroup "Position Numbering")**

This command will draw a position number tag or a bolt tag.

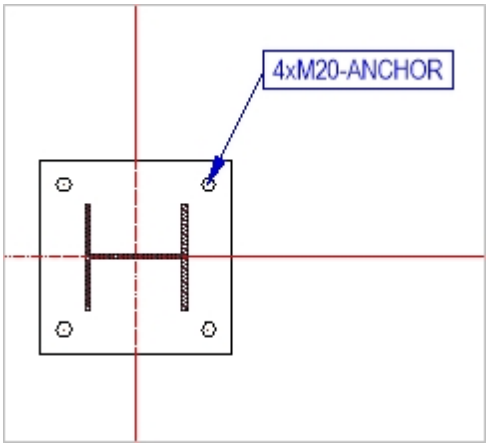
The annotations are meant to show additional information about a part on the 2D sheet.

You can also place annotations in model space between the 3D parts, but drawing them this way is not recommended.

After activating this command, you should first select the part to be annotated.

After that, the tag will attach itself to the mouse pointer : select a position and right click.

The appearance of the annotation can be adjusted by referring to Annotation Styles.



Assembly Number Tag

Command - **(Prb_TagGroup "Mark Numbering")**

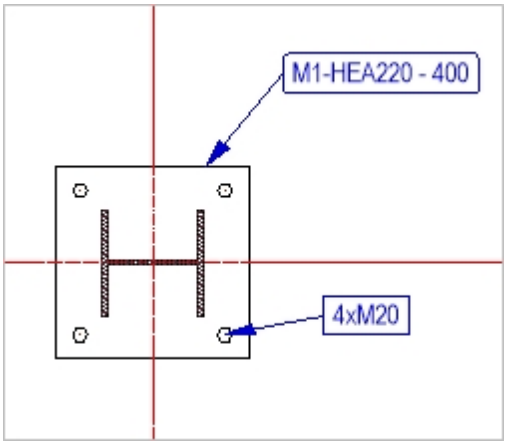
This command will draw an assembly number tag or a bolt tag.

The annotations are meant to show additional information about a part on the 2D sheet

After activating this command, you should first select the part to be annotated.

After that, the tag will attach itself to the mouse pointer : select a position and right click.

The appearance of the annotation can be adjusted by referring to Annotation Styles.



Part Name

Command - **(Prb_TagGroup "Part name")**

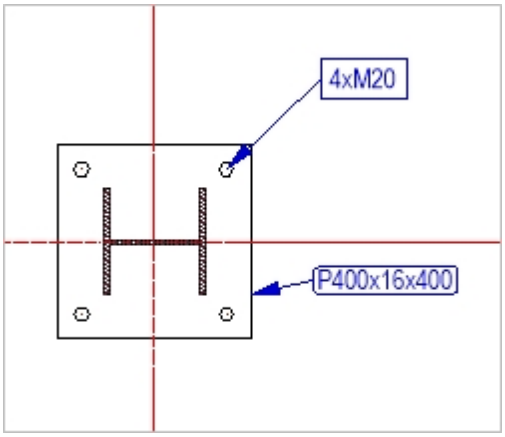
This command will draw a part name tag or a bolt tag.

The annotations are meant to show additional information about a part on the 2D sheet.

After activating this command, you should first select the part to be annotated.

After that, the tag will attach itself to the mouse pointer : select a position and right click.

The appearance of the annotation can be adjusted by referring to Annotation Styles.



Comment

Command - **(Prb_TagGroup "Comment")**

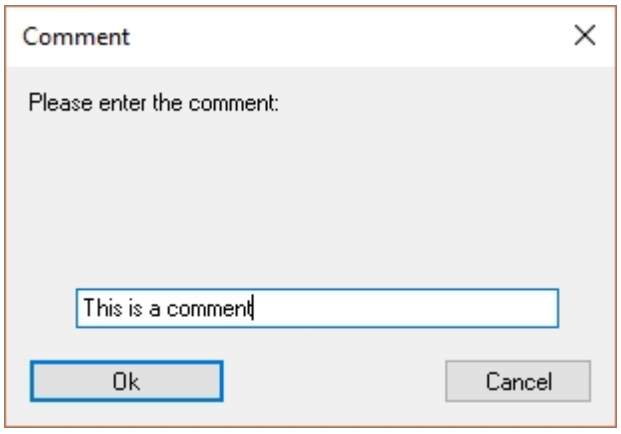
This command will draw a comment tag.

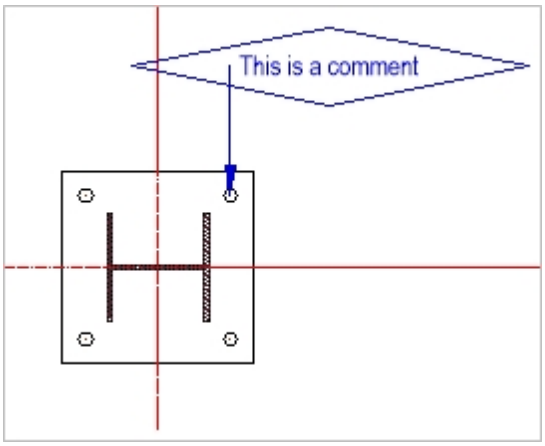
The annotations are meant to show additional information about a part on the 2D sheet.

After activating this command, you should first select the part to be annotated.

After that, the tag will attach itself to the mouse pointer : select a position and right click.

The appearance of the annotation can be adjusted by referring to Annotation Styles.





Other Annotations

Command - **PrB_Tag**

This command will draw a tag of a different group, for which no icon exists.

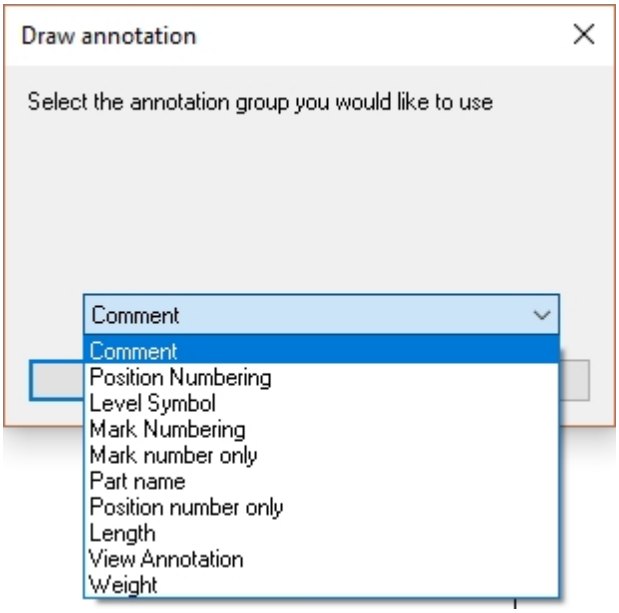
Having this tool, one can draw tags based on custom tag styles created by the user.

The annotations are meant to show additional information about a part on the 2D sheet.

After activating this command, you should first select the part to be annotated.

After that, the tag will attach itself to the mouse pointer : select a position and right click.

The appearance of the annotation can be adjusted by referring to Annotation Styles.



Level

Command - **(Prb_TagGroup "Level Symbol")**

This command will draw a level annotation.

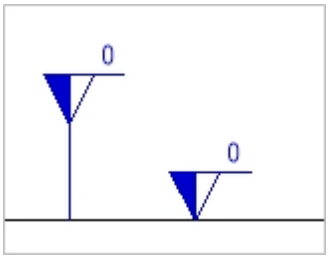
The annotations are meant to show additional information about a part on the 2D sheet.

After activating this command, you should first select the part to be annotated, at the location where the height should be measured.

After that, the tag will attach itself to the mouse pointer : select a position and right click.

The appearance of the annotation can be adjusted. For more information see the Annotation

Styles topic.



Weld Symbol

Command - **PrB_DrawWeldTag**

For more information about this object, seen the Weld Symbol topic.

Edit Weld Symbol

Command - **PrB_EditWeldTag**

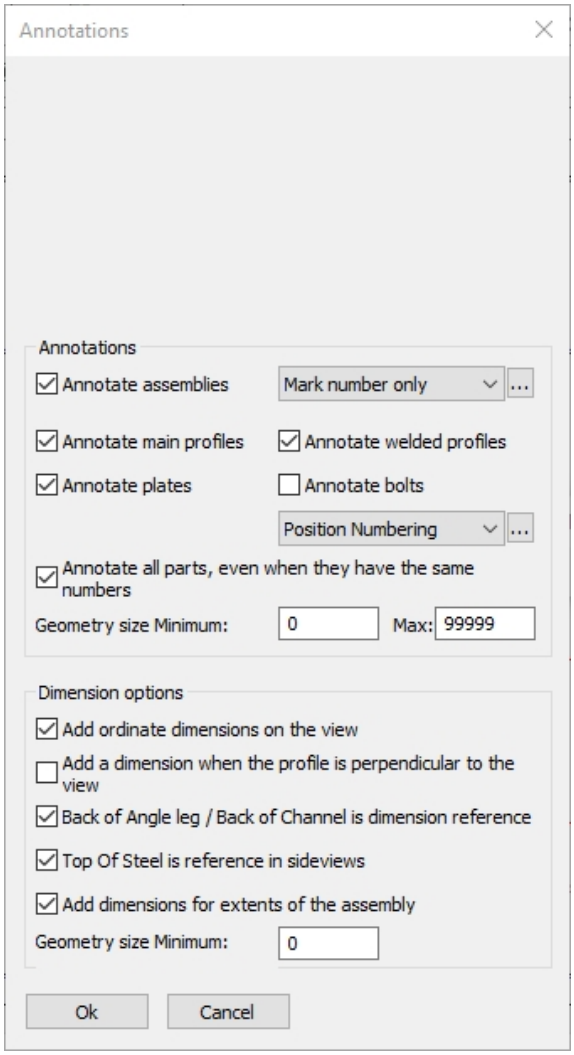
For more information about this object, seen the Weld Symbol topic.

Auto Tags on View

Command - **PrB_DrawTagsOnView**

This command can automatically annotate all parts on a 2D view.

After activating this command, you will first have to select the view, and the following dialog will appear:



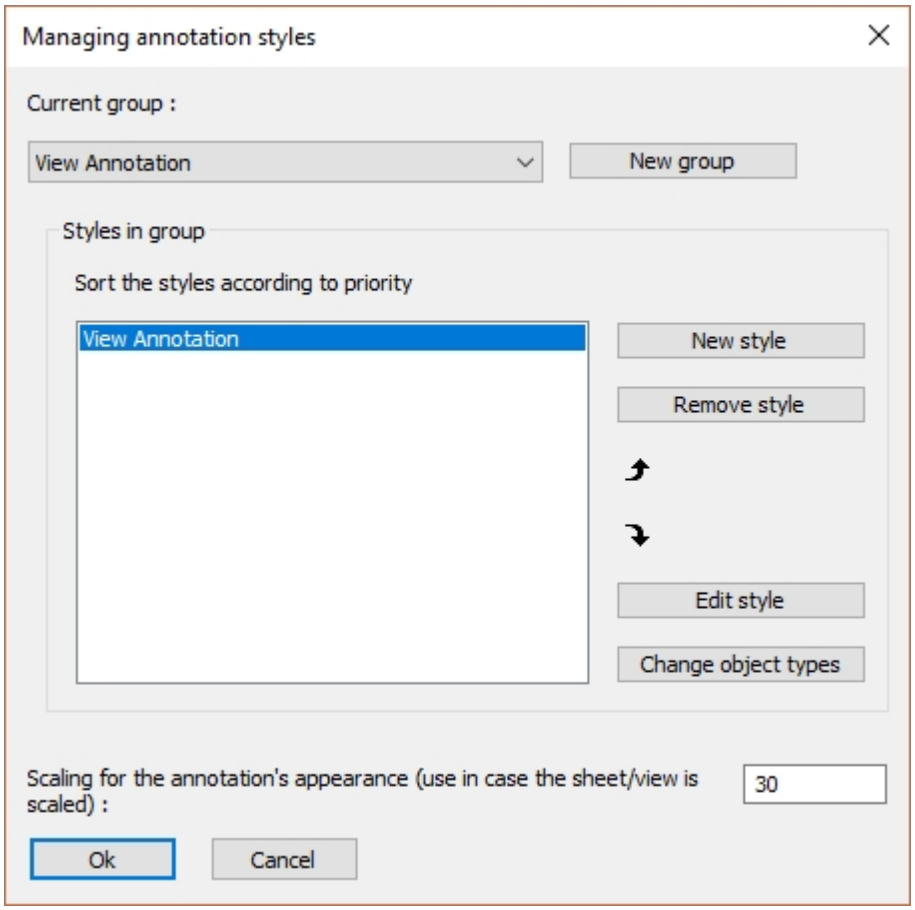
For more information about the options in this dialog, see the Add new Plan view topic.

Just a note of warning: Select only the options that you feel are necessary to achieve your intent. Activating all options may unnecessarily 'clutter' the drawing requiring you to spend time editing the finished result.

Annotation styles

Command - **Prb_TagSettings**

Activating this command will open the **Managing annotation styles** dialog, which can also be accessed from the Sheet Properties dialog.



This dialog allows you to create and modify annotation styles. Each annotation style is part of an annotation group.

The purpose of the annotation group is to allow us to access multiple styles from a single icon.

If we take for example the Position number group, which is accessed through this icon  : we can use this icon to draw a part number tag or a bolt tag, both of which have a completely different annotation style.

This is accomplished by adding multiple annotation styles in the group, and also by correctly setting the *Object types* per style (explained below).

The group name can be used directly on the command line to draw annotations using the command name as follows :

(Prb_TagGroup "Group name")

By using the above line as the command line command in a new icon, it is possible to create custom icons that access custom annotation styles.

Alternatively, we can also access custom group names from the Other Annotations command in Parabuild. Creating a new icon is not necessary thanks to this command.

The actions in this dialog are explored below :

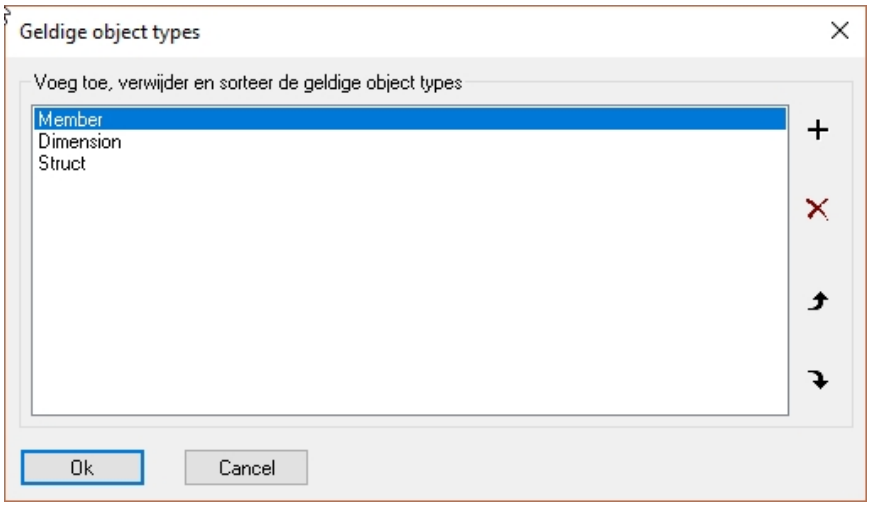
Current group and **New group** - These allow you to manage the groups

New style - This will create a new style within the current group

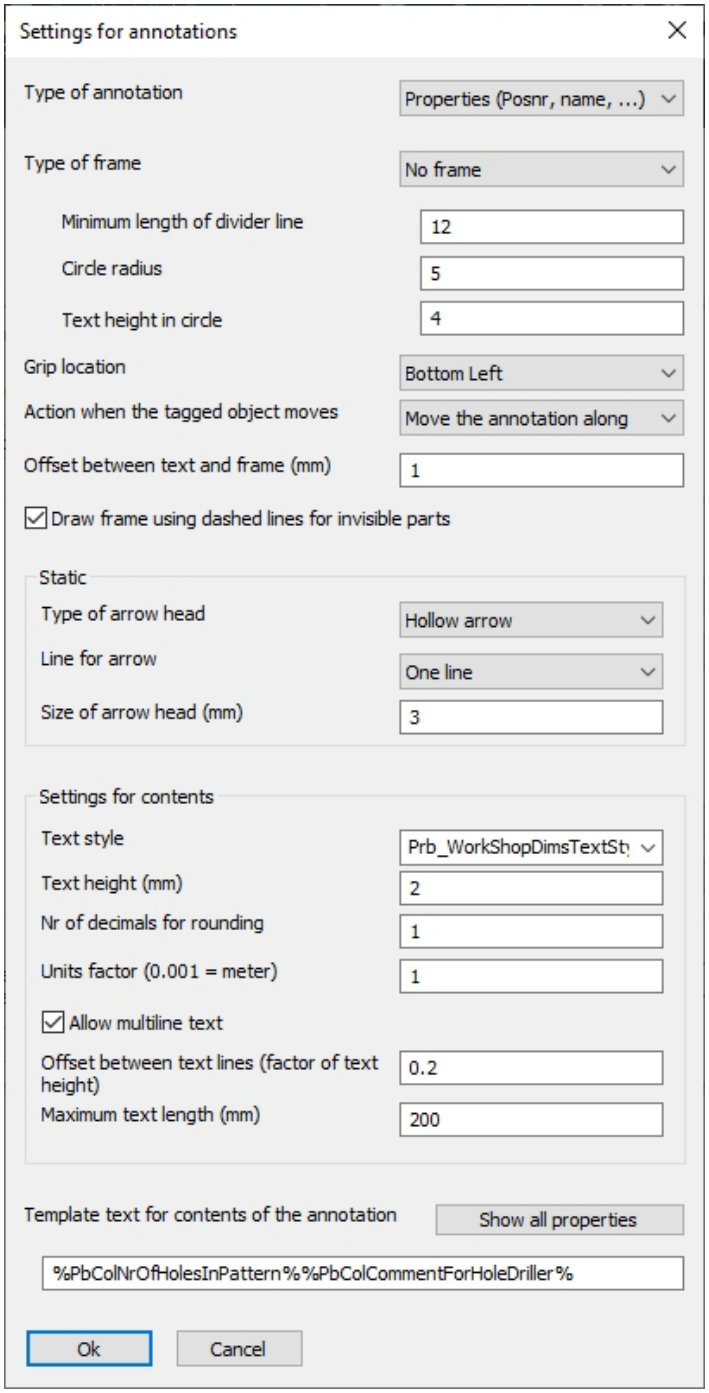
Remove style - This will remove the selected style

Edit Style - This will open the Settings for annotations dialog

Change object types - This allows us to change the object types for the currently selected style. As explained above, the object types in this list allow us to dictate which style should be used for each different object type.



Settings for property annotations



Many of the options in this dialog box are self-explanatory, but we will explore the less obvious options :

Type of annotation - The following annotation styles are available :

- **Properties** - This is the typical annotation that displays one or more properties of a 3D part
- **Comment** - This annotation type's purpose is to show a manually written comment. In essence there is no difference with the *properties* type.
- **Property through proxy object** - This annotation type is the same as the *Properties* type. But a difference was created here to allow the user to differentiate between an annotation that shows the property of a part directly, or 'by proxy' when the annotation is touching the part indirectly.
  An example could be an annotation touching a dimension. Usually, the annotation would not depict properties of the dimension but rather properties of the part that is dimensionned.

- **Arrow for section view** - This type has some unique properties that only apply to section annotations. A section annotation is used to depict the viewing direction and location of a section view. For more about the options, see below.

- **Level annotation** - This type has some unique properties that only apply to level annotations. For more about these options, see below.

Type of frame - All of the frame types should be obvious except for *Text above and below a divider line*.
This frame is an annotation with 2 or 3 text fields. For more information about this, see the Minimum *length of divider line* and *Circle radius options*.

Minimum length of divider line - This option will only work if the text %HorDivider% is used in the *Template text* (see below). The purpose of this is to draw an annotation with a horizontal divider that has text above and below the divider line. The contents before the %HorDivider% will be displayed above the divider line, the text after the %HorDivider% will be drawn below the divider line

Circle radius - This option will only work if the text %BubbleText% is used in the *Template text* (see below). The text that is written after the %BubbleText% will be drawn inside a circle. The circle will be drawn on the left-hand side of the divider line.

Text height in circle - This is the height of the text inside the special %BubbleText% circle.

Grip location - This is the location where the cursor attaches to the frame when we try to move an annotation frame.

Action when the tagged object moves - What Parabuild should do when the tagged part moves. Not implemented yet.

Draw frame using dashed lines for invisible parts - Not implemented yet.

Line for arrow - At the time of writing, only the options *None* and *One flexible line and a fixed line* are supported.

Allow multi-line text - When enabled, the text will be shown on multiple lines if necessary. The below options will decide when new lines are introduced :

Offset between text lines - This is measured as a factor of the text height

Maximum text length - If the text field would become larger than this value, then a new line will be introduced.

Template text for contents of annotation - This is the most important option for the annotations. With this option we can decide the contents that should be written in the annotation.
We need to use variable names to set the properties that should be depicted in the annotation.
But it is still allowed to write regular 'static' text in the same string.
The variables are always entered between % symbols to make this possible.
Some examples annotation texts that demonstrates this mix :

The template text for a hole annotation :

ø%PbColBoltDiamater%

Could result in the text :

ø16

The template text for a profile annotation :

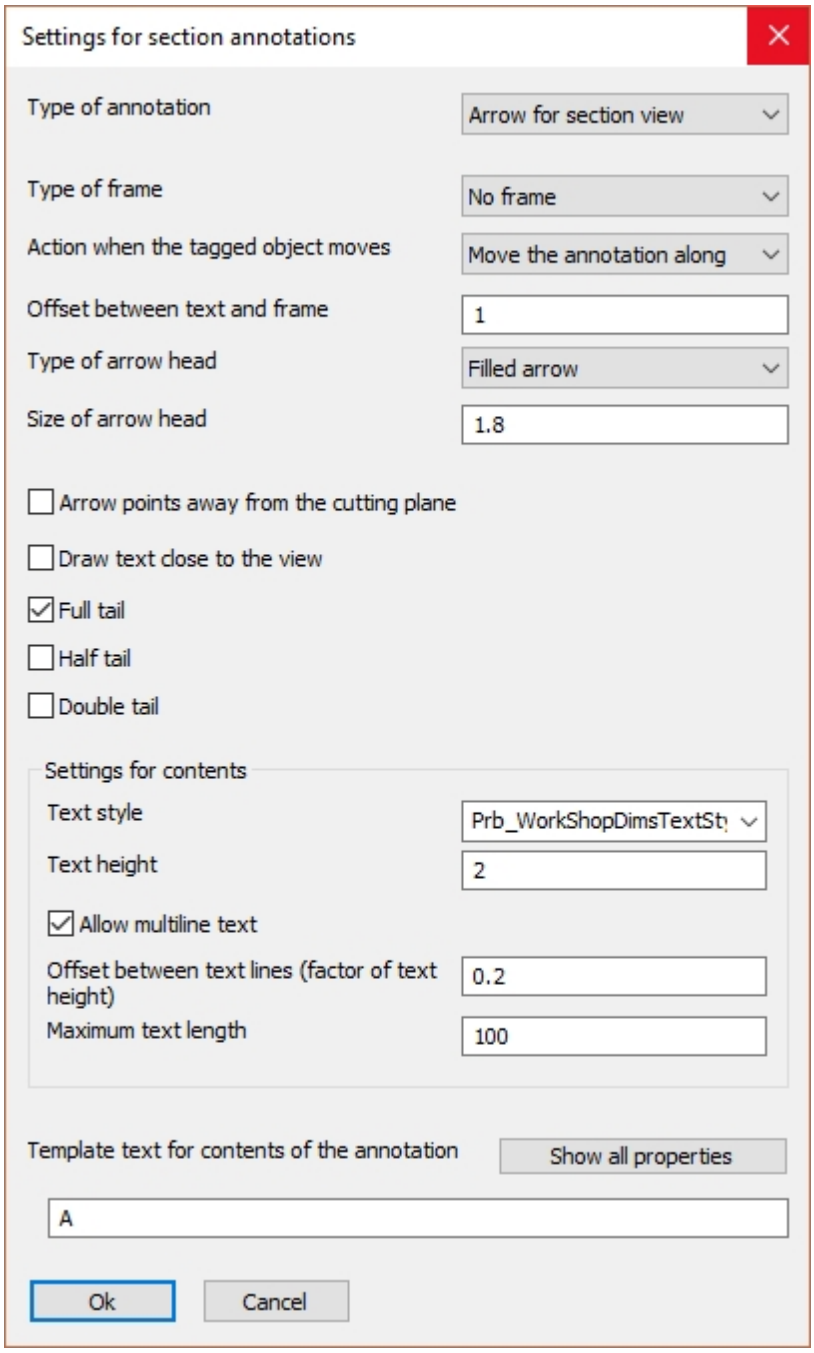
%PbColPosNumber%-%PbColName% L%PbColLength%

Could result in the text :

PR1-IPE200 L3590

Use the button **Show all properties** to display all the variables that Parabuild supports.

Settings for section annotations



Many of the options in this dialog box are the same as the regular annotations dialog box, except these:

Arrow points away from the cutting plane - Not implemented yet.

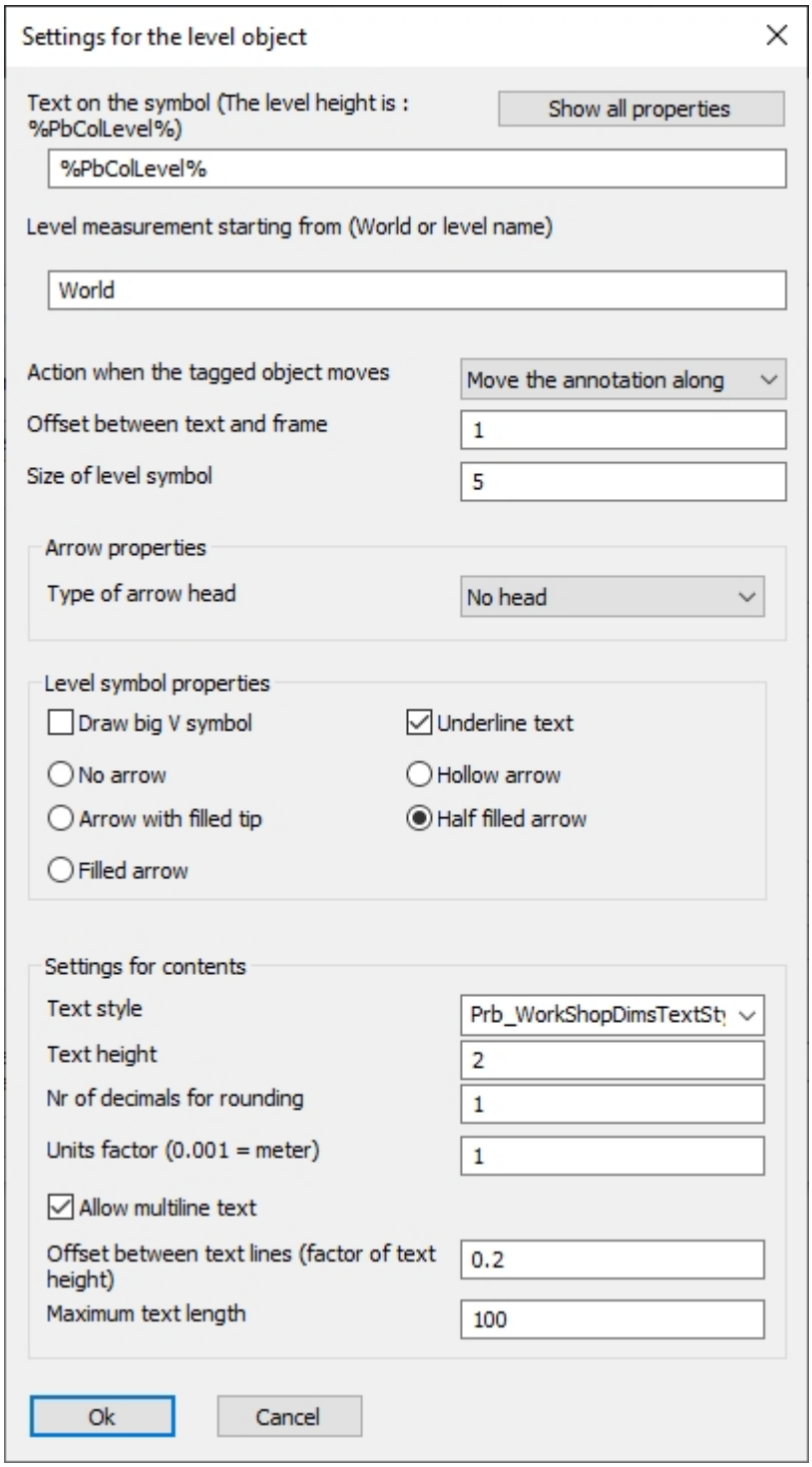
Draw text close to the view - Not implemented yet.

Full tail :

Half tail : 

Double tail - Not implemented yet.

Settings for level annotations



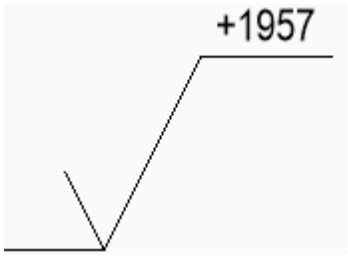
Many of the options in this dialog box are the same as the regular annotations dialog box, except

these:

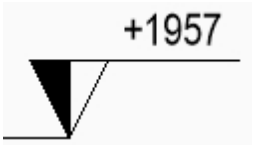
Text on the symbol - This works the same as the *Template text* of the property annotations. The only difference is that for this annotation type the variable %PbColLevel% can be used.

Level measurement starting from - This should be "World", or the name of the level that should be used as starting plane for the height measurement. Levels can be reviewed in the Modify levels topic.

Draw big V symbol :

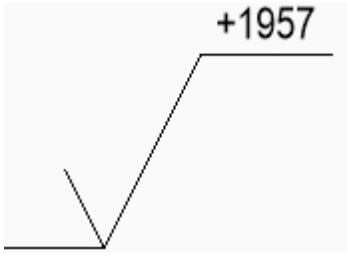


*An example level annotation with **Draw big V symbol** enabled*

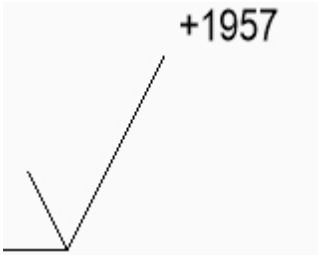


*An example level annotation with **Draw big V symbol** disabled*

Underline Text :

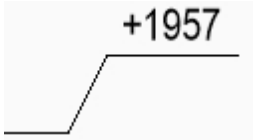


*An example level annotation with **Underline text** enabled*

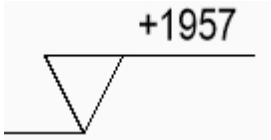


*An example level annotation with **Underline text** disabled*

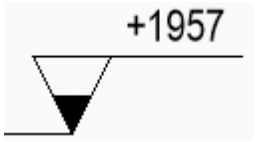
Tip type or arrow type :

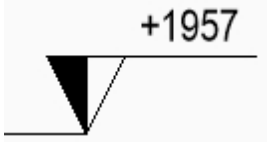


*Tip set to : **None***

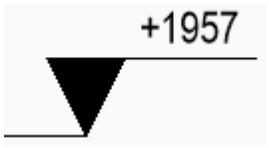


*Tip set to : **Hollow***





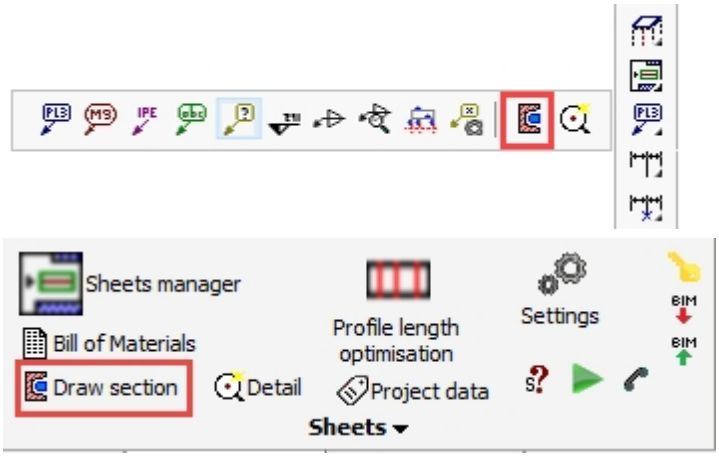
Tip set to : ***Filled tip*** *Tip set to :* ***Half filled***



Tip set to : ***Fully filled***

Draw Section

Command - **(Prb_TagGroup "View Annotation")**



This command will do different things depending on the space in which you start it :

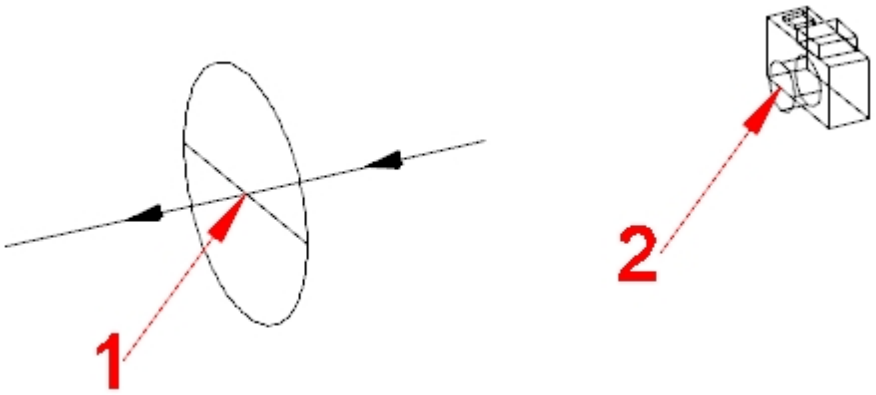
- If you start this command from the 3D model, then it will create a new camera.
- If you start this command from a 2D sheet, then it will draw a new section view together with a camera in 3D. The new section view is connected to the new camera because the camera defines the viewpoint of the view.

The first point that you indicate will become the center of the camera.

The second point will determine the viewing direction as well as the depth of the view limitation.

Drawing the camera in the 3D model

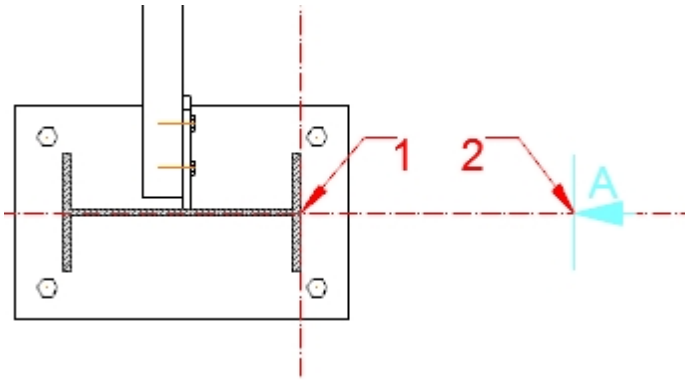
The command will ask for two points, as indicated in the illustration :



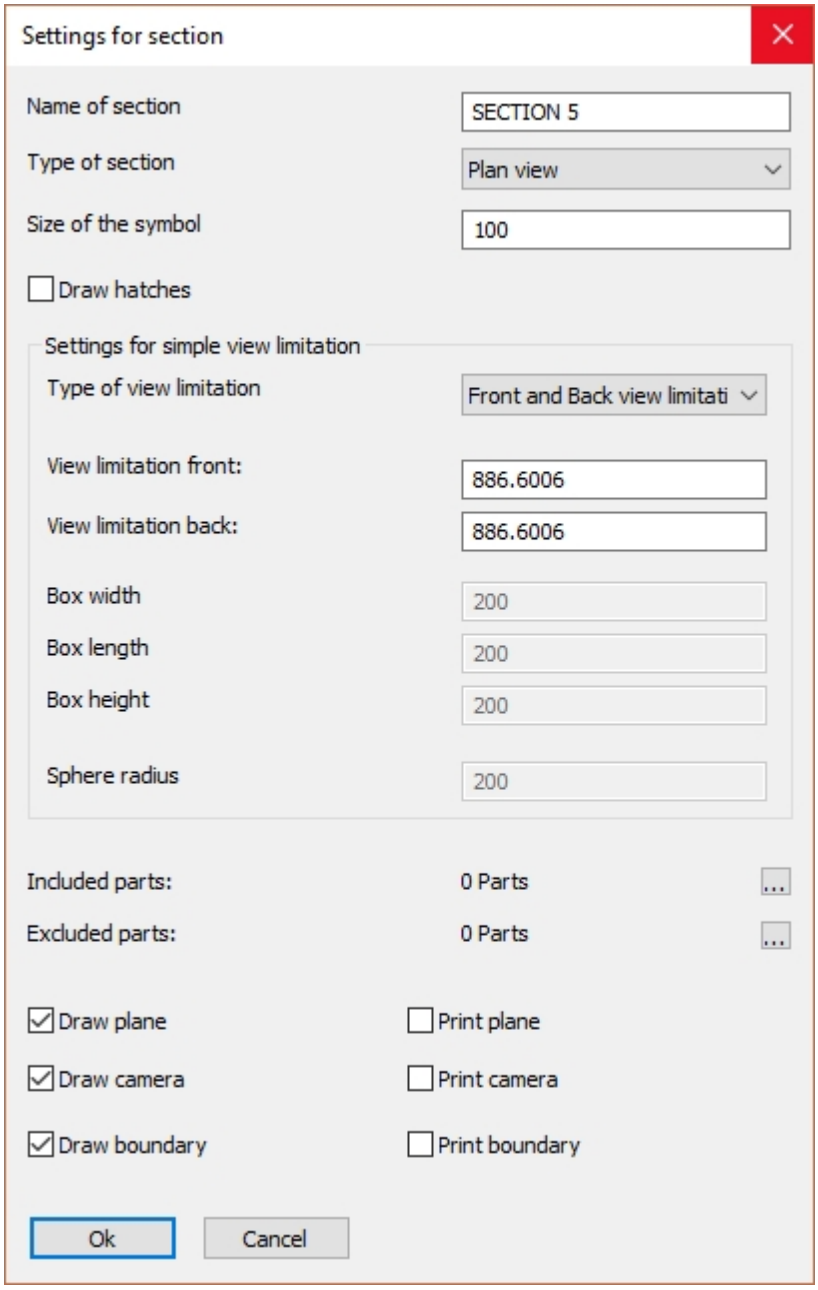
The center location (1) and the viewpoint location (2) of the camera as it is drawn in the 3D model.

Drawing a section view on a sheet

The command will ask you to indicate two points on an existing sheet. You should indicate them as in the illustration :



Where to indicate the center location (1) and the viewpoint location (2) for the new view in 2D.



Name of section - This is the name for the new camera but also for the new view. This name should be unique for the 3D model. If you do not enter a unique name, Parabuild will make it unique automatically by adding a digit at the end of the text string.

Size of the symbol - This is the size of the camera's appearance in the 3D model : the camera and the circular plane.

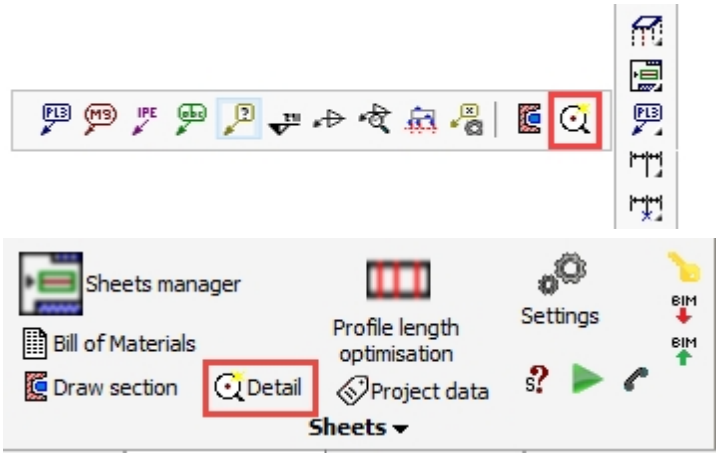
Draw plane/camera/boundary - When active, the camera will be drawn on the 2D sheet. By default detail camera's are drawn on 2D sheets as a rectangle to indicate the boundary of the detail.

Print plane/camera/boundary - Same as above, but these options refer to printing.

For more information about the other options in this dialog box, see the Camera properties topic.

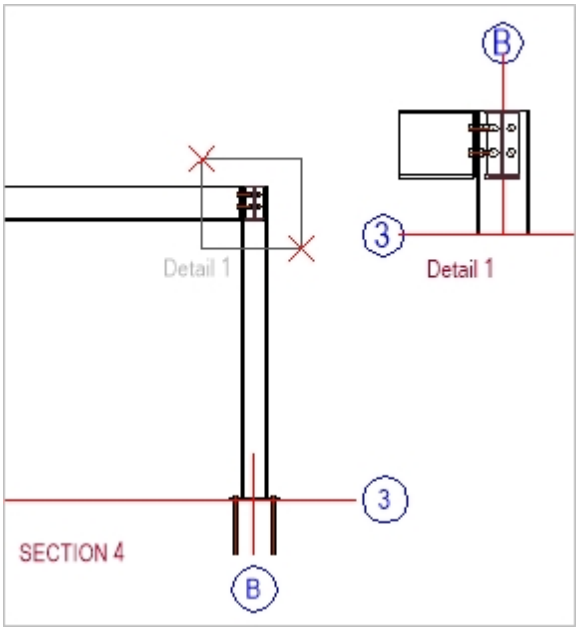
Detail in G.A. Drawing

Command - **PrB_CreateDetail**

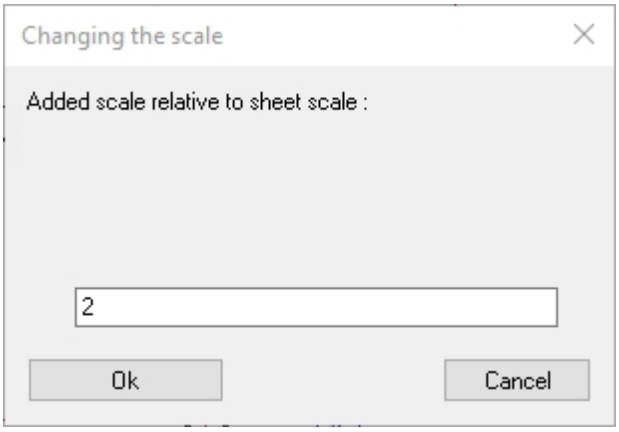


This command will draw a new detail view based on an existing view.

When you start this command you first need to indicate two points that form the rectangle of the detail :

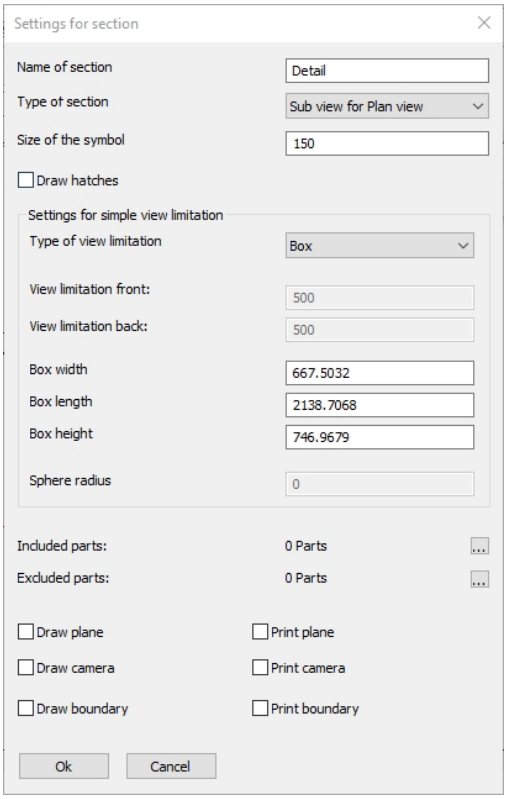


After that you will be asked to enter the scaling for the new detail view :

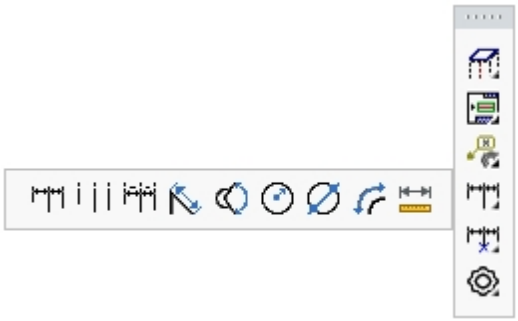


Parabuild will automatically show correct 3D measurements on the dimension values of scaled 2D views.

The 'Settings for section' dialog will appear. The options in this dialog are explained in the Draw Section topic..



Dimensioning



All dimensions that are drawn automatically by Parabuild are regular BricsCAD or AutoCAD dimensions.

You can change these dimensions and draw new ones using the default BricsCAD/AutoCAD commands.

In some cases these tools are not the most efficient, especially for drawing **chain dimensions**, **ordinate dimensions** and **oblique dimensions** where it can often take too many steps to draw such dimensions.

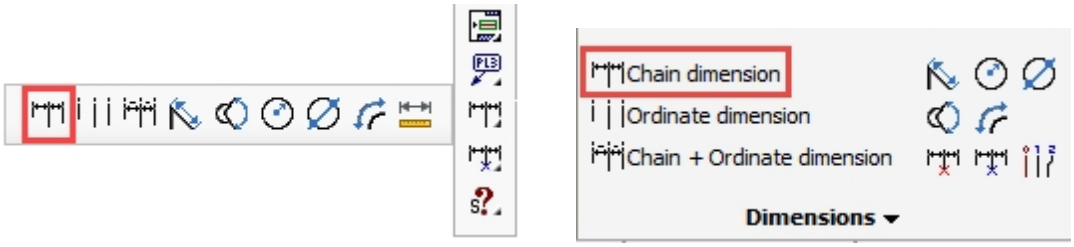
For this reason, some commands were added to Parabuild to intercept these shortcomings.

These commands will however draw standard CAD dimensions, so you can still use the properties and dimension styles on the dimensions drawn by Parabuild.

- Chain dimension
- Ordinate dimension
- Chain + ordinate dimension
- Aligned dimension
- Angular dimension
- Radius dimension
- Diameter dimension
- Arc length dimension
- Measure distance

Chain Dimension

Command - **Prb_ChainDim**

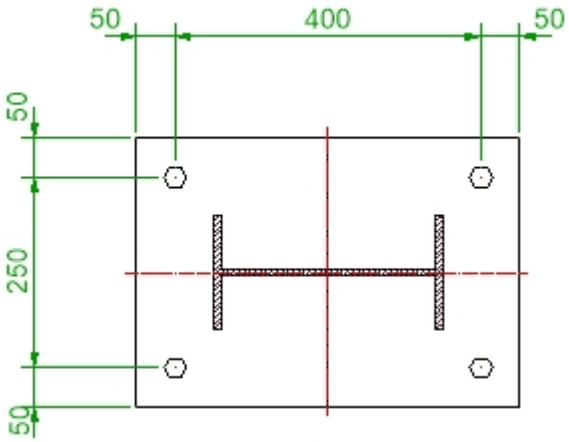


This command will draw a continuous set of dimensions that share the same baseline.

The command input is almost the same as the AutoCAD/BricsCAD Aligned Dimension command.

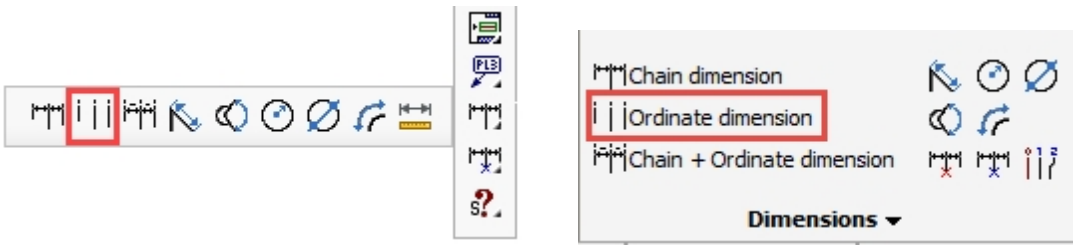
With the difference that it will ask you for more than 2 dimension points.

It also allows you to draw oblique dimensions. To do that, press <Enter> right after starting the command. Then select an oblique line in the view to which you want the new dimension to be aligned to.



Ordinate Dimension

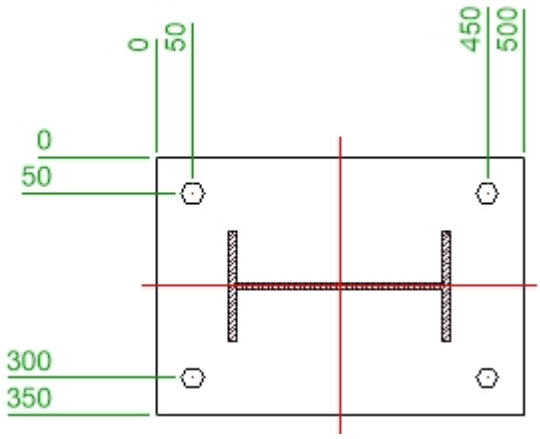
Command - **Prb_OrdinateDim**



This command will draw a set of ordinate dimensions.

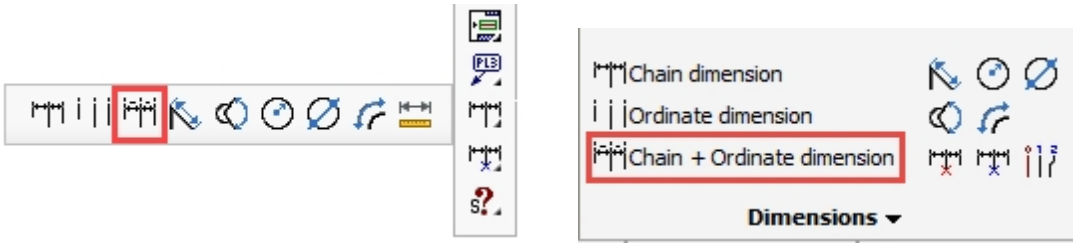
The command input is almost the same as the AutoCAD/BricsCAD Aligned Dimension command.

It also allows you to draw oblique dimensions. To do that, press <Enter> right after starting the command. Then select an oblique line in the view to which you want the new dimension to be aligned to.



Chain + Ordinate Dimension

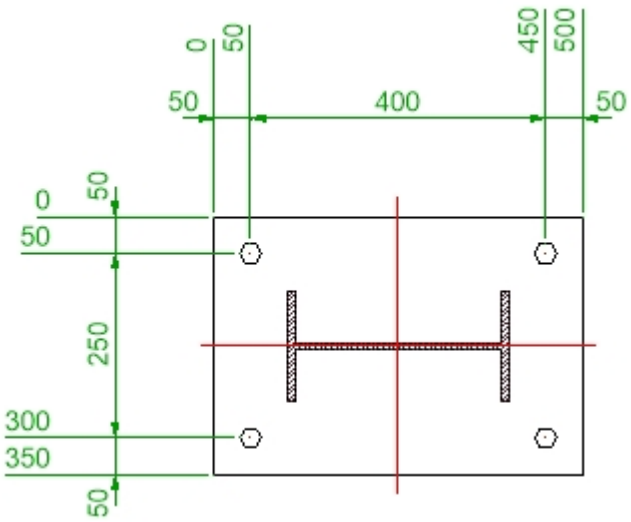
Command - **Prb_ChainOrdinateDim**



This command will draw a chain dimensions set combined with an ordinate dimensions set.

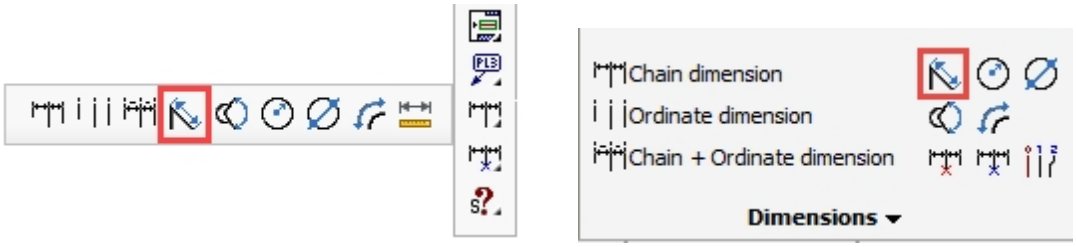
The command input is almost the same as the AutoCAD/BricsCAD Aligned Dimension command.

It also allows you to draw oblique dimensions. To do that, press <Enter> right after starting the command. Then select an oblique line in the view to which you want the new dimension to be aligned to.



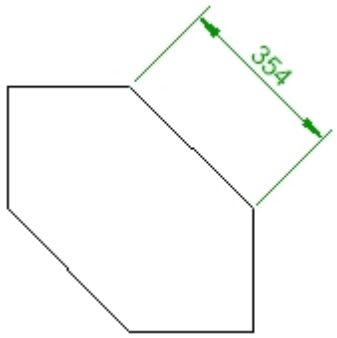
Aligned Dimension

Command - **_dimaligned**



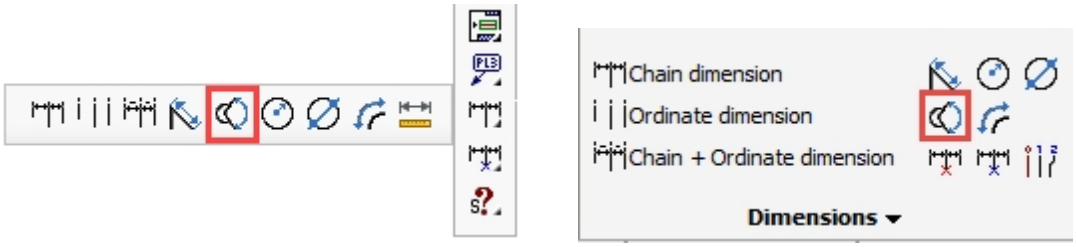
This is the default AutoCAD/BricsCAD command for drawing a single aligned dimension.

The icon was added to this toolbar for your convenience.



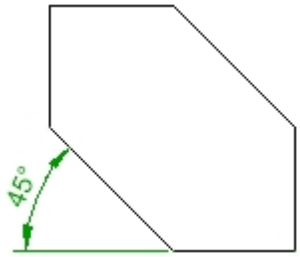
Angular Dimension

Command - **_dimangular**



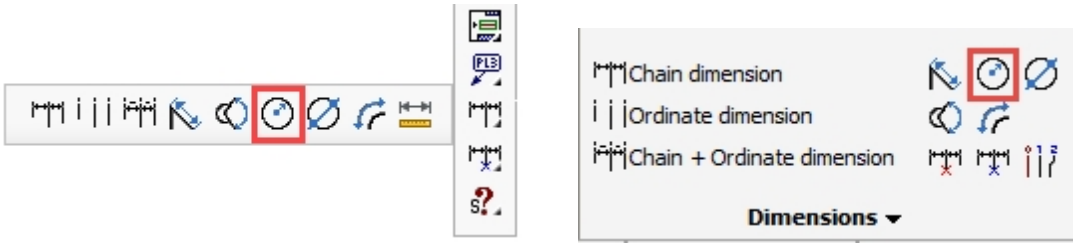
This is the default AutoCAD/BricsCAD command for drawing an angular dimension.

The icon was added to this toolbar for your convenience.



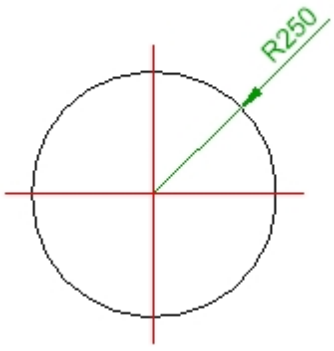
Radius Dimension

Command - **_dimradius**



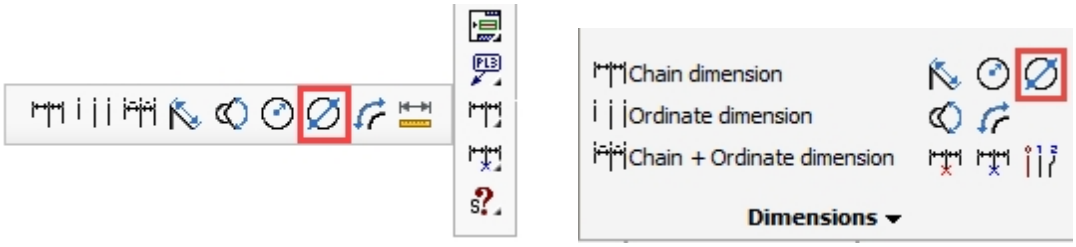
This is the default AutoCAD/BricsCAD command for drawing a radius dimension.

The icon was added to this toolbar for your convenience.



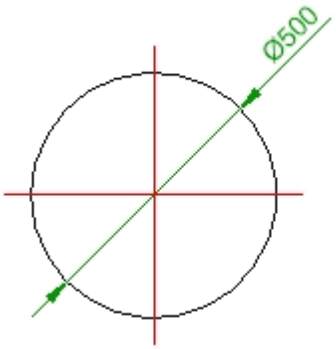
Diameter Dimension

Command - **_dimdiameter**



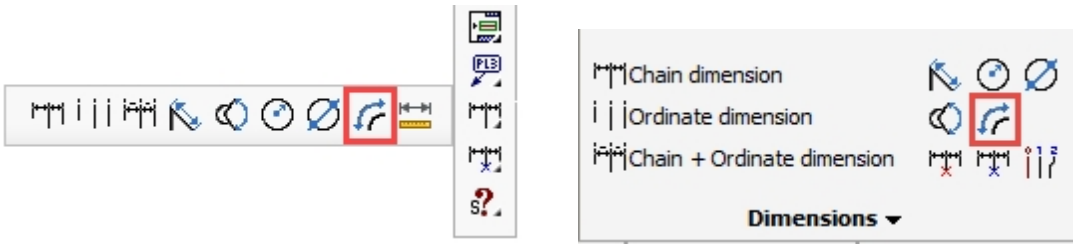
This is the default AutoCAD/BricsCAD command for drawing a diameter dimension.

The icon was added to this toolbar for your convenience.



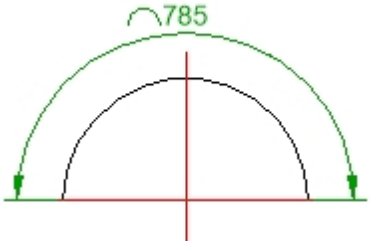
Arc Length Dimension

Command - **_dimarc**



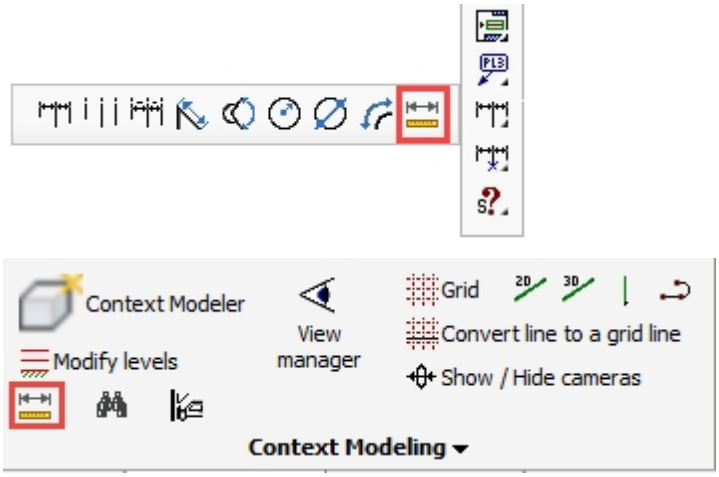
This is the default AutoCAD/BricsCAD command for drawing an arc length dimension.

The icon was added to this toolbar for your convenience.



Measure Distance

Command - **'_dist**



This is the default AutoCAD/BricsCAD command for measuring the distance between 2 points in 3D or 2D.

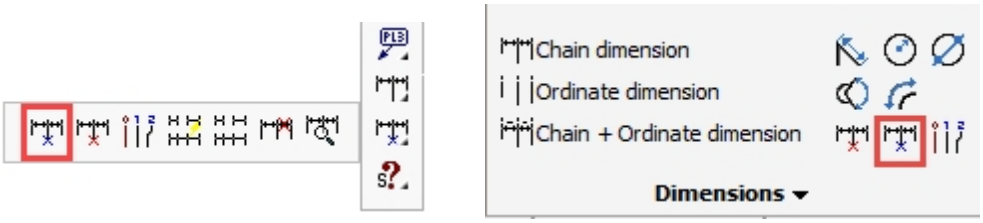
The icon was added to this toolbar for your convenience.

Modifying Dimensions

- Add extension line to chain dimension
- Remove extension line to chain dimension
- Change origin of ordinate dimension
- Search for dimensions and merge
- Manually merge into chain dimension
- Disconnect dimensions from chain
- Modify settings of chain dimension

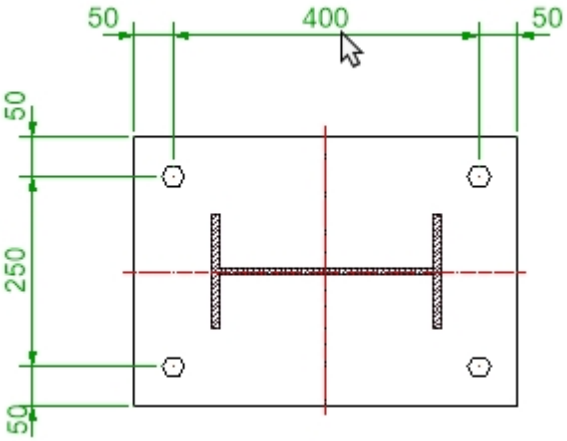
Add Ext. Line to Chain Dim.

Command - **PrB_AddToDims**

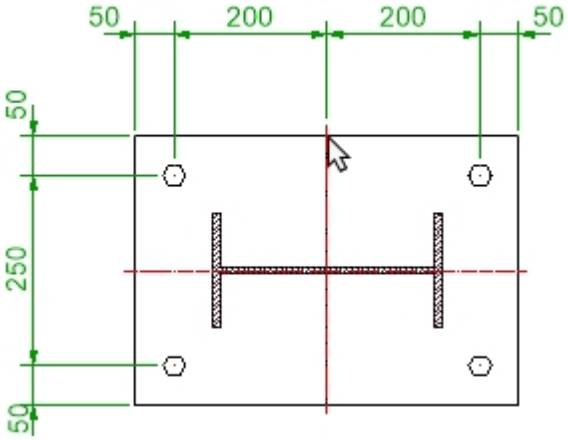


This command allows you to add more dimension points to an existing dimension.

It works on a chain dimension, on single aligned dimensions and also on ordinate dimensions.



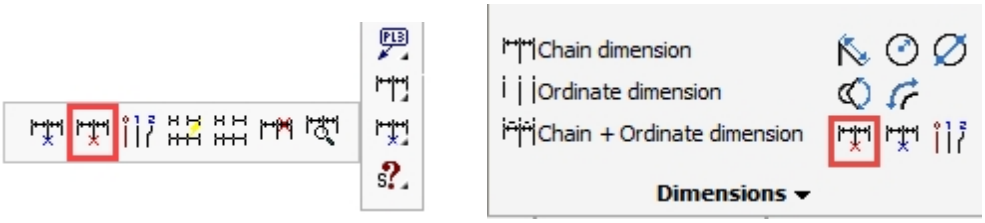
Selecting the dimension to which you want to add more dimension lines



Selecting the location for the new dimension line and the result

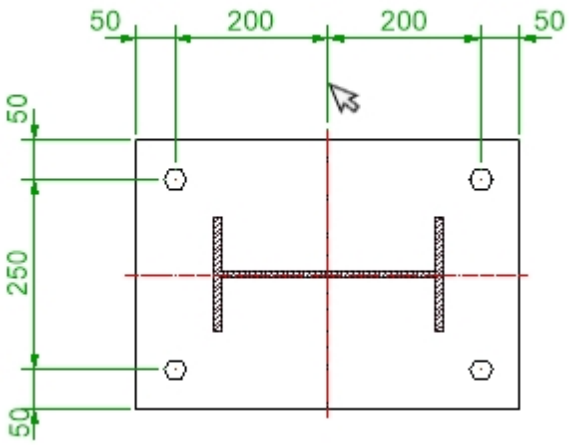
Remove Ext. Line from Chain Dim

Command - **PrB_RemoveDimLine**

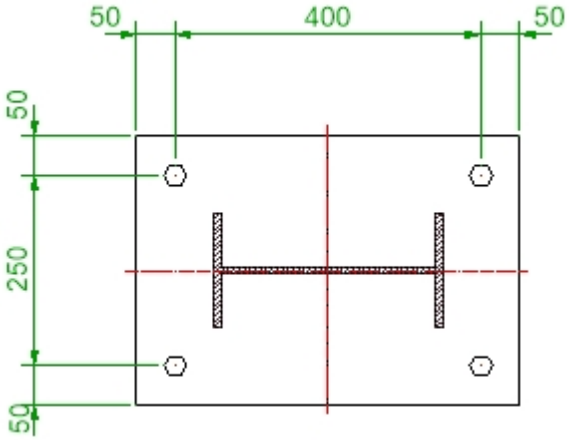


This command allows you to remove dimension lines from existing dimensions.

It works on chain dimensions and on ordinate dimensions.



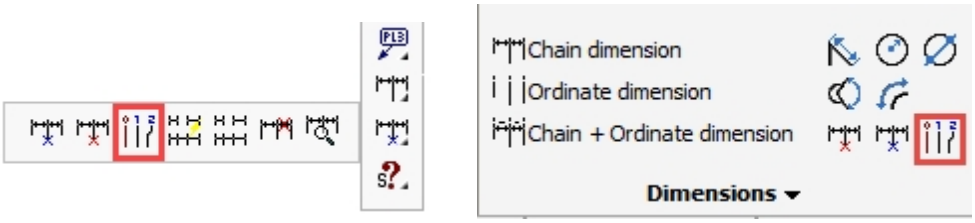
Selecting the dimension line that should be removed



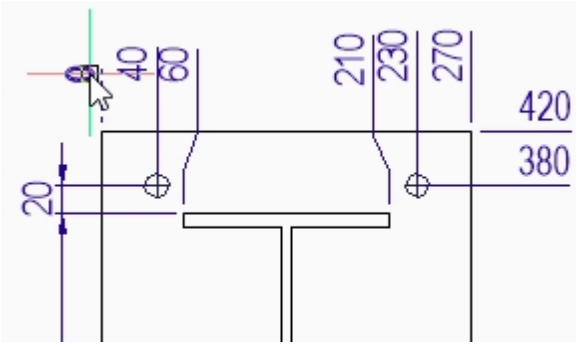
The result : the 2 dimensions are merged into 1

Change Origin of Ordinate Dim.

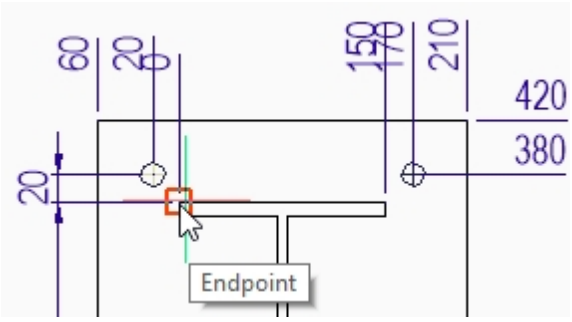
Command - **Prb_SetDimOrigin**



This command allows you to change the origin point of ordinate dimensions.



Selecting the ordinate dimension that needs to be modified

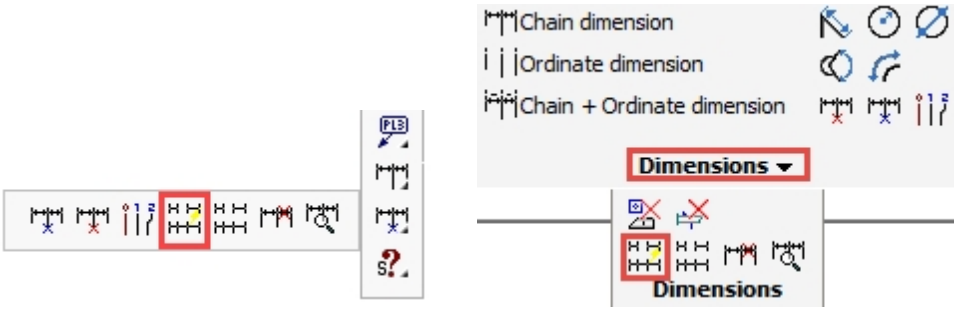


Selecting the new origin point and the end result

The origin (0) will be moved to the indicated point, and all ordinate dimensions in the same set will be automatically updated.

Search for Dims. and Merge

Command - **Prb_MergeDims**

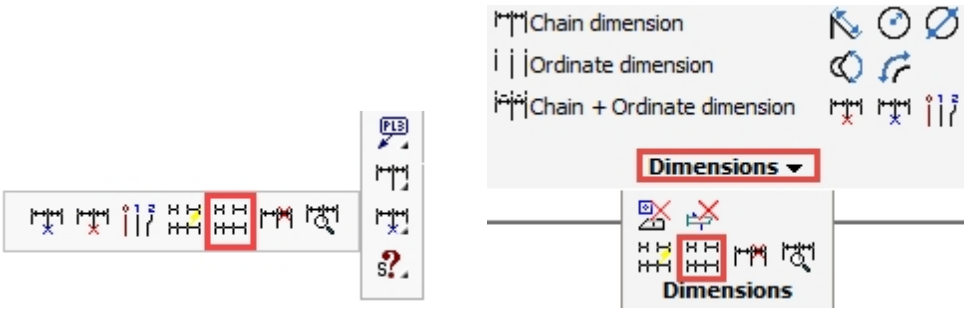


This command will automatically try yo merge several single dimensions into a chain dimension.

It works proximity based : dimension that are close to each other and that share the same base line position will be merged.

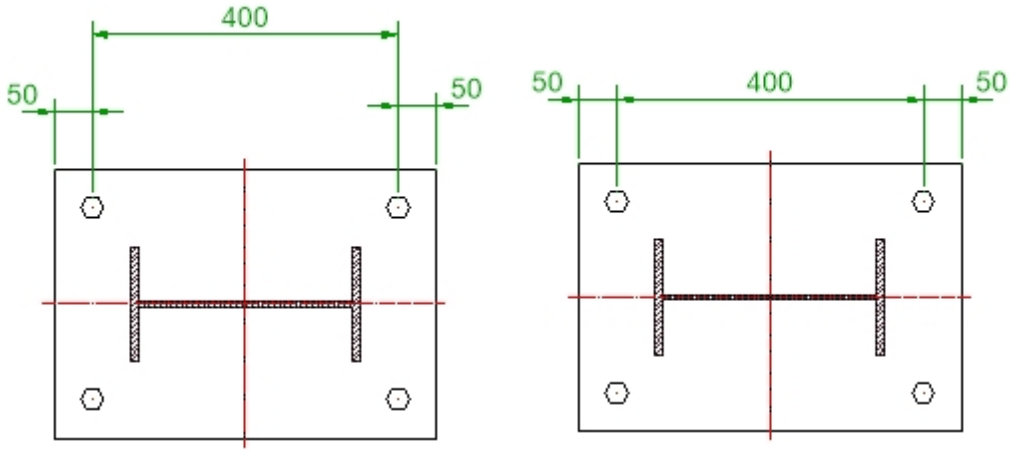
Manually Merge into Chain Dim.

Command - **PrB_ManualMergeDims**



This command allows you to manually merge dimensions into a chain dimension.

It works on dimensions that do not share the same baseline position.



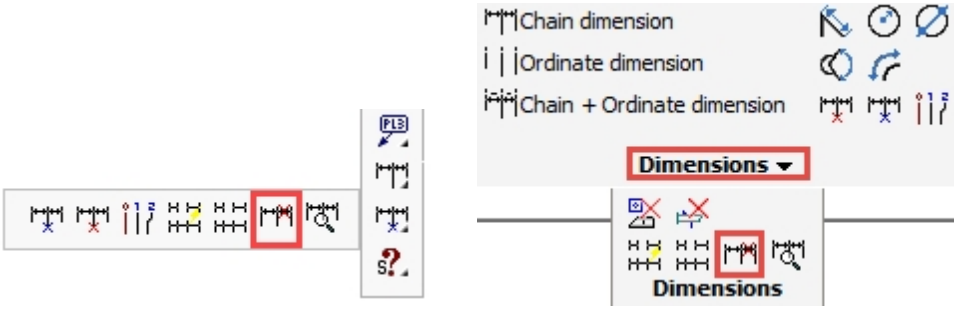
Separate dimensions before using the

The result after merging the dimensions

merge command on them

Disconnect Dims. from Chain

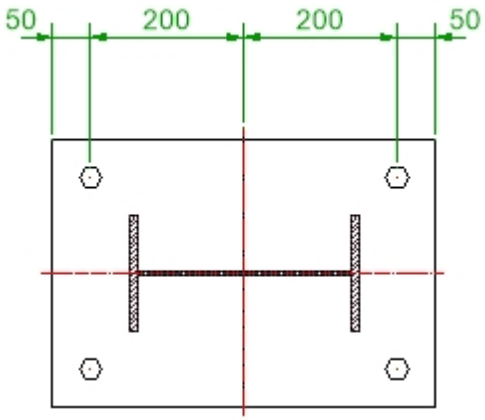
Command - **PrB_DetachDims**



This command allows you to disconnect dimensions from a chain.

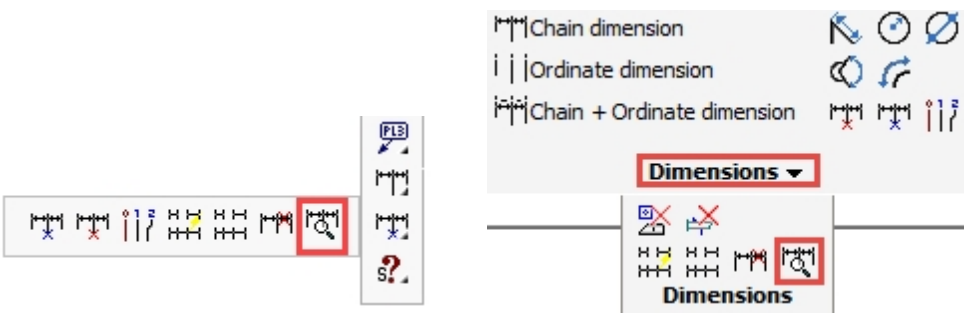
This can be useful if you do not want the dimensions to react as a single chain dimension.

To learn more about the chain dimension's capabilities, see the Modify chain dimension settings topic.

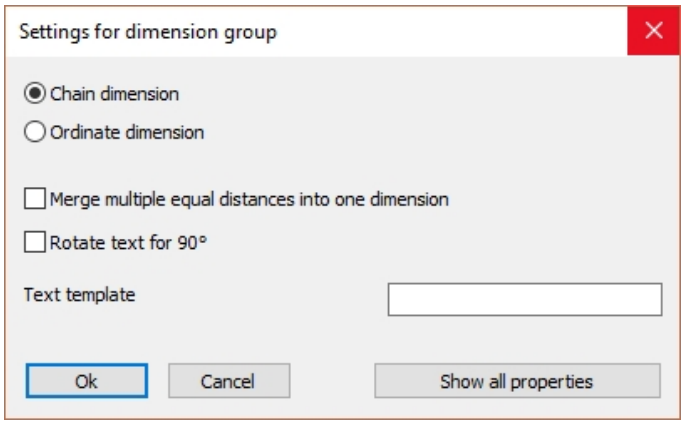


Modify Settings of Chain Dim.

Command - **PrB_DimOptions**

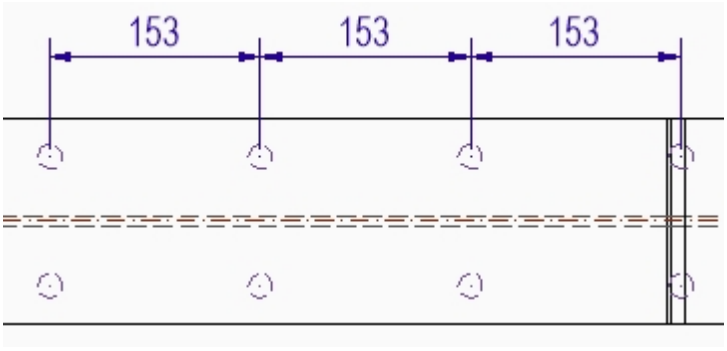


This command will show the following dialog box when you select a linear dimension :

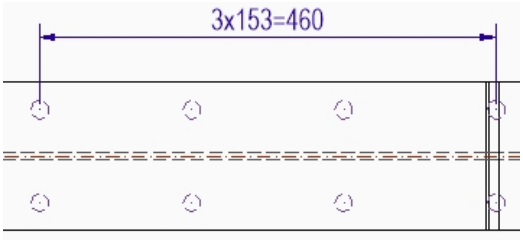


We will explore the options in this dialog :

- Convert the **Chain dimension** to an **Ordinate dimension** or vice versa by activating the radio button
- **Merge multiple equal distances into one dimension** - This option will merge several repetitive dimension into a single dimension. This option is particularly useful for repetitive parts or holes. The number of instances is then shown in the merged dimension :



The dimensions before activating the merge option



The resulting dimension after activating the merge option

- **Rotate text through 90° -** Activate this to rotate all of the dimension text fields 90°
- **Text template -** You can use this field to compose the text for the fields of the dimension(s). Use this field the same way an annotation's text field is set. Refer to the Annotation styles topic to learn how to do this.
  This could be used for example to show the section size of the dimensioned part.
  The actual measurement value can be entered here as the variable **%Measure%**.

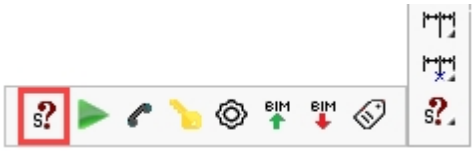
Parabuild General

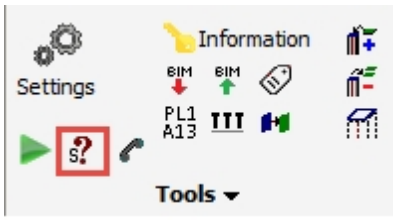
- Help
- Exercises

- Allow remote control
- Project data
- Parabuild Information
- Settings
- Export BIM to file

Help

Command - **PrB_ArxHelp**

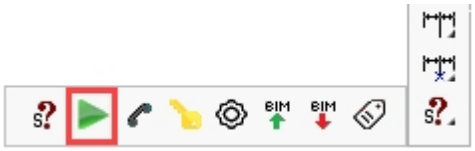


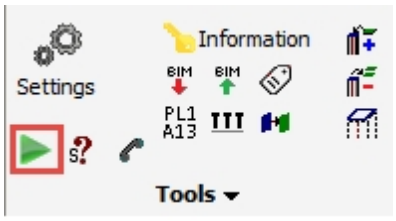


This command will open this manual which was stored during installation of Parabuild.

Exercises

Command - **PrB_Exercises**

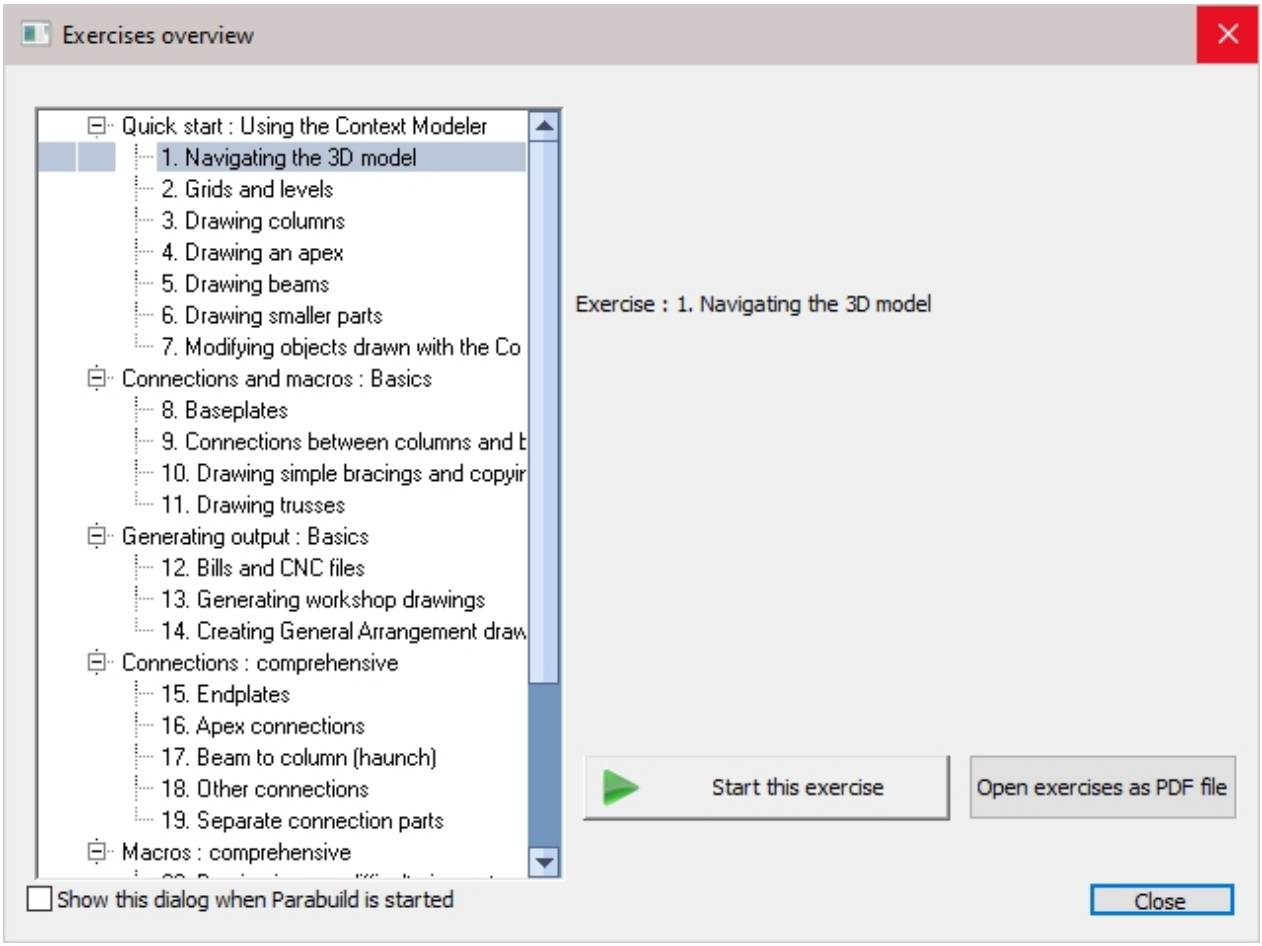




These exercises are designed to familiarize the user with the basic functionality of Parabuild.

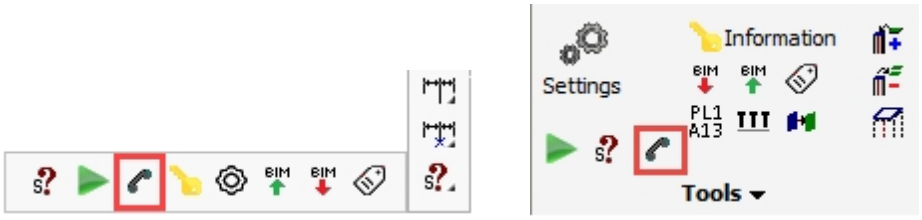
They will take you, step-by-step from drawing a typical 3D structural model, adding connections and finishes, to generating General Arrangement drawings, workshop details, Bills-of-material, and DSTV NC files for CAM machines.

You may work on the exercises directly from with the Parabuild interface - or the PDF version of the exercises may be opened for printing, which can sometimes be easier to use.



Allow Remote Control

Command - **PrB_RemoteControlHelp**



This tool allows a support technician to take control of your computer.

By giving the ID and password to someone else, they can see your screen and execute actions on your computer.
It is a great tool to train and help you remotely, and also to resolve technical issues.

However be careful who you give access to your computer!

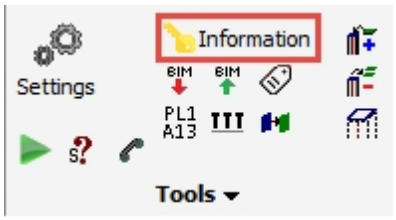
Generally, if you contacted Parabuild or a Parabuild reseller yourself for assistance, then you know who will take control of your computer.

However if someone calls or emails you and comes up with an excuse for needing access to your computer, then you are most likely dealing with a scammer. After gaining access to your computer, such people might lock your computer and all of it's data and require you to pay a ransom before unlocking your computer.

Access to your computer with this tool is only possible if you start this tool and then give the ID and password to a person.

Parabuild Information

Command - **PrB_Info**

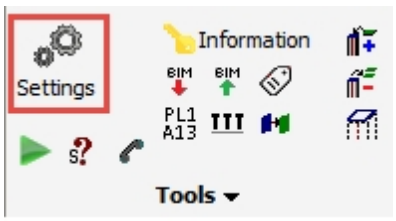


Here you will find information about your current Parabuild license and version.

See the License management topic for more information about this.

Settings

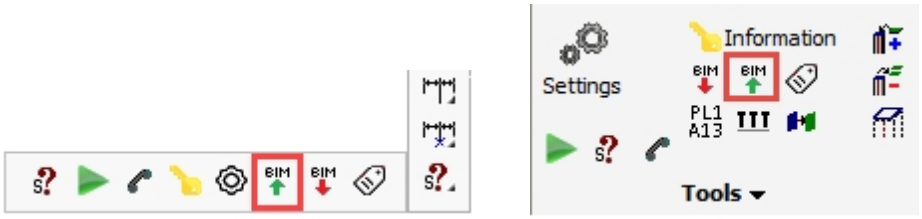
Command - **PrB_Settings**

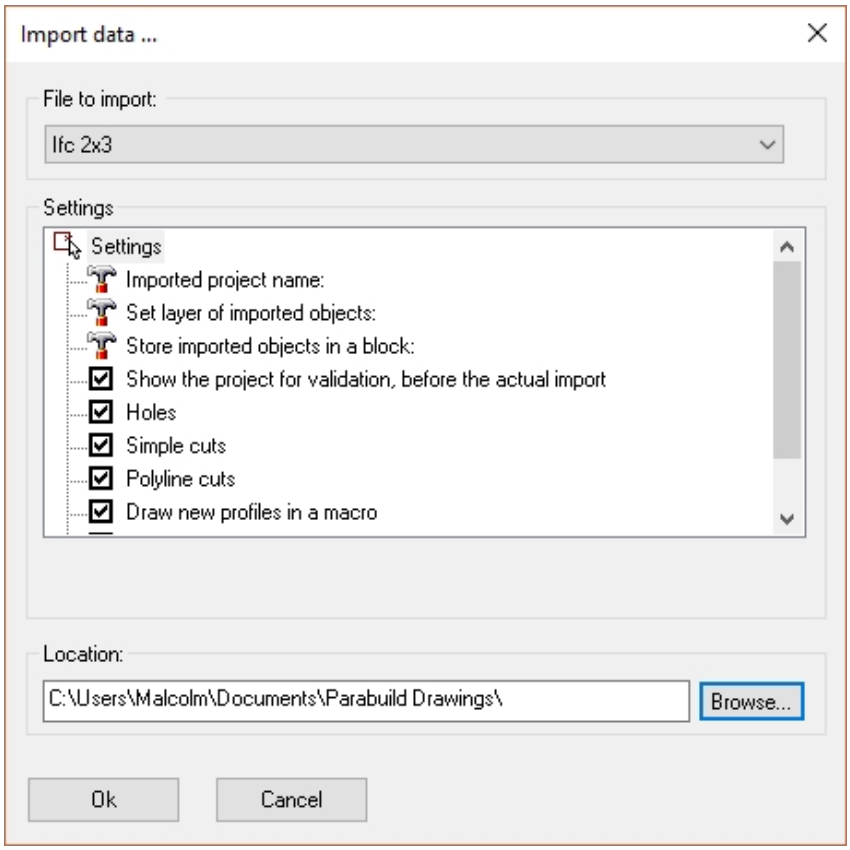


For more information about this dialog box, see the Global Settings topic.

Import File

Command - **PrB_Import**





With this command you can read files of types that are not recognized by AutoCAD.

The following file types can be read:

- **Ifc 2x3** : Allows you to read files coming from for example Revit or ArchiCAD, with BIM data. It is used widely in the EU.
- **CIS/2.1 LPM6** : This format is used widely in the US. It has been tested to work together with CADWorx Structure. The format is not actively being expanded anymore, and if possible it is recommended to switch to Ifc.
- **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 stands for **B**uilding **I**nformation **M**odeling.

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 is known as BIM data. Parabuild saves this data in the drawing together with the geometric data.

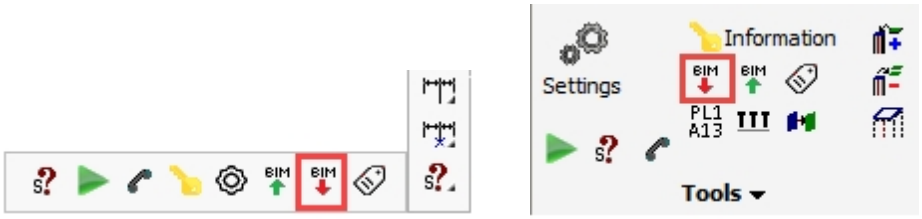
BIM data can be exchanged between different applications with the help of the IFC format.

Only the Ifc and CIS/2 formats support exchanging of BIM data.

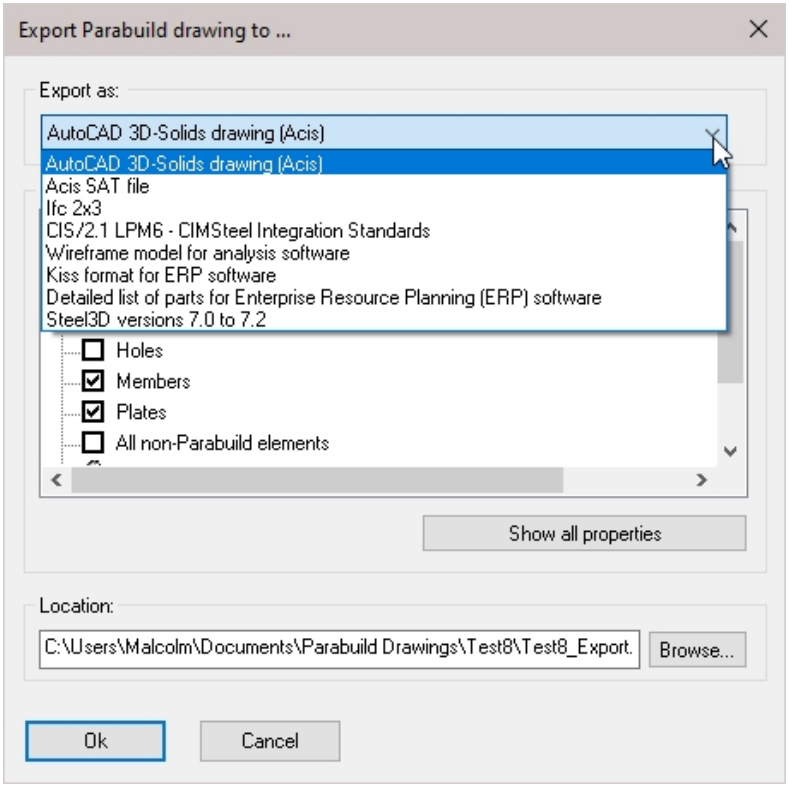
The other formats support only the exchanging of geometry.

Export BIM to File

Command - **PrB_Export**



With this command you can export the current Parabuild drawing to a range of file types that AutoCAD cannot write directly.



The following file types can be written:

AutoCAD 3D-Solids drawing

Will export to an AutoCAD 3D-Solids (DWG) file so that each Parabuild part will become a 3D Solid without any built-in intelligence.

Elements to export:

Template for bolts - Default templates - M%PbColBoltDiameter% and %PbColBoltAssemblyNorm%

Template for plates and members - Default templates - %PbColPosNumber%, %PbColMarkNumber% and %PbColName%

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 recognize the DWG format.

Elements to export:

Template for bolts - Default templates - M%PbColBoltDiameter% and %PbColBoltAssemblyNorm%

Template for plates and members - Default templates - %PbColPosNumber%, %PbColMarkNumber% and %PbColName%

Ifc 2x3

Allows you to deliver the Parabuild drawing with BIM data to the client for checking and planning purposes.

Allows you to write files that are readable in Revit or ArchiCAD, with BIM data. It is used widely in the EU.

Elements to export:

Add profiles with a curved path - This should be disabled if Autodesk 'Advance Steel' will be used to read the file.

Use counter clockwise PolyLines for sections - This should be enabled if Autodesk 'Advance Steel' will be used to read the file.

CIS/2.1 LPM6 - CIM Steel Integration Standards

This format is used widely in the US. It has been tested to work together with CADWorx Structure. The format is not actively being expanded anymore, and if possible it is recommended to switch to Ifc.

WireFrame model for analysis software

Used for exporting the Wire-frame model in DWG format - not widely used.

Elements to export:

Template for plates and members - Default templates - %PbColPosNumber%, %PbColMarkNumber% and %PbColName%

KISS format for ERP software

The **KISS** format (Keep It Simple Steel) is a format for exchanging construction data with ERP/MIS software.

Elements to export: **Also export field bolts** : Shop bolts are always exported
Export nuts and washers : Nuts and washers get separate 'detail parts'
Also export labor info : Will calculate and export labor data for all holes, cuts and burns
Allow accumulation of labor : Should normally be disabled: according to KISS specification the counts should be per piece. But Kiss files written by 'Tekla structures' are accumulated so that is why this is available as an option.
For the main part, use the assembly number instead of the main part's position number
Export type - Options : Estimate / Advanced bill / Final bill / Revised bill / Change order
Sequence template : Optional, to make the sequence field a combination of phase&sequence. Currently supports %Sequence% and %Phase% ex: "%Sequence% - %Phase %"
Section name column keywords : Columns with these keywords will be used for the section shape's name. These keywords are accessible in the profile section tables.

Assembly number for sheet name when the assembly does not have a shop drawing : When no sheet was found with this assembly on it, the number could serve as a good place holder (default on)
Default drawing number when assembly has no shop drawing : This is used when the option '*Use assembly number for sheet name*' is not active

Detailed list of parts for Enterprise Resource Planning (ERP) software

This will generate a simple text file (.txt) that contains a complete list of all parts in the 3D model. All of the data fields that Parabuild has for the parts are also stored.
It can be used for ERP software that only supports reading of simple text files.

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.

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.

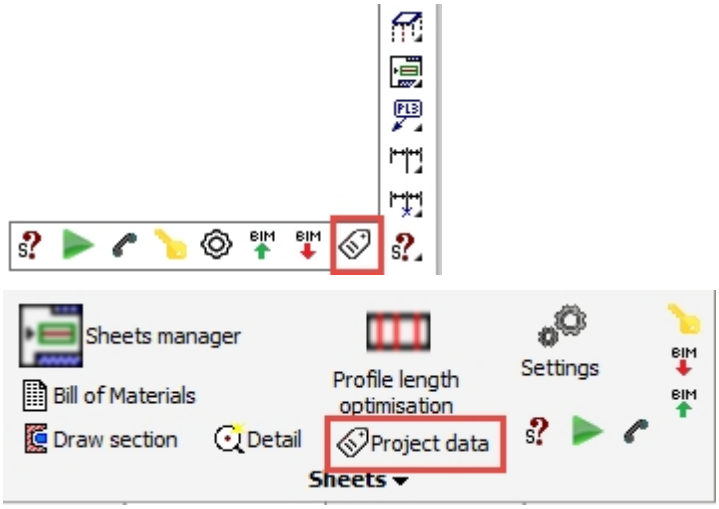
BIM data can be exchanged between different application with the help of the IFC format.

Only the IFC format supports exchanging of BIM data.

The other formats support only the exchanging of geometry.

Project Data

Command - **Prb_ProjectSettings**



Whenever Parabuild generates a new 2D sheet, it will insert a title block into it.

The text that should be inserted in the fields of these title blocks can be entered in this dialog box.

The 2D sheets will always guard the project data. So changing the project data in this dialog box will not automatically update the text fields in all the 2D sheets.
To update the project data in 2D sheets, right-click on the sheet to Change Scale / Format / Project data. See the Right-click on a 2D Sheet topic for more information about this.

More about the project data fields in this dialog box

In this dialog box the first 15 fields are fixed and always there, no matter what the title blocks contain.

These 15 fields are used not only for filling title blocks in sheets, but also for writing project data to KISS format files for example.

These 15 fields will work across all languages. If you would change the Parabuild language then the field values will stay intact.

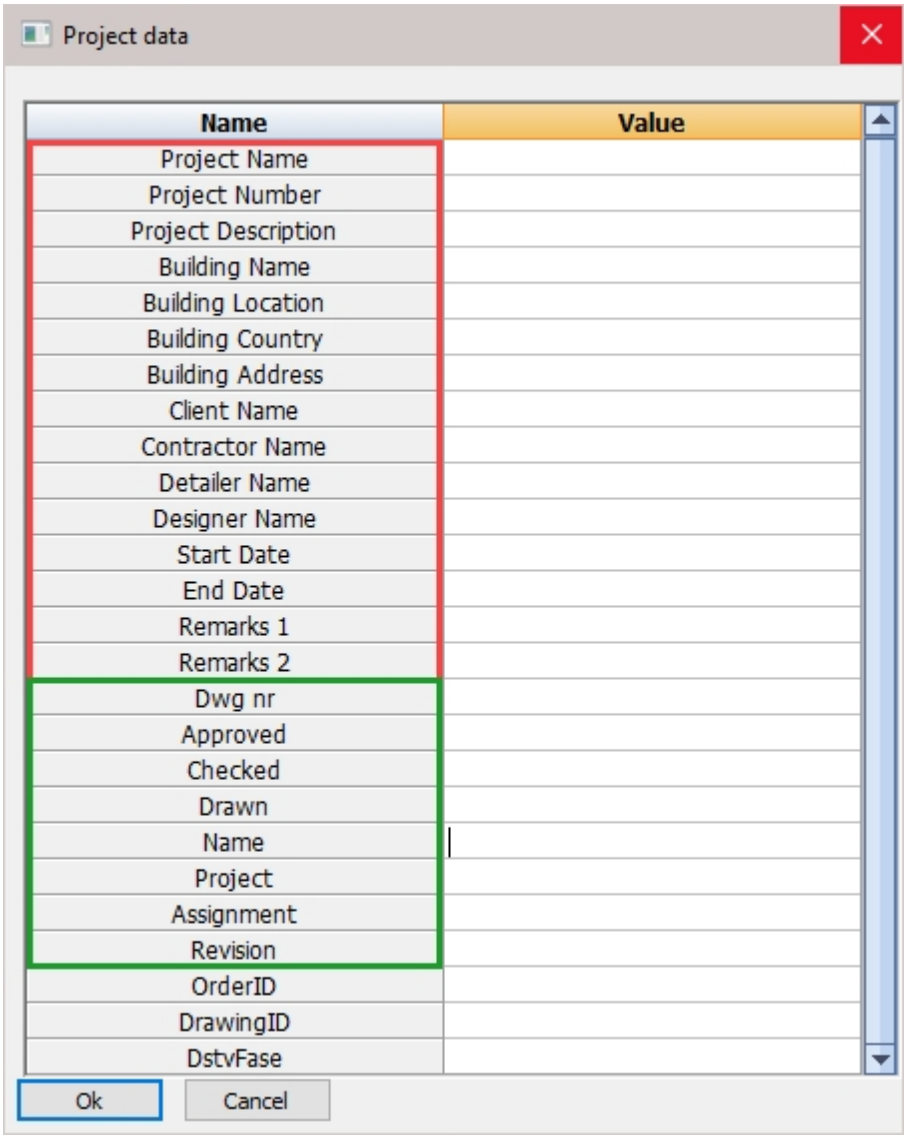
In the below image, the fixed fields are indicated in red.

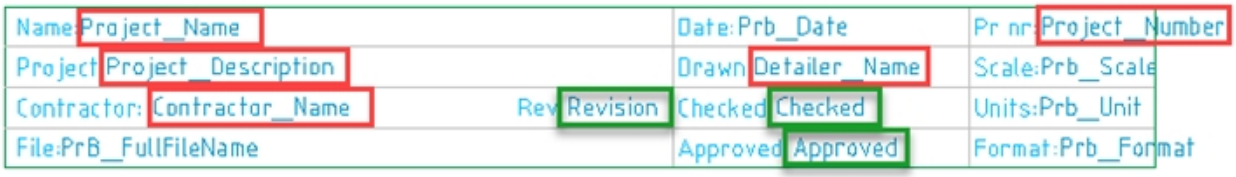
When using these fields in a title block, it is possible to use the text "Project Name" or "Project_Name". Both texts will be replaced by the actual project name.

The dynamic fields are indicated in green. These fields can be entered freely inside title blocks.

More dynamic fields can be added by simply typing them in a title block, and saving the dwg file.

When you open the Project data dialog box, all of the title blocks that start with the current language (English*.dwg) will be loaded and all text fields are analyzed. The non-standard fields are then added to the list of fields.





Miscellaneous

Center of Gravity

Command - **PrB_GetCenterOfGravity**

This command can work on a single or on multiple parts.

It will draw a plane object at the center of gravity location of the all the parts combined.

Phases

Large projects may need to be divided into phases so that each phase will conform to the planning schedule.

By dividing into phases, it's possible to determine in advance which parts of the overall project need to be fabricated and delivered to site in a specified sequence.

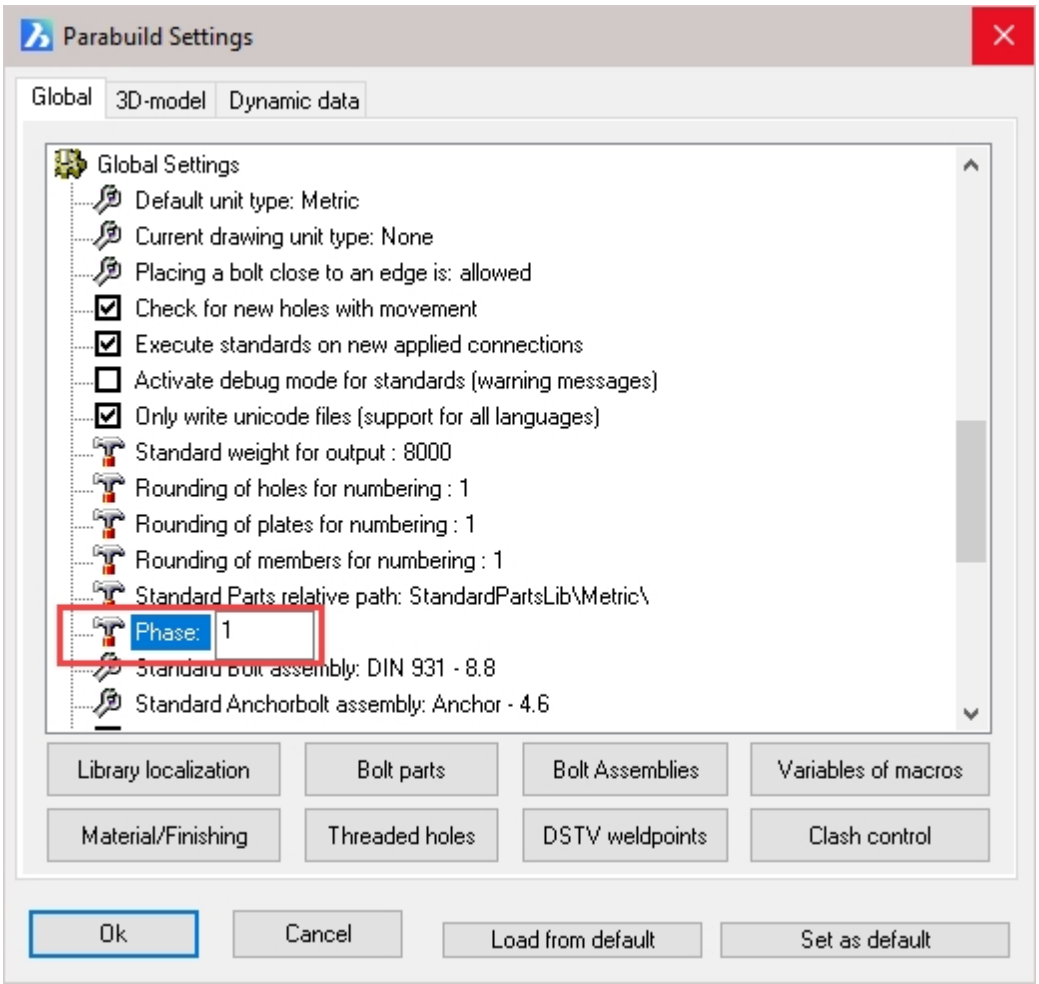
In Parabuild, it's possible to split any project into any number of phases.

Organizing the phases

The the default phase is set at **1** - This can be adjusted in the Global Settings dialog - any adjustment made here will define the phase for all following component parts of the structure.

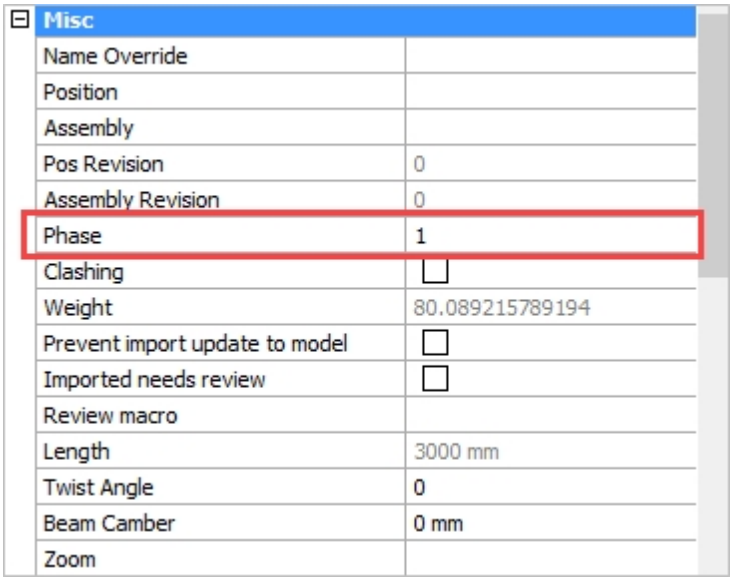
If for example, a structure comprises of multiple levels, with the levels to be built in succeeding phases - it will be necessary to adjust this default setting for each phase.

You can change the Phase for individual components by adjusting the Properties of objects.



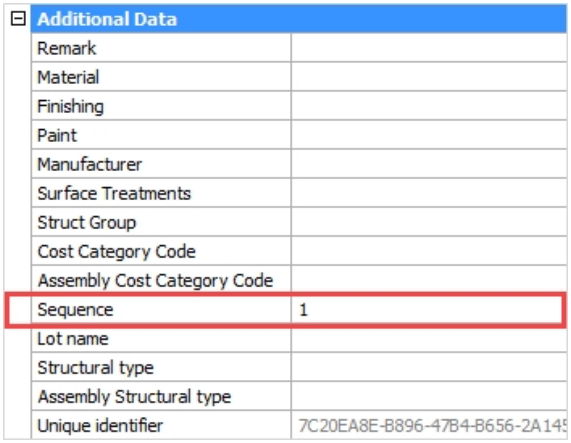
Changing the Phase in 'Properties'

To change the phase for individual parts, click on **Modify** > **Properties** from the top menu and select the member to change. Adjust the value in the dialog.



Erection Sequence

All plates, profiles and structures have a sequence property



The sequence of a part has the following capabilities:

- The reference assembly of a bolt can be determined by the sequence number of the assemblies
- The sequence is exported to KISS file
- It should be available in bills when you make the columns : %PbColSequence% (doesn't work yet)
- A planning tool for erection is a future addition

Advanced - 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. 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 wireframe of a building.

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

To learn how 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 draftsman 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 draftsman modifies them.

- Geometric rules-based drafting: The draftsman 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.

All of the macro commands are accessible from a hidden ToolBar.

To activate this hidden toolbar, you will need to right click any Parabuild toolbar, and then select **Parabuild English >** Activate the toolbar **Geometric Rules**

The commands reading from left to right include:

- Create new macro
- Create an array of macros
- Set macro as current
- Calculate all macros
- Edit macro
- SmartCopy settings
- Edit macro groups

- Create dialog boxes
- Edit automatically translated text
- Copy a macro
- Manually copy a macro without geometric help

Geometric Rules

- Parallel
- Perpendicular
- Coincident
- Concentric
- Tangent
- Equal radius
- Distance between
- Angle between
- Radius
- Create bolts pattern
- Create coordinate system
- Create plane help geometry

Macros and Modules

Introduction to macros

Geometric constraints determine and control the relationships of objects with respect to each other. An object that is constrained to another object, cannot move independently of the object to which it's constrained.
An object may be constrained against another profile, a plate, a line, a point, or a combination of each.

Geometric constraints are the foundation principle of parametric drafting.

Macros in Parabuild are represented green spheres. The Macro is a collection of Modules containing data which defines the shape, size, and form of it's components parts, together with their relationship to each other. This is done by the application of Geometric Rules.

Macros

A macro is a collection of modules. The Geometric Rules are contained within the Modules. It's not compulsory to create a macro using more than 1 module - all data can be compiled into one module, but it's not advised, for reasons given in the next chapter (Modules)

As an example, a typical beam to column connection may be comprised of an end-plate on the

beam connecting to a column with web stiffeners - making the connection complete would be the bolts fitted through both the end-plate and column (in this instance) flange. The component parts of this connection are contained within the modules making up the macro.

This macro may be re-used in all situations where the supporting column and incoming beam are of similar profiles, or groups, because the same Geometric Rules will apply, however it will not work if one or other of the profiles are different to that defined by the macro. For example, if the column has been replaced by a channel section, the stiffeners cannot be applied.

Modules

A module is a collection of Geometric Rules: if a Geometrical Rule is created, it will always be added to a module. A module will split the connection macro into a number of logical parts, which in doing so, offers several advantages:

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

To prevent modules from contradicting each other, there is a list for each module that contains the various elements used by the module and which of these elements it adapts.

If a module needs a certain element (or a part of it), its possible with one of the 3 “Possession Degrees”:

- **Fixed** - This means that the module **uses** the element **as a basis** and the module cannot change or move the element.
- **Rigid** (=to move) - The module cannot change the size of the element itself, but can **move** it.
- **Flexible**. The module can **adapt and move** the element.

These 'Possession Degrees' ensure that modules will never contradict each other, but there are rules:

1. An element can only be '**Flexible**' or **'Rigid'** in 1 module in 1 macro. And that is the case for the entire 3D drawing.
2. There is no restriction on the number of '**Fixed**' elements in a module

The '**Rigid**' Possession Degree is only needed if an element will never be adapted intelligently, but should be moved.

All elements in a module are added to the module 'Element list' with one of the 3 possession degrees, the exception being '**Profiles**'

A profile may be split by means of its Cuts, it's therefore possible that a module does not posses the 'Flexible' possession degree in 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 end cuts and internal cuts separately.

Parabuild calculates modules in series. If a macro is calculated, then each module in it is calculated in sequence. Parabuild will choose automatically which module must be calculated first. This depends on which modules are dependent on which modules.

As an example, assume a plate with bolts:

There is a module '**plate'** that entirely defines the plate (width, thickness,…). There will be a second module '**bolts**' that defines the bolt pattern on the plate. The bolts module will become dependent on the plate module because the bolts module uses the plate as '**fixed'**.

Parabuild will calculate the plate first, followed by the bolts module. We could call the plate module the parent module of the bolt module.

Due to this serial manner of calculating the modules, a module can never adapt an element upon which one of it's parent modules is dependent. This would be a circle of dependence, which can never be calculated.

Parabuild detects such bad dependencies automatically : it will refuse to add geometric rules that do not comply to this rule.

Geometric 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.

Parallel

Command: **Prb_Parallel**

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

Perpendicular

Command: **Prb_Perpendicular**

Perpendicular planes: For example to obtain a rectangular plate

Coincident

Command: **Prb_Coincident**

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 be used between a cylinder and a plane, a cylinder and a point,... Cylinders appear when we use rules on round tubes or bended profiles.

Concentric

Command: **Prb_Concentric**

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

This rule can also be used to set a line concentrix with a cylinder's axis (or vice versa).

Tangent

Command: **Prb_Tangent**

This rule can be used only in the following combinations:

- Plane vs Cylinder
- Line vs Circle

Equal radius

Command: **Prb_EqualRadius**

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.

Note! 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.

An alternative for angle dimensions could be a sin/cos equation.

Radius

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

Geometries

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

- point
- Line
- Circle
- Plane
- Spline
- Ellipse
- Cylinder
- Helix (Diagrid)
- Cone
- Torus
- Sphere

Create Geometric Rules

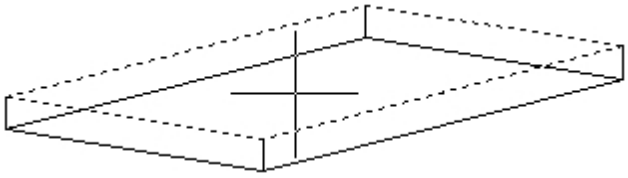
For creating geometrical rules there are icons on two locations. The first range icons are in the hidden ToolBar Geometric rules.

The second range has been processed in the dialog box Edit macro

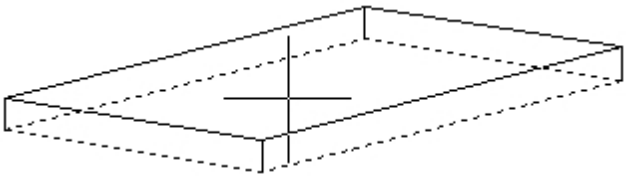
The icons are all the same: they were added twice for the convenience of the user.

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 we 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 inside the area of the plane and click once on the left mouse button. Take care that your cursor is not located 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 cycle through all the planes under the cursor.



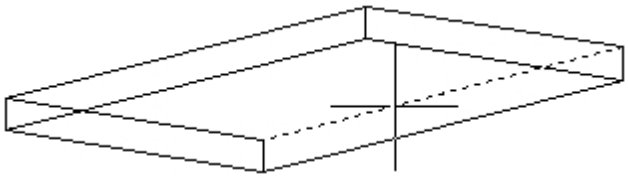
Selection of the upper plane



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

Selection of a line

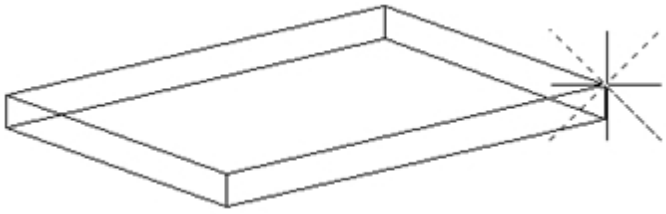
To select a line you need to move the cursor over a line and click on the left mouse button. On the screen you see immediately which line you have selected as a dashed line. If the selection is wrong you can try simply again with the left mouse button.



Selection of a line

Selection of a point

To select a point you need to move the cursor to an endpoint. On the screen you see that a diagonal 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 click 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.

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 are coincident on each other will put more geometric restrictions than two lines that are coincident. 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 and intuitively.

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.

Another example of too many degrees of freedom can be found in the Calculate all Macros topic.

Create New Macro

Command - **PrB_CreateMacro**

This command will create a single new, empty macro.

Create an Array of Macros

Command - **PrB_CreateArray**

This command will create an array.

Before starting this command you will need to create a macro that is ready for being arrayed.

That means the macro should :

- Define 1 or more elements
- The position of these element(s) should be influenced by a variable.
  This variable should be a user-modifiable variable.
  The array will need the variable name, because it will use this variable to position each macro copy's position
- Optionally the macro can also contain a total distance variable.
  This would be a variable determined by Parabuild (so not modifiable by the user).
  The array can use this variable to know the total distance over which macros should be copied.

Set Macro as Current

Command - **PrB_SetCurrentMacro**

This command can be used to set the active macro and module.
The active module is the module in which we want to do modifications.

Because a geometrical rule always must be placed in a module, you must indicate which macro and which 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 opened. Therefore you only need this command if you use one of the 10 geometrical rules commands in the toolbar.

This command also regulates 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 base geometry is adapted. That automatic adaptation is temporarily suspended for all macros in the drawing. You can recalculate the macros with the command Calculate all macros. Now you decide when macros are recalculated.

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

Calculate all Macros

Command - **PrB_Evaluate**

This command recalculates all macros that were modified in the entire drawing (geometrical rules added, element position changed,…)

You will need this command while you are creating geometrical rules.

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

The automatic calculation of macros can be suspended by the Set Macro as Current command and the Edit Macro dialog box.

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).

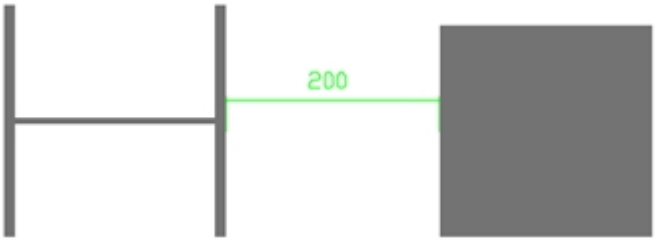
It can however happen that when your elements were not defined entirely and you recalculate, that something unexpected happens with your element (ex. the element is placed some meters further away so that it disappears from your screen).

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 location.

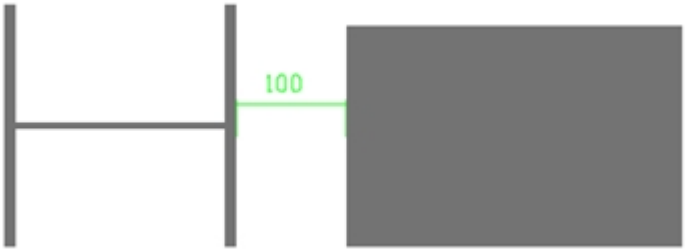
We explain this problem a bit more closely by means of a concrete example:

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

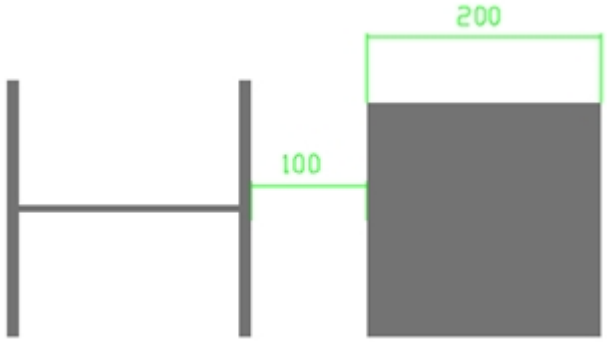
The profile is fixed (we do not adapt). The plate is flexible (move + adapt).



In this image, only one geometric rule was added - the dimension between the plate and profile



In this image, the value of the dimension was changed and the macro was recalculated. But as the plate itself was not defined, the solution was left undetermined, which means either the plate could be moved, or in this instance, made longer.



The solution: 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 skillful manner to test your solution while you are placing geometrical rules.

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

Edit Macro

Command - **PrB_DlgEditMacro**

This dialog box is useful for having an overview of what a macro does, and also for editing the 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

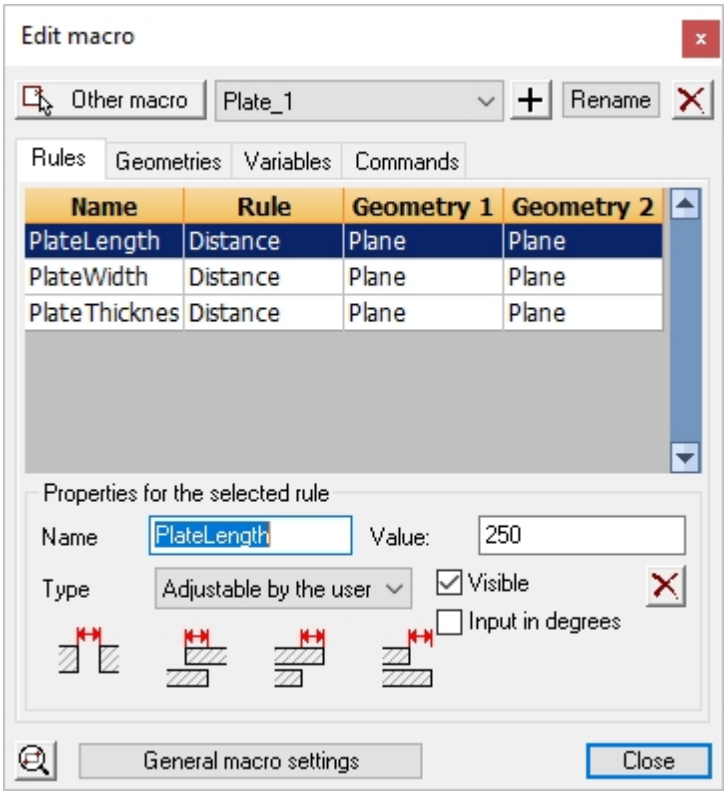
Other macro: Click on this if you want to edit another macro.

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.

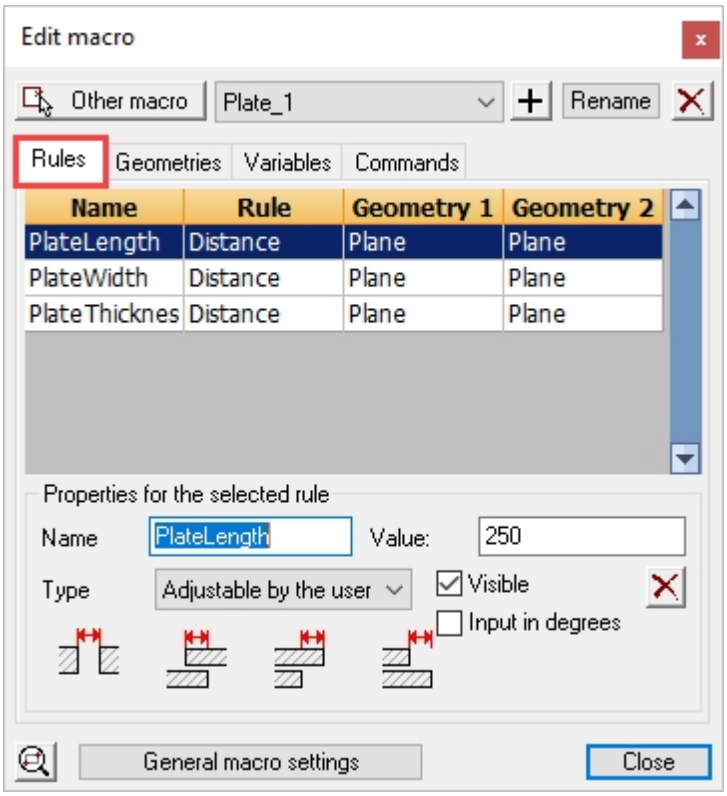
X: Removes the module that was selected.

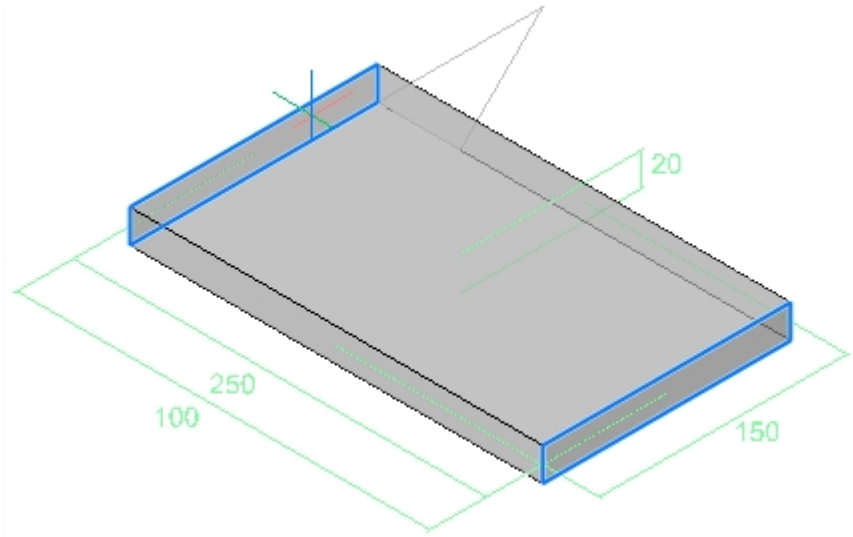


See Also:

- The Rules tab
- The Geometries Tab
- The Variables tab
- The Commands tab

The Rules Tab



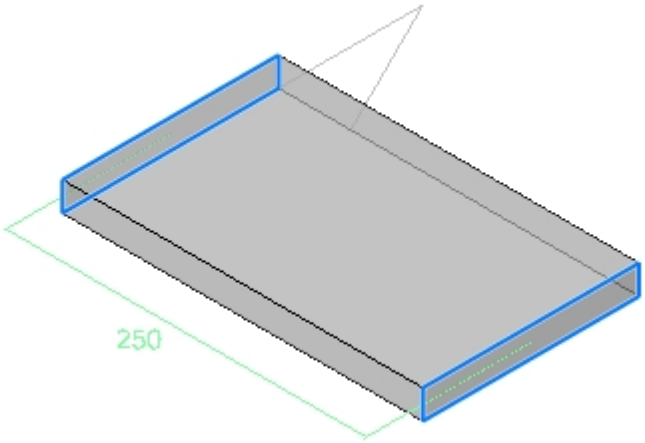


This tab contains a list with an overview of all geometrical rules in the chosen module.

One line in the list presents one rule. The list contains 4 columns. The first is the dimension name - the second column is the type of rule, while the third and fourth columns describe the geometries connected to the rule.

In selecting a rule:

- Options of that rule are shown such as dimension name, value,...
- In the drawing itself the geometries of the rule will be highlighted. In the above example 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: This is actually a variable name that is linked to the dimension. We use the variable 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 variable visible in the Review macro dialog box. You can hide dimensions that contain formulas 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 enter a number (no formula) 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 an equation. For example "Length1+Length2".

You can use the following mathematical symbols to create equations:

- **+-/***: To add, subtract, 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.
- **asin() acos() atan()** : Calculates the inverted sinus, cosinus or tangent.
- () **^**2: Would return the power of 2
- **A % B** : Returns the remainder that results from dividing A by B
- **sqrt ()**: Return the square root
- **log()** : Returns the natural logarithm (base-e logarithm) of the argument
- **log10()** : Returns the common logarithm (base 10 logarithm) of the argument
- **exp()** : Returns the exponential (For example the exponential value of 5 is 148.413159)
- **abs()** : Returns the absolute value. So this will always strip off the minus sign of a value (-5 will become 5).
- **sign()** :
- **int(**) : Returns an integer value even when the input is a comma-separated value.
- **frac()** :
- **dtr()** : Converts the degrees value into radians
- **rtd()** : Converts the radian value into degrees
  NOTE - Internally Parabuild always works in **radians**. Therefore any angle variable calculated by Parabuild will always contain radian values. Care should be taken when variables are used as input or output in the Review macro dialog box.
  All the dimension rules have a Input in degrees checkbox property, which activates the automatic degrees/radians conversion in the *Review macro* dialog box.
  Also all the variables that are created in the Variables tab have an option to do the degrees/radians conversion automatically.
- **Conditions can be used inside equations**
  For example a condition could look like this :
  offset = (useTopSide>0)?0:-thickness
  This equation demonstrates how a switch for the position of a plate can be achieved, for example to switch the plate to switch from left/right side or top/bottom side.
  So if the switch *useTopSide* is larger than 0, the displacement is 0. If *useTopSide* is 0, then a displacement of the thickness of the plate is applied.

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

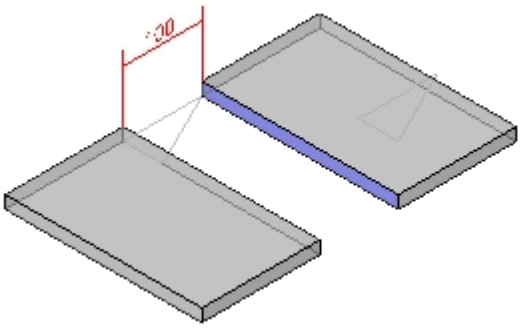
- **Adjustable by the user**: The value of the dimension is a number and can be modified during the use of the macro (the dimension will be adjustable in the Review macro dialog box!)
- **Fixed value**: the value must be a number. This value is not adjustable afterwards during the use of the macro.
- **Flexible value**: You must use this if other dimensions can influence the value of the dimension. Or it can be used to just 'measure' a distance. If you enter for example an equation then this option must be used (this is generally already done automatically for you).

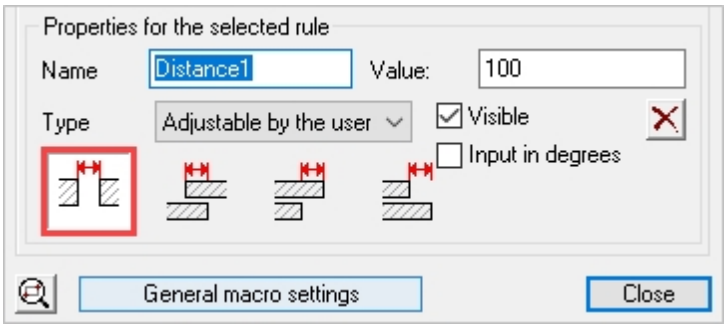
Directions of rules

The directions refer to the relationships between objects - of which there are 4 options:

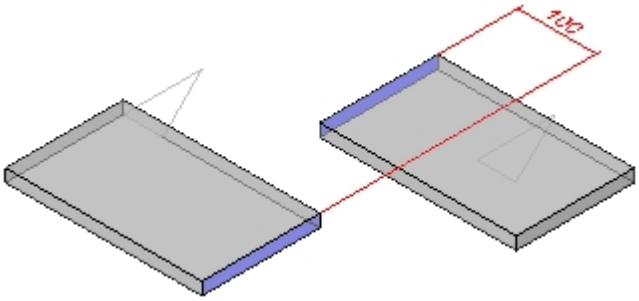
1. Gap between Planes
2. Planes passing through each other
3. Planes are aligned - Plane 1 extends beyond Plane 2
4. Planes are aligned - Plane 2 extends beyond Plane 1

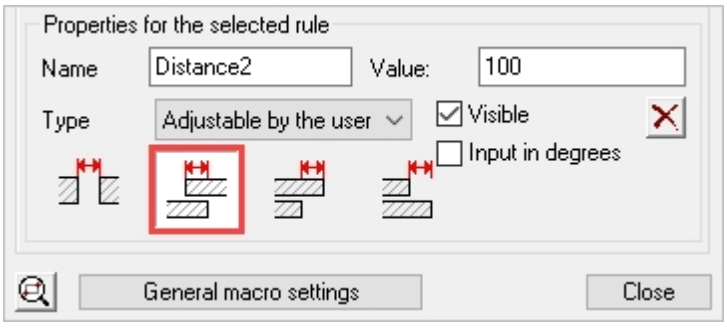
The following examples illustrate the directions:



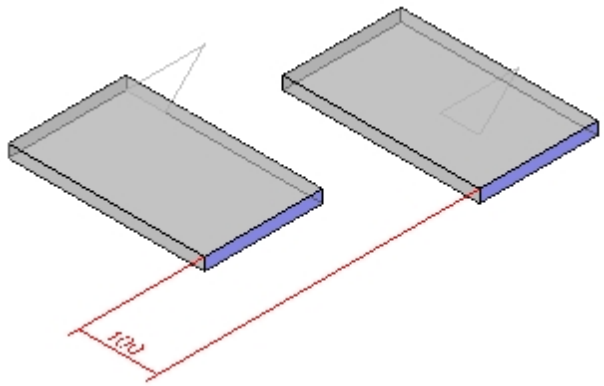


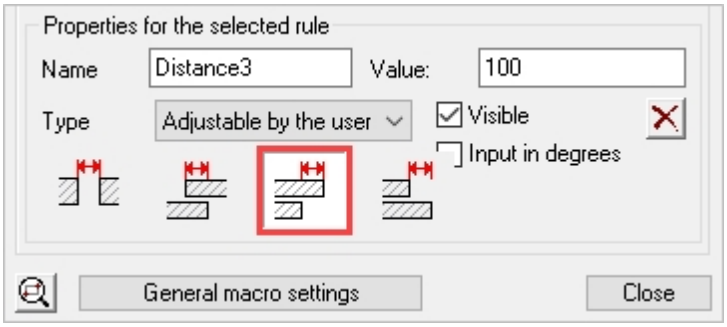
Here the gap between planes of the plates have been set at 100



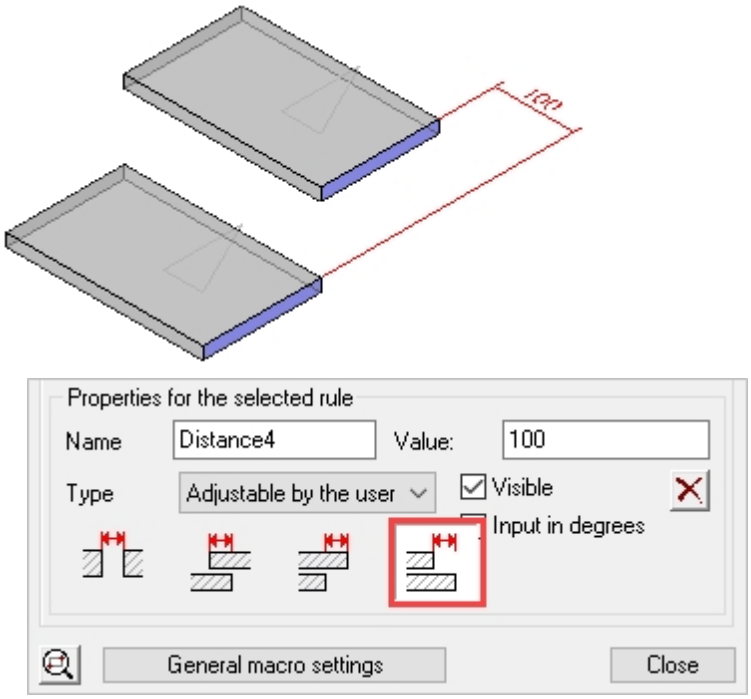


Here the plates overlap one another over a given distance

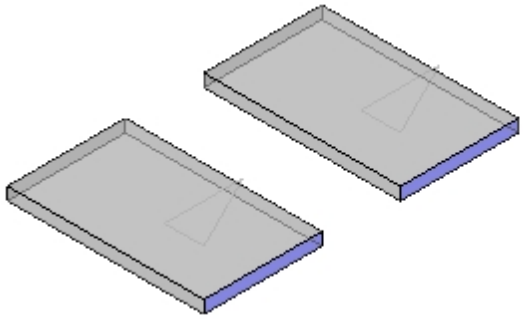


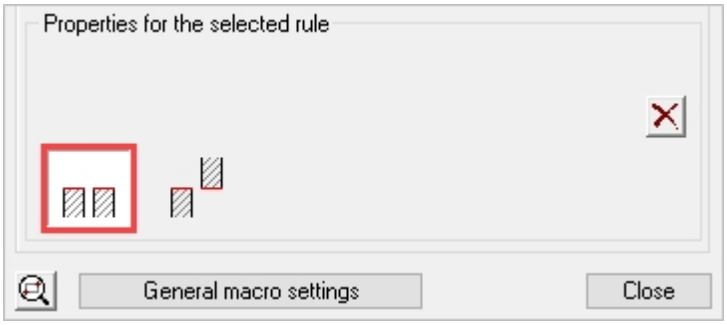


The one plate is put here beyond the other. Which plate is put beyond which plate depends on the order in which the sub geometries were selected during creation of the rule.

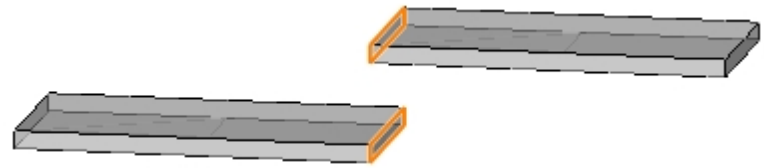


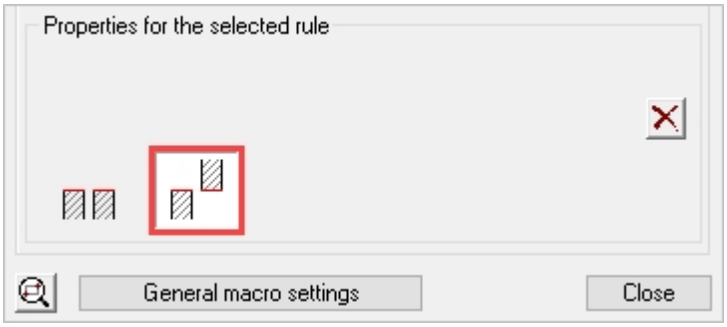
Again one plate is put beyond the other but the roles are switched.





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.

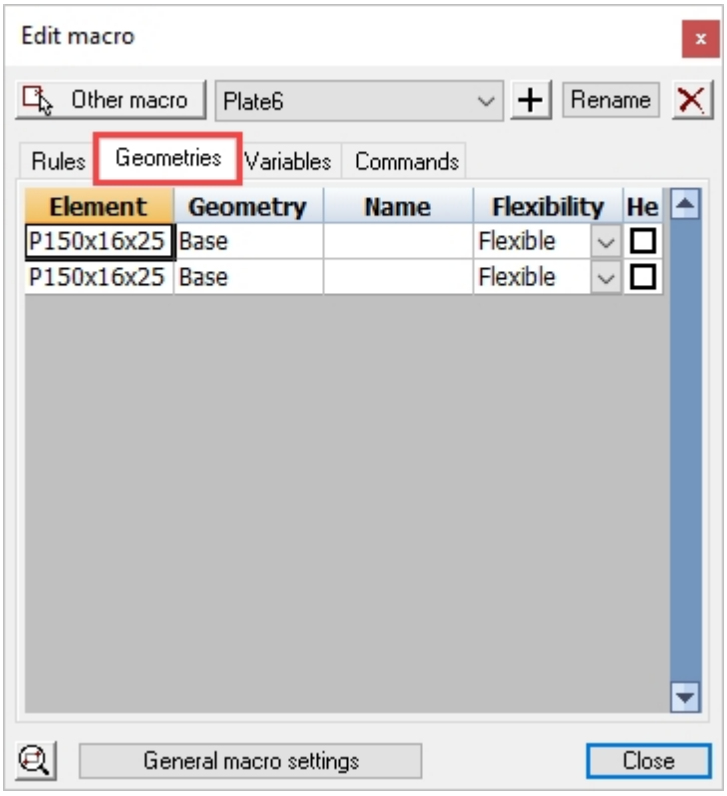




The planes in the middle become placed on each other with their body in opposed direction.

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

The Geometries Tab



This list contains all (sub) geometries of which the module is dependent 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.

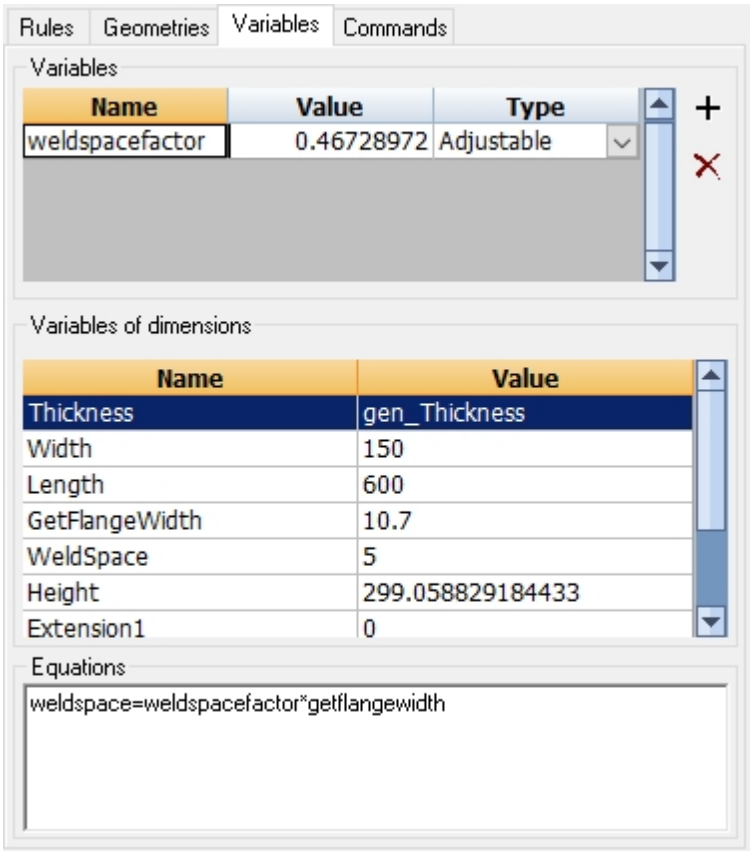
Flexibility:

- Flexible: The element can be adapted and moved.
- Rigid: The element cannot be adapted, only moved.
- Rigid fixed to base: This option only works on cuts. The cut will be rigid relative to the profile itself.
  The purpose of this option is to make an entire profile rigid together with all of it's cuts.
  Using the regular rigid option on a cut would still allow the cut to be moved.
  Using the fixed option on a cut would make the cut fixed in world coordinates. In this case the cut would not move together with the profile.
- Fixed: The element is only used, cannot be adapted/moved.

Helper 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 or plane that determines the intersection point between the two beams.

The Variables Tab



When adding new dimension rules, we are already automatically using variables and equations. The name of each dimension is, as it happens, is a variable.

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

We can only add 1 equation per dimension rule and that is sometimes not enough.

This variables tab can be used if you need to add more variables and equations for doing specific intermediate calculations.

In the above example, the value of the variable *weldspacefactor* is calculated by the equation *weldspace=weldspacefactor*getflangewidth*

The variable *weldspacefactor* can now be used in several places (for example for the value of a dimension).

We will explore the different variables type :

- **Adjustable** - Use this when you want the end-user of the macro to be able to change the value of the variable in the Review macro dialog box
- **Fixed** - Use this when the value of this variable should be fixed at all time. So the end-user can't change the value, but the macro calculation can't change it either. This variable will be visible in the Review macro dialog box.
- **Flexible** - Use this when the value of this variable should not be modifiable by the end-user,

but it should be calculate by the macro. It should thus be calculated by an equation. This variable will be visible in the Review macro dialog box.

- **Fixed Invisible** - Same as *Fixed*, but the variable will not be visible in the Review macro dialog box.
- **Flexible Invisible** - Same as *Flexible*, but the variable will not be visible in the Review macro dialog box.
- **Adjustable invisible** - Same as *Adjustable*, but the variable will not be visible in the Review macro dialog box.
- **Adjustable degrees** - Same as *Adjustable*, but the variable contents filled in by the macro will be converted into degrees before the value is shown in the Review macro dialog. The degrees that the user enters for this variable in the Review macro dialog will be converted into radians because macros always work in radians.
- **Flexible degrees** - Same as *Flexible*, but the variable contents filled in by the macro will be converted into degrees before the value is shown in the Review macro dialog

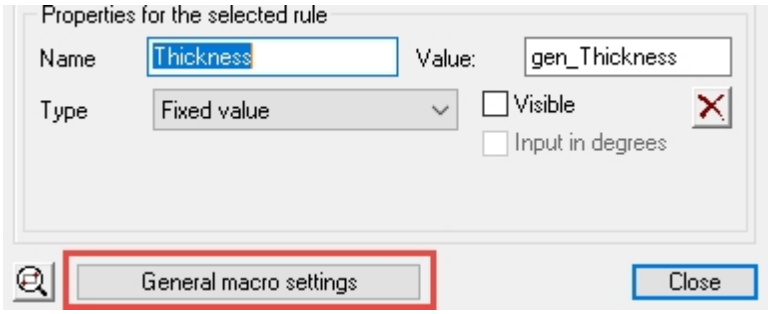
NOTE : Internally Parabuild macros always use radians internally. If we want to input something in degrees, the conversion will have to be done by enabling the degrees conversion option in the variable or in the dimension rule.

For more information about equations and the operations that can be done in equations, see the Rules tab topic.

The Commands Tab

With these commands you can add new geometrical rules or patterns to the module.

General macro settings



This dialog box can be started with a button at the bottom of the Edit macro dialog box.

This dialog box contains 5 tabs that all influence the behavior of the macro in the Review macro dialog box and also the behavior when the macro is inserted, smartcopied, copied by an array and propagated.

We will explore all the options of each tab.

The two tabs **Filters on angles** and **Filters on distances** contain options that are used during the application/insertion of the macro from the library using one of the connection icons.
With these options it will be decided if the macro is appropriate for the base profiles that the user has selected or not.

Filters on angles

It is not obligated to fill any of the filters. You should fill in these 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 of its options will become visible. Only the filters that you activate will later be used.

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.

An 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 quick filters that the user enabled.

The properties are stored per 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 design. 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 topic Create Dialog boxes.

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 will always be adjustable in the Review 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 entire 3D drawing. A practical example of this is the variable for the welding gap between welded elements.

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

You should change the general variables in the drawing that contains the macro and you would 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 that 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 add this gen_ prefix to 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 Review macro dialog box when several macros are selected: it allows editing these macros simultaneously (those with the same short name).

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

Each module will automatically receive a tab in the Review macro dialog box.

But 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 can give both modules the group name “Stiffeners” and the variables of both modules will be merged in one tab called “Stiffeners”.

When tabs are merged, the variable names are also merged :

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

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

Smart Copy Settings

Command - **PrB_AutoApplySettings**

With this command you can 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.

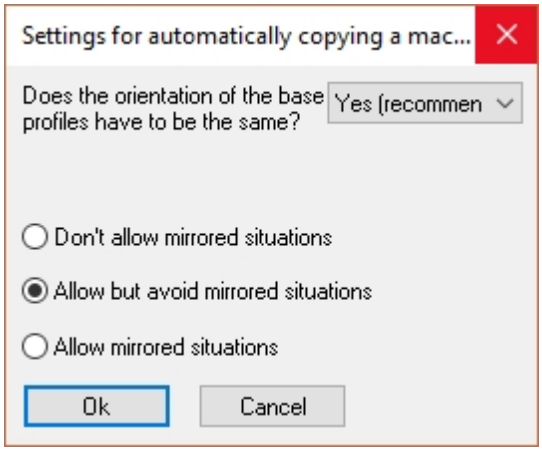
When starting this command you are first asked to select the macro of which you want to adapt the options. These options are stored inside each macro separately.

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:



We will explore the options in this dialog box :

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.

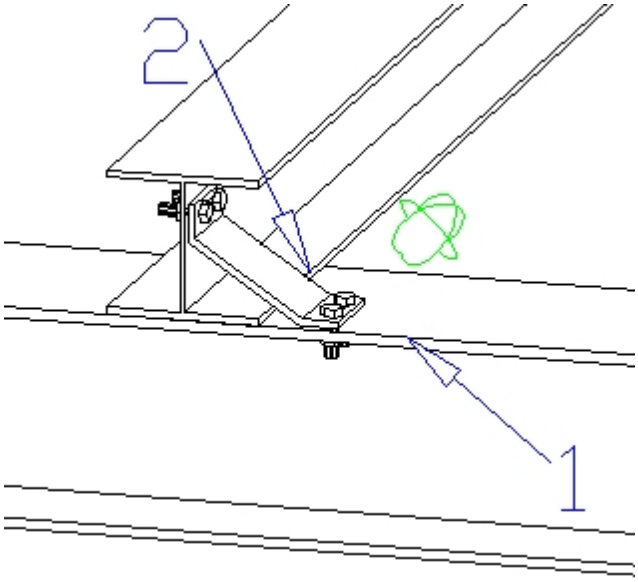
Allow but avoid mirrored situations

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.

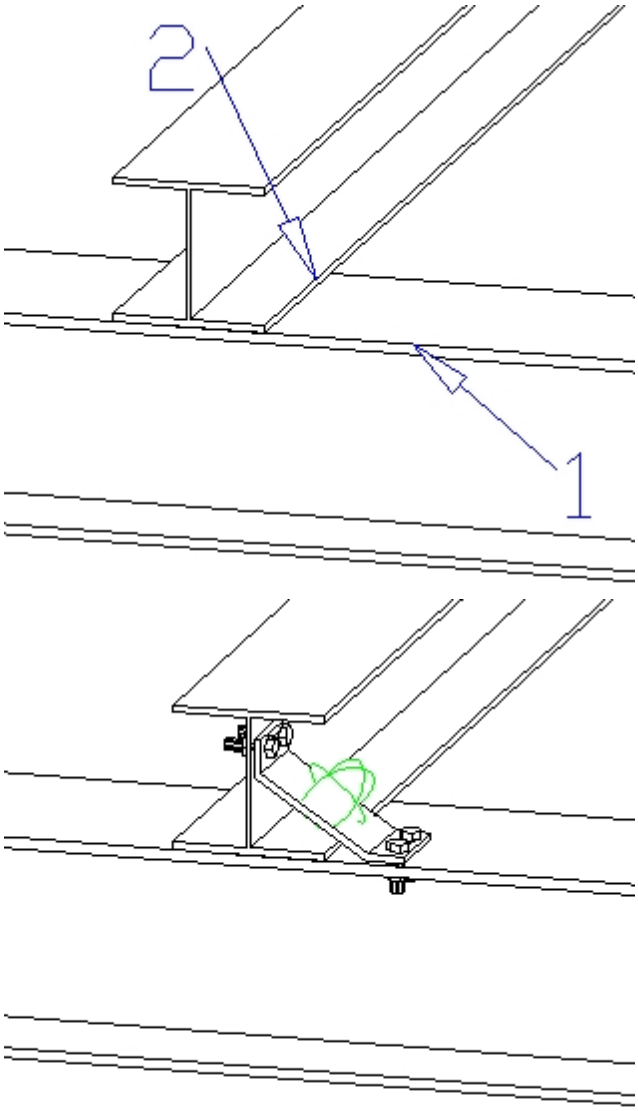


The above image is the image of the macro source itself (the drawing that resides in the macro library). The indicated points 1 and 2 were clicked in the **Macro apply settings** command. The bottom profile is therefore the first; the upper is the second base profile. The above image only illustrates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

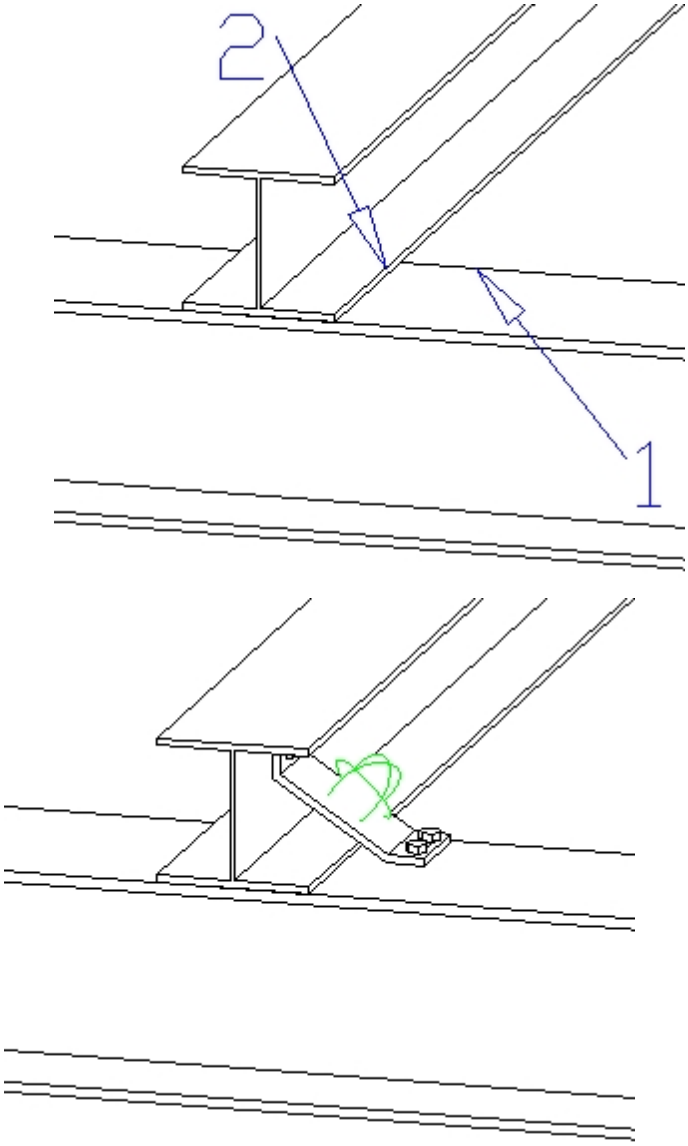
For each scenario the left image shows the selection that was made for applying of the macro. The right image is the result after the macro was applied with those selections.

Scenario 1:



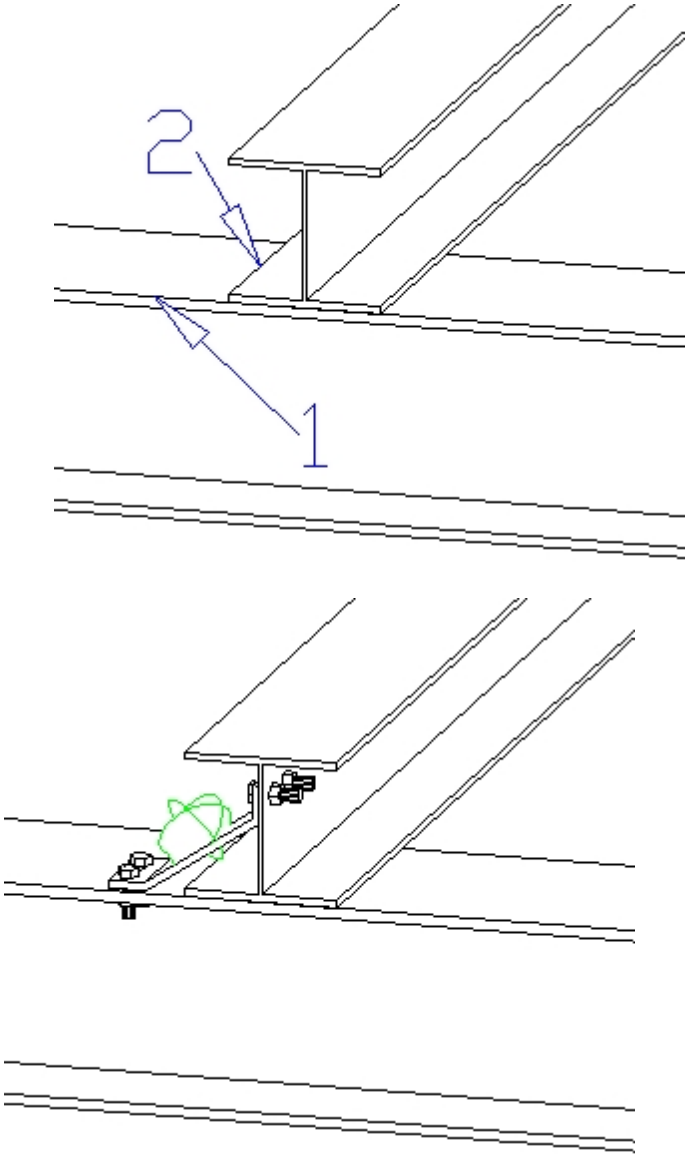
The result of this scenario is no surprise: the base profiles were selected on exactly the same spots as those of the macro source. The macro is copied to exactly the same place as the source macro.

Scenario 2:



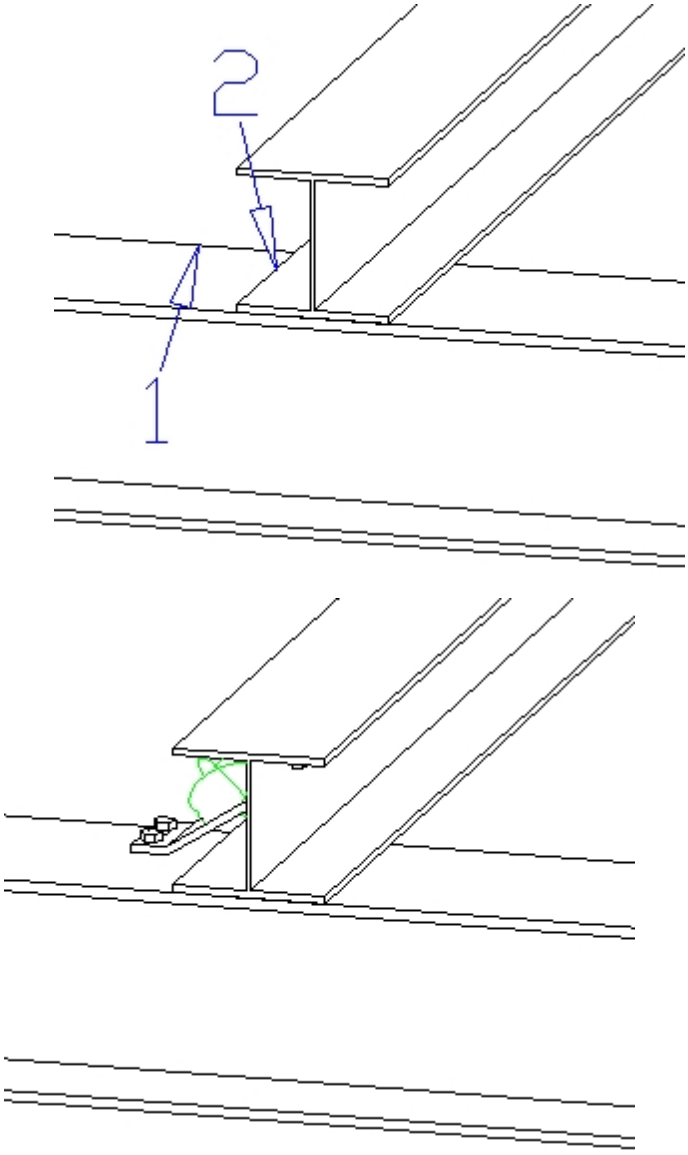
In this scenario the first base profile was selected on another spot, on the other side of the upper flange. The result is that the bent plate is also placed on that side of the flange. The bent plate was mirrored over 1 axis.

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.

Edit Macro Groups

Command - **PrB_EditMacroGroups**

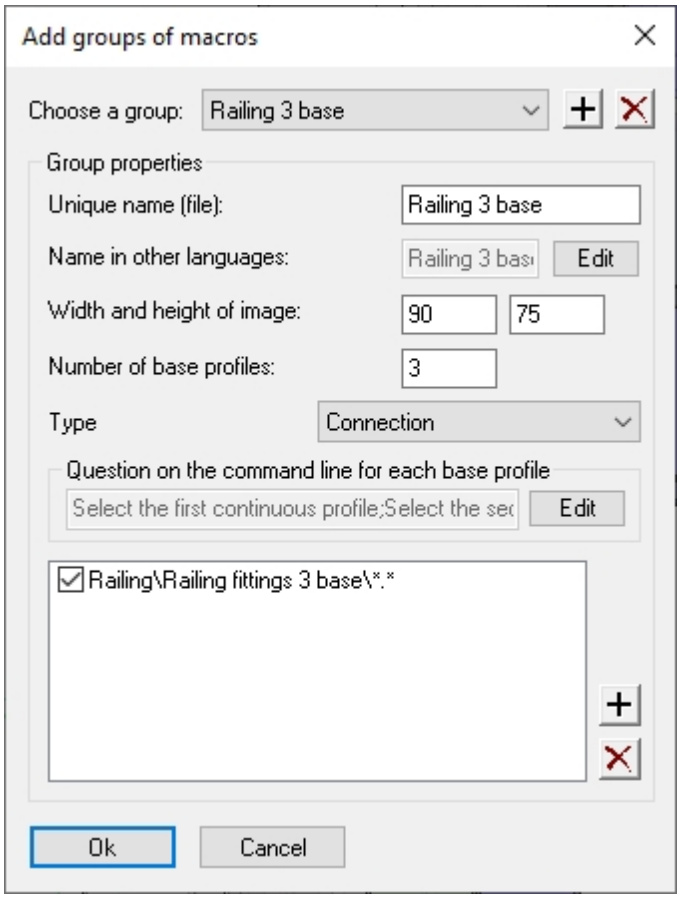
As you already know we have access to the macros library by means of several icons.

Behind each icon a "group" of similar macros can be accessed from the library.

With this command you can edit these groups and create 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 see 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 should contain one main macro that will be applied automatically).

Besides drawings you can 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 creating a new icon that starts the following command:

(PrB_MacroGroup "Haunch connections")

The brackets and quotation marks are obliged! Also add a space at the end of the last bracket : a space is the same as <Enter>.

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).

Alternatively if you do not wish to spend the time on creating a new icon for your new group, you can use the New connection from group command to draw a new connection from any group.

Create Dialog Boxes

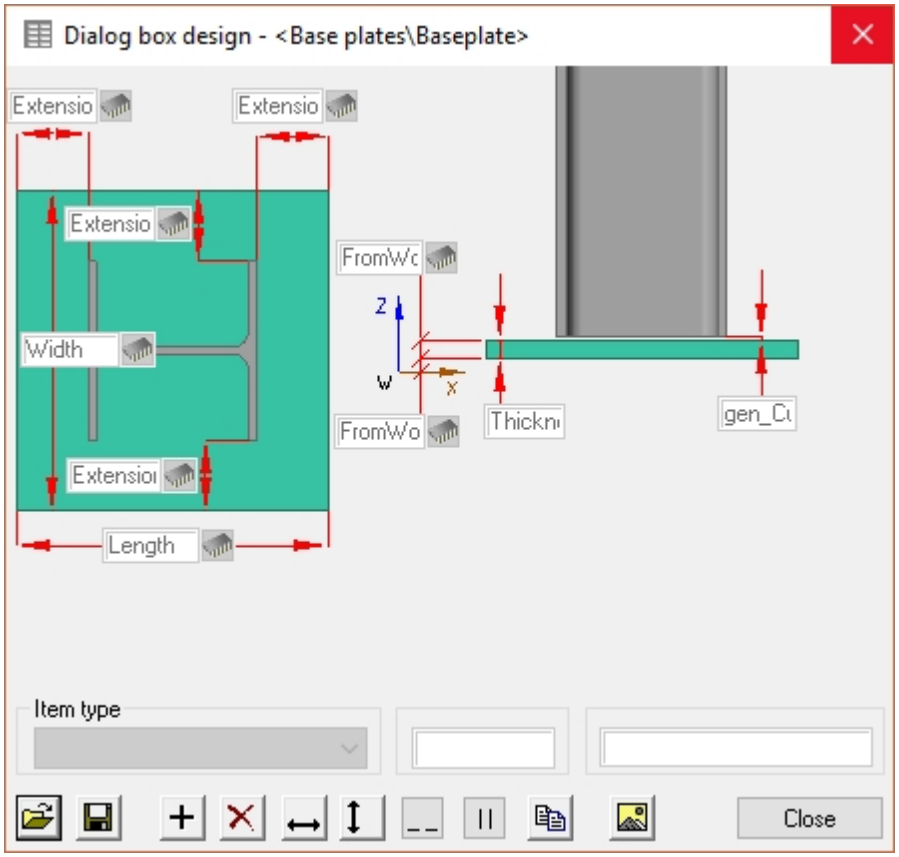
Command - **PrB_DlgDesigner**

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

This design dialog can also be opened directly from within the Review macro dialog box. Just press the <F12> function key, and the dialog of the module that is currently active will be opened in the design dialog box.

To couple a dialog design with a macro/module, one should activate it in the macro. The option for doing this is in the General macro settings dialog, on the tab **Module properties**.

We will explore the capabilities of the dialog box :



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.

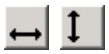
Also the properties of the control below in the dialog will become adjustable.

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.

 With these save/open buttons you can store the complete dialog box under a filename.

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

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

 In order to create a new dialog, one first needs to create a bitmap file (with extension .bmp). Then the bitmap can be opened with this button. If you then add controls to the dialog and save the design with the button  , then the dialog is created.

 Use these buttons to make 2 or more controls coincident horizontally or vertically. This is very practical to align a flex/fix button with it's edit control.

 This button will copy all of the currently selected controls.

Automatic Text Translation

Command - **PrB_MacroLanguages**

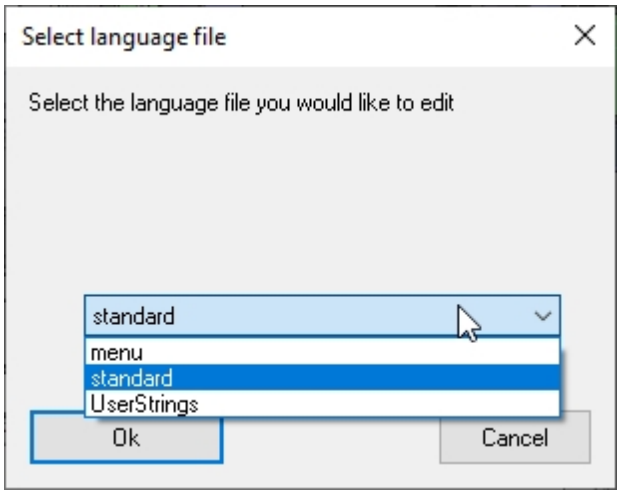
Parabuild contains a lot of dwg files with macros that contain text strings.

These strings are always written in English, so that they can be automatically translated to other languages with the help of the tables inside this command.

This command allows you to manage the text translations of :

- Texts used in macros and shown in the Review macro dialog box
- Macro names shown in the macro selection dialog box. Parabuild attempts to translates the dwg filename with the help of the tables in this command.
- Texts used in the ribbon and toolbars of Parabuild

When first starting the command, you are prompted for the table that you want to edit :



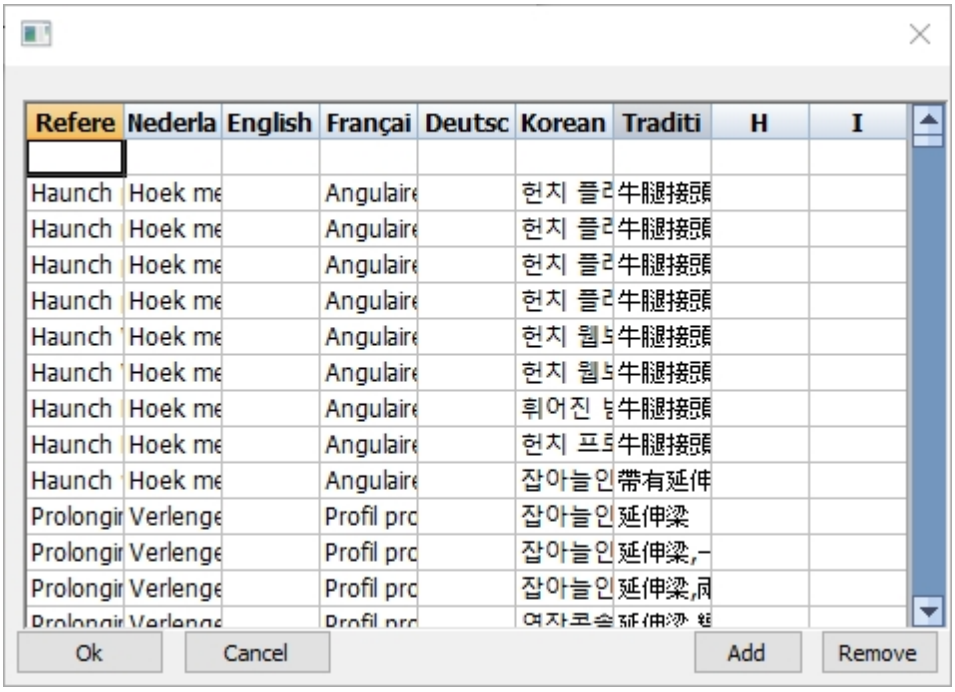
The purpose of each table is as follows :

Menu - Do not modify this table. It can be overwritten after performing a Parabuild update! This table is specifically used for translating all the texts used in the ribbon and toolbars. The purpose is to have just 1 CUI file for Parabuild in English. The CUI files in other languages are generated automatically based on the English CUI file, with the help of this table.

Standard - Do not modify this table. It can be overwritten after performing a Parabuild update. This table is used for translation of macro dialog boxes and macro names.

UserStrings - This table is reserved for end users. It will not be overwritten by a Parabuild update.

After choosing the desired table you will see the table :



In the very first column you fill in the reference name. This is the name of a macro or a variable in the reference language (English).

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

Manually copy a Macro

Command - **PrB_ApplyMacro**

At the time of writing, this command was not yet supported in BricsCAD.

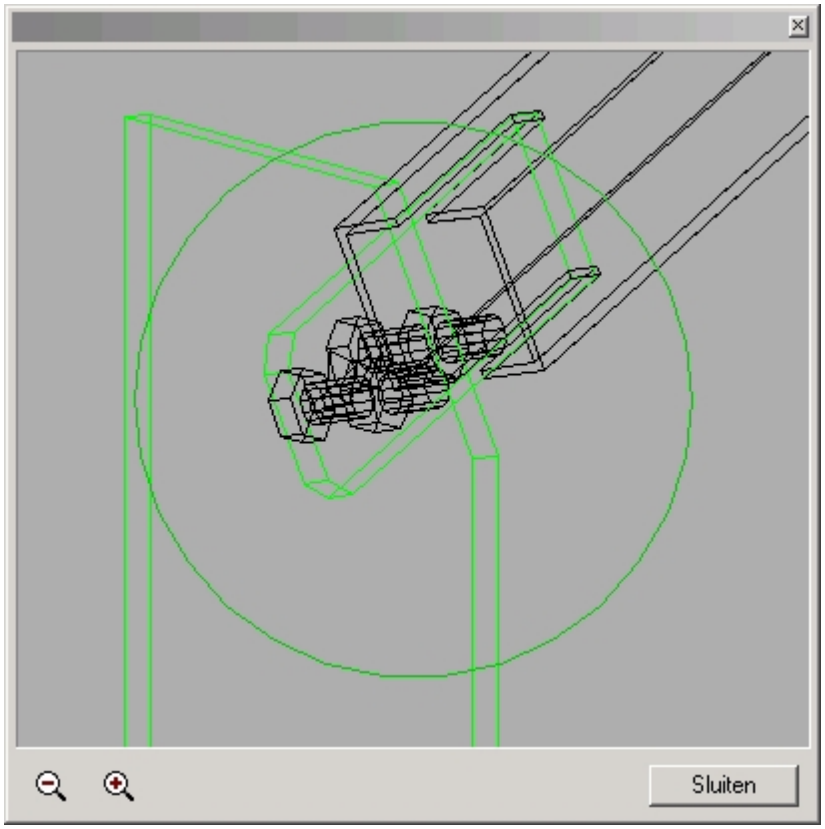
This command does approximately the same such as Smartcopy, only you have more control concerning the way the macro is applied.

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 in those situations you can remedy this restriction with this command.

You are not being asked to select destination base profiles, but 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 and out in this window.

In this window 3D-orbit is active: you can rotate 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 next 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.

Manually copy a Macro with more control

Command - **PrB_ApplyMacroManually**

At the time of writing, this command was not yet supported in BricsCAD.

This command does approximately the same as Copy a macro automatically, but it serves as a backup in case the latter fails.

The latter may abort the macro copy when the sub-geometries that you selected differs too much.

In that case you can use this command to redo the manual copy operation.

This command differs from the other macro copy command in the following ways :

- It will prompt for you to select more sub-geometries because it does no assumptions
- It has no geometric rules checking mechanism that aborts on rule combinations that are impossible to calculate.
  This is not a problem as long as you test your macros thoroughly and make sure there are no degrees of freedom!

Create Bolts Pattern

Command - **PrB_AddPattern**

This command will add a new bolt pattern to the currently active module.

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 create 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 Review macro dialog box.

There are 3 types of 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 Review 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.

Each pattern type as a plane that determines somewhat the location of the bolt heads. You should set the location of this plane with a geometric rule.

Create Coordinate System

Command - **PrB_CreateCoordsys**

If you are looking for an object that calculates 'intermediate' geometries, then you should definitely use the Plane helper geometry. The plane object is far better suited for that purpose.

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 type of object is almost never used, except for baseplate-like connections.

The baseplate connections use this coordinate system to stick the macro to the drawings' World coordinates.

As you have already seen, each 3D drawing has such a coordinate system that is fixed on the World of the drawing and that you cannot remove.

The baseplate connections have geometric rules to this world coordinate system.

Whenever the baseplate macro is copied or inserted, the geometric rules are fixed to the World coordinate system of the 3D drawing.

This ensures a fix to the WCS coordinates across all drawings.

Create Plane Help Geometry

Command - **PrB_CreatePlane**

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

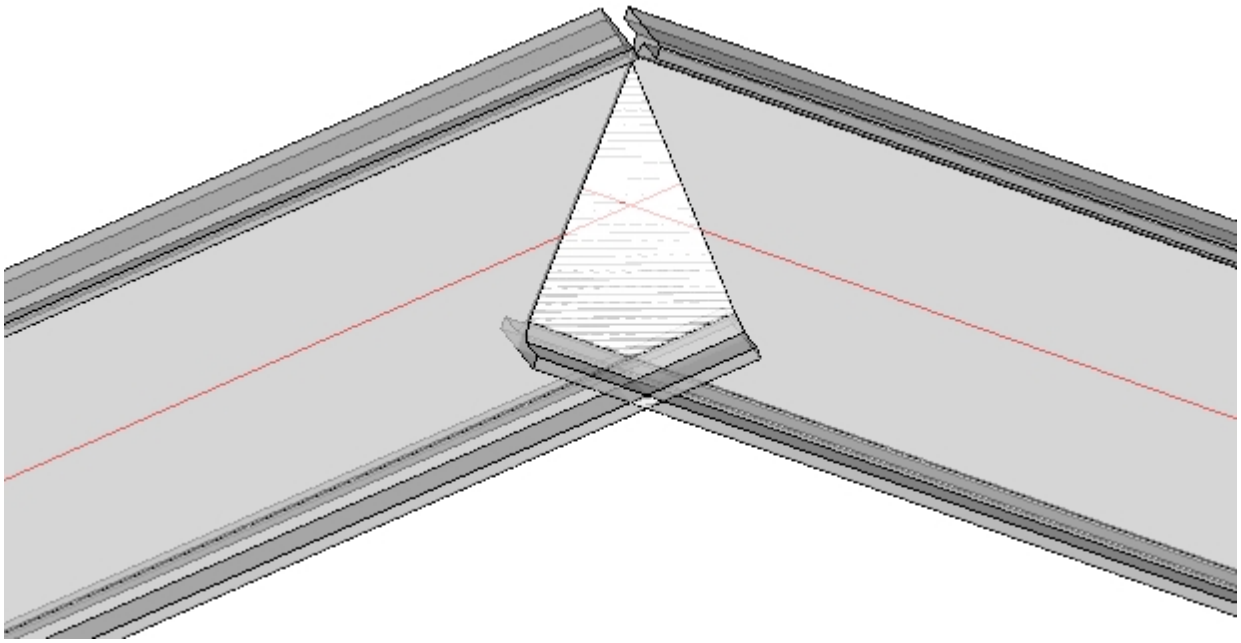
A plane object has no practical use for the output of the drawing itself (this object will never appear in a bill of material or workshop drawing).

This object contains a planar surface, a line and a point. You can use all of them to add geometrical rules.

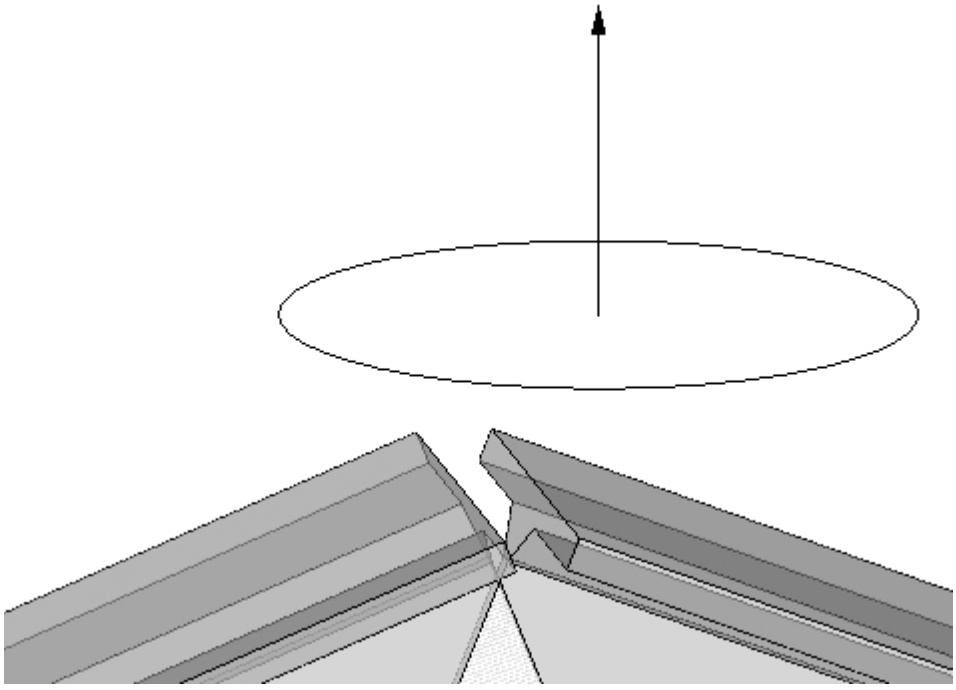
The plane object is useful in modules with parts that are hard to achieve with geometric rules alone.
The plane can be used to calculate intermediate geometries, that can then be used to complete the rules for the wanted part.

An example use for a plane : calculating the intersection of 2 profiles

In this example we want to calculate the intersection position of 2 profiles, but not on the axis but on the top of steel.

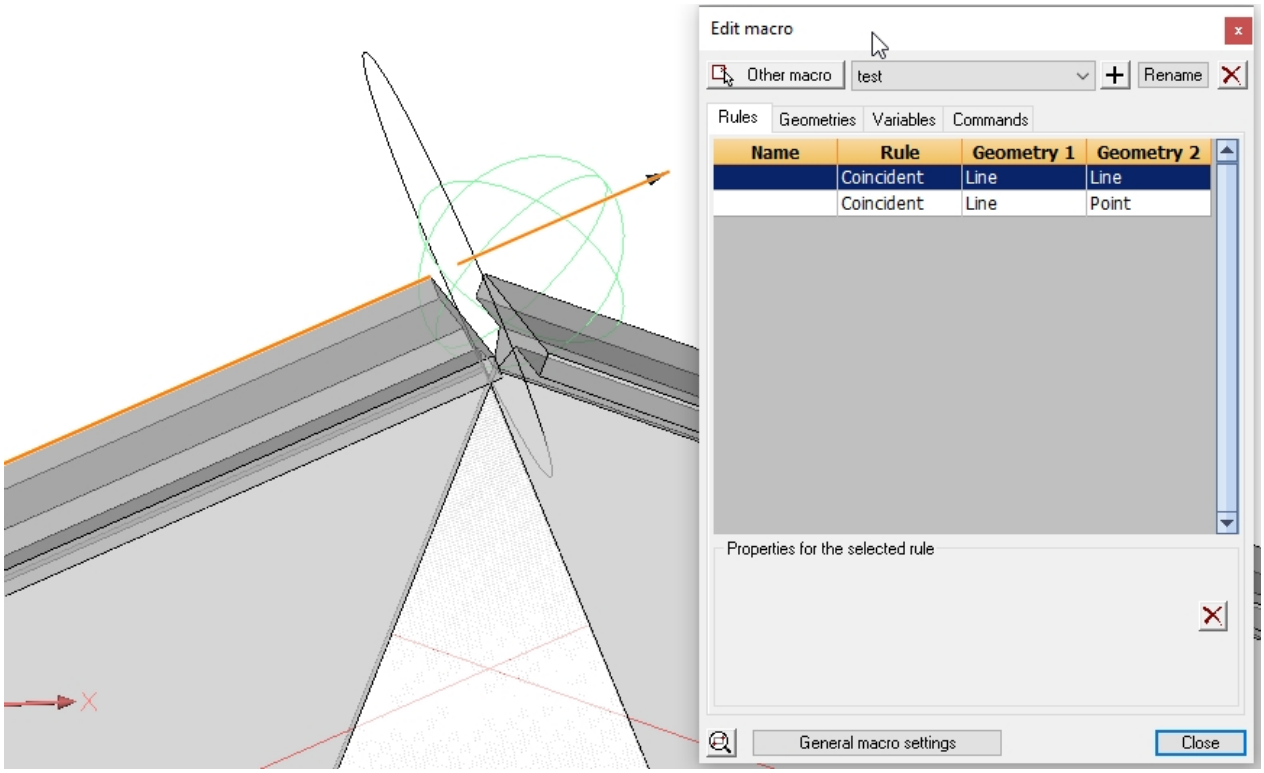


The plane is a perfect tool to quickly calculate a position (or surface).

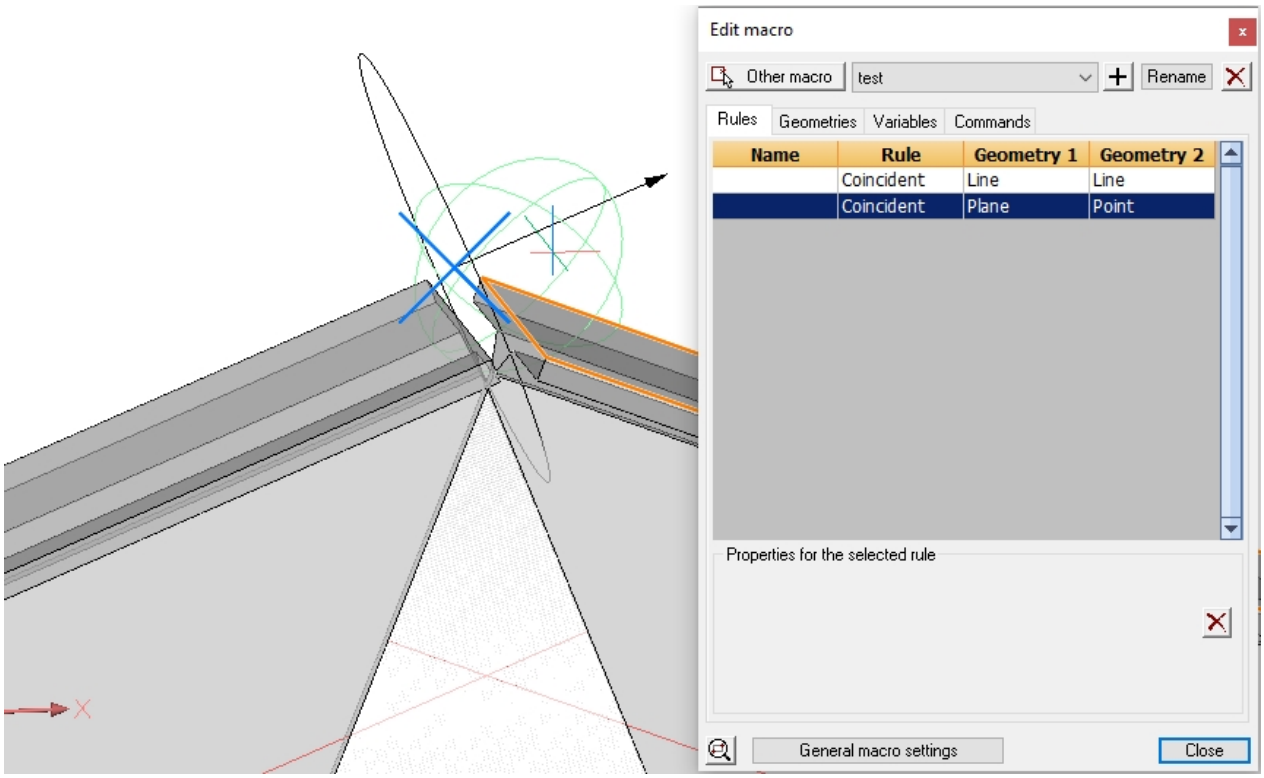


The plane that needs to be constrained to the intersection of the profiles.

We need just 2 constraints to fix the center of the plane to the intersection at the top of the profiles :



A coincident rule between an edge of the first profiles and the line of the plane



A coincident rule between the top plane of the second profile and the center of the plane

Choosing the top plane is a better choice than choosing the edge of the second profile.
If the 2 profiles are not perfectly aligned, the position of the plane can still be calculated by Parabuild.

If the edge of the second profile was chosen and the profiles are not perfectly aligned, the macro will fail to calculate the plane's position.

This plane object can now be used as a basis for other parts, for example 2 endplates between the profiles.

Do note that the plane is not fully defined. With these rules the direction of the plane was not defined.

But the macro is not 'complaining' about this either. There is no "1 degrees of freedom" warning when calculating this macro.

Plane objects are special in this regard :the macro allows for some underconstrained degrees, especially the direction of the plane.

There are many safeguards in the macro calculation routines that will keep the plane's orientation always consistent.

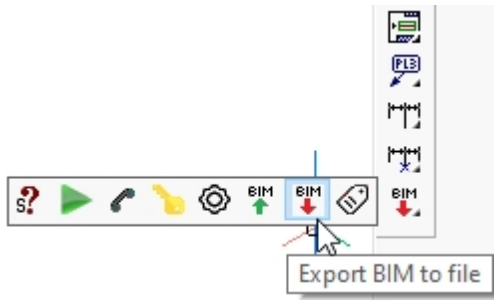
Release Notes

Version 5.1

Export to KISS Format

The **KISS** format (Keep It Simple Steel) is a format for exchanging construction data with ERP/MIS software.

The new file format can be written with this tool :



Here's a short explanation for each option :

Also export field bolts : Shop bolts are always exported

Export nuts and washers : Nuts and washers get separate 'detail parts'

Also export labor info : Will calculate and export labor data for all holes, cuts and burns

Allow accumulation of labor : Should normally be disabled: according to KISS specification the counts should be per piece. But Tekla Kiss files are accumulated so that is why this is available as an option.

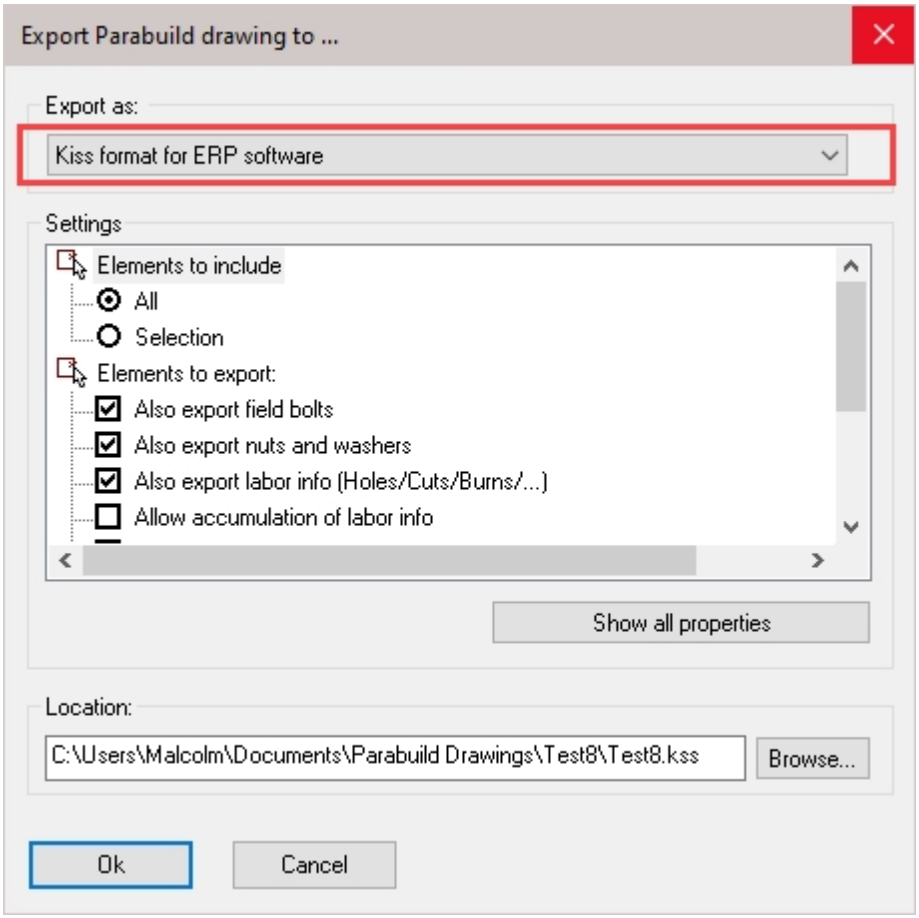
Sequence template : Optional, to make the sequence field a combination of phase&sequence.

Currently supports %Sequence% and %Phase% ex: L"%Sequence% - %Phase%"

Section name column keywords : Columns with these keywords will be used for the section shape's name. These keywords are accessible in the profile section tables.

Use assembly number for sheet name when the assembly does not have a shop drawing : When no sheet was found with this assembly on it, the Assembly number could serve as a good place holder (default on)

Default drawing number when assembly has no shop drawing : This is used when the option '*Use assembly number for sheet name*' is not active



Shop Bolts

If you enable this bolt property, then the bolt will be drawn on the assembly drawing.

The bolt will act as if it is welded to the assembly.

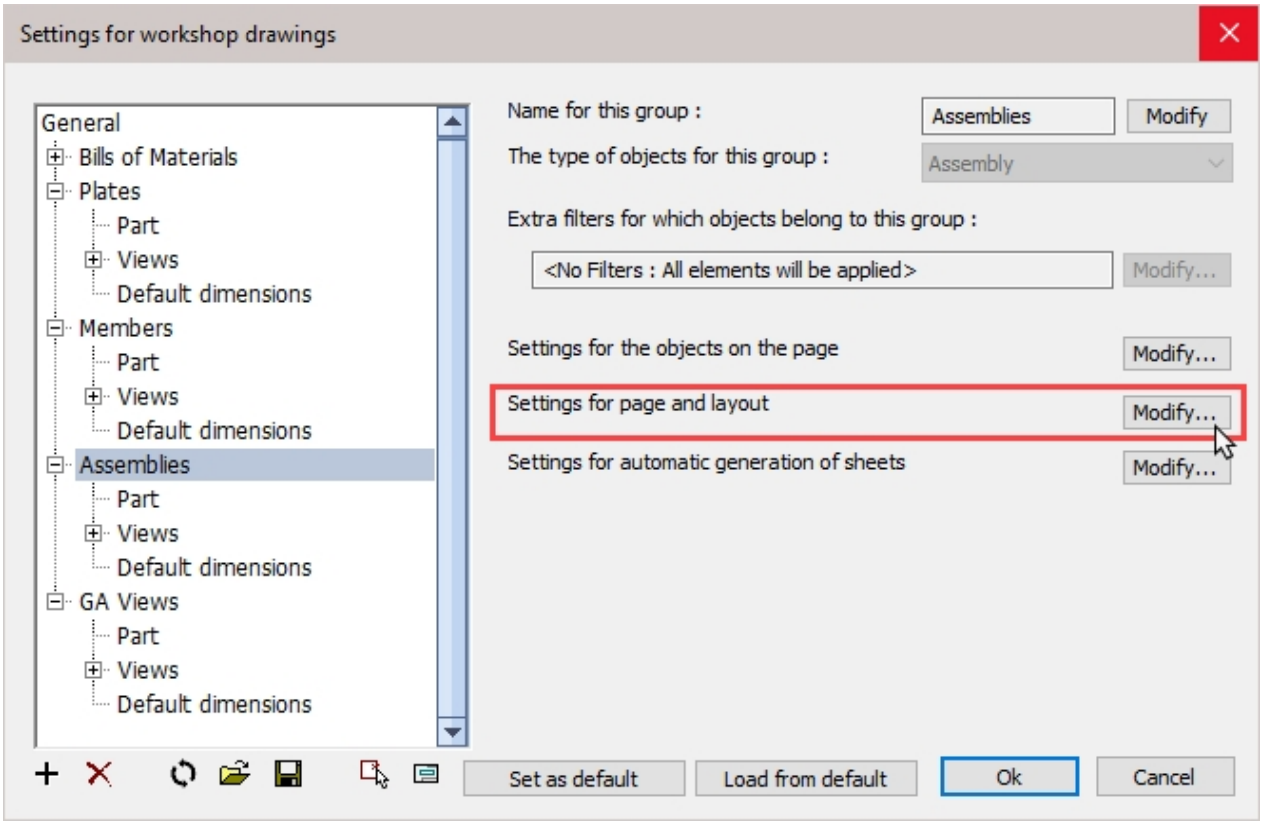
This property will also influence the number of the assembly.

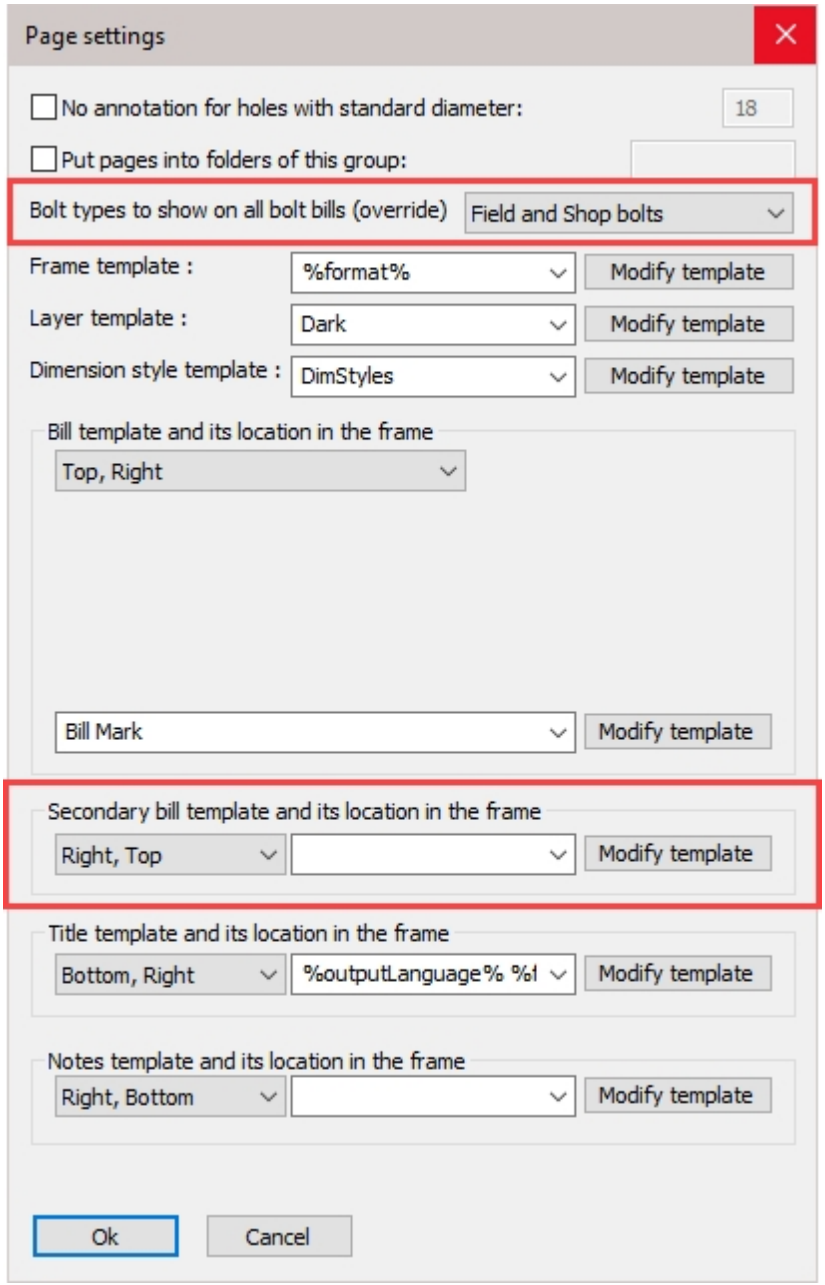
Misc	
Phase	1
Revision	0
Cost Category Code	
Shop Bolted	☐
Structural type	
Orientation	
Attach to assembly	Not set
Attached to assembly nr	M5
Clashing	☐
Zoom	
Review macro	
Unique identifier	F6738E16-7F43-4A21-9CFB-077F5

Add Bolts bill on assembly drawings

A new secondary bill can be added to the shop drawings. By default, this is a bolts bill on the assembly drawings.

You can change the bill template file, and also the types of bolts to be inserted here :



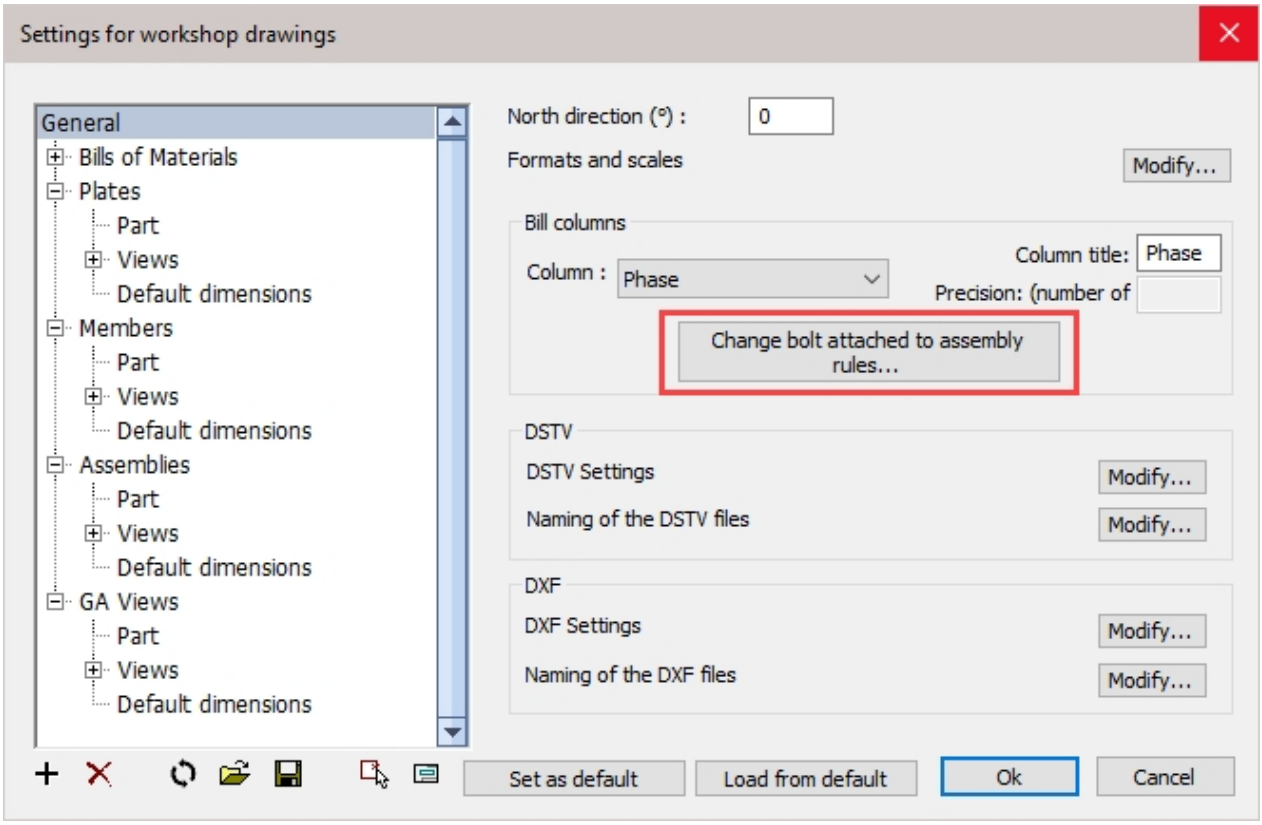


Influencing to which assembly a bolt is attached

A bolt is always attached to 1 reference assembly, even if it connects 2 or more assemblies together.

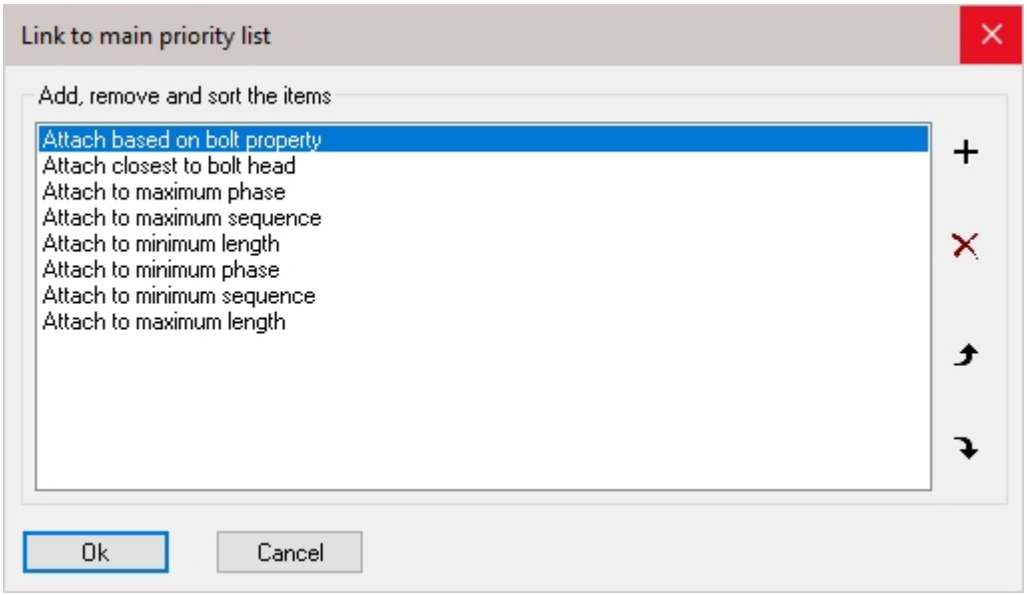
We shall call this the reference assembly of the bolt.

This reference assembly is important to have because it will determine on which assembly drawing the bolt will appear.
(a bolt will never appear on 2 assembly drawings to avoid counting the bolt more than once)



With this priority list you can influence the automatic assignment of reference assembly of all bolts - the list is ordered according to priority.

you can add, delete, and sort all items in the list.



In exceptional cases where the automatic choice taken by Parabuild is not as desired, it is still possible to manually set the reference assembly in the properties of the bolt:

Attach to assembly - In this property we can choose the assembly from the drop-down menu: **Closest to bolt head** or **Closest to Bolt end**

This will effectively override the main priority list

Attached to assembly nr - This will indicate to which assembly the bolts are attached

Misc	
Phase	1
Revision	0
Cost Category Code	
Shop Bolted	☐
Structural type	
Orientation	
Attach to assembly	Not set
Attached to assembly nr	M5
Clashing	☐
Zoom	
Review macro	
Unique identifier	F6738E16-7F43-4A21-9CFB-077F5

Erection Sequence Property

All plates, profiles and structures now have a new sequence property

Additional Data	
Remark	
Material	
Finishing	
Paint	
Manufacturer	
Surface Treatments	
Struct Group	
Cost Category Code	
Assembly Cost Category Code	
Sequence	1
Lot name	
Structural type	
Assembly Structural type	
Unique identifier	7C20EA8E-B896-47B4-B656-2A145

The sequence of a part has the following capabilities:

- The reference assembly of a bolt can be determined by the sequence number of the assemblies
- The sequence is exported to KISS file
- It should be available in bills when you make the columns : %PbColSequence% (doesn't work yet
- A planning tool for erection is a future addition

New features for Dxf files

The automatic location of text on the plate is now much better : It is drawn close to the contours, and making sure that the text does not collide with contours.
Here we outline the new options that were added to the Dxf dialog box :

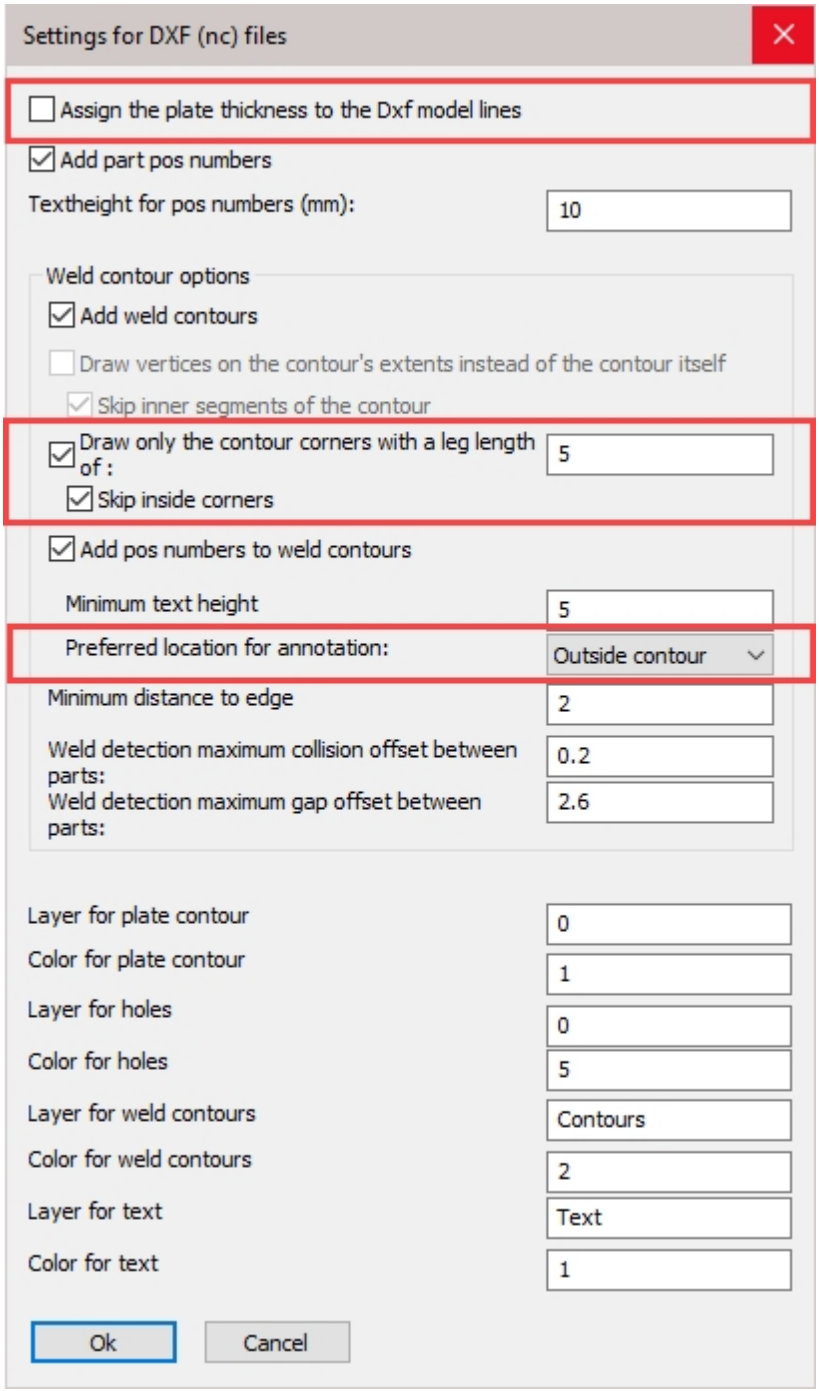
Assign the plate thickness to the Dxf model lines : When active, the thickness of the plate will be assigned to the model lines in the dxf files. This creates a semi 3D model and the machine can know the required thickness of the plate this way. Some machines can't read the dxf file when this option is active.

Draw only the contour corners with a leg length of : When this is active, only the corners of the contour are scribed to save the machine some time with scribing work.

Preferred location for annotation : Normal behavior is to place this annotation outside of the contour. You can choose outside of the contour, but if the text is too large to fit in the contour then

it will be placed outside of the contour anyway.

Skip inside corners : Inside corners will occur when an I-shaped profile is welded with the I shape to the profile. This case would have 4 inside corners. You can skip these corners with this option.



New features for Dstv file generation

- Countersunk and blind holes are now implemented in Dstv (509)
- The automatic location of text on the parts is now much better : It is drawn close to the contours, and making sure that the text does not collide with contours.

Here we outline the new options that were added to the Dstv dialog box :

Default view for this annotation : You can choose on which side the part's number should be scribed

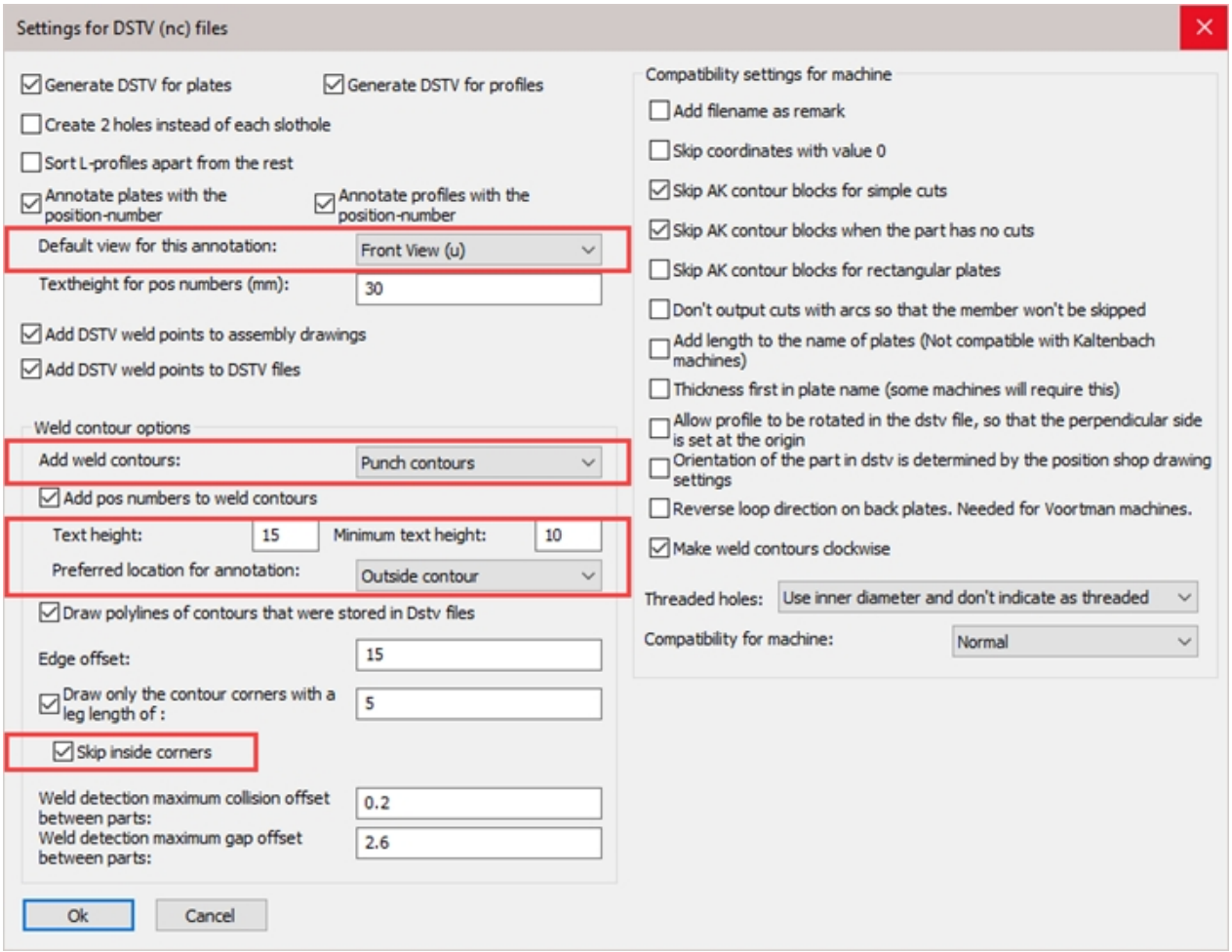
Add weld contours : You can now choose between either punch, powder, or both. If you choose both, then all of the contours are added double to the dstv file.

Weld contour text height : Sets the default height of welded part numbers text (this is a separate value from the main part's number).

Weld contour minimum text height : If the part number text would fit somewhere (especially inside the contour) if it were of a smaller size, then a reduced height will be used. With this option you can choose the minimum text height that the function is allowed to use for this purpose.

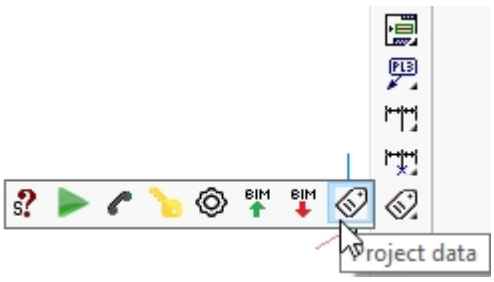
Weld contour preferred location for annotation : Normal behavior is to place this annotation outside of the contour. You can choose outside of the contour, but if the text is too large to fit in the contour then it will be placed outside of the contour anyway.

Skip inside corners : Inside corners will occur when an I-shaped profile is welded with the I shape to the profile. This case would have 4 inside corners. You can skip these corners with this option



New project data for the title block in shop drawings

The title blocks have now received 15 fixed fields.
These fields can be modified in the Project data dialog box :



In this dialog box the first 15 fields are fixed and always there, no matter what the title blocks contain.

These 15 fields are used not only for filling title blocks in sheets, but also for writing project data to KISS format files for example.

These 15 fields will work across all languages. If you would change the Parabuild language then the field values will stay intact.

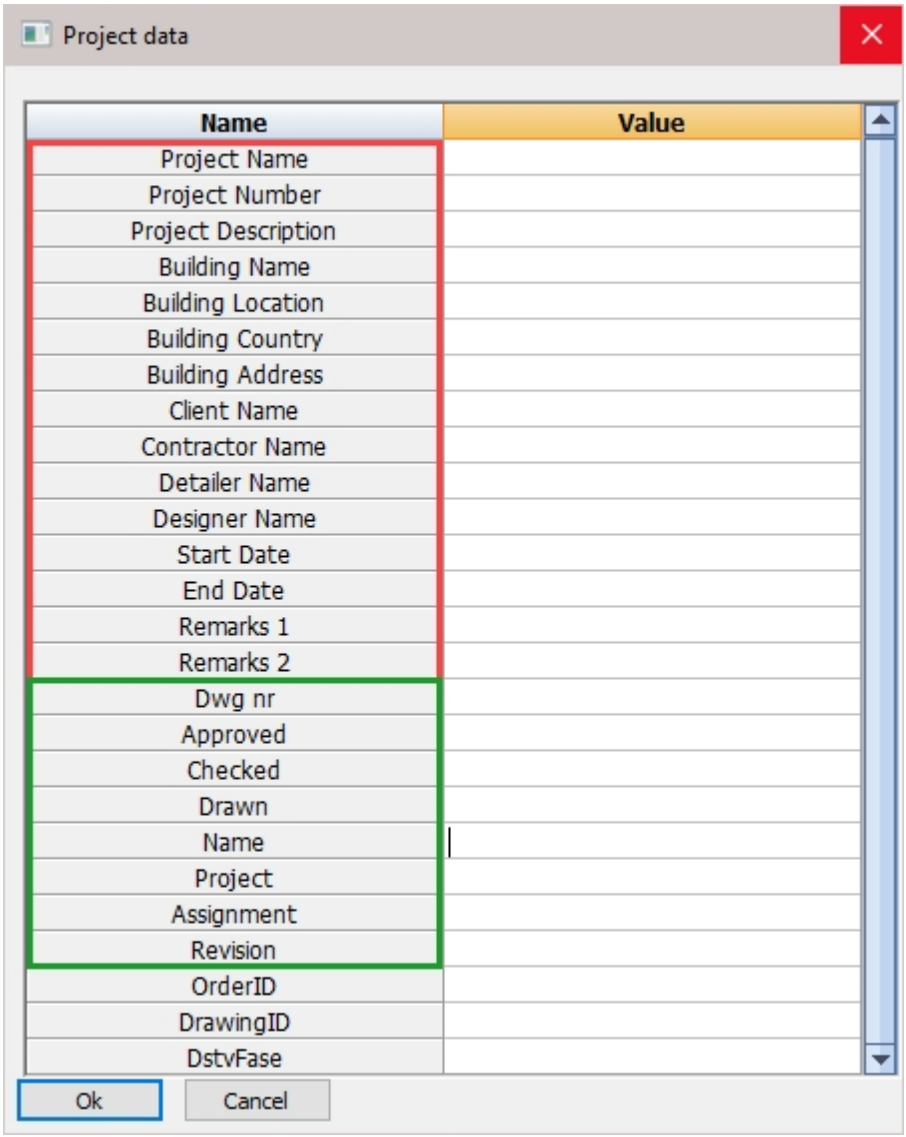
In the below image, the fixed fields are indicated in red.

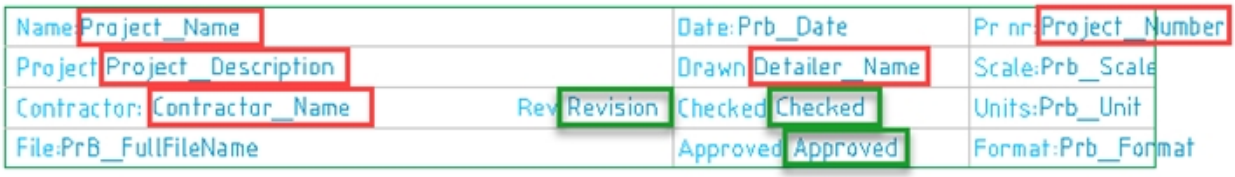
When using these fields in a title block, it is possible to use the text "Project Name" or "Project_Name". Both texts will be replaced by the actual project name.

The dynamic fields are indicated in green. These fields can be entered freely.

The dynamic fields can be added by simply typing them in a title block, and saving the dwg file.

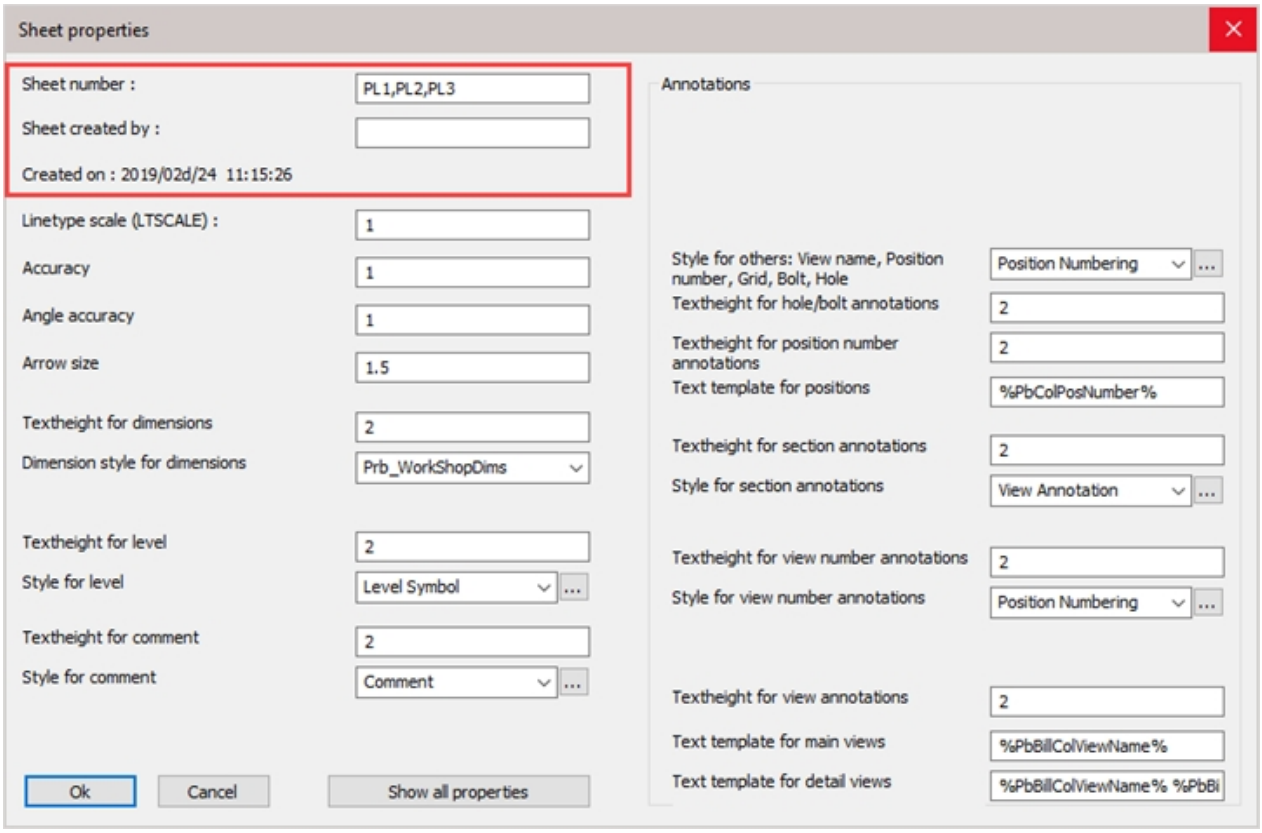
When you open the Project data dialog box, all of the title blocks that start with the current language (English*.dwg) will be loaded and all text fields are analyzed. The non-standard fields are then added to the list of fields.





New sheet properties

There are 3 new properties available for sheets



By default, the sheet number is the same as the sheet name. But this number can be changed manually.

The created on date and time is filled automatically.

All of the 3 new properties are written in KISS files.

New plate / profile properties

Cost Category code : These codes are usually determined while a quote for the project is made. This code can then later be assigned to the 3D model when the project was ordered. This way the parts can be traced from estimation to production and to completion inside ERP/MIS software. This value is exported to KISS files. (PbColCostCategoryCode and PbColAssemblyCostCategoryCode)

Assembly Cost Category code: See Cost Category code

Sequence: The erection sequence of the part. This value is exported to KISS files. (PbColSequence)

Lot name: The lot that the part belongs to. This value is exported to KISS files. (PbColLotName)

Structural type: To be used in the future. For example : Bracing/Rafter/Stringer/Handrail/Post/etc.... (PbColStructuralType)

Assembly Structural type: To be used in the future. For example : Stair/Railing/etc.... (PbColAssemblyStructuralType)

Beam camber (profiles only): The camber (distance) to be used for the beam. This value influences the position number of the beam. This value is exported to KISS files. (PbColBeamCamber)

Additional Data	
Remark	
Material	
Finishing	
Paint	
Manufacturer	
Surface Treatments	
Struct Group	
Cost Category Code	
Assembly Cost Category Code	
Sequence	1
Lot name	
Structural type	
Assembly Structural type	
Unique identifier	7C20EA8E-B896-47B4-B656-2A14

New bolt properties

Structural type : to be used in the future

Cost Category code : important for ERP/MIS software (PbColCostCategoryCode)

Misc	
Phase	1
Revision	0
Cost Category Code	
Shop Bolted	☐
Structural type	
Orientation	
Attach to assembly	Not set
Attached to assembly nr	M5
Clashing	☐
Zoom	
Review macro	
Unique identifier	F564674D-F316-4166-A1E1-B625

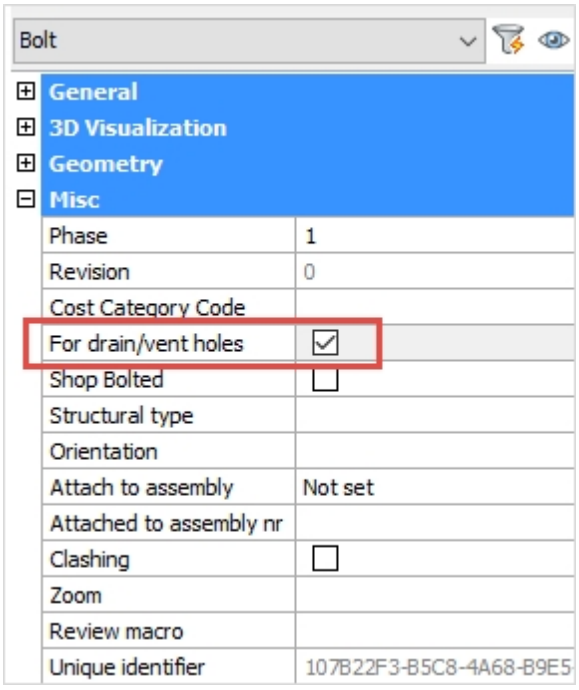
Hole annotation override

A new property was added for holes, which allows you to set a custom annotation text for a hole.
The property is only modifiable from the properties of the bolt.
This text will appear on **position** shop drawings

Properties for hole 1	
Hole type	Normal
Tolerance	1 mm
Diameter	22 mm
Annotation override	
Comment	
Threaded	☐
Remove hole	☐

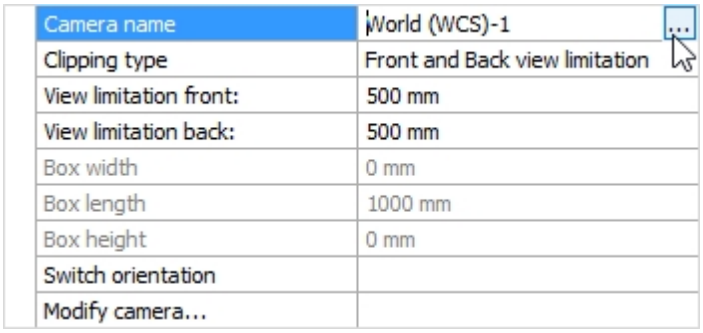
Skip holes for drain / vent holes in shop drawings

When active, all 'drain / vent' holes will not receive dimensions on the **position** shop drawings.



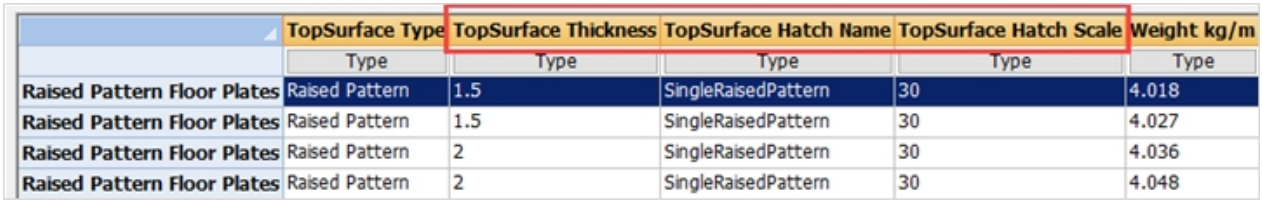
Connect view to a camera

It is now possible to connect a view to a different camera.
This is a useful tool to fix broken views



Raised pattern floor plates

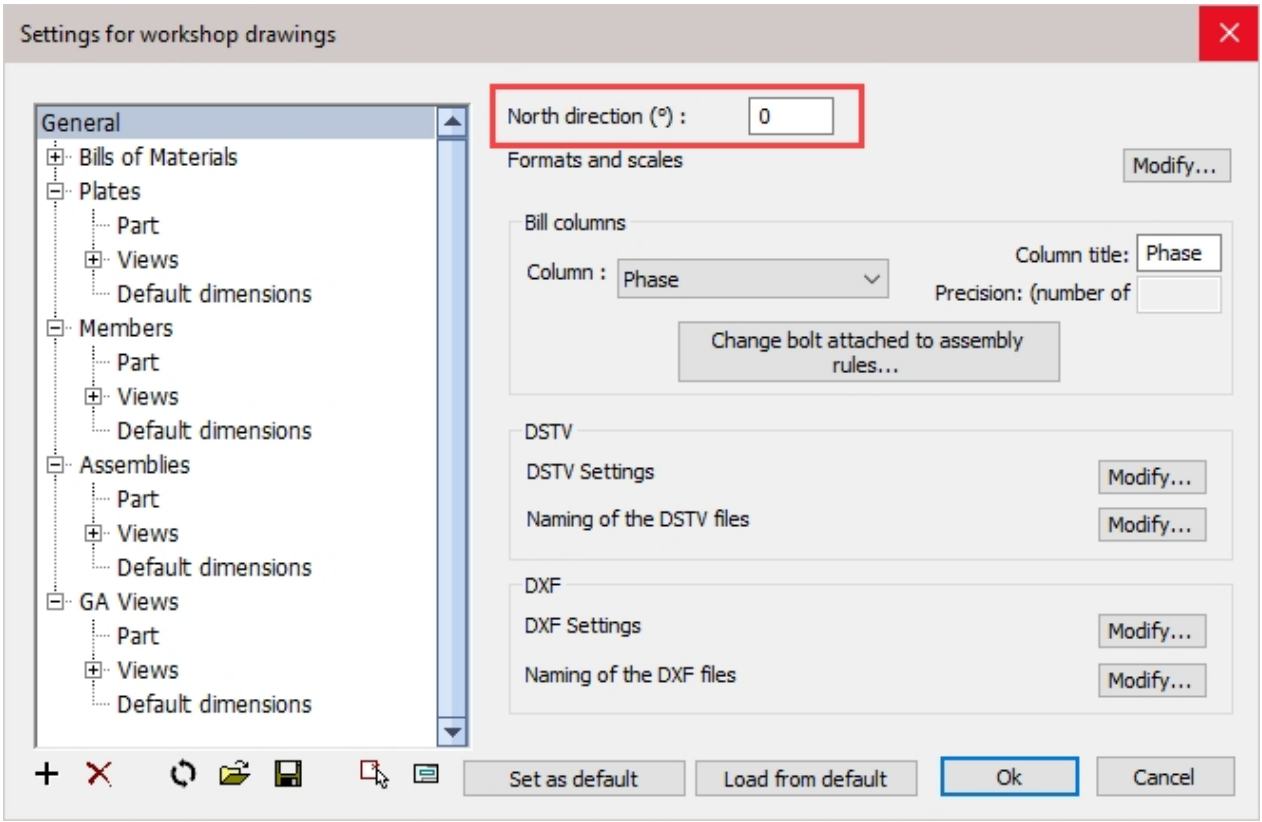
The hatch pattern in the profile tables work now.
New profiles drawn with a table that has these special hatch columns will now automatically get a surface treatment on 1 face.
(this features is only supported for flat bar shapes)



	TopSurface Type	TopSurface Thickness	TopSurface Hatch Name	TopSurface Hatch Scale	Weight kg/m
	Type	Type	Type	Type	Type
Raised Pattern Floor Plates	Raised Pattern	1.5	SingleRaisedPattern	30	4.018
Raised Pattern Floor Plates	Raised Pattern	1.5	SingleRaisedPattern	30	4.027
Raised Pattern Floor Plates	Raised Pattern	2	SingleRaisedPattern	30	4.036
Raised Pattern Floor Plates	Raised Pattern	2	SingleRaisedPattern	30	4.048

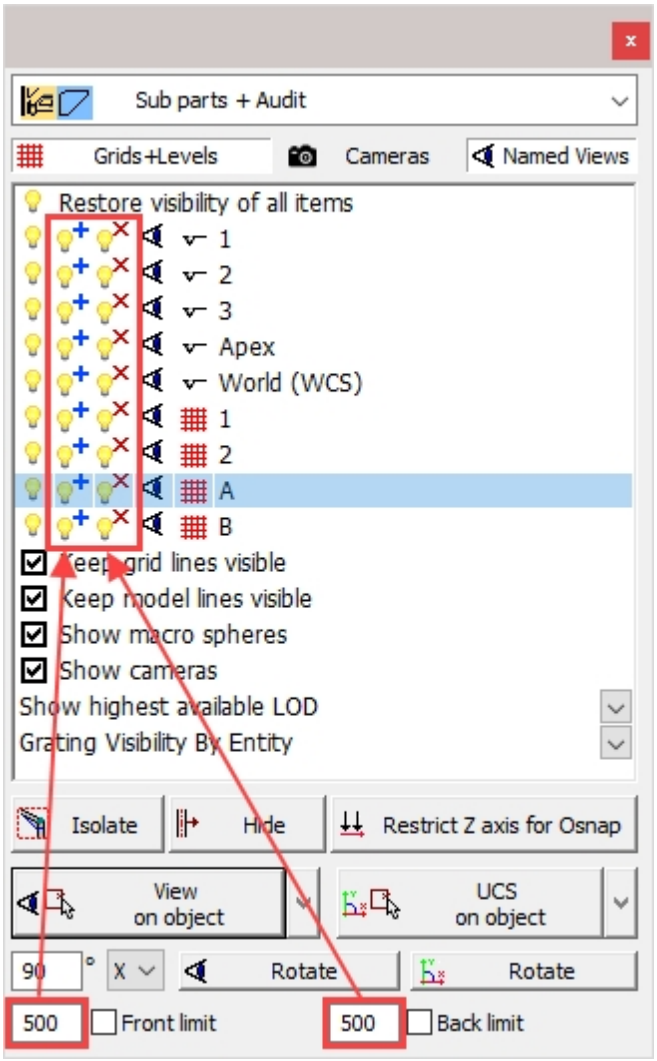
Cardinal directions for 2D drawings

The shop drawings can optionally use the cardinal directions for views (this is nothing new).
Parabuild uses the current drawing's NORTHDIRECTION variable to know this direction.
An option in the shop drawings dialog box was added to more clearly show this relation:



Additional visibility options

The isolate lamp buttons will now use the front/back values to determine how many parts should be isolated


Sub parts + Audit
Grids+Levels
Cameras
Named Views
Restore visibility of all items
1
2
3
Apex
World (WCS)
1
2
A
B
Keep grid lines visible
Keep model lines visible
Show macro spheres
Show cameras
Show highest available LOD
Grating Visibility By Entity
Isolate
Hide
Restrict Z axis for Osnap
View on object
UCS on object
90
X
Rotate
Rotate
500
Front limit
500
Back limit