

ATHENATUTORIALS

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A Introduction

Here you will find step-by-step instructions on various topics.

Requirements

Good basic knowledge of AutoCAD and ATHENA is required.

In the tutorials only the processes are described which are specified in the respective section heading or introduction. It is assumed that you are familiar in working with the AutoCAD/ATHENA user interface and that you in particular have knowledge of the command options, command modifiers (e.g. OSnap) and dialog boxes.

Topics:

- **Managing objects**
- **Acquiring master data**

1 Managing objects

In this Chapter you will learn how to save ATHENA objects and how you can use saved objects.

Topics:

- **Introduction and preliminary Remarks**
- **Saving objects**
- **Using saved objects**
- **Working with libraries**
- **Design environment and catalog**

1.1 Introduction and preliminary Remarks

For most ATHENA objects a Manager section is available which enables you to save drawing objects with specific properties.

On one hand you can use this range of functions to maintain your internal company standard. On the other hand you can also save project-specific standards, thus ensuring a consistent presentation of your project drawings.

This is a particular advantage when projects are being processed in a planning team. The individual planner or designer no longer needs to consider with which properties the specific drawing object has to be produced, but can instead access previously saved objects.

Saved ATHENA objects are only available in the current drawing.

With the aid of libraries however saved ATHENA objects can be used in other drawings. For faster access you can save several libraries as a catalog in a design environment and load them as required or automatically.

1.2 Saving objects

In the Manager section you can save ATHENA objects with specific properties.

This tutorial shows you how you define a membrane with specific properties and then save it. The procedure can also be used on other ATHENA drawing objects.

Task definition

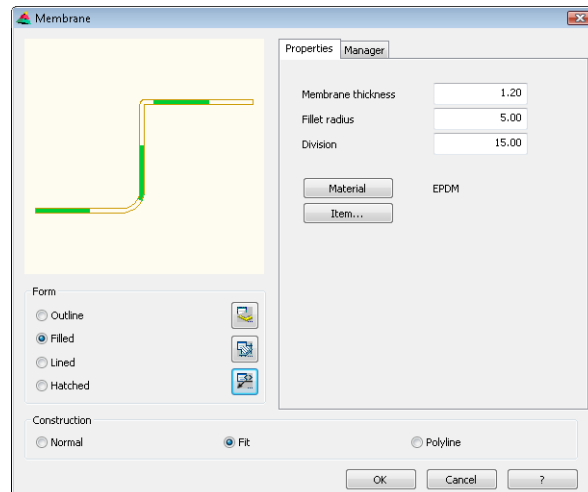
Define a membrane with the following properties:

- Membrane in EPDM (with appropriate material layer)
- Membrane thickness: 1.2
- Fillet radius: 5
- Division: 15
- Labeling: NOVOPROOF FA EPDM 1.2 / N mm

Save the membrane and draw it.

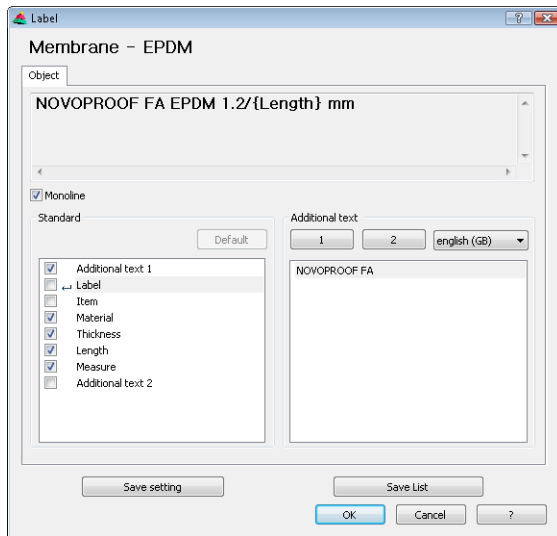
Steps

1. Start the Membrane command and carry out the appropriate settings.

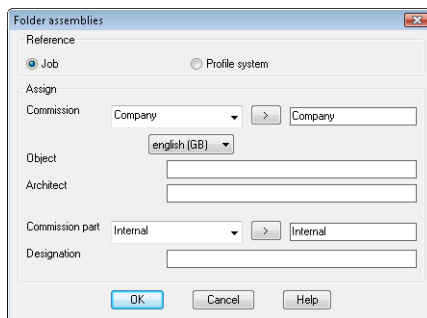


2. Click the Label button and define the label properties for the membrane. Terminate the Label dialog box with OK.

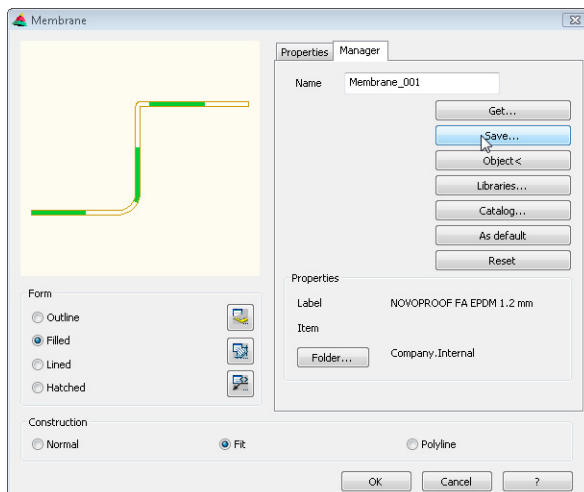
Note: Do not forget to save the new label texts which you have produced (click the button Save list).



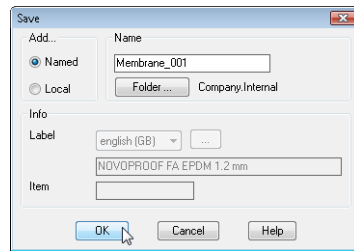
3. Change to the Manager tab and enter a unique name into the input field.
4. Click the Folder button and specify a folder structure. Close the dialog box Folder assemblies with OK.



5. Click the Save button.



6. Confirm saving in the following dialog box by clicking OK.



7. Close the Membrane dialog box with OK and draw the membrane with the defined settings.

Note: *The saved object is only available in the drawing. In order to be able to use it in other drawings you must copy it via a library into another drawing. The procedure is explained in a separate tutorial.*

In the following sections you will learn how you can access saved objects in your drawings.

1.3 Using saved objects

ATHENA objects saved in the Manager section can be brought into the respective dialog box and used in the drawing.

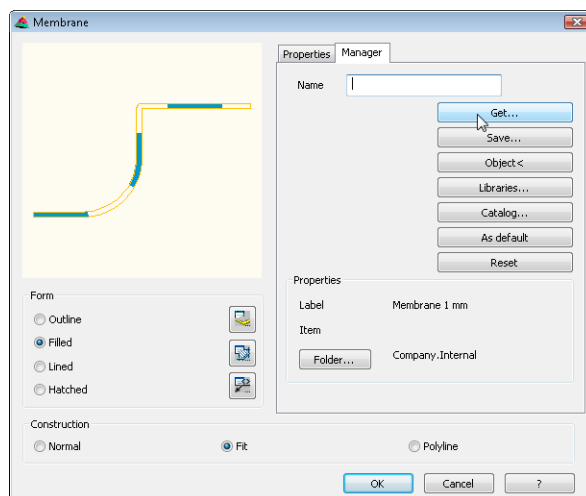
This tutorial shows you how you can get a saved membrane and use it in the drawing. The procedure can be used on other ATHENA drawing objects.

Task definition

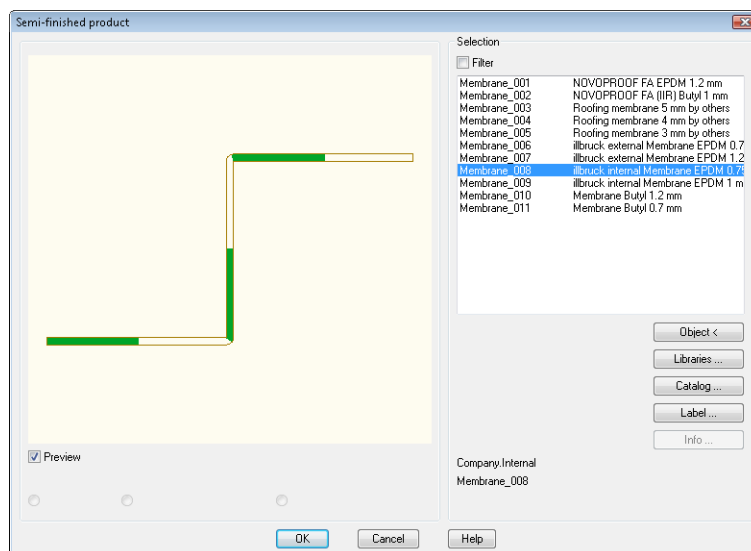
Bring saved membrane with certain properties into the dialog box and draw it.

Steps

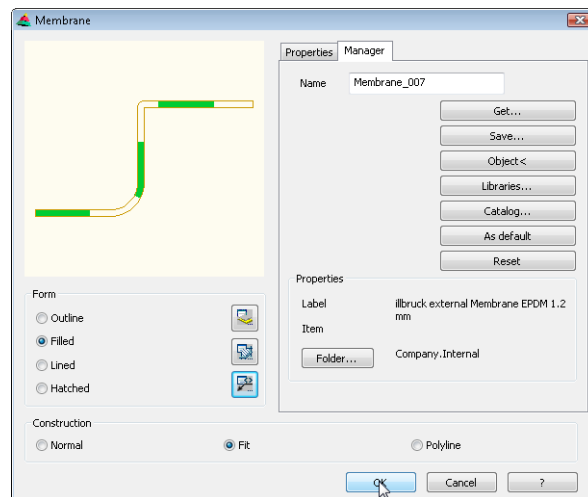
1. Start the Membrane command, change to the Manager section and click the button Get.



2. Choose the required membrane from the list and close the dialog box with OK.



3. The selected membrane is displayed. Also close the dialog box Membrane with OK and draw the membrane.



In the following section you will learn how you store saved drawing objects in libraries in order to be able to use them in other drawings.

1.4 Working with libraries

When you save ATHENA objects in the Manager section, they are only available in the drawing in which they were saved. With the library function you can save these objects independently and use them in other drawings.

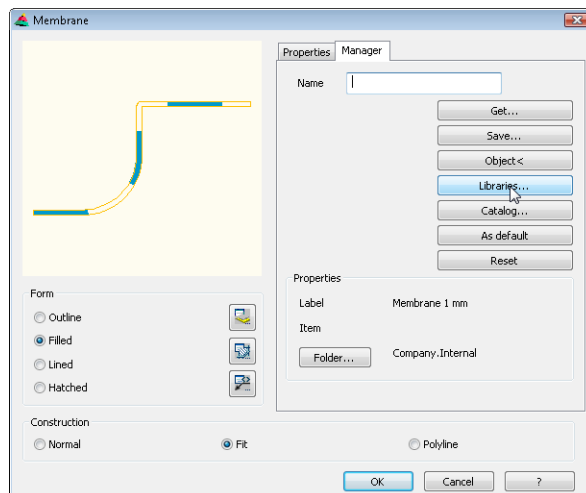
This tutorial shows you how you copy into a library various membranes which have been saved in a drawing. In the second part of this tutorial you will learn how you can open a library and copy the ATHENA objects it contains into a new drawing.

Task definition

Transfer several membranes which have been saved in the drawing into a new library and save them. Open a drawing and transfer the membranes from the library into the drawing in order to use them

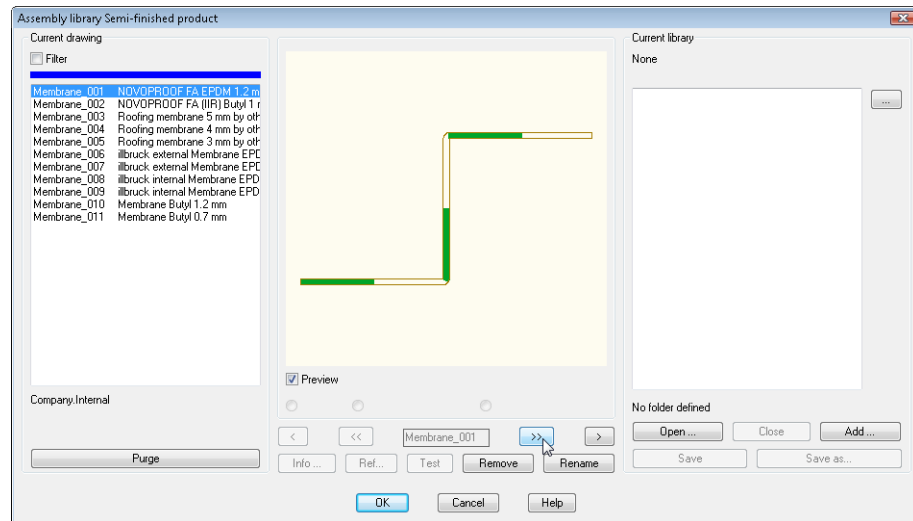
Steps

1. Start the command Membrane, change to the Manager section and click the button Libraries to gain access to the dialog box Assembly library.



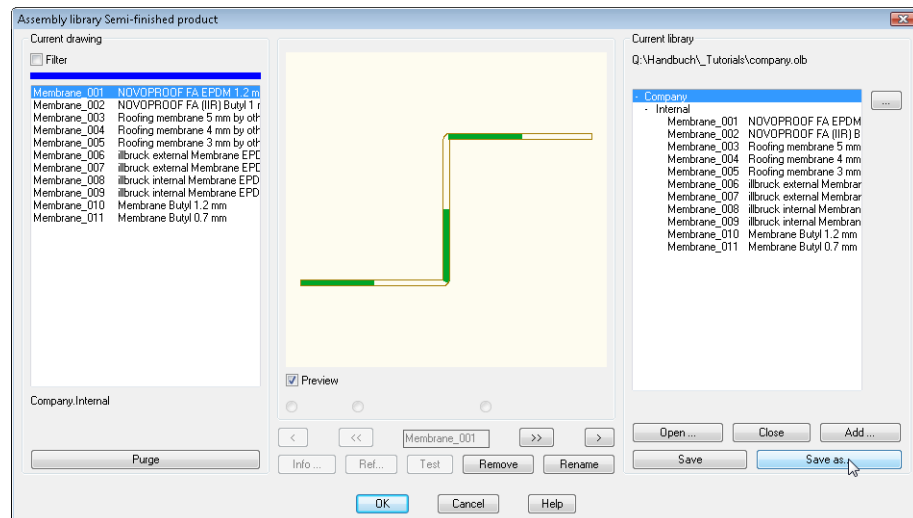
2. On the left side of the dialog box the objects in the current drawing are listed. The list on the right side is empty, because no library has been opened. Now click on the [>>] button to copy all objects in the drawing into a new library.

Note: Alternatively you can also copy individual objects with the [>] button.

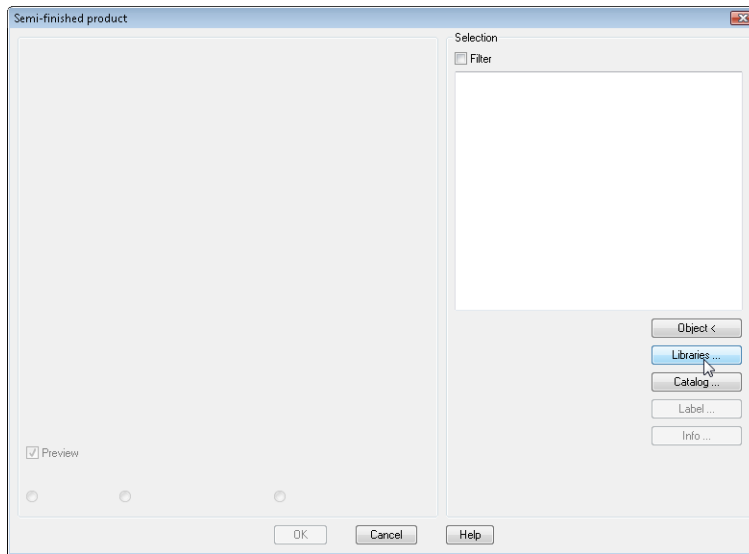


3. Click on the button Save as to save the library.

Note: A standard dialog box is used for saving, where you can specify the storage location and filename. The path and the filename of the saved library are now displayed in the dialog box section Current library.

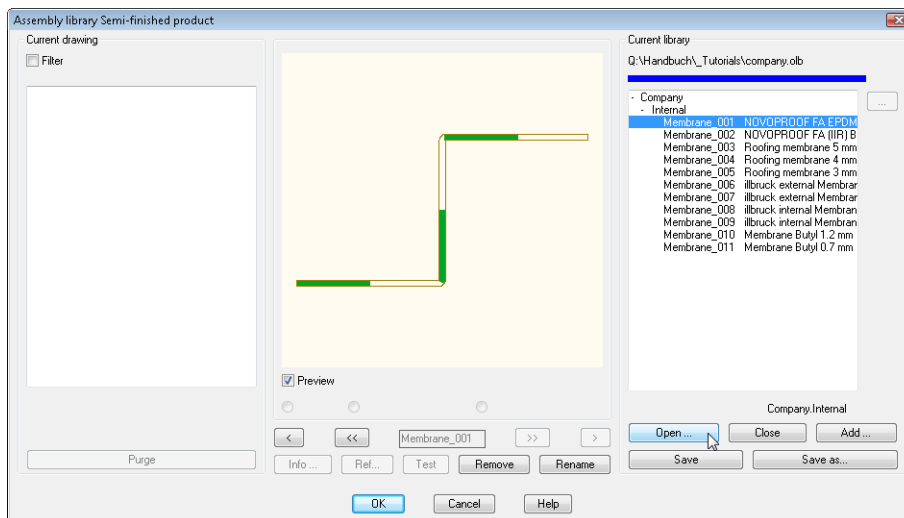


4. Now open a new (or different) drawing. Here, start the Membrane command again, change to the Manager section and click the button Get. The dialog box Semi-finished product opens.



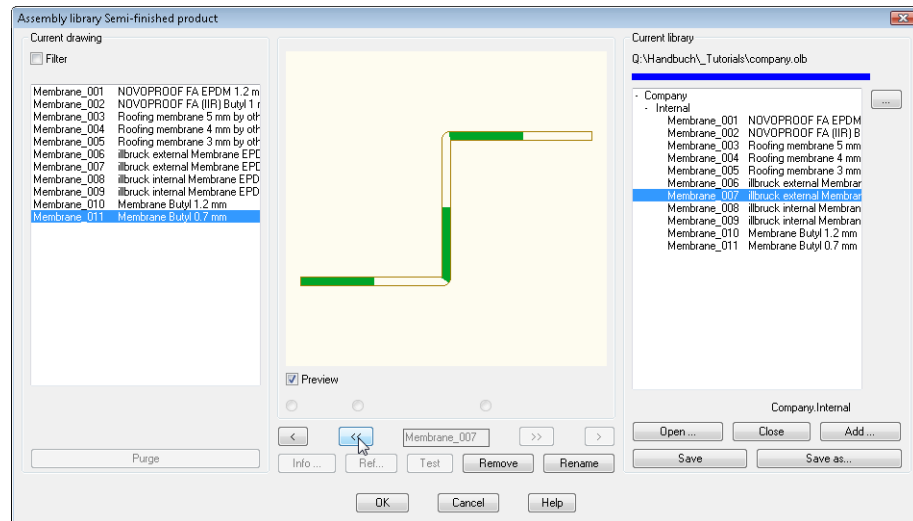
5. Now click on the Libraries button to open the dialog box Assembly library. On the right side click on the button Open to open the previously saved library.

Note: The standard dialog box for file selection is used for opening.



6. Now click on the [<<] button to copy all objects from the library into the current drawing.

Note: An object must be marked in the dialog box section Libraries, otherwise no objects can be copied.



7. Now close the dialog box Assembly library with OK. Then, in the dialog box Semi-finished product choose a membrane and again close the dialog box with OK. The selected membrane is now displayed in the dialog box and can be used in the drawing.

In the following section you will learn how you can combine several libraries into a catalog and save it as the design environment.

1.5 Design environment and catalog

With the aid of the design environment you can combine, save and load libraries (as well as other settings files and programs) appropriate to the task. Using the catalog you can access all objects within the loaded libraries without having to tediously search for individual libraries.

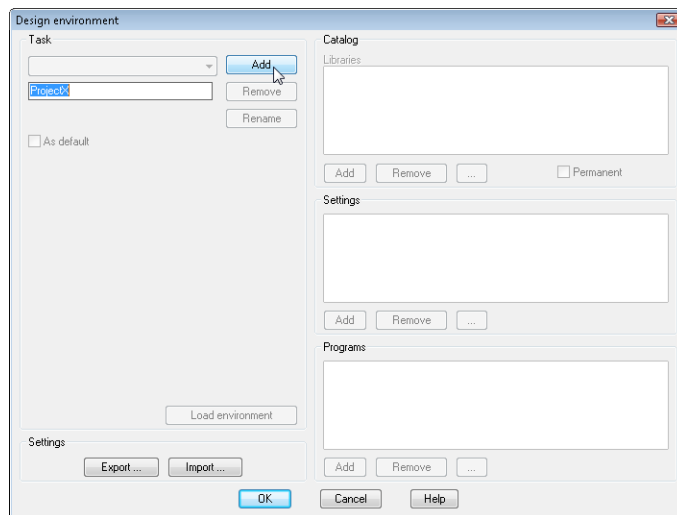
Working with design environments is then a good idea if you need to access different libraries for various projects.

Task definition

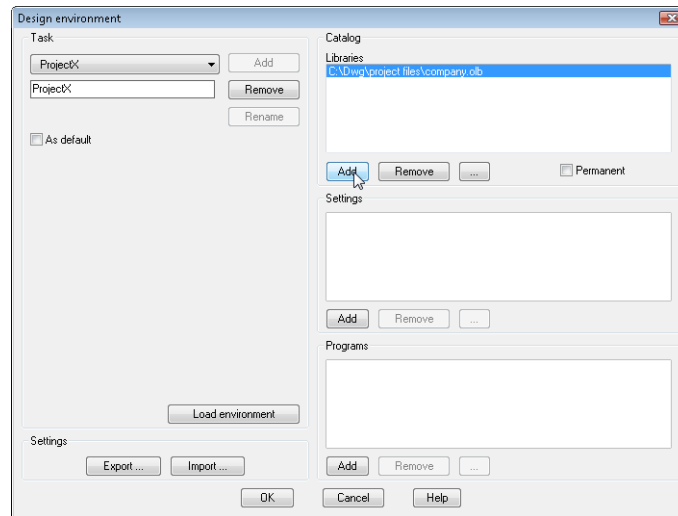
Create a task with the name ProjectX and add several libraries. Save this task as a construction environment and load it. Use the objects from the catalog in the drawing.

Steps

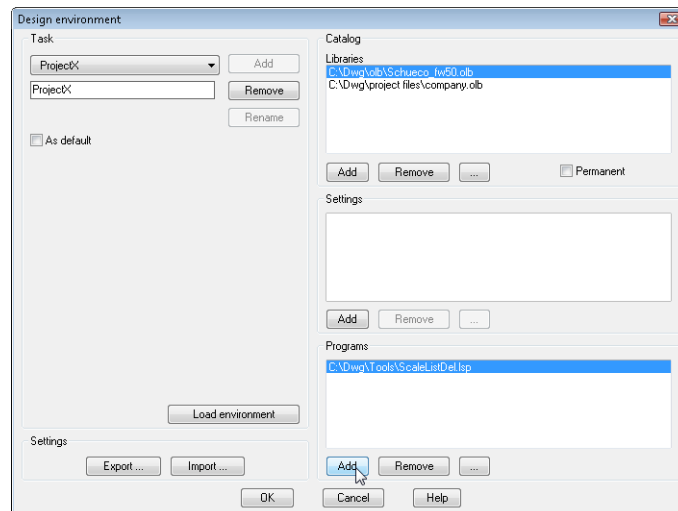
1. Start the ATHENA options and click on the button Design environment to open the dialog box with the same name.
2. Write the project name in the input field Task, confirm the input and click the button Add.



3. In the section Catalog click the button Add and choose the library which you need for your task. For this a standard dialog box is used for file selection. Repeat this step if you want to supplement further libraries.

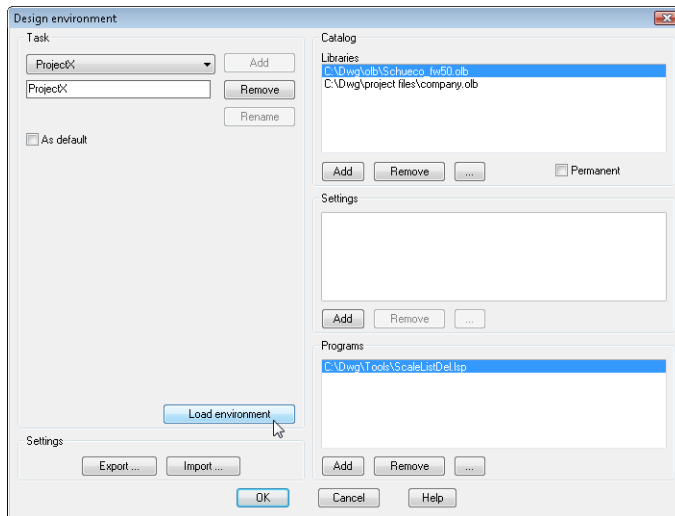


4. Optionally, you can also select settings files or programs. To do this, click on the button Add in the respective dialog box section.



5. Click on the button Produce environment to produce the task just created as a design environment. In this way the task is saved and the dialog box Design environment terminated.

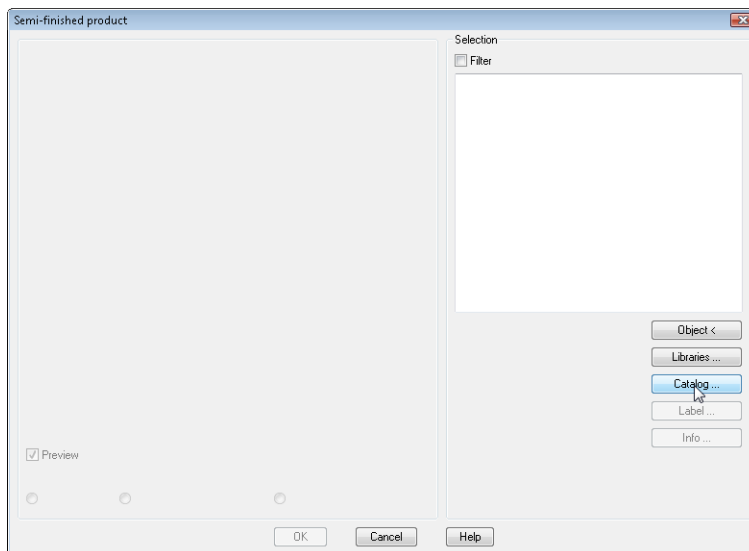
Note: When you need the environment in the current drawing, you can also terminate the dialog box with OK, thus saving the task.



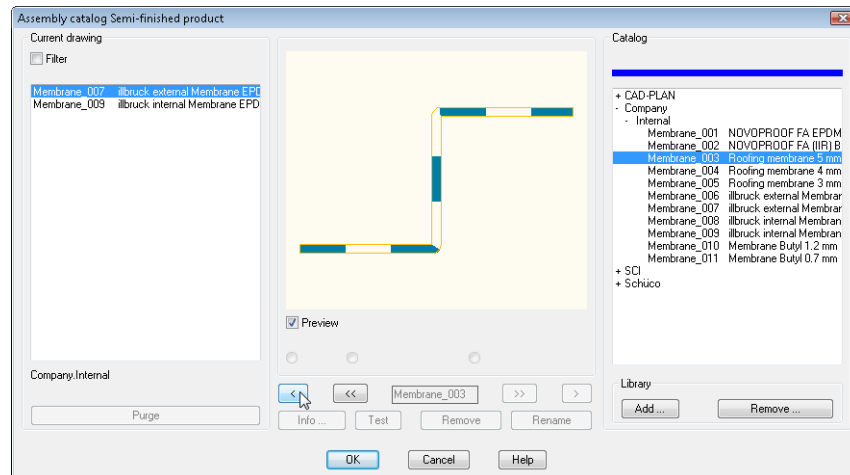
6. Now close the ATHENA options dialog box again. The loaded libraries, settings files and programs are listed in the command line.



7. Start the command Membrane and change to the Manager section.
8. Click on the button Get and then on the button Catalog in the dialog box Semi-finished product.



9. In the dialog box, Assembly catalog, which now follows on the right-hand side all parts of the libraries loaded using the design environment are displayed in the respective folder structure.
In the folder structure select the required membranes and transfer them with the arrow keys [<] or [<<] into the current drawing.



10. Close the dialog box Assembly catalog with OK. In the dialog box Semi-finished product choose the required membrane and close this dialog box with OK. The selected membrane is now displayed in the dialog box of the same name and can be used in the drawing.

2 **Acquiring master data**

In this chapter you learn how you acquire master data and define properties and rules for this data.

Topics:

- **Introduction and preliminary Remarks**
- **Bar assembly with a single profile**
- **Bar assembly with notch**
- **Bar assembly with an angled profile**
- **Profile combinations with variants**
- **Glazing profile combination**

2.1 Introduction and preliminary Remarks

Master data (here bar assemblies) consist of one or more outlines. Through the definition of specific properties and rules they can be more or less intelligent and variable.

Preliminary considerations

Before you create master data, you should get to know the respective profile system and make yourself familiar with its special features. You should for example beforehand consider in which situations the data are to be used and which base points (reference planes) might be practical. This is particularly important for variable bar assemblies (e.g. variable angles or variable insets).

Consider also that master data are most likely to be used over a longer period of time and that the effort in modification increases with the complexity of the bar assemblies.

2.2 Bar assembly with a single profile

Generally we recommend that all profiles are acquired singly (a profile assembly consists of just one component part) and that they are later combined by Reference to form profile combinations. This not only increases the flexibility, but is also a condition for producing variable bar assemblies.

For creating component parts you can use single polylines or closed polylines contained in blocks (no block structures). It is important that all segments are contiguous and do not overlap. The layer allocation is irrelevant. Closed outlines may contain islands. Outlines may be ATHENA objects (sheet metal, spacer, insulation, ...).

This tutorial shows you how you acquire and save a single profile. As an example a Schüco profile is taken which is present as a polyline outline in the WCS of the drawing. A cutting outline has been drawn beforehand as a closed polyline around the profile. Furthermore, the base point of the profile has been marked with a circle.

Task definition

You create a new, single bar assembly with the following properties:

- Name SCI_322270
- Designation mullion 85 mm
- Article 322270
- Material aluminum
- Cutting BASIC

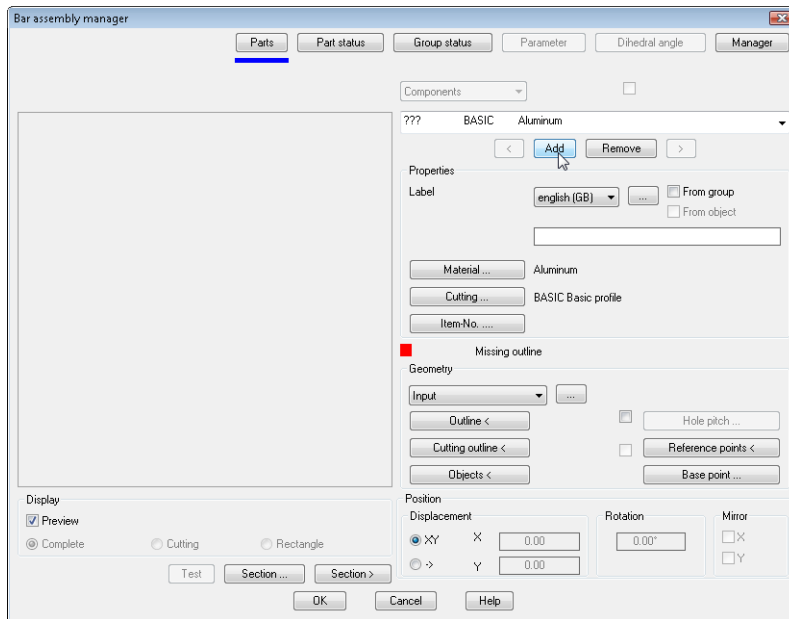
Define the outline and the cutting outline and save the bar assembly.

In order to be able to follow this tutorial you can also use the drawing cp_tutorial_sample_01.dwg.

Note: *In this example the adoption of blocks has been dispensed with.*

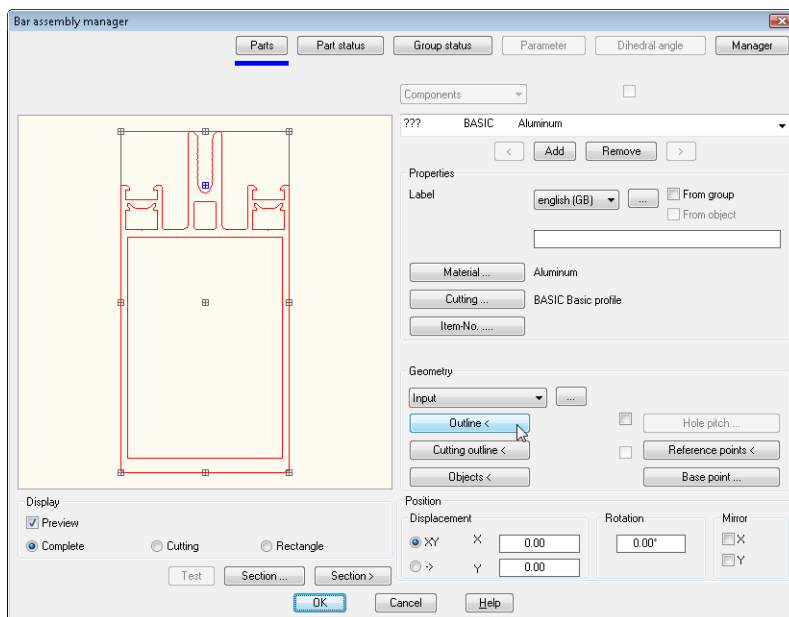
Steps:

1. Call the Bar assembly manager and a reset the dialog box if necessary.
2. Add a new, empty component to the definition via the button Add.
Make sure that the list entry in the dialog box section Geometry is set for input.



3. Select the button Outline < in the dialog box section Geometry to change to the Model section and to access the outline of the profile.
Follow the instructions of the input request, determine the outlines and specify the base points.

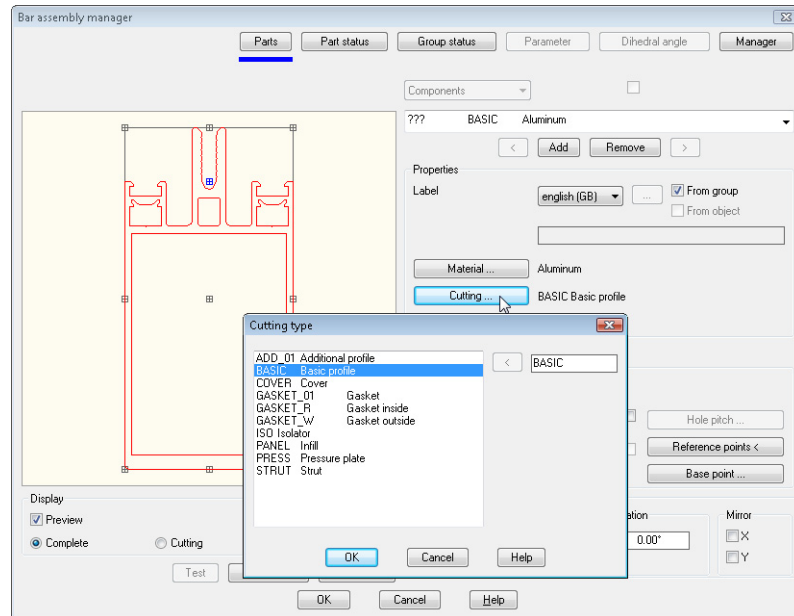
Note: The basic point of the component specifies its insertion point. The base point of the assembly specifies the displacement or position of the component for inserting the group. In many cases in which only one component exists in the group both points are identical.
During the adoption of an outline in a block its insertion point would be automatically adopted as the insertion of the component.



4. Once you have defined the outline, return to the Bar assembly manager.
Activate the tick box From group to use the group name as the component name.

Note: The tick box *From group* is only available for a component. If the outline is an *ATHENA* object the tick box *From object* can be used.

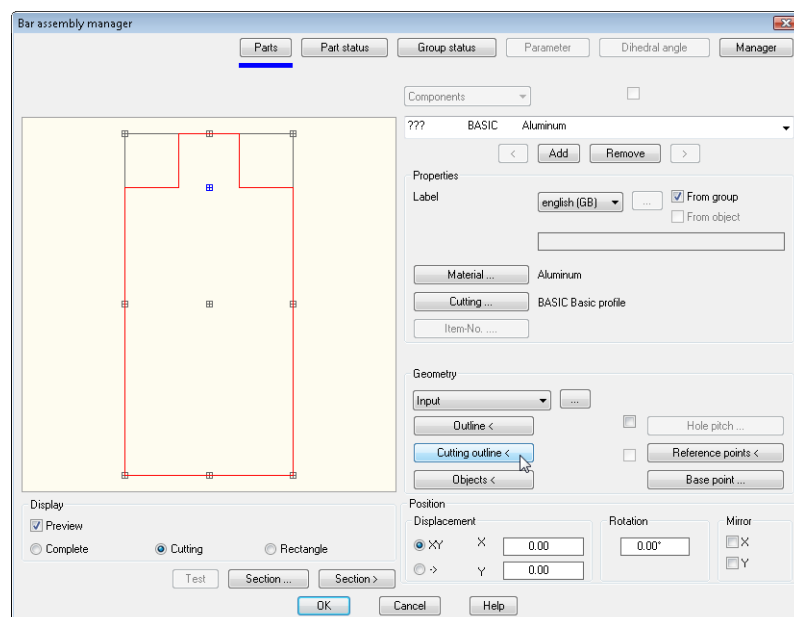
5. Assign a material and a cutting (important!) to the outline.



6. Display the saved cutting outline with the option *Cutting* in the dialog box section *Display*.

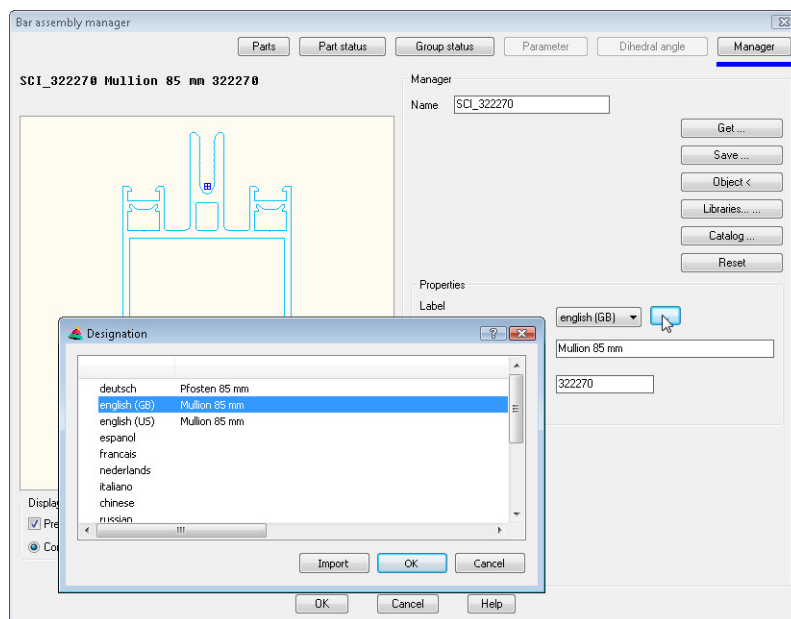
Note: The outline for cutting operations was initialized automatically to the external outline of the geometry when reading in the component.

7. Use the button *Cutting outline <* in the dialog box section *Geometry* to change to the *Model* section in order to define a new cutting outline. Specify the base point of the assembly to temporarily display the previous cutting outline and then select the new outline for the cutting. You are then automatically returned to the Bar assembly manager.

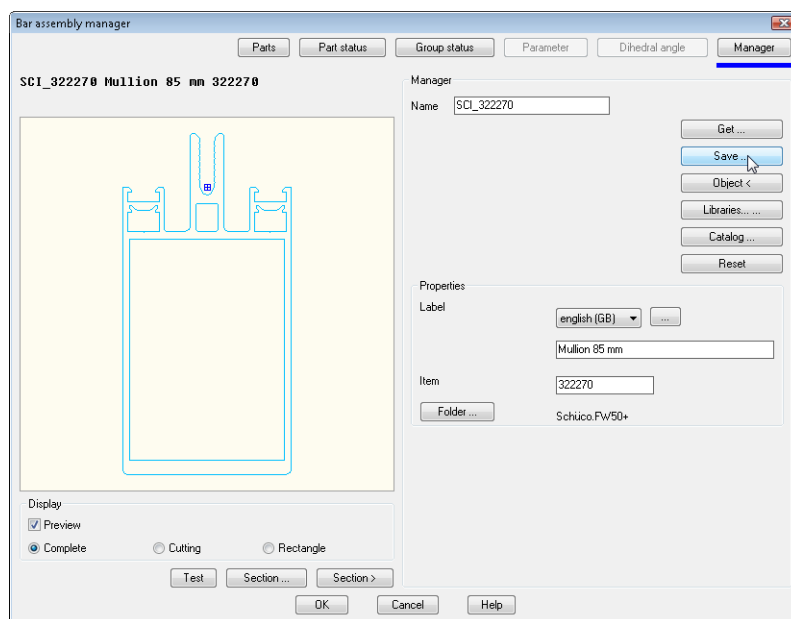


8. Change if required to the section Part status and check the possible statuses for the evaluation and display of solids.
9. Change if required to the Group status section and check the format of the label using the button of the same name in the dialog box section Assembly properties.
10. Change to the Manager section and give the definition a unique storage name such as for example a combination of the manufacturer's abbreviation and the article number (SCI_322270).
Give the definition one or several (multilingual) designations as well as an article number and a folder affiliation.

Note: The affiliation to a folder helps during the archiving of structured acquired master data.



11. Save the definition with the button of the same name.



12. Change to the section Parts and check whether the group name has been adopted for the component. Now terminate the Bar assembly manager with OK.

As a trial, you can use the command Use bar assembly to use the bar assembly just saved as a section or solid in the drawing.

2.3 Bar assembly with notch

For various situations profiles must be notched at the crossing point (e.g. Schüco planes). In the ATHENA Bar assembly manager you can produce components for this and define them as notches.

Definition: A notch is a process aligned longitudinally on the bar which (in its length) is limited by the cross-section of the joining bar. The volume which is subtracted from the Bar assembly is added to the joining bar assemblies.

Task definition

Expand an existing bar assembly and define the notch.

A completely saved bar assembly must be already present; refer to the previous tutorials.

In order to be able to follow this tutorial you can also use the drawing cp_tutorial_sample_01.dwg.

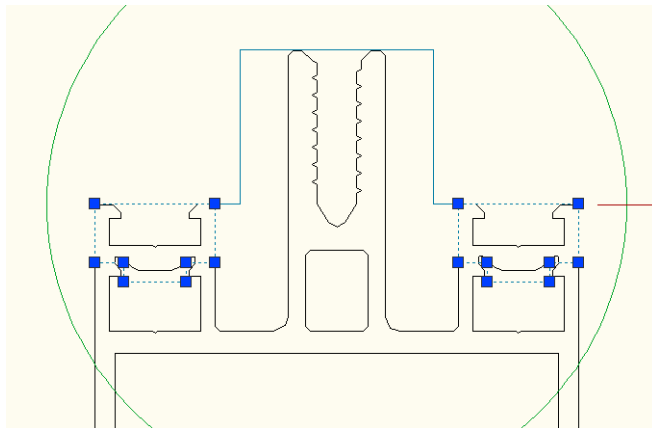
Steps

1. Start the command Arrange bar assembly and insert where applicable the saved bar assembly as a section.

Tip: Mark the base point of the bar assembly e.g. with a circle.

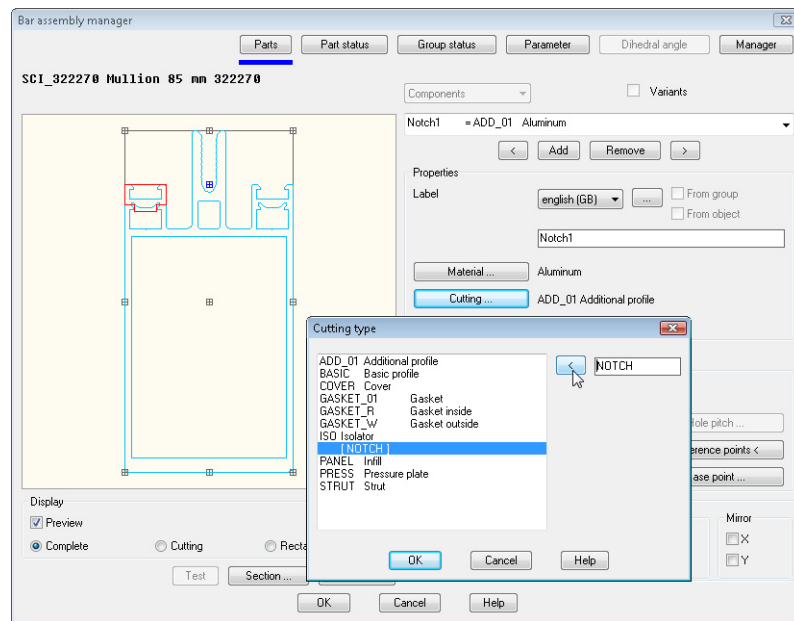
2. Draw the outlines of the notches as closed polylines.

Note: The notch is subtracted as a process from the cutting outline. If the notch is too small or does not border the edge it may possibly be ineffective.



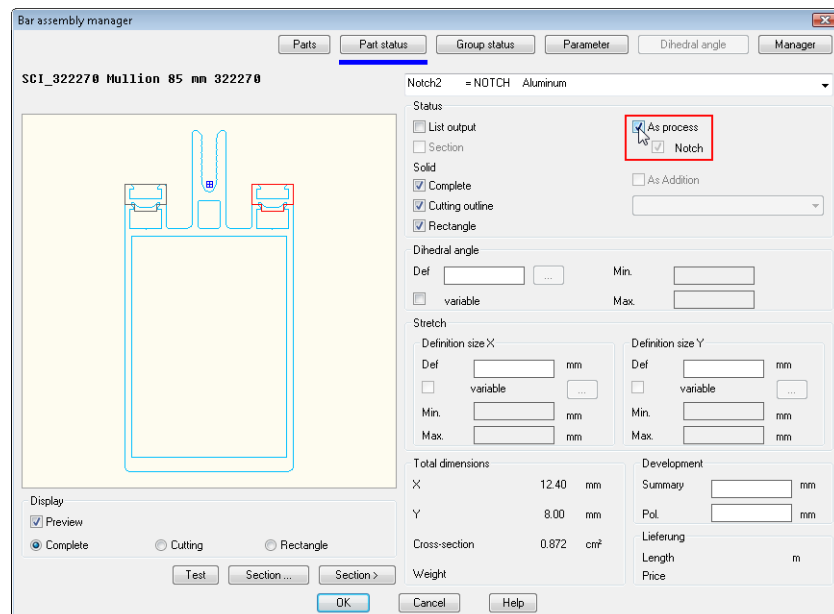
3. Call the Bar assembly manager, change to the Manager section and bring the bar assembly just positioned in the Model section into the dialog box.
4. Change to the Parts section and add a new component using the button Add, designating it as a notch.
5. Using the button Outline in the dialog box section Geometry assign to it one of the notch outlines drawn in the Model section.
6. Assign the cutting NOTCH to the component.

Note: The cutting for the notch must be unique and must not be used by any other component.



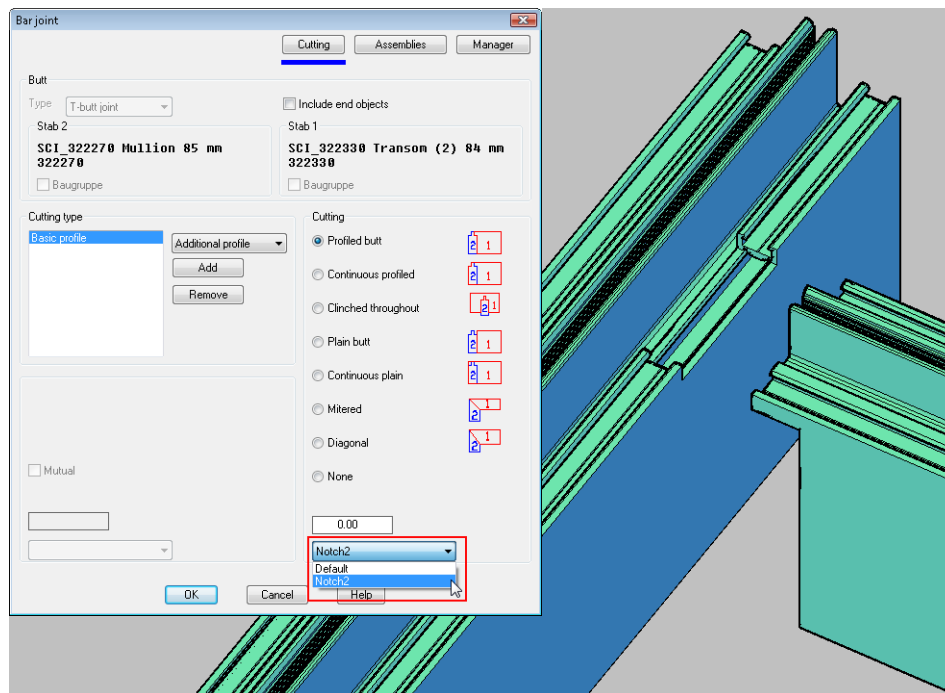
7. Repeat the previous steps for the second notch.
8. Change to the section Part status, in the upper pick list consecutively select the notches and in the dialog box section Status activate in each case the tick box As process.

Note: A notch will then be detected as such and set automatically when the component is on one hand labeled as a process and on the other hand possesses a cutting which, apart from other notches, is unique within the definition.



9. Change to the Manager tab and save the assembly.

An application note: During use the notch can be selected with the bar joint and in each case acts on the mullion in the region of the crossing point.



2.4 Bar assembly with an angled profile

For variable corners often profiles with an angled glass plane are used. ATHENA offers you the possibility of defining bar assemblies for variable angles. With these profiles the seal receptacle is angled according to the manufacturer's data.

Note: *Folding, bending and cutting of components is only possible with pure outlines (no references) and ATHENA objects.*

Task definition

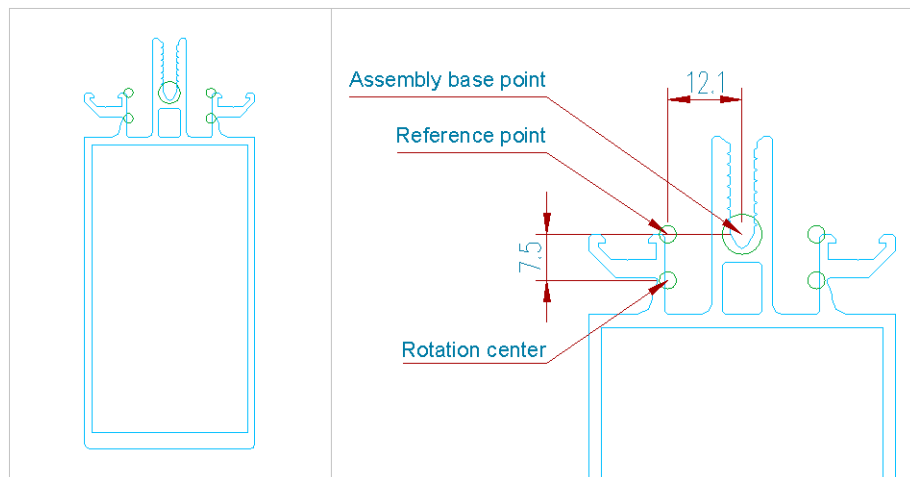
You expand an existing bar assembly and define an angled kink. A completely saved bar assembly must already be present; refer to the previous tutorials.

In order to be able to follow this tutorial you can also use the drawing cp_tutorial_sample_01.dwg.

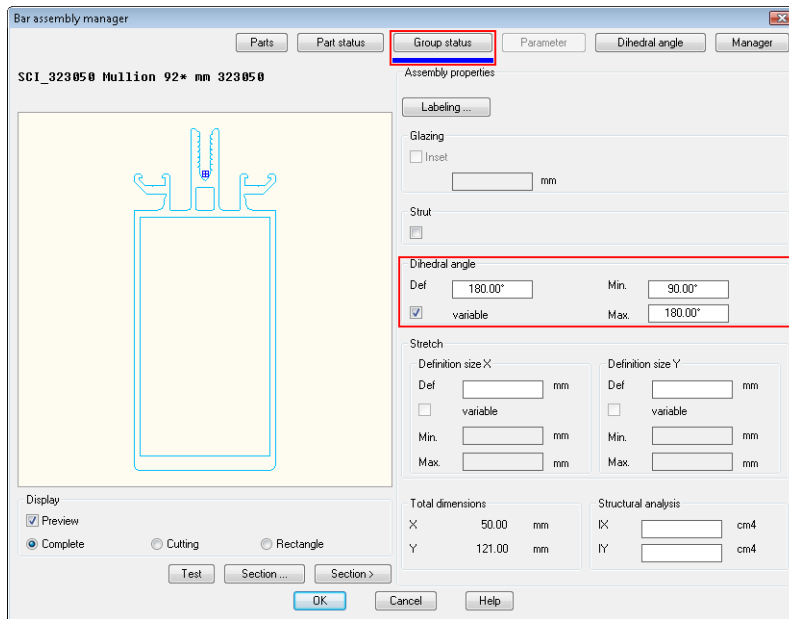
Steps

1. Start the command Use bar assembly and insert where applicable the saved bar assembly as a section. Mark the base point of the bar assembly e.g. with a circle. Also mark the rotation and reference points for the angled kink of the seal receptacles.

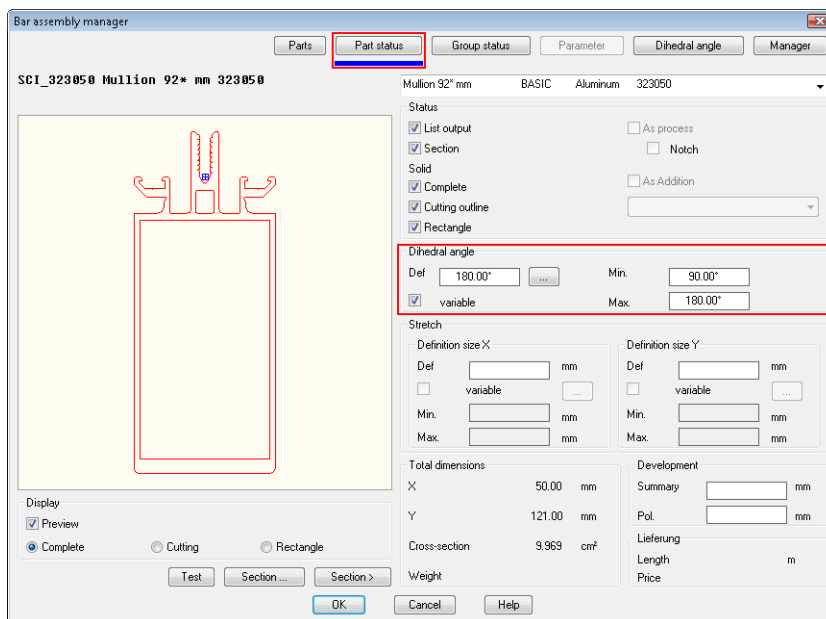
Note: *For technical reasons the rotation point must not lie within the outline to be modified. Therefore, choose a point which lies on or adjacent to the outline. The resulting rotation point is calculated as a plummeting point onto the outline.*



2. Start the Bar assembly manager and bring the bar assembly just inserted into the dialog box.
3. Change to the section Group status. In the dialog box section Dihedral angle enter 180° as the definition angle in the input field. Activate the tick box Variable and enter 90° for the minimum value.

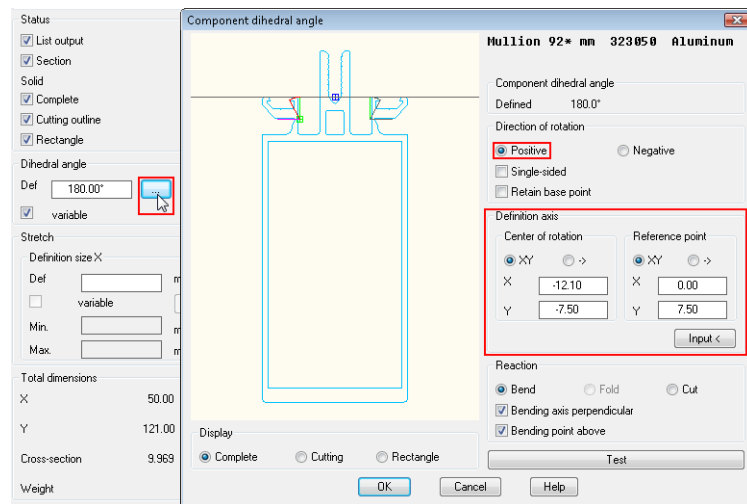


4. Change to the section Part status and in the dialog box section Dihedral angle enter 180° into the input field as the definition angle. Activate the tick box Variable. In the input field Min. enter the value of 90° and 180° in Max.

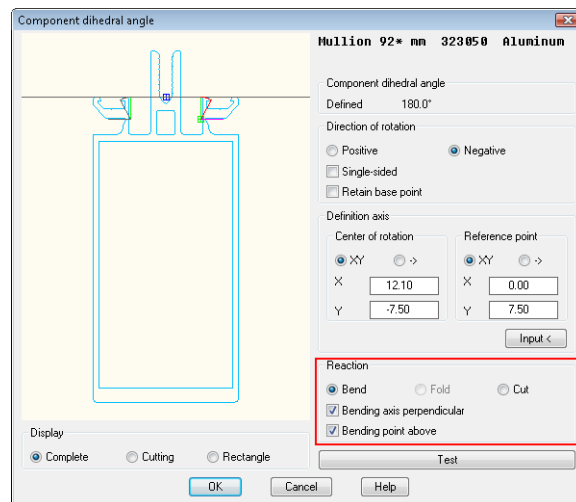


5. With a click on the button [...] in the dialog box section Dihedral angle start the subdialog box Component dihedral angle.
6. Define the kink point for the left seal receptacle as follows:
Select the option Positive in the dialog box section Direction of rotation.
Click on the button Input < to determine the center of rotation and the reference point for the left seal insert. Follow the instructions in the command line until you again access the dialog box.

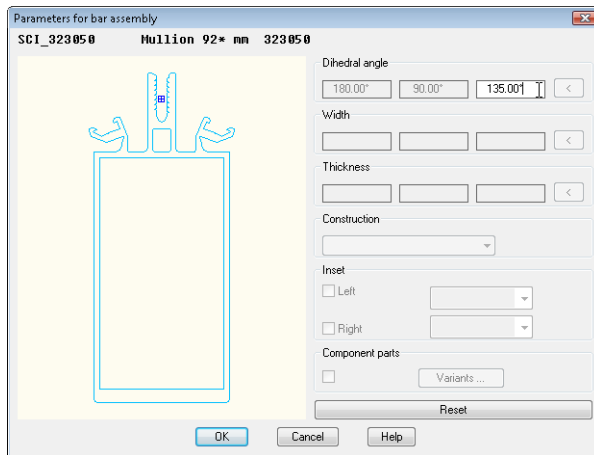
Note: You can also enter the distance values directly into the input fields.



7. You define the kink point for the right seal receptacle analogously to the left one:
Select the option Negative in the dialog box section Direction of rotation. Click the button Input < to determine the center of rotation and reference point for the right seal insert. Here too, follow the instructions in the command line until you again access the dialog box.
8. Select the option Bend in the dialog box section Reaction and activate the tick boxes Bending axis perpendicular and Bending point above.



9. With a click on the button Test start the subdialog box Parameters for bar assembly and check the reaction with an angle of 135°.



10. Terminate the dialog box Parameters for bar assembly with OK and change to the section Manager. Assign the storage name and designation and, where applicable, the article number and folder affiliation.
11. Save the bar assembly and terminate the Bar assembly manager with OK.
12. As a trial, you can apply the command Use bar assembly to use the bar assembly just saved as a section or solid in the drawing.

2.5 Profile combinations with variants

A bar assembly can consist of several components. If existing bar assemblies (with only one component) are used for this, the term References is used. The advantage with references of bar assemblies is that variants can be formed and these bar assemblies can then be used more flexibly for various applications.

Task definition

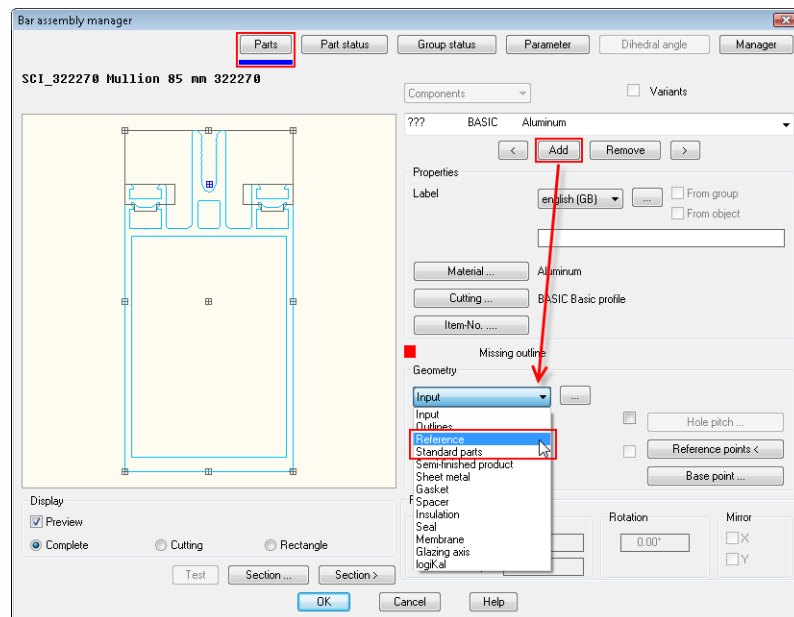
You expand the existing mullion bar assembly SCI_322270 in that you add a further component as a reference. You define a variant for the referenced component.

In order to be able to follow this tutorial you can also use the drawing cp_tutorial_sample_01.dwg.

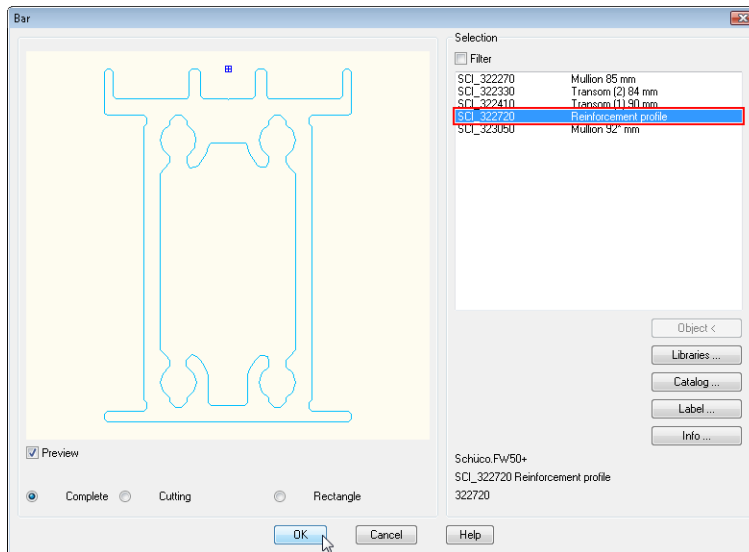
Note: Assemblies can only be referenced when they have previously been saved under their own name. Variants can only be set up with referenced assemblies.

Steps

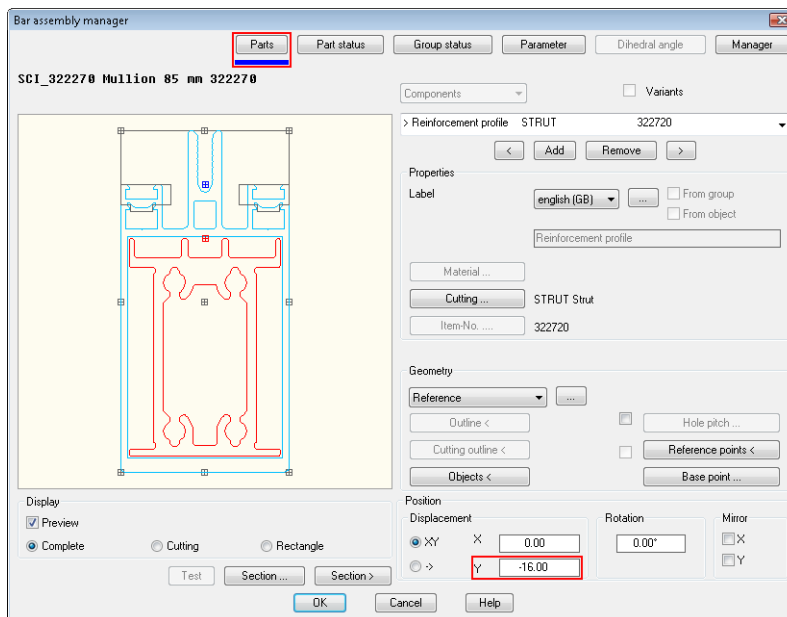
1. Start the Bar assembly manager, change to the Manager section and bring the mullion assembly already saved into the dialog box.
2. Change to the Parts section, add a new component and in the dialog box section Geometry select the option Reference from the pulldown list.



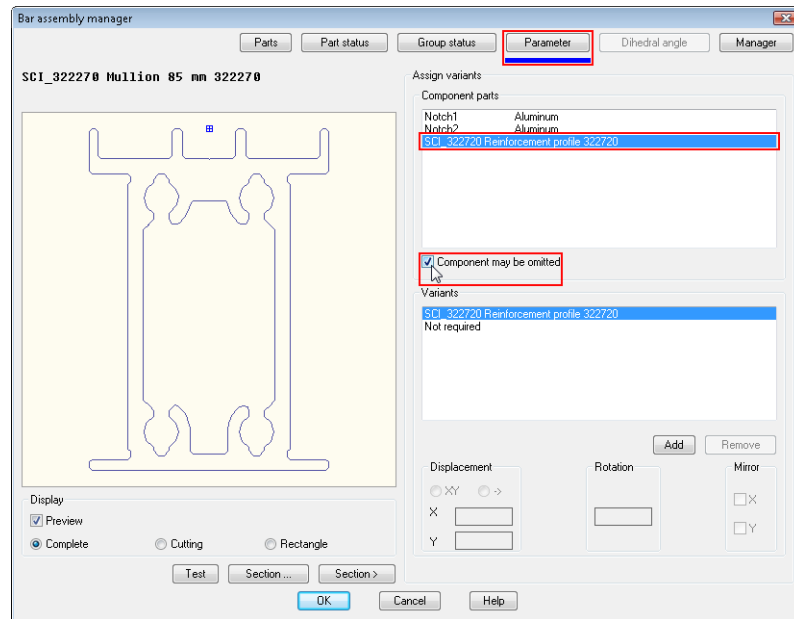
3. In the dialog box, Use bar, which then follows select the insertion profile and close the dialog box with OK to return to the Bar assembly manager.



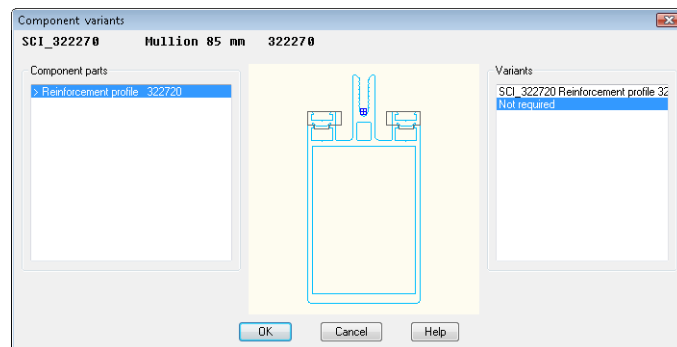
4. You must now insert the insertion profile into the mullion chamber. To do this, in the dialog box section Displacement change the Y value to -16.



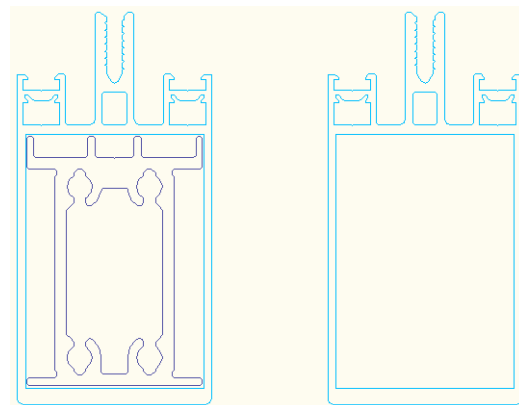
5. Change to the Parameter section.
Since the insertion profile is only needed when the structural analysis requires it, it should be optionally omitted. This should be controlled via a variant definition.
6. In the component list choose the insertion profile and activate the tick box Component may be omitted in the dialog box section Assign variants The variant Not required is now displayed in the variant list.



7. You can now test the function of the variants. To do this, click the button Test and the button Variants in the following dialog box Bar assembly parameters.
8. In the following dialog box, Component variants, the component is displayed on the left and its variants on the right. You can now select the respective variants and test the reaction in the preview.



9. Close all dialog boxes until you again access the Bar assembly manager, change to the Manager section and save the assembly.
10. As a trial, you can apply the command Use bar assembly to use the bar assembly just saved as a section or solid in the drawing.



2.6 Glazing profile combination

A glazing is a special, also parameterizable, bar assembly with referenced components. You can define glazing for different thicknesses of glass. Depending on the selected glass thickness, components can be automatically replaced (for example seals), supplemented (for example glass groove reduction) or changed in their position.

Task definition

You produce a new bar assembly and supplement the components in that you select assemblies inserted into the drawing. You define this assembly as glazing and supplement the variants for other glass thicknesses.

In order to be able to follow this tutorial you can use the drawing `cp_tutorial_sample_01.dwg`.

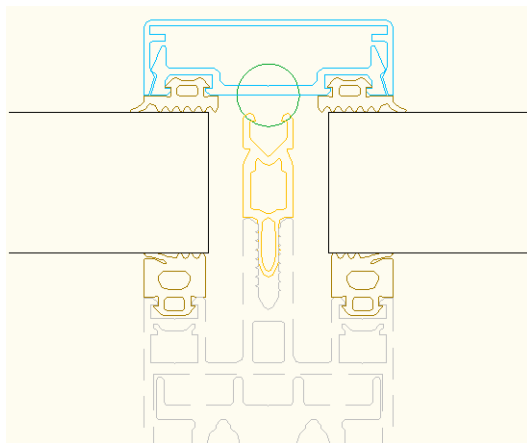
Note: In *ATHENA* glazing can only be defined if all the components are of the type reference or axis symbol. Therefore, all components must initially be available individually; in this respect refer to the previous tutorials.

It is not advisable to doubly acquire components just because they are differently orientated. For this use the options Mirror, Rotate and Displace.

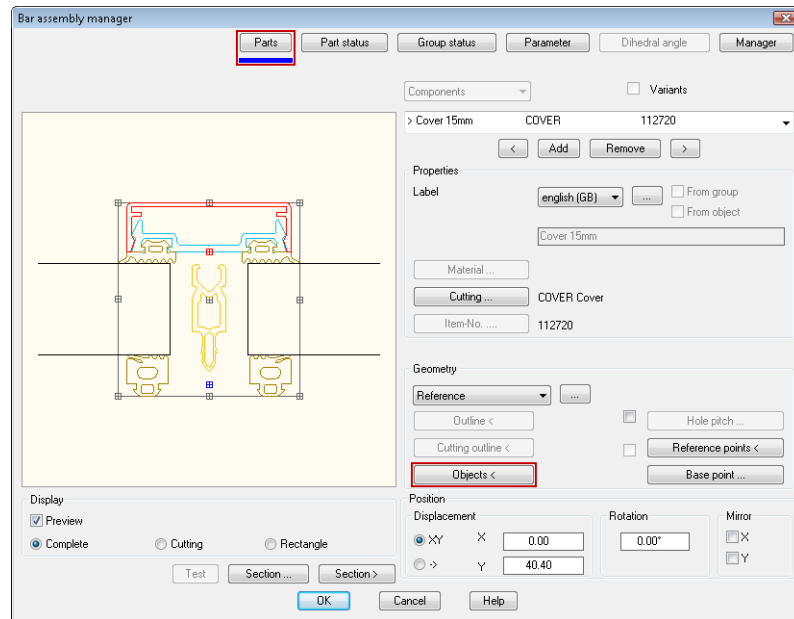
Steps

1. With the command Arrange bar assembly insert all the required components individually as sections into the drawing and bring the profile combination together accordingly.
2. Use the command Axis symbol (*ATH_AXIS*) to insert an infill position as a placeholder for the infill element.

Tip: Assume the greatest possible inset as the definition dimension. This simplifies the work afterwards.



3. Start the Bar assembly manager and if necessary reset the dialog box to start a new bar assembly definition.
In the section Parts click on the button Objects <, select the components positioned in the drawing and specify the insertion point of the profile combination.

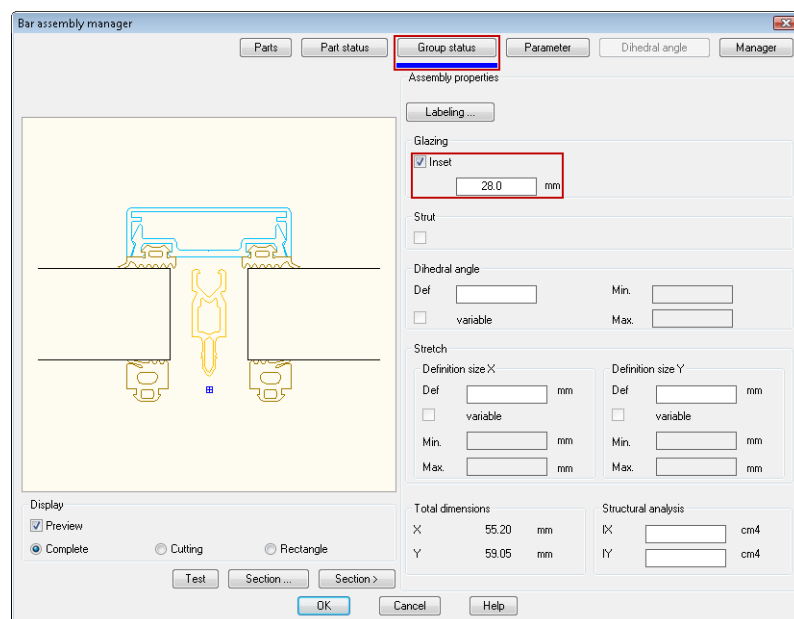


4. Check the cuttings of all components and modify these if necessary according to your requirements.

Note: If the referenced assembly consists of only one component, its cutting is initialized. If the assembly consists of more than one component (which we do not recommend) or if the cutting of the current definition deviates from the reference, it must be readjusted.

5. Change to the section Group status. To define a glazing activate the tick box Inset and enter 28 mm into the input field as the inset thickness for the current definition. Confirm the following query with Yes.

Note: From this point in time the insertion and removal of components in the Parts section is no longer possible. At the same time the functionality in the Parameter section has changed.



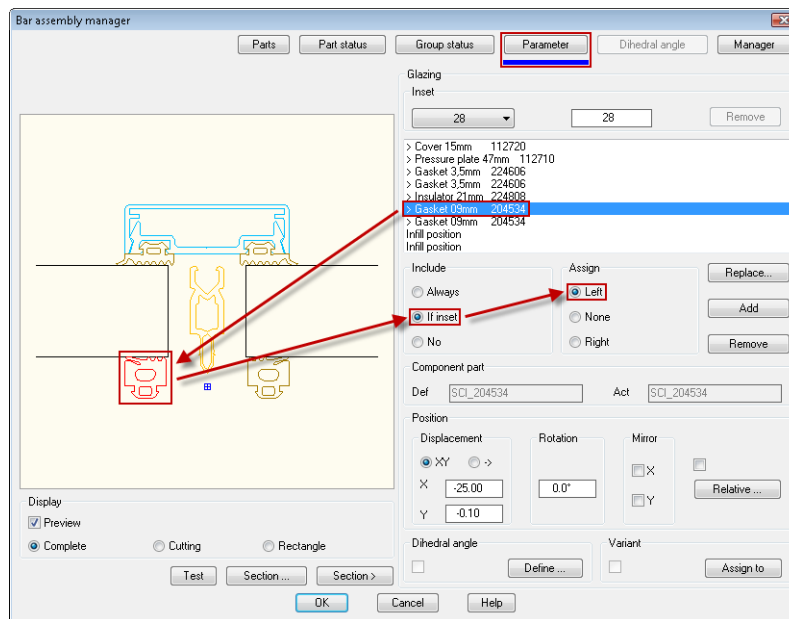
- Change to the section Parameter to describe the reaction of individual components to the possible insets.

Note: In principle the position, assignment and visibility of each component is described for each component. In order to achieve a certain automatic operation, all components of an inset are described and with the production of a further inset copied and only slightly corrected.

- Initially, the properties Include and Assign should be defined for all components. You set the properties as follows:

Designation	Include	Assign
Insulator	Always	None
Glass seal	Inset	Left
Glass seal	Inset	Right
Seal, external	Always	Left
Seal, external	Always	Right
Clamp profile	Always	None
Cover section	Always	None
Infill position	Always	Left
Infill position	Always	Right

Tip: As a check use the test button regularly which is not only located in the main dialog box. In this way you can check parameters such as inset, angle, side affiliation and variants already in the definition phase.

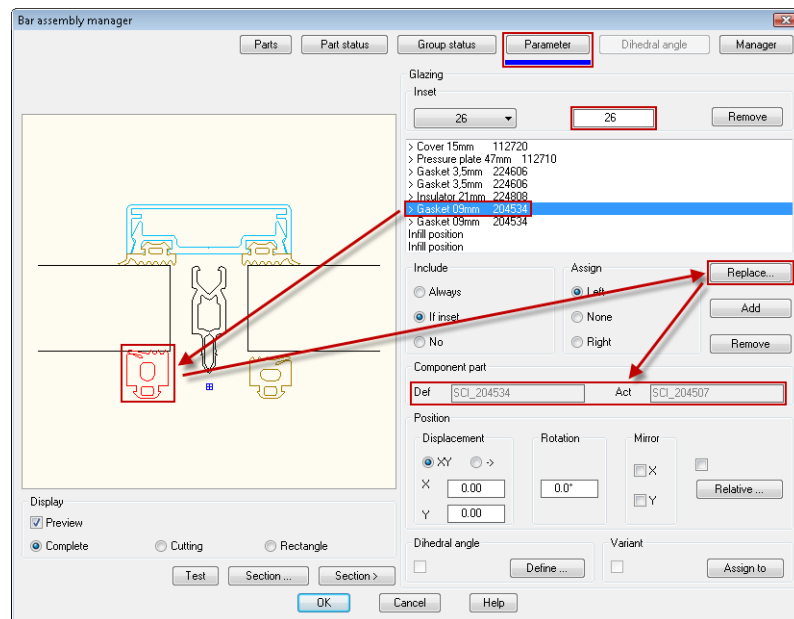


- Now you define the next inset thickness To do this enter the value 26 in the Inset input field.

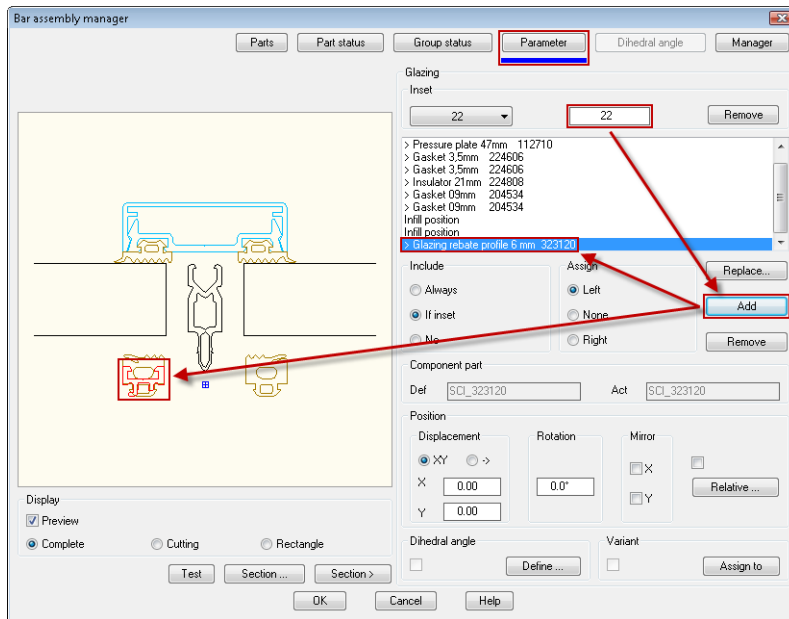
Note: The infill positions change their thickness automatically. The new situation requires a different internal seal with larger geometry.

- Mark the list entry for one of the internal seals and click on the button Replace ... to replace the 9 mm internal seal by an 11 mm seal. Repeat this step for the second internal seal.

Note: If the component which is replaced has an adapted orientation (rotation, displacement, mirroring) the new component adopts it automatically. In the dialog section Component part you will now see the storage name of the component in the defined and actual states.

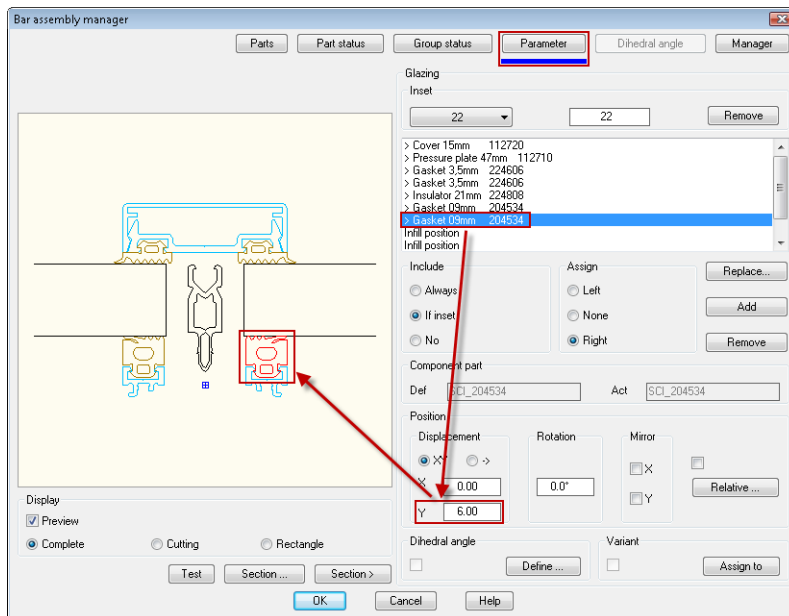


- Now define the inset thickness as 24 mm. Replace the two 9 mm internal seals by 13 mm internal seals according to the previously described method.
- Then define the inset thickness as 22 mm. For this inset thickness a glass reduction profile must also be inserted. For this, mark the list entry for one of the internal seals, click the button Add ... and insert the 6 mm glass reduction profile.



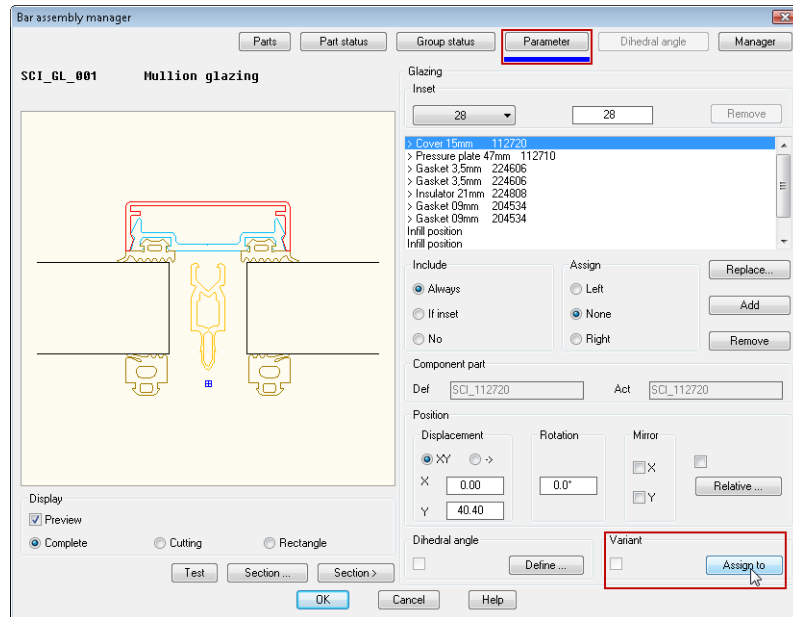
12. Mark the last selected internal seal and move it by entering the Y value 6 mm in the section Displacement.
(If required: Replace the current internal seal by the 9 mm internal seal.)
Repeat this step for the second internal seal.

Tip: If components for an inset are being added, the position, orientation, inclusion and assignment of the current selection are copied. Makes use of this feature.



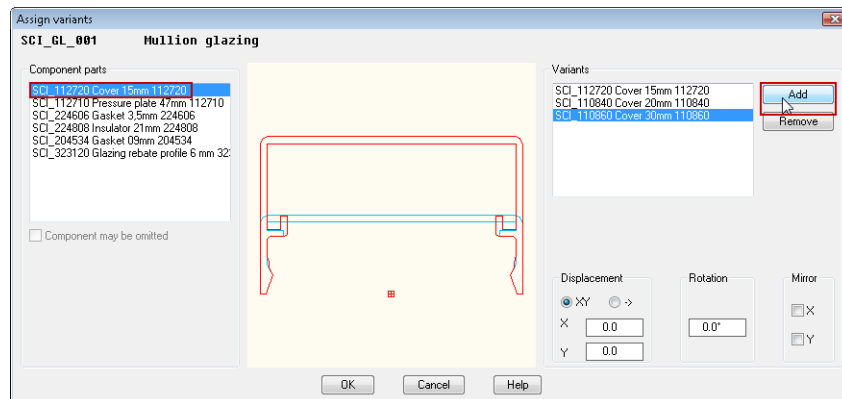
13. Repeat the last steps for the inset thicknesses which are still missing: 20 mm, 18 mm, 16 mm, 14 mm, 12 mm, 10 mm, 8 mm and 6 mm. Supplement the respective glass reduction profiles, replacing and removing the seals.
14. Save the bar assembly.
15. Optionally, you can now define variants for individual components (e.g. the cover section).

To do this, click on the button Assign in the dialog box section Variants to access the subdialog box Assign variants.



16. In the left pick list select the cover section and, using the button Add in the section Variants, assign another, already saved cover section.

Note: In order to avoid component displacement with variants it is recommended that identical base points are defined for similar components. If you have not done this, you can adapt the orientation of the assigned component variants.



17. Repeat the last step until you have added all the required cover sections and terminate the dialog box Assign variants with OK.

18. Simulate the previously defined variants. For this, click on the test button and in the dialog box Parameters for bar assembly carry out the appropriate settings.

19. Now change to the section Manager and save the bar assembly.

20. As a trial, you can apply the command Use bar assembly to use the bar assembly just saved as a section or solid in the drawing.

B Definition of terms

Component (bar)

An element within a bar assembly consists of a cross-section geometry which is located in a bar assembly application as a sectional representation or solid.

Reference plane

Imaginary horizontal plane through the base point of the bar assembly.

Reference point

Perpendicular point from the center of rotation onto the reference plane.

Center of rotation

Point about which outlines are rotated, folded, bent or cut.

Reference

Link to an existing bar assembly (with a single component) for further use. This means that complicated and structured object definitions built up on single, saved objects are possible. Objects for referencing may be assemblies and bar assemblies.

Bar assembly

Grouping of individual bar cross-sections which are compiled using rules and, controlled with parameters, can be used as sections and solids.

Glazing

A glazing is a special, also parameterizable, bar assembly with referenced components. It is used for automated application on subconstructions (bar assemblies with glazing axes), whereby position and orientation are determined / adopted.

Cutting

This is the type of characteristic for an applied cutting operation. With bar assemblies the following operations are currently available:

- Profiled butt
- Continuously profiled
- Clinched throughout
- Plain butt
- Continuous plain
- Mitered
- Diagonal

With assemblies these are the operations

- Union
- Difference
- Intersection

Cutting type

The property of a component which serves as the feature for a cutting operation of two elements. Only elements with the same cutting type can carry out an operation with one another.

Cutting outline

This is a property of a component of a bar assembly which as a solid can be used for cutting operations along the axis of the joining bar.

Appendix

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