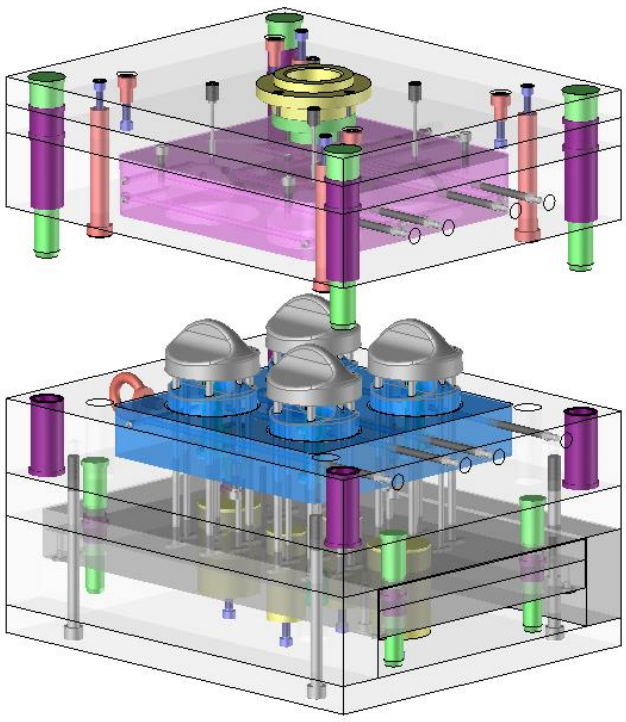
TopSolid Mold

Training Guide

TopSolid'Mold



MASTER YOUR MANUFACTURING PROCESS

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Note: If you are experiencing problems using this training guide, please feel free to send your feedback and comments to edition@topsolid.com.

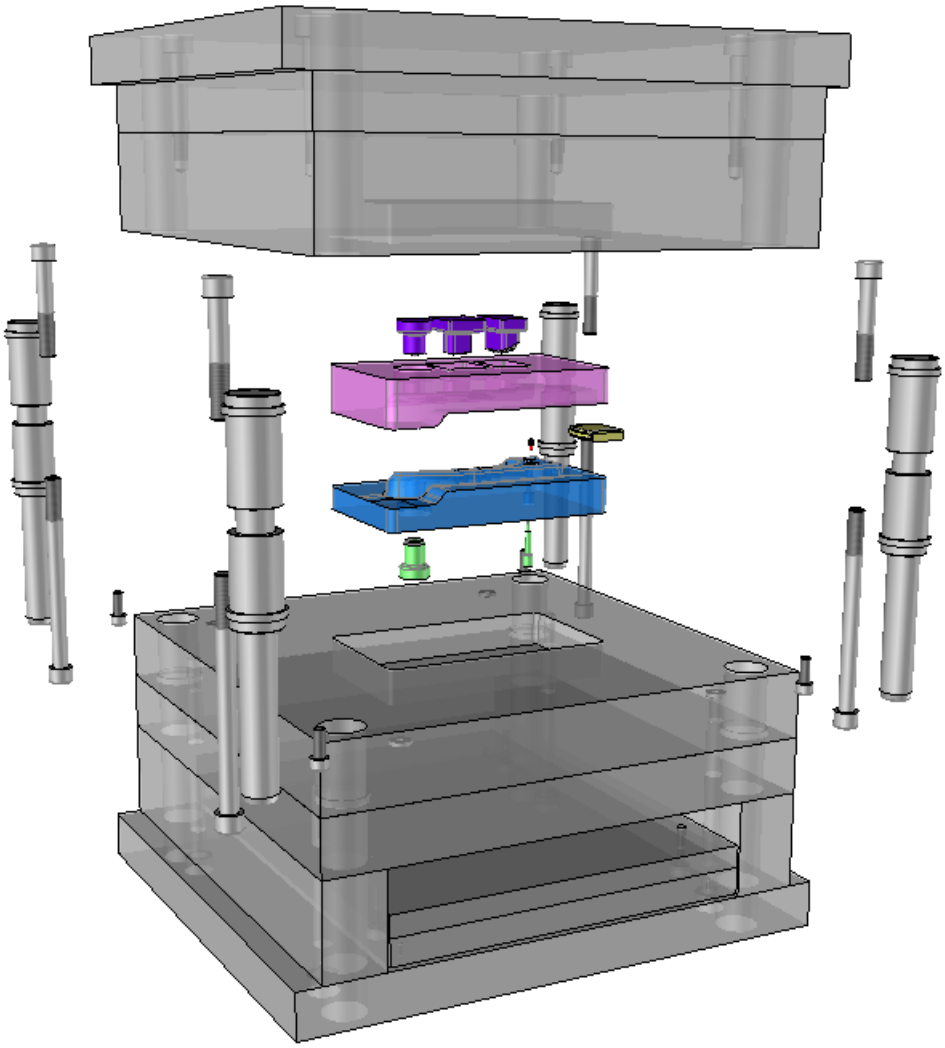
Contents

Single-cavity Mold

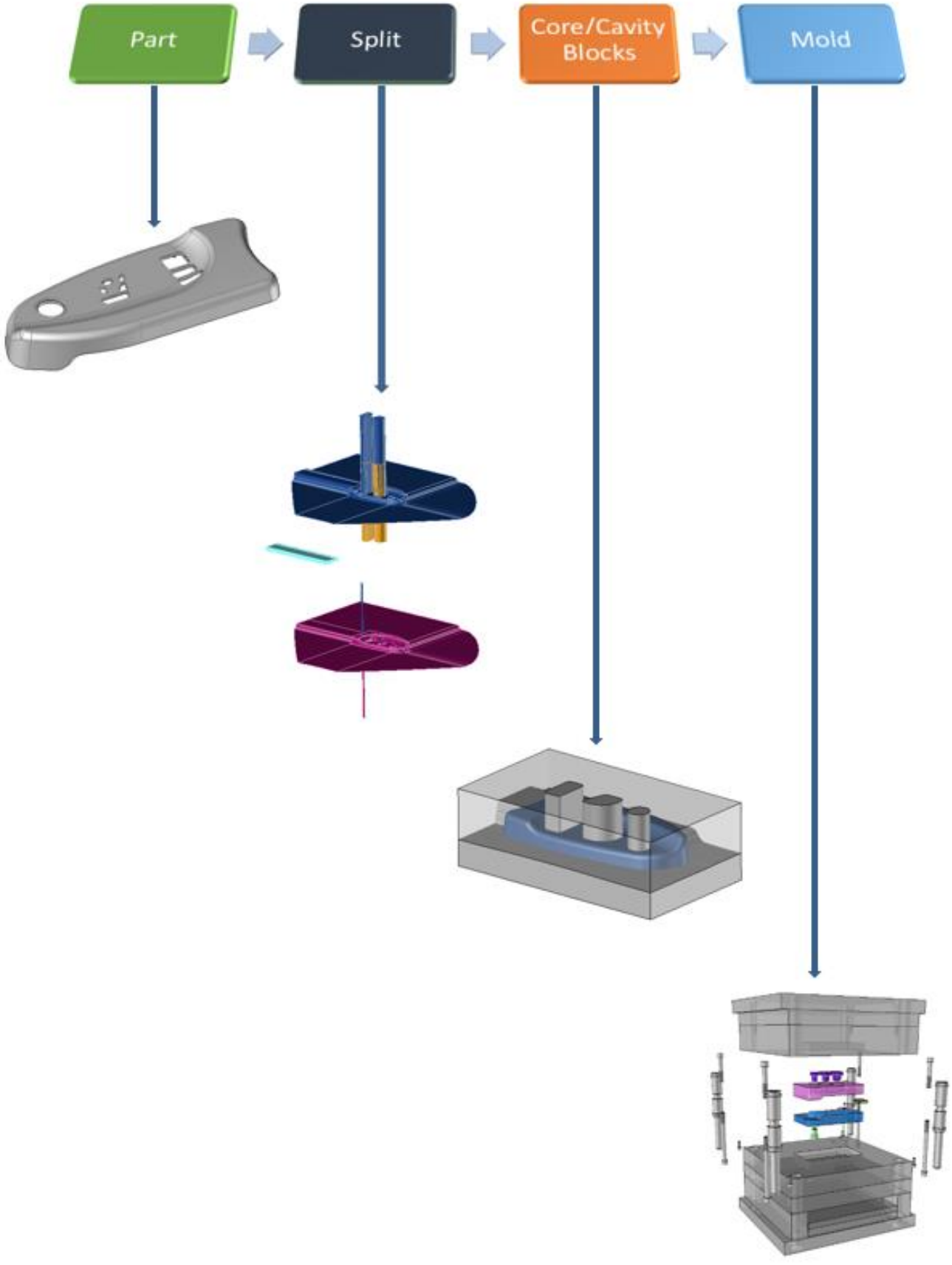
The purpose of this first part of the guide is to understand how the different types of documents related to the Mold document interact with each other, but also to know how to position basic elements such as the assembly resulting from the Split, and a standard mold base.

Concepts addressed:

- Types of documents linked to the Mold document
- Positioning the Split elements and the standard mold base
- Cavity process
- Handling and viewing the sets
- Drafting and bill of materials



Structure of documents



Including an assembly resulting from the Split process

Importing the package

- From the **Home** tab, **import the project** named *TopSolid'Mold Training D1.TopPkg*.

Note: This package contains all the documents needed to complete all of the exercises in this training guide.

Warning: Do not confuse the **Import Project** command (formerly known as **Import Project as Replication**) which creates a new project or synchronizes your documents (data exchange between the PDM server and the local PDM for example) with the **Import/Export** > **Import Package** command which creates new documents. You can only access the latter command by right-clicking on the node (root or folder) of a project. Halfway between these two commands, you can find the **Import Project as Distinct Copy** command (**TopSolid** icon > **File**) which creates a new project and thus new documents.

Including the assembly in a new mold document

Note: There are many ways to include an assembly in a mold document:

- Right-click on the assembly document's tab and select the **Mold** command.
- Or right-click on the assembly document in the Project tree and select the **Mold** command.
- Or create a new **document**, select **Mold**, and then drag and drop the assembly document into the new mold document from the Project tree.

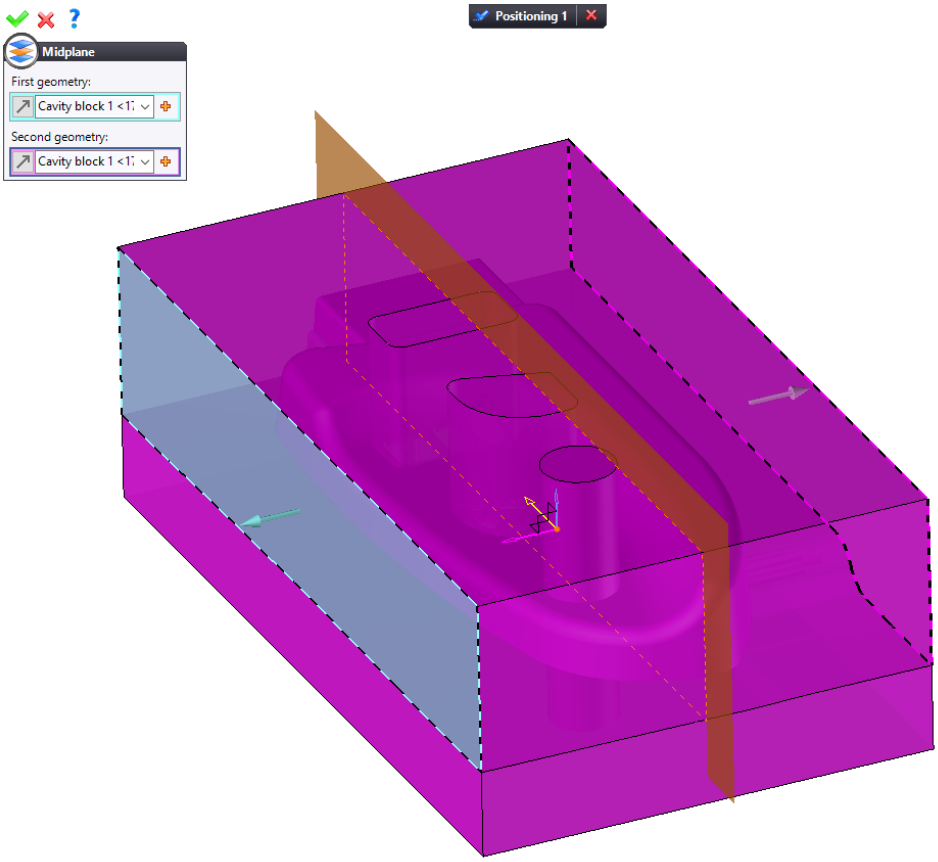
- From the Project tree, open the *Ex01 - Simple mold* folder, right-click on the *DoorArmrest Assembly From Split* assembly document and select the **Mold** command.
- Select **Blank Template** and click on to **confirm**.
- From the Project tree, rename the new mold document *My first mold*.

Repositioning the core and cavity blocks

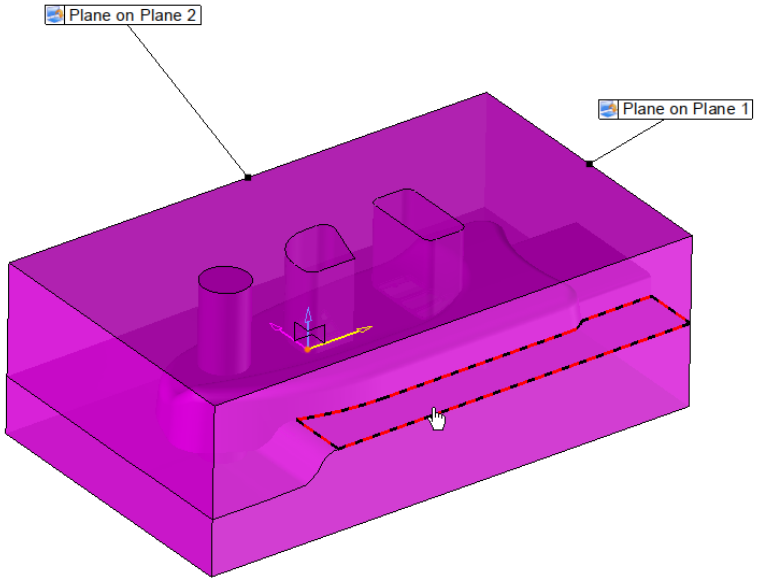
By default, during inclusion, **TopSolid** positions the assembly's opening frame (from the Split) on the absolute frame of the new mold document. In this exercise, we want to position the center of the core and cavity blocks and a face given on the mold document's absolute frame.

- From the Entities tree, open the **Frames** folder, right-click on **Absolute Frame** and select the **Show** command.
- Right-click on one of the assembly shapes and select the **Edit Positioning** command from the **Core Cavity Blocks Inclusions** section.
- Right-click on the **Frame on Frame 1** positioning constraint's label and select the **Delete** command.
- From the **Assembly** tab, select the **Plane on Plane** command and pin the dialog box using the icon.

- In the **Source plane** field, click on the icon and select the **Midplane** command. Select the front and back faces as shown below, then click on to **confirm**.



- In the **Destination plane** field, select **Absolute YZ Plane** from the drop-down list.
- For the second **Plane on Plane** constraint, repeat the previous steps to constrain the middle plane of the left and right side faces with the **absolute XZ plane**.
- For the last **Plane on Plane** constraint, select the face as shown below and constrain it with the **absolute XY plane**.



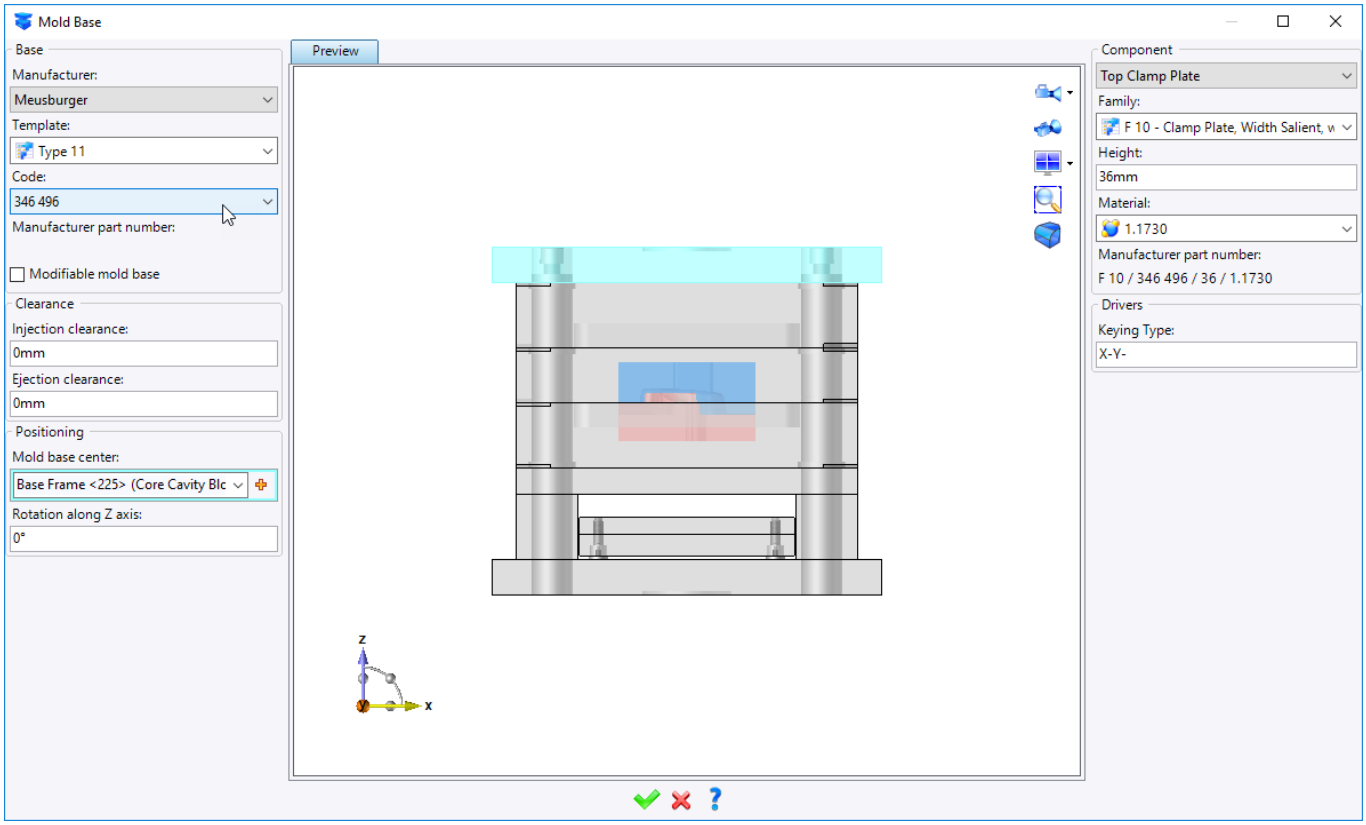
- **Confirm** positioning 1.

Including a manufacturer's standard mold base

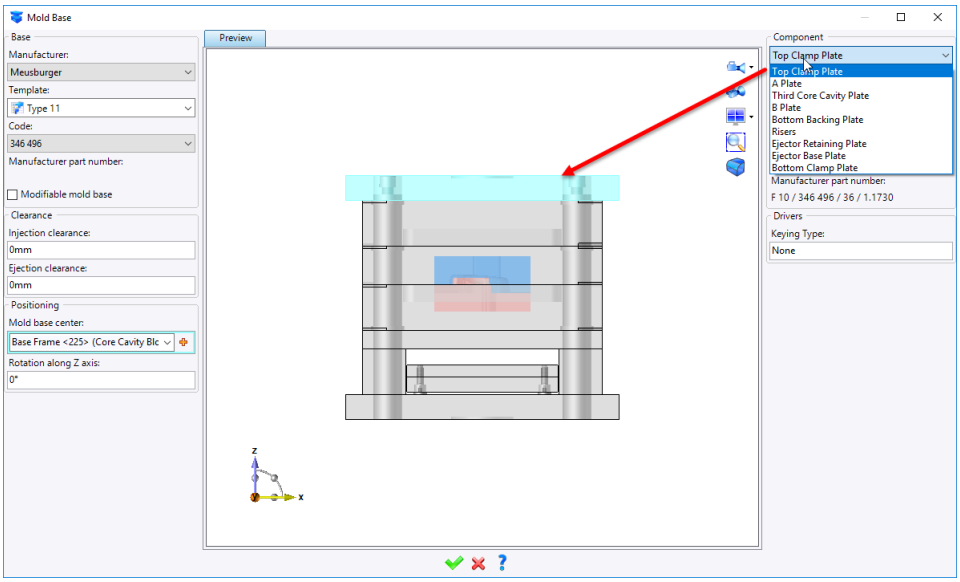
- Make sure that the manufacturer's library is referenced in the project. To do this, open the **References** node in the Project tree and make sure that the **TopSolid Meusburger Tooling** library is available. If not, right-click on **References**, select the **Reference Library** command and select the **TopSolid Meusburger Tooling** library.
- From the **Mold** tab, select the **Mold Base Inclusion** command.

The parameters on the left of the dialog box are the general parameters of the mold base.

- Adjust the parameters as shown below.



The parameters on the right of the dialog box are the parameters for a plate of the mold base. You can select the plate from the **Component** drop-down list.

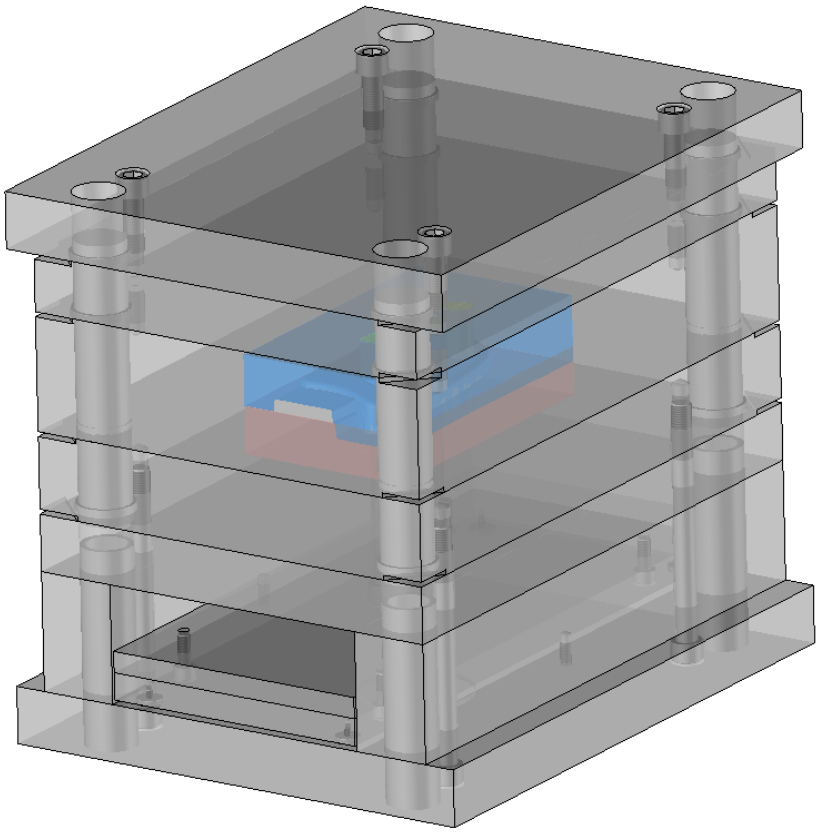


Note: You can also select the plate by clicking directly on the plate in the graphics area.

- Adjust the following parameters for each plate.

Component	Family	Height	Width	Material
Top Clamp Plate	F15	46mm	-	1.1730
A Plate	F50	46mm	-	1.1730
Third Core Cavity Plate	F50	96mm	-	1.1730
B Plate	F50	66mm	-	1.1730
Bottom Backing Plate	F60	46mm	-	1.1730
Risers	F70	96mm	62mm	1.1730
Ejector Retaining Plate	F80	17mm		1.1730
Ejector Base Plate	F85	22mm		1.1730
Bottom Clamp Plate	F15	46mm	-	1.1730

- Click on to **confirm**.



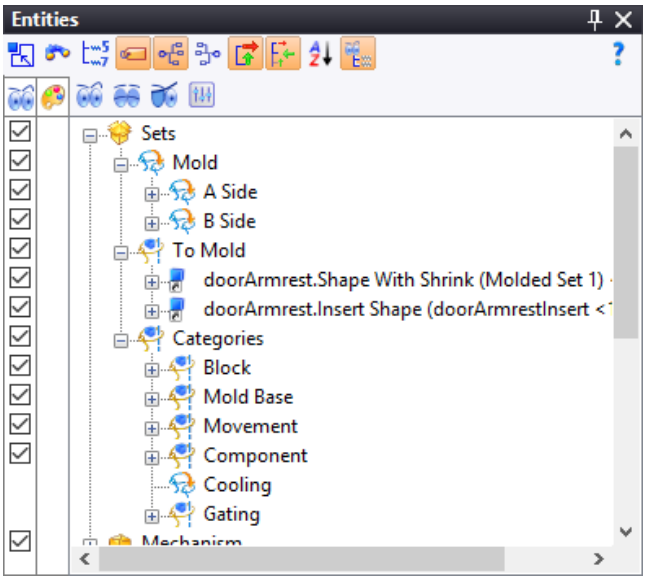
Note: At any time you can return to the mold base dialog box by editing the command. From the Operations tree, you only have to right-click on the **Mold Base Inclusion** operation and select the **Edit** command.

Viewing the sets

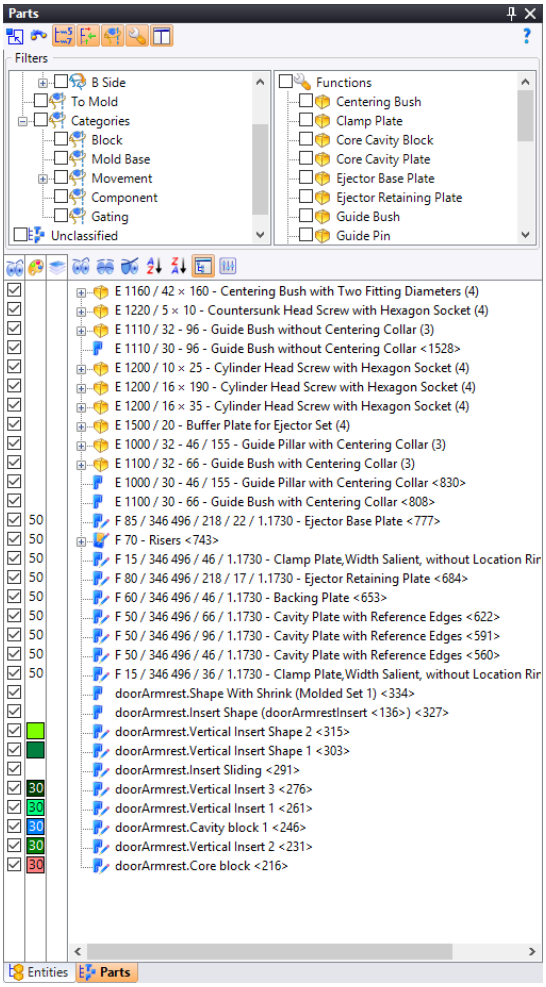
TopSolid automatically creates three sets and its subsets in a mold document:

- the **Mold** set contains the **A Side** and **B Side** subsets;
- the **To Mold** set contains the plastic part(s);
- the **Categories** set contains subsets of the mold elements such as the blocks, the mold base, etc.

These sets and subsets can be used in the bills of materials and drafting documents. It can be interesting to filter, hide or display them in order to facilitate certain tasks.
These sets and subsets are available in the Entities tree's **Sets** folder.



They are also available in the Parts tree. As well as being able to view the sets and parts, you can also filter the elements by **function** (plate, ejector base plate, guide pin, screw, inserts, etc.).

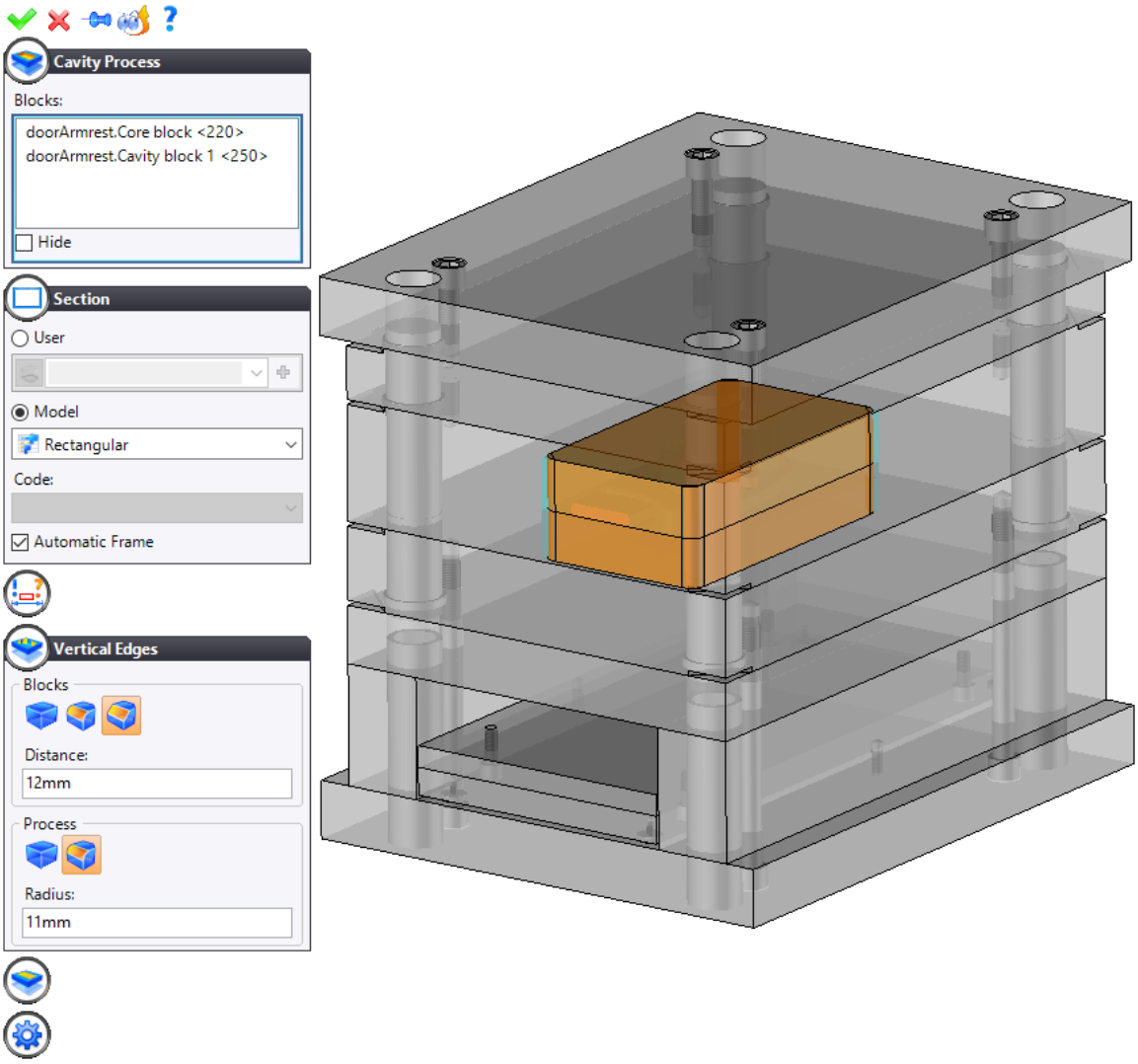


The icons on the bottom right of the graphics area allow quicker access to the main sets.

Creating a cavity process

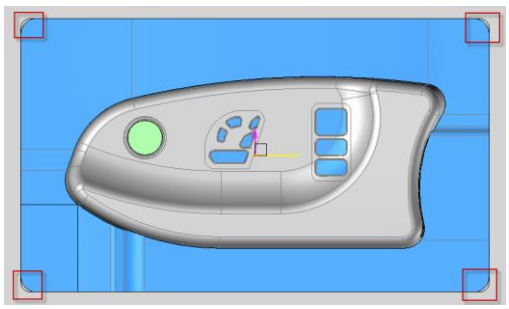
The **Cavity Process** command quickly removes the material of the core and cavity blocks from the mold base's plates, and allows you to directly apply fillets/chamfers onto the vertical edges of the blocks.

- Select the **Cavity Process** command and adjust the following parameters. Make sure you select the **Rectangular** model document.



- Click on to **confirm**.

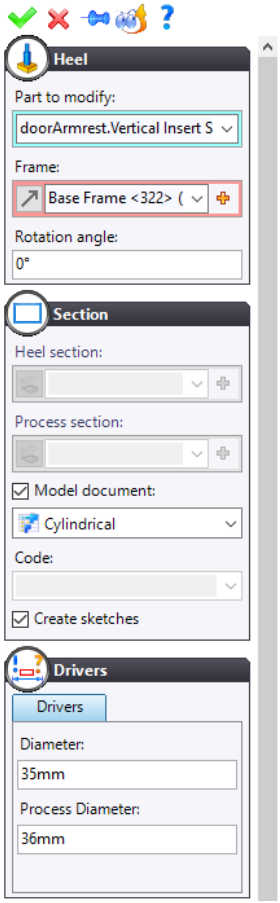
Note: From the Parts tree, you can filter and hide the **A Side** subset to view the process result.

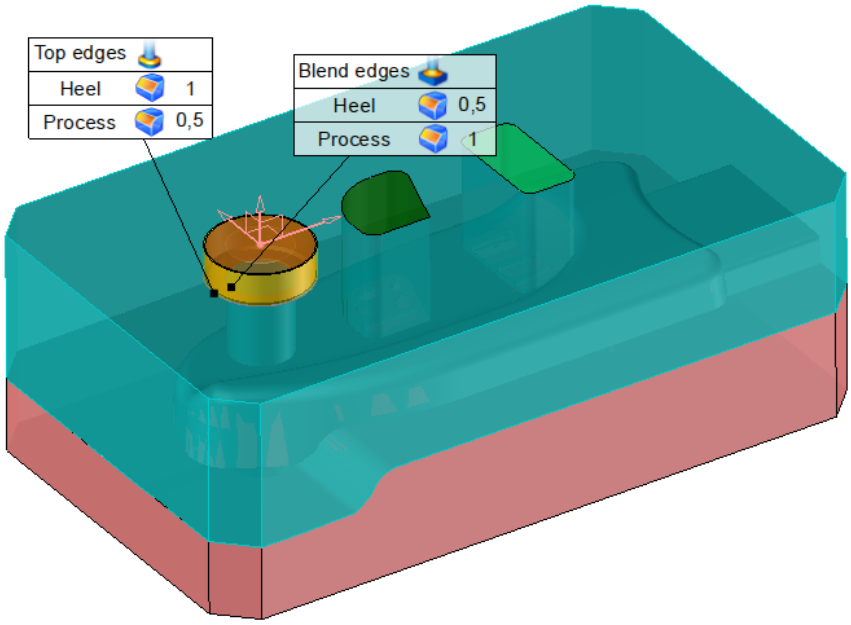


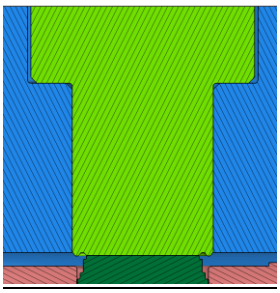
- Hide the mold base. To do this, click on the **Hide/Show Mold Base** icon at the bottom right of the graphics area.

Creating the heels

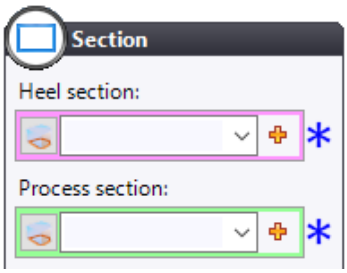
- From the **Mold** tab, select the **Heel** command and adjust the parameters as indicated below:
  - **Heel**: Select the cylindrical insert of the cavity block as the **part to modify**.
  - **Section**: Check the **Model document** box and select the **Cylindrical** model.
  - **Drivers**: Enter **Diameter** = *35mm* and **Process Diameter** = *36mm*.
  - **Height**: Enter **Heel** = *12mm*.
  - **Top Edges**: **Heel** = *1mm*, **Process** = *0.5mm*.
  - **Blend Edges**: **Heel** = *0.5mm*, **Process** = *1mm*.



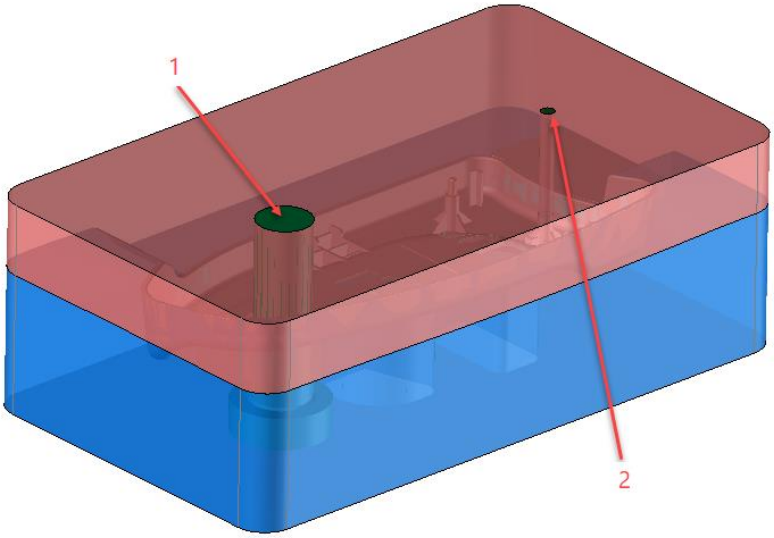




Note: You can create the sketches on the fly or select existing profiles for the heel and process sections using the special inputs.



- Create **heels** on the cylindrical inserts of the core block by adjusting the following values.



First cylindrical insert on the core block:

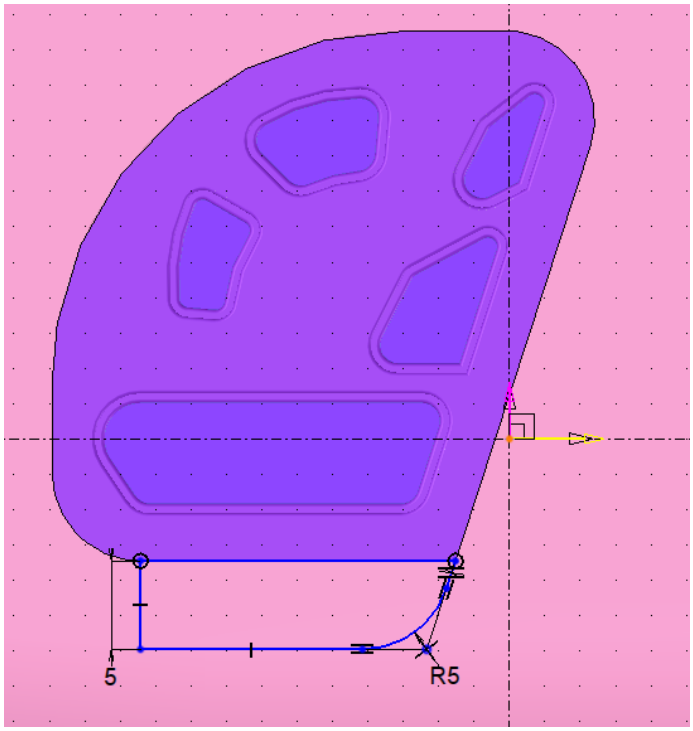
- **Drivers**: Adjust the **heel diameter** to *30mm* and the **process diameter** to *31mm*.
- **Height**: Adjust the **height** to *12mm*.
- **Top and Blend Edges**: Enter the same values as for the first heel.

Second cylindrical insert on the core block:

- **Drivers**: Adjust the **heel diameter** to *10mm* and the **process diameter** to *11mm*.
- **Height**: Adjust the **height** to *12mm*.
- **Top and Blend Edges**: Enter the same values as for the first heel.

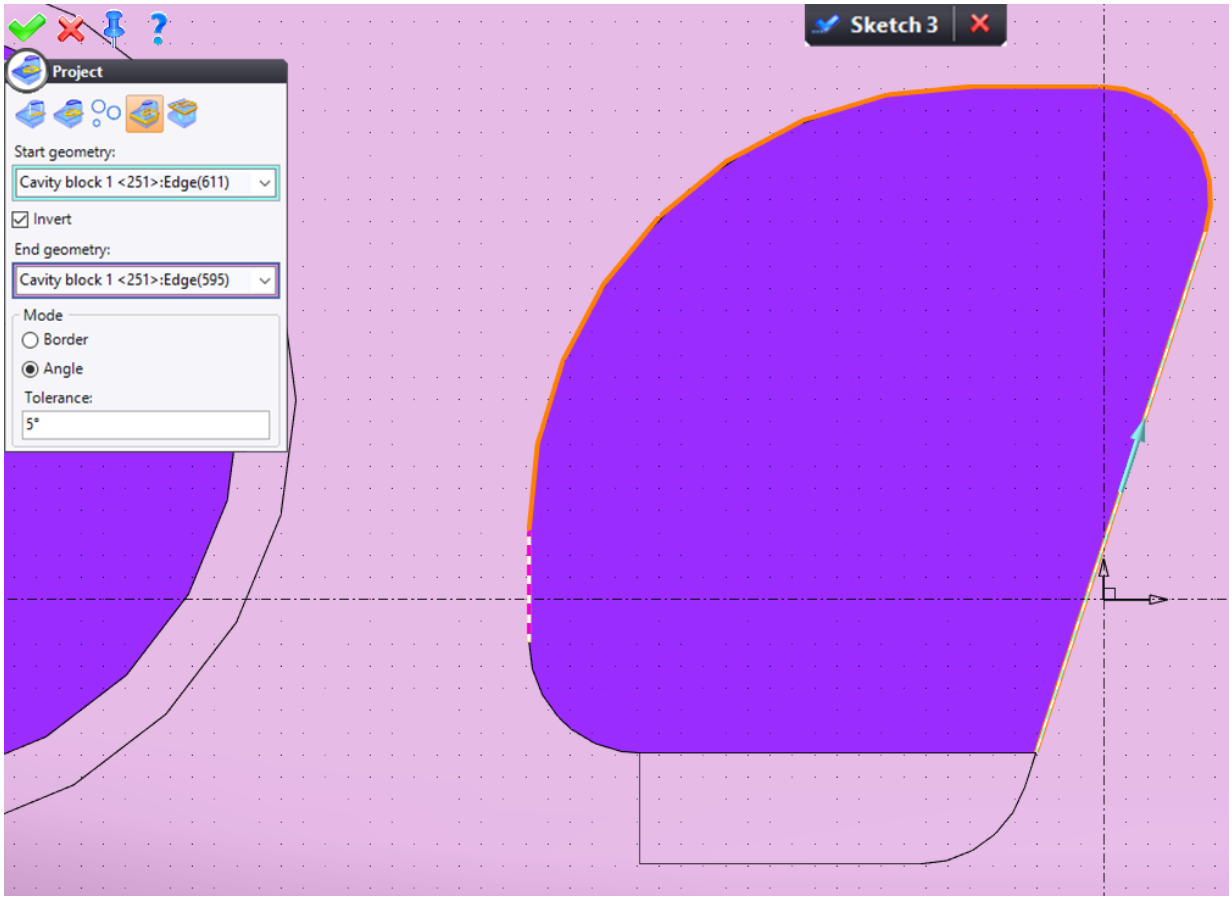
Note: If the frame is not created automatically, you can create it on the fly using the special inputs. In this exercise, you can use a **frame on plane**.

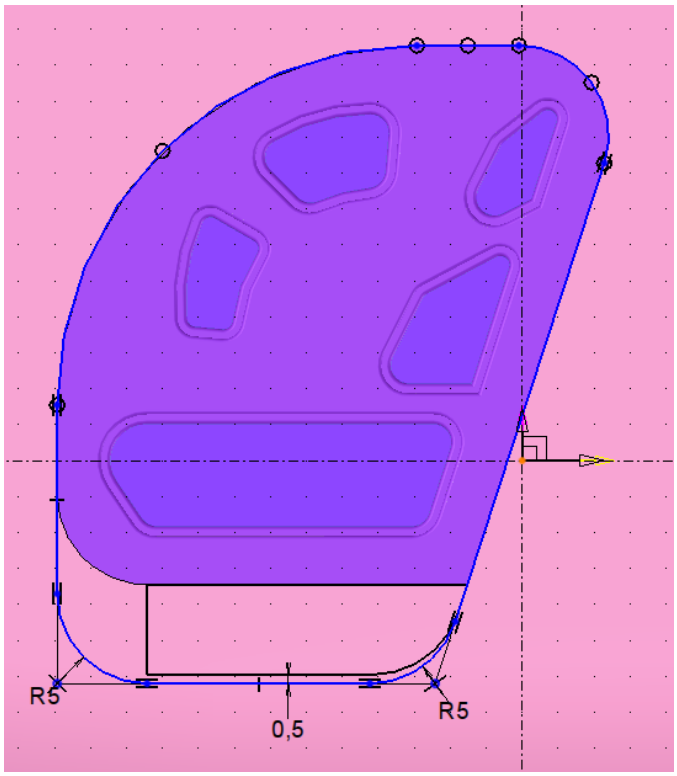
- Create the following sketch on the top face of the cavity block's center insert, then **confirm** the sketch. This sketch represents the shape of the heel.



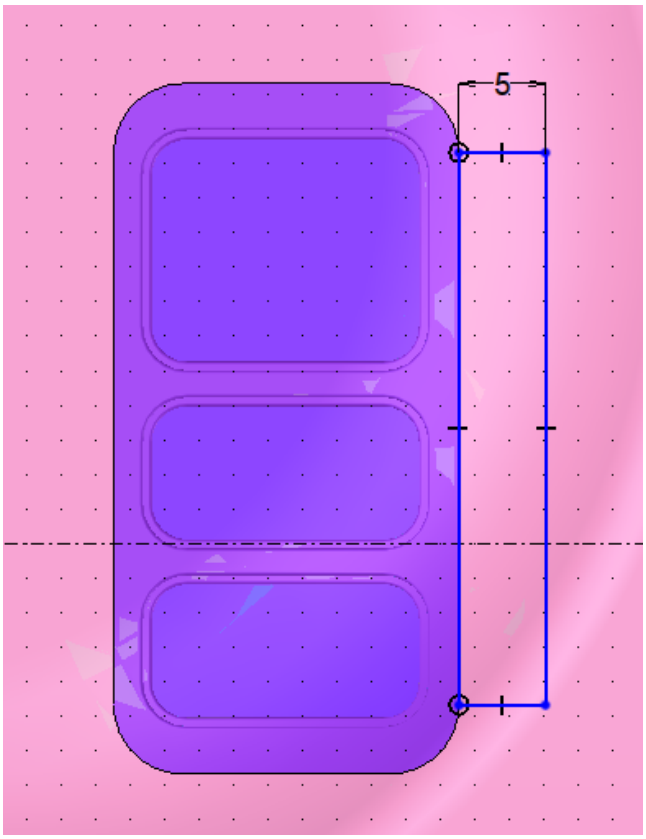
- Create a second sketch as shown below on the same top face of the cavity block's center insert, then **confirm** the sketch. This sketch represents the shape of the process.

Note: You can use the projection in  **Path between two edges** mode.

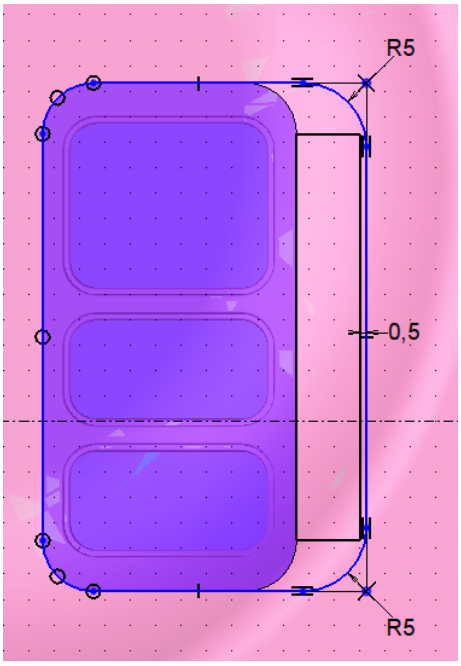




- Create a new sketch as shown below on the top face of the cavity block's rectangular insert, then **confirm** the sketch. This sketch represents the shape of the heel.



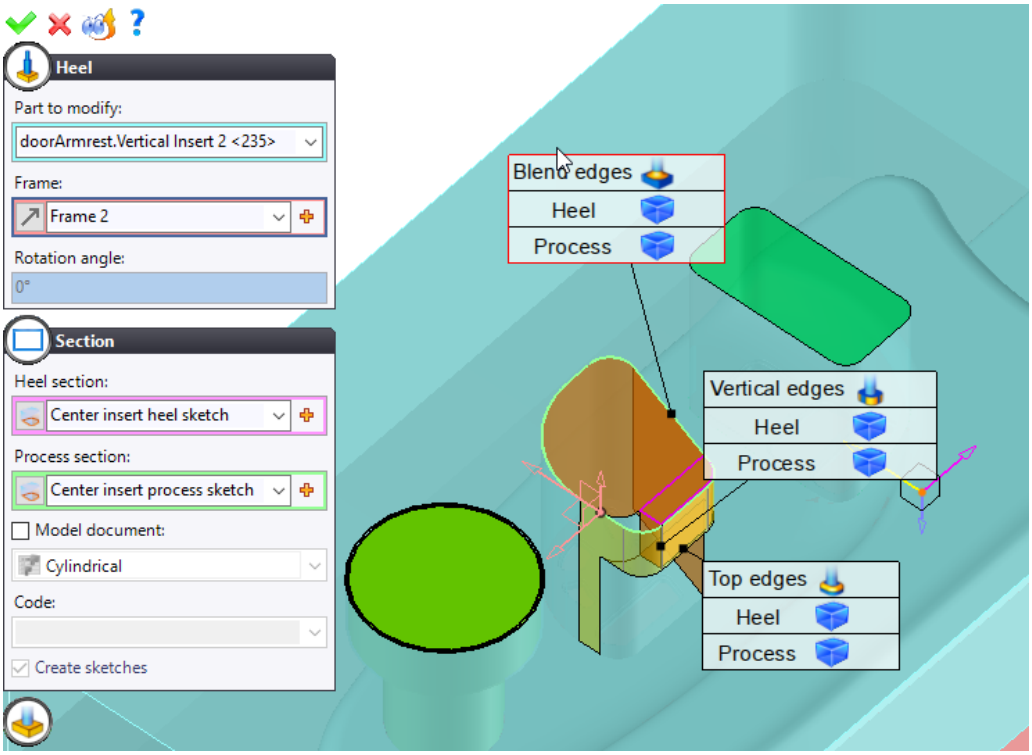
- Create a second sketch as shown below on the same top face of the cavity block's rectangular insert, then **confirm** the sketch. This sketch represents the shape of the process.



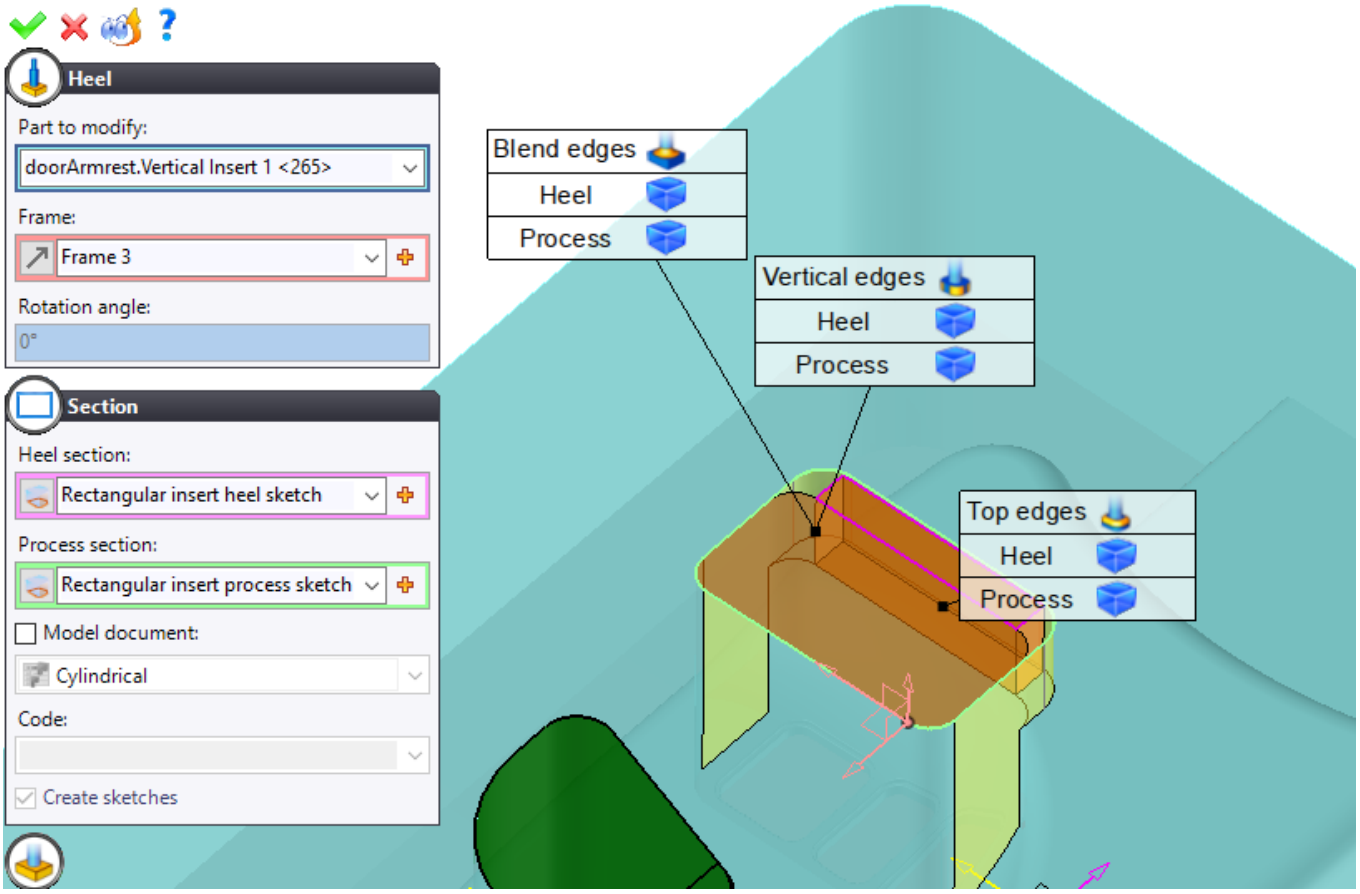
Note: We recommend that you rename the previously created sketches to identify them more easily for future manipulations.
To do this, open the Entities tree's **Sketches** folder and rename the four previously created sketches. For example:

- *Center insert heel sketch*
- *Center insert process sketch*
- *Rectangular insert heel sketch*
- *Rectangular insert process sketch*

- From the **Mold** tab, select the **Heel** command and adjust the parameters as indicated below.
Select **None** for the **top**, **vertical** and **blend edges**.



- Create a **heel** on the last insert of the cavity block using a rectangular profile.
  Select **None** for the **top**, **vertical** and **blend edges**.

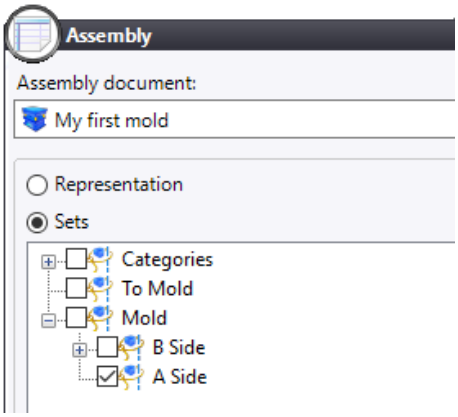


- **Save** the mold document.

Creating the bill of materials

- Right-click on the mold document's tab and select the **Bill of Material** command.
- Select the **Id.Qty.Des.PN.Mat.Mass** template from **Standard Templates - United States**, then click on to **confirm**.
- **Confirm** the default assembly document.

Note: You can create a bill of materials by sets/subsets by selecting the **Sets** option.



- Add the following two columns: **Manufacturer** and **Manufacturer Part Number**. To do this, select the **Columns** command from the **Bill of Material** tab. From the list on the left, select the **Manufacturer (General)** and **Manufacturer Part Number (General)** variables and click on the icon to move them to the list on the right.
- Click on to **confirm** the operation.

Start Page | My first mold | My first mold

ID	QTY	DESCRIPTION	PART NUMBER	MATERIAL	MASS	MANUFACTU...	MANUFACTURER NUMBER
	1						
1	1	Backing Plate		1.1730	59.6kg	Meusburger	F 60 / 346 496 / 46 / 1.1730
2	4	Buffer Plate for ...		1.1730	0.0kg	Meusburger	E 1500 / 20
3	1	Cavity block 1		Steel	10.3kg		
4	1	Cavity Plate wit...		1.1730	59.6kg	Meusburger	F 50 / 346 496 / 46 / 1.1730
5	1	Cavity Plate wit...		1.1730	75.4kg	Meusburger	F 50 / 346 496 / 66 / 1.1730
6	1	Cavity Plate wit...		1.1730	114.0kg	Meusburger	F 50 / 346 496 / 96 / 1.1730
7	4	Centering Bush...		1.7131	0.7kg	Meusburger	E 1160 / 42 × 160
8	1	Clamp Plate,Wi...		1.1730	53.5kg	Meusburger	F 15 / 346 496 / 36 / 1.1730
9	1	Clamp Plate,Wi...		1.1730	68.4kg	Meusburger	F 15 / 346 496 / 46 / 1.1730
10	1	Core block		Steel	9.3kg		
11	4	Countersunk H...		Class 10.9	0.0kg	Meusburger	E 1220 / 5 × 10
12	4	Cylinder Head ...		Class 12.9	0.0kg	Meusburger	E 1200 / 10 × 25
13	4	Cylinder Head ...		Class 12.9	0.1kg	Meusburger	E 1200 / 16 × 35
14	4	Cylinder Head ...		Class 12.9	0.1kg	Meusburger	E 1200 / 16 × 190
15	1	Ejector Base Plate		1.1730	18.5kg	Meusburger	F 85 / 346 496 / 218 / 22 / 1.1730
16	1	Ejector Retainin...		1.1730	14.4kg	Meusburger	F 80 / 346 496 / 218 / 17 / 1.1730
17	1	Guide Bush wit...		1.7131	0.4kg	Meusburger	E 1100 / 30 - 66
18	3	Guide Bush wit...		1.7131	0.4kg	Meusburger	E 1100 / 32 - 66
19	1	Guide Bush wit...		1.7131	0.5kg	Meusburger	E 1110 / 30 - 96
20	3	Guide Bush wit...		1.7131	0.5kg	Meusburger	E 1110 / 32 - 96
21	1	Guide Pillar wit...		1.7131	1.5kg	Meusburger	E 1000 / 30 - 46 / 155
22	3	Guide Pillar wit...		1.7131	1.6kg	Meusburger	E 1000 / 32 - 46 / 155
23	1	Insert Sliding		Steel	0.1kg		
24	2	Riser		1.1730	20.7kg	Meusburger	F 70 / 346 496 / 62 / 96 / 1.1730
25	1	Vertical Insert 1		Steel	0.3kg		
26	1	Vertical Insert 2		Steel	0.2kg		
27	1	Vertical Insert 3		Steel	0.0kg		
28	1	Vertical Insert S...		Steel	0.1kg		
29	1	Vertical Insert S...		Steel	0.2kg		

- **Save** the bill of materials.

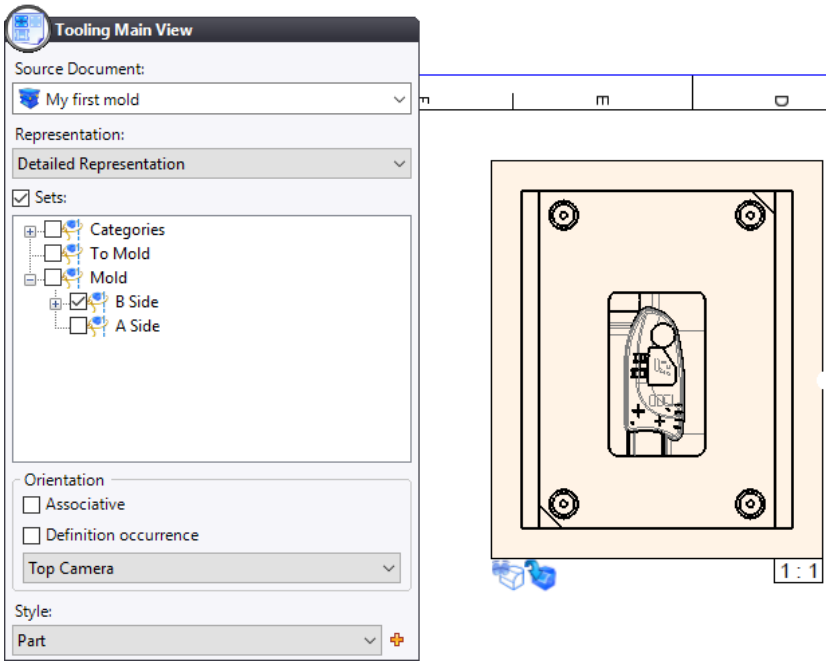
Creating the drafting document

A Side and B Side views

- Right-click on the mold document's tab and select the **Drafting** command. Select the **Assembly A1 ISO Landscape** template from **Standard Templates - United States**.

Note: The **Tooling View** command is automatically displayed and you can select the set to be added. This command is available in the **Mold** tab.

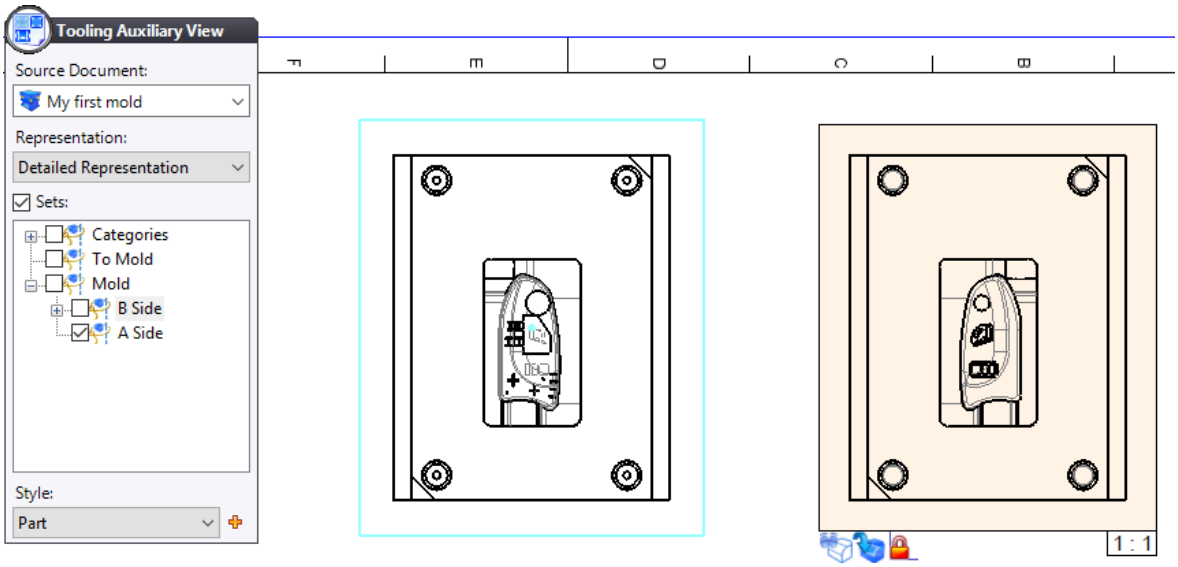
- Check the **Sets** box, then open the **Mold** set and check the **B Side** subset.



- Place the view in the drawing and click on to **confirm**.

You can then create an auxiliary view for the A side view.

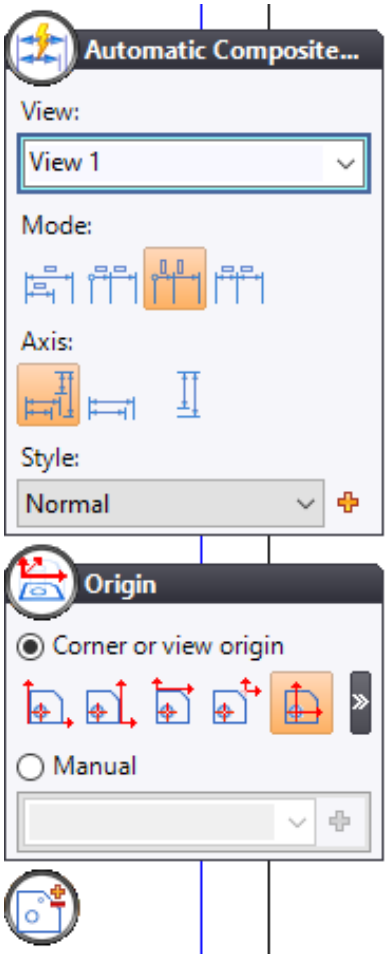
- Uncheck the **B Side** subset and check the **A Side** subset.

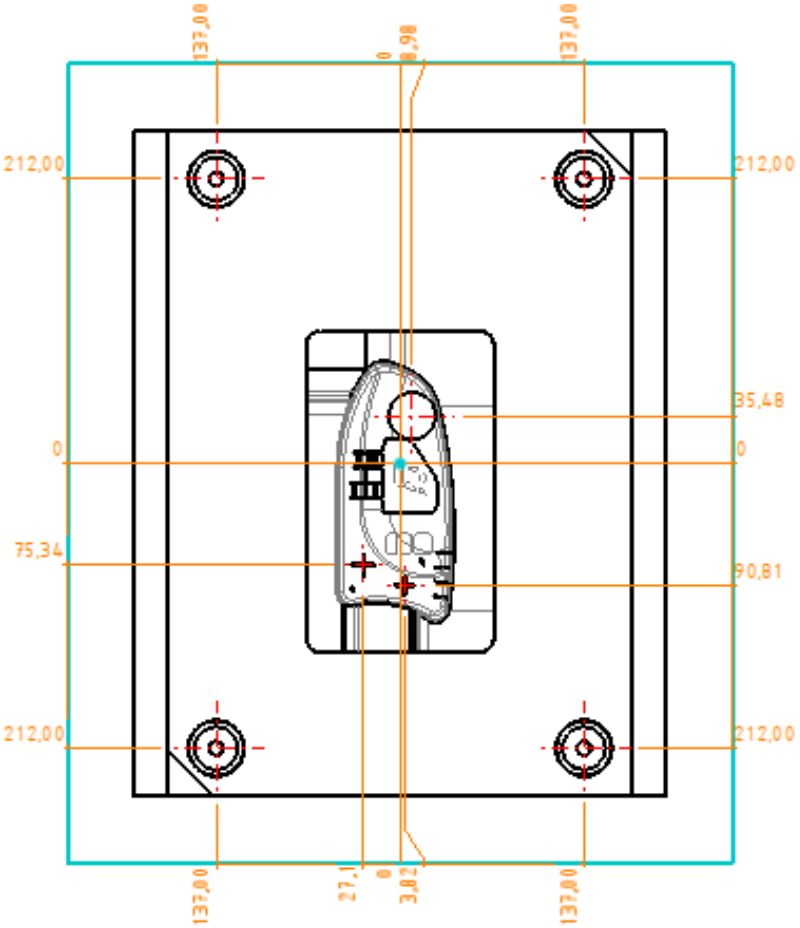


- Place the view in the drawing and click on to **confirm**.
- **Close** the dialog box.

Detailing the views

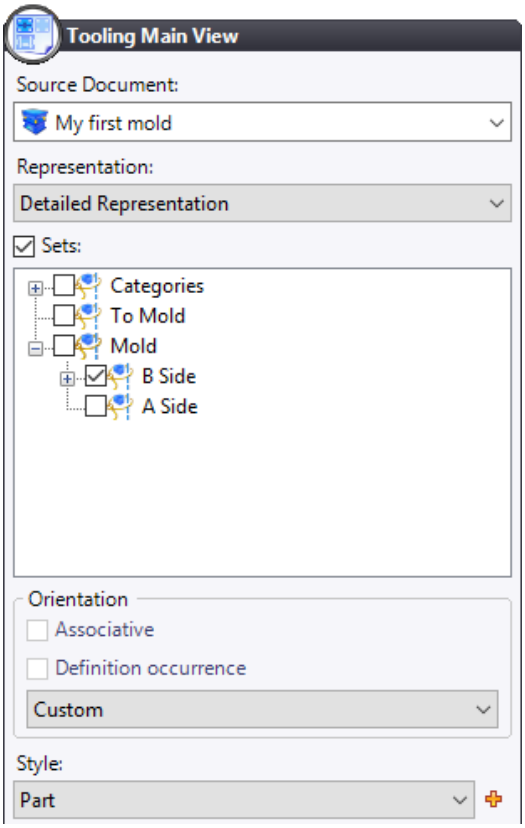
- Right-click on the main view, select the **Automatic Axes** command, and then click on to **confirm**. Repeat the operation for the auxiliary view.
- For each view, select the **Automatic Composite Dimensions** command from the **Detailing** tab and adjust the parameters as shown below.





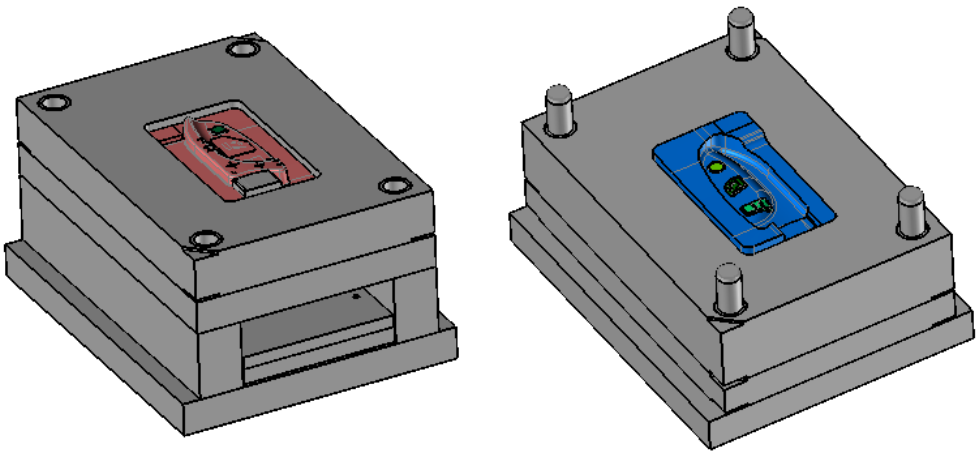
View with custom orientation

- From the **Mold** tab, select the **Tooling View** command.
- Check the **Sets** box, then open the **Mold** set and check the **B Side** subset.



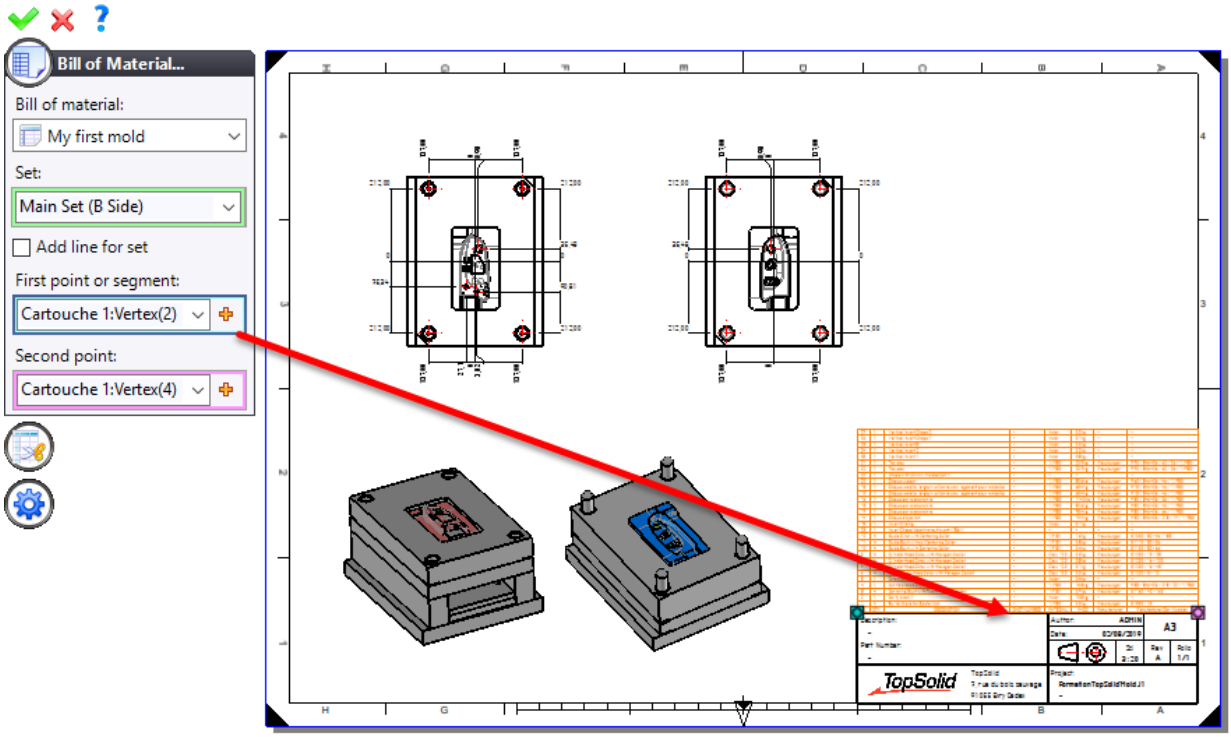
- Place the view in the drawing, regardless of orientation.
- From the **View** tab, deactivate the **Snapped Camera** option located on the right of the icon bar.

- Use the mouse wheel to orientate the view as you wish. You can also adjust the visibility of the edges and the render mode.
- Click on to **confirm** the operation.
- Repeat the previous operations for the **A side**.

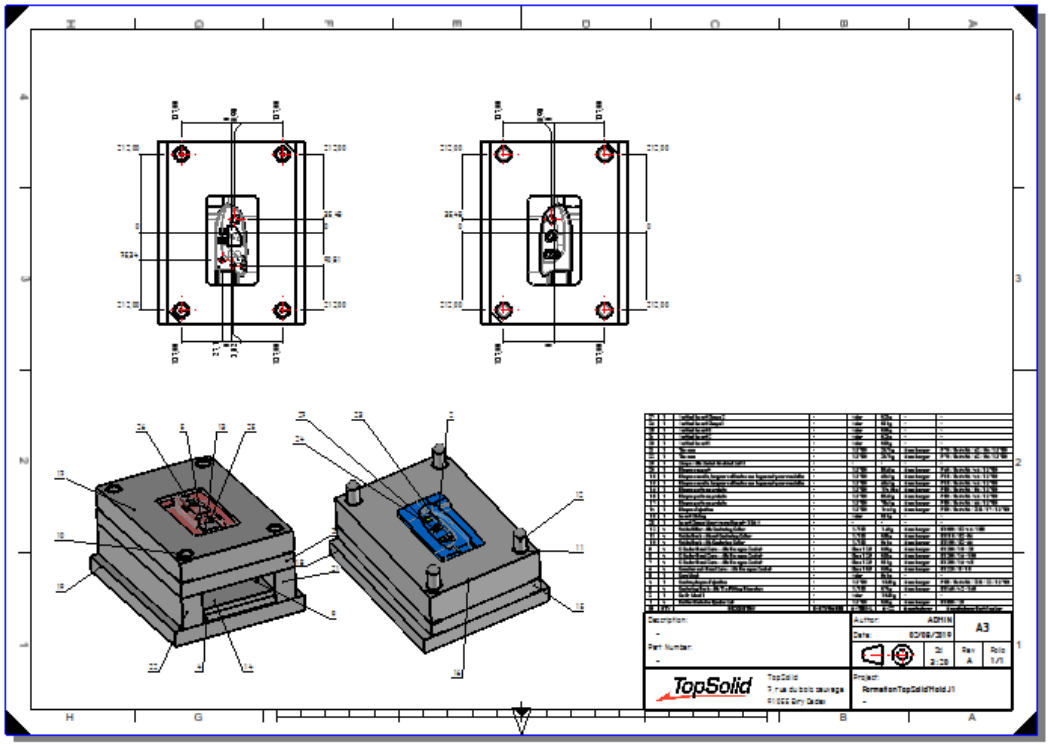


Inserting the bill of materials and automatic indexing

- From the Project tree, drag and drop the BOM document into the drafting document.
- In the **Set** field, select **Main Set** from the drop-down list.
- In the **First point or segment** field, click directly on the title block's upper line in the graphics area.



- Click on to **confirm** the operation.
- From the **Detailing** tab, select the **Automatic BOM Index** command. Select the **B Side** view.
- Click on to **confirm** the operation.
- Repeat the previous operation for the **A side**.

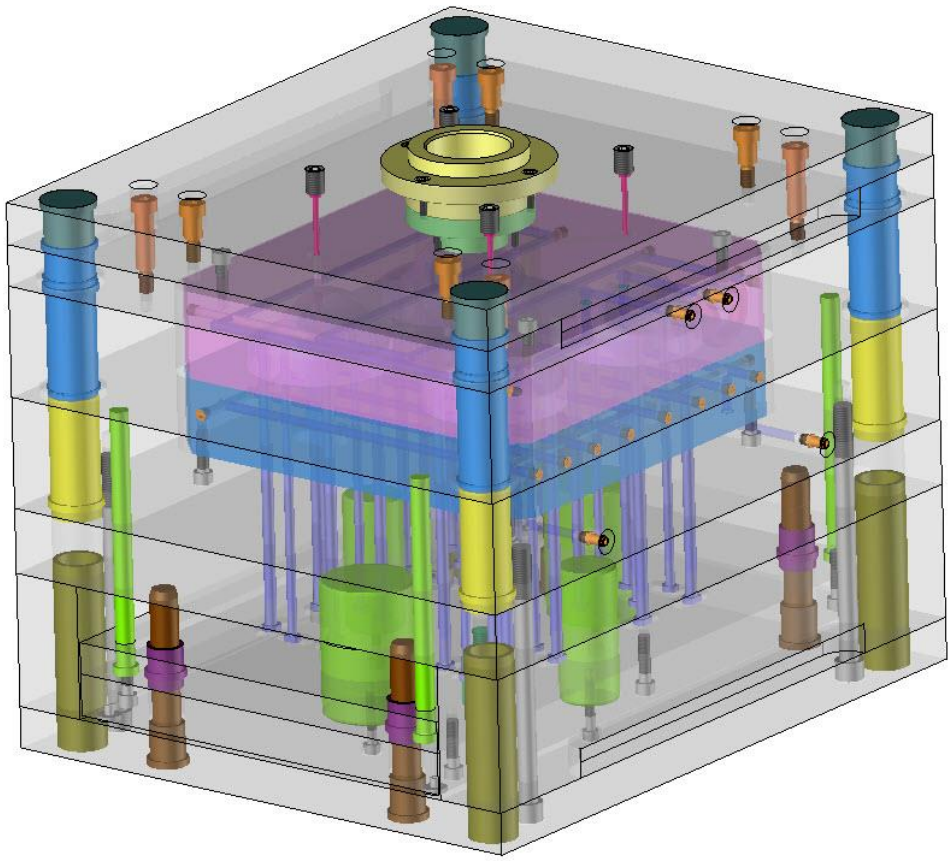


Multi-cavity Mold

In this second part of the guide, you will learn how to manage a multi-cavity mold and insert various components using the processes, wizards and families.

Concepts addressed:

- Including an assembly resulting from the Split process
- Multi-cavities and union
- User mold base
- Cavity process
- Positioning pins
- Inserting components (pins, screws, flange sets…)
- Cooling circuit
- Runner circuit
- Drafting and bill of materials



Including an assembly resulting from the Split process

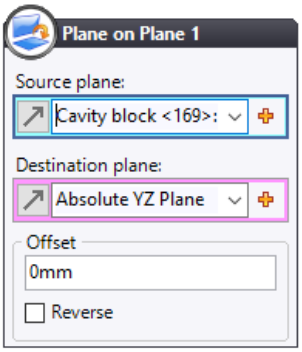
Including the assembly in a new mold document

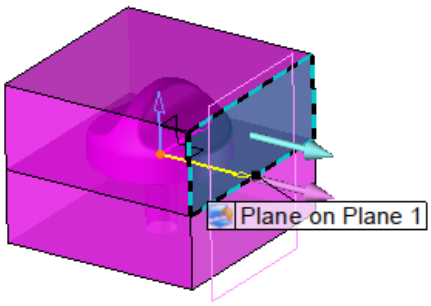
- From the Project tree, open the *Ex02 - Multi-cavity mold* folder, right-click on the *Plug* assembly document and select the **Mold** command. Select **Blank Template** and click on to **confirm**.
- From the Project tree, rename the new mold document *My second mold.*

Repositioning the core and cavity blocks

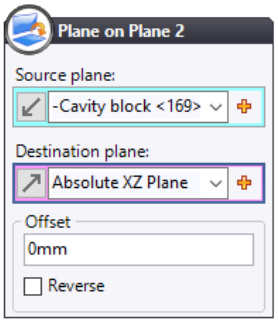
As in the previous exercise, during the inclusion, **TopSolid** positions the assembly's opening frame (from the Split) on the absolute frame of the new mold document by default.
In this exercise, we want to position the core and cavity blocks differently.

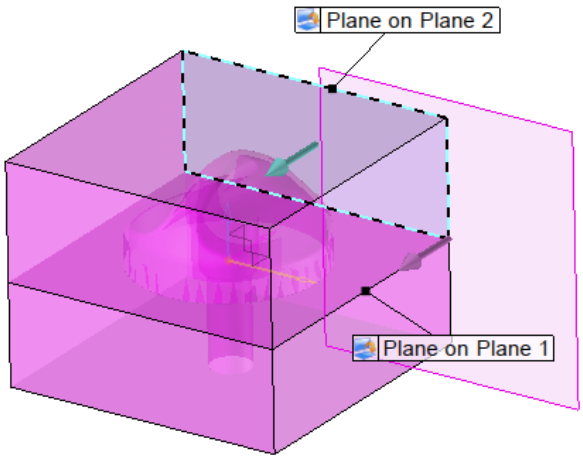
- From the Entities tree, open the **Frames** folder, right-click on **Absolute Frame** and select the **Show** command.
- Right-click on one of the assembly shapes and select the **Edit Positioning** command from the **Core Cavity Blocks Inclusions** section.
- Right-click on the **Frame on Frame 1** positioning constraint's label and select the **Delete** command.
- From the **Assembly** tab, select the **Plane on Plane** command and pin the dialog box using the icon.
  - **Source plane**: Select the side face as shown below.
  - **Destination plane**: Select **Absolute YZ Plane** from the drop-down list.



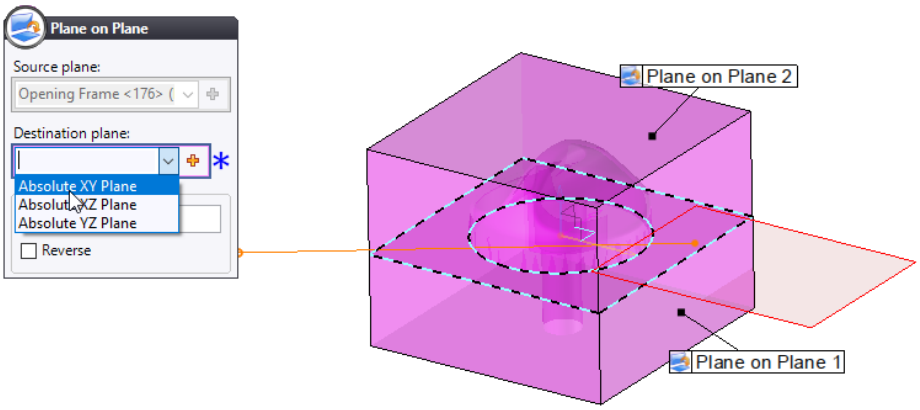


- For the second **Plane on Plane** constraint, do the following:
  - **Source plane**: Select the rear face as shown below.
  - **Destination plane**: Select **Absolute XZ Plane** from the drop-down list.

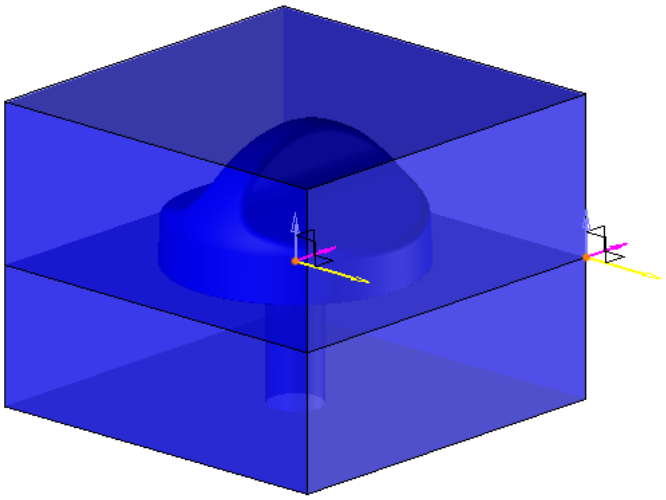




- For the last **Plane on Plane** constraint, do the following:
  - **Source plane**: Select the face of the parting line as shown below.
  - **Destination plane**: Select **Absolute XY Plane** from the drop-down list.



The absolute frame will represent the mold zero point. It is located this way because we will subsequently repeat the core and cavity blocks to make a total of four.

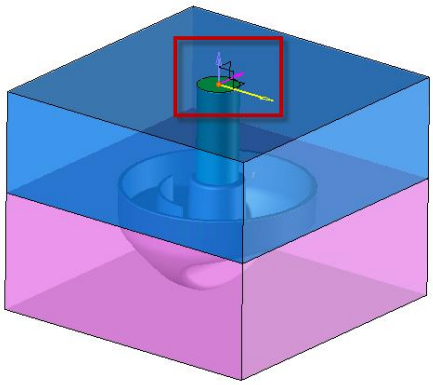


- **Confirm** positioning 1.

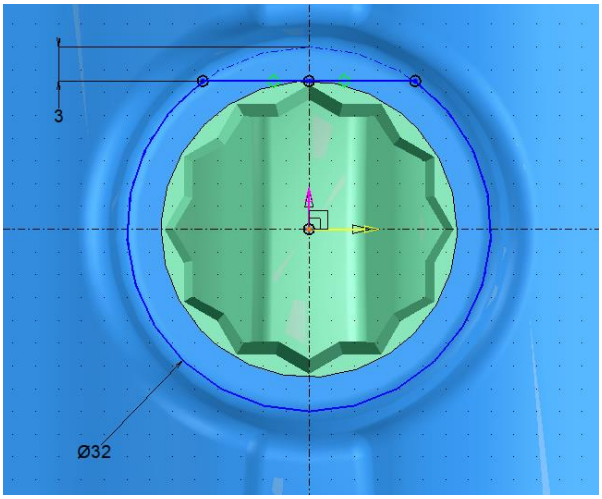
Creating the heel on the insert

Heel and process sketches

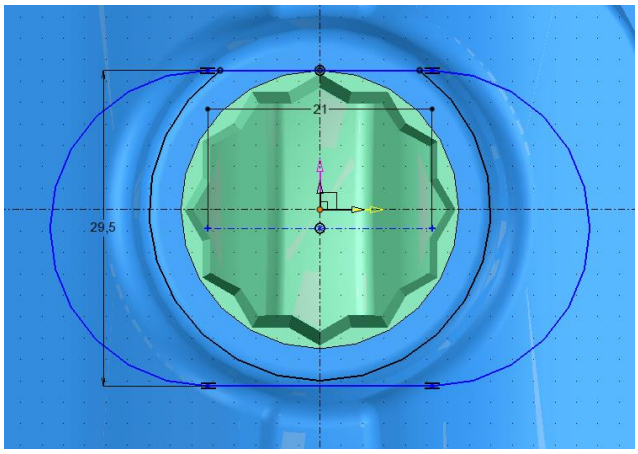
- Create a frame on the core block's bottom face, with the origin centered on the insert.



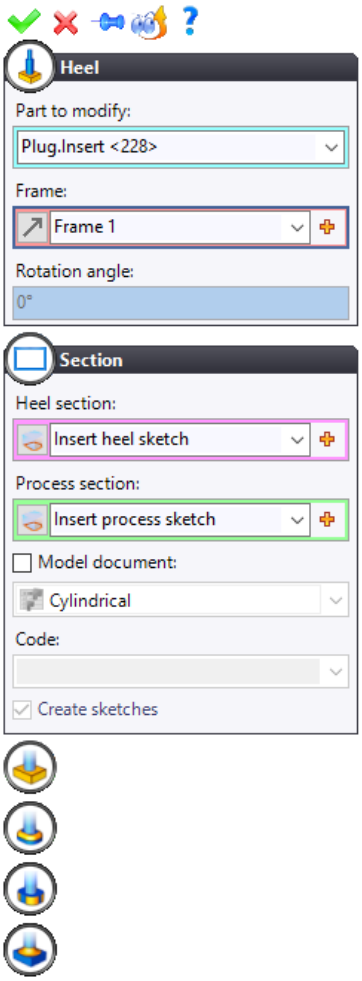
- Draw the following new sketch on the XY plane of the previously created frame. To do this, select the **Half Moon** standard profile and adjust the following values:
  - **Circle Diameter**: *32mm*
  - **Distance Between Circle Center and Flat**: *3mm*

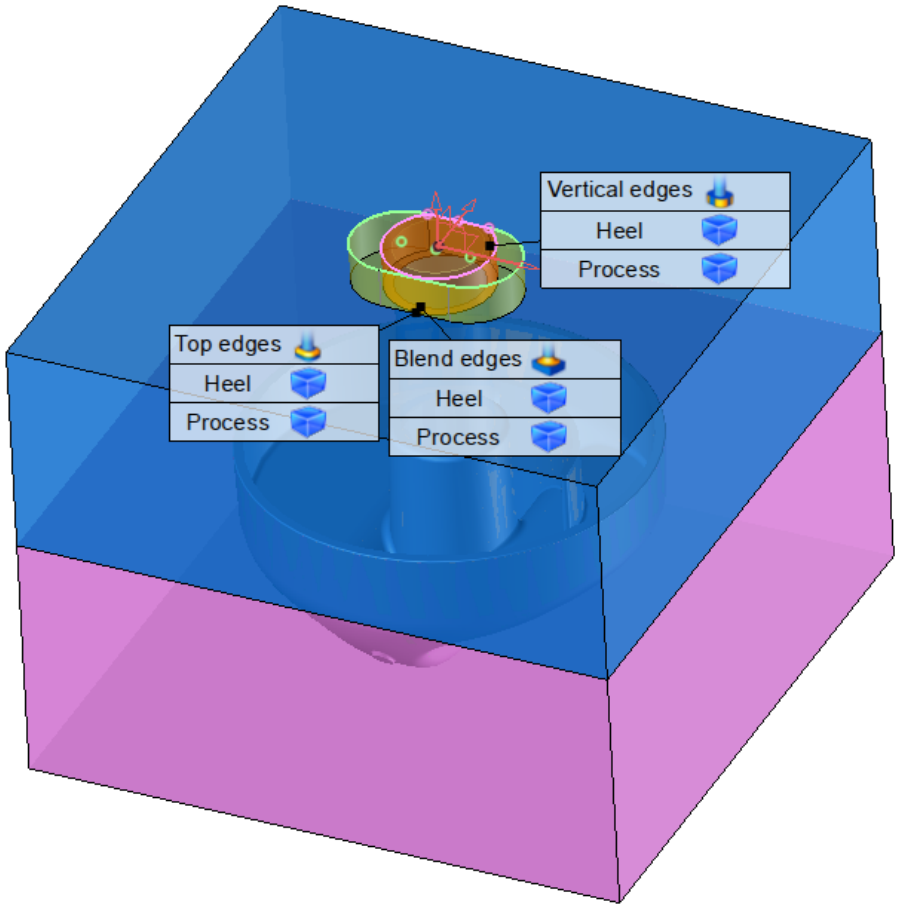


- Draw the following new sketch on the same XY plane. Select the **Obround** standard profile and adjust the following values:
  - **Internal Length**: *21mm*
  - **Thickness**: *29.5mm*



- From the **Mold** tab, select the **Heel** command and adjust the parameters as indicated below.





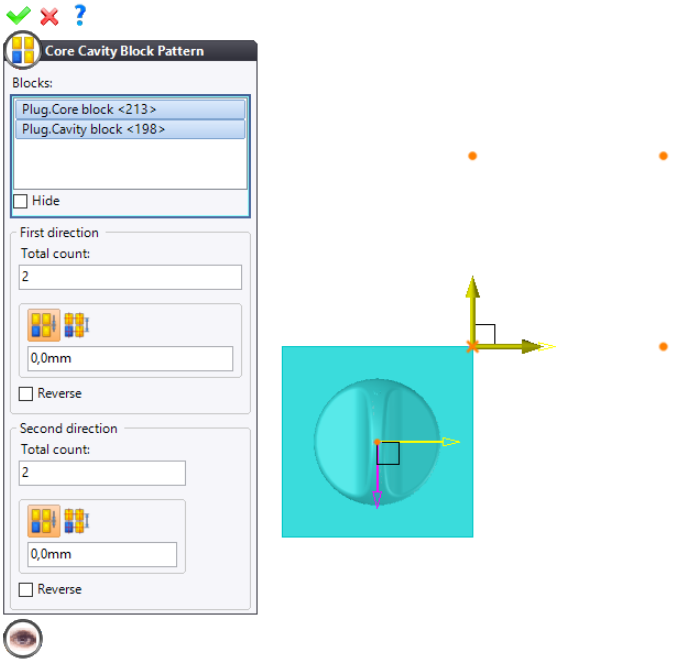
- Click on to **confirm** the operation.

Creating the multi-cavities

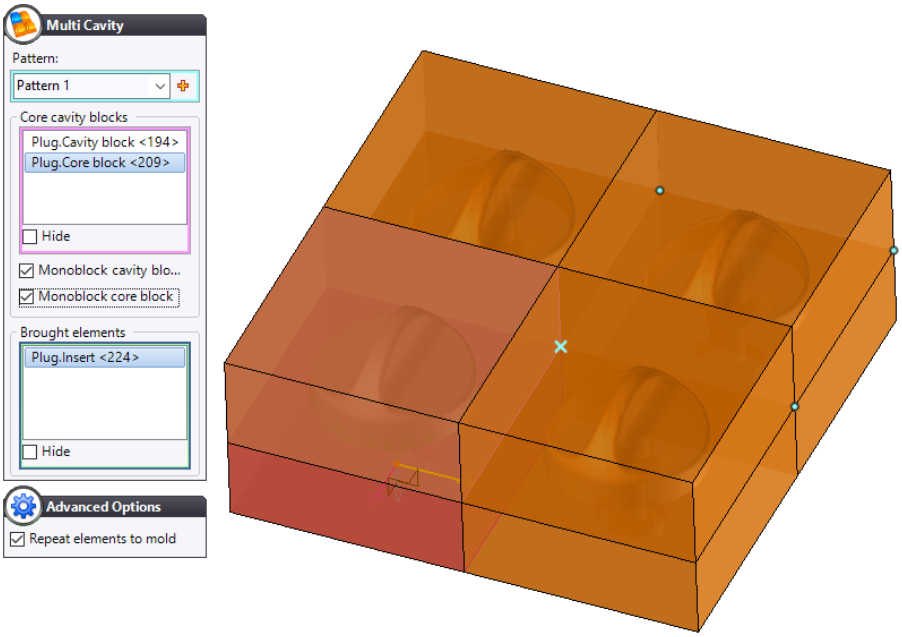
- From the **Mold** tab, select the **Multi Cavity** command.
- Click on the icon to select the **pattern** to be used.

There are several repetition modes: circular, linear, symmetrical, etc.
For this exercise, we will choose the **core cavity block pattern** which allows you to perform a linear matrix repetition with automatic calculation of the step. You only have to indicate the direction of repetition for the axes and the total count of core and cavity blocks on each axis.

- Select **Core Cavity Block Pattern** and adjust the following parameters.



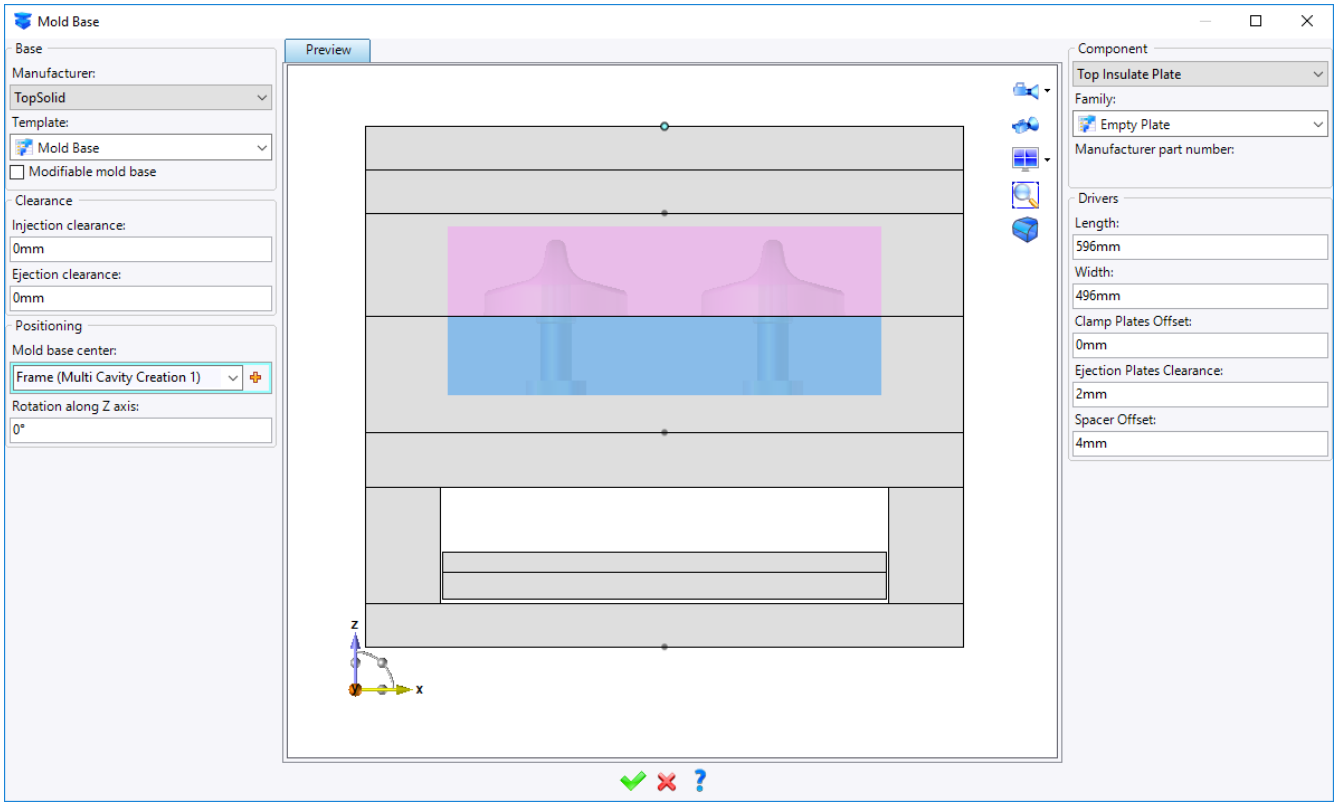
- Click on to **confirm** the operation.
- Complete the multi-cavity dialog box as shown below.



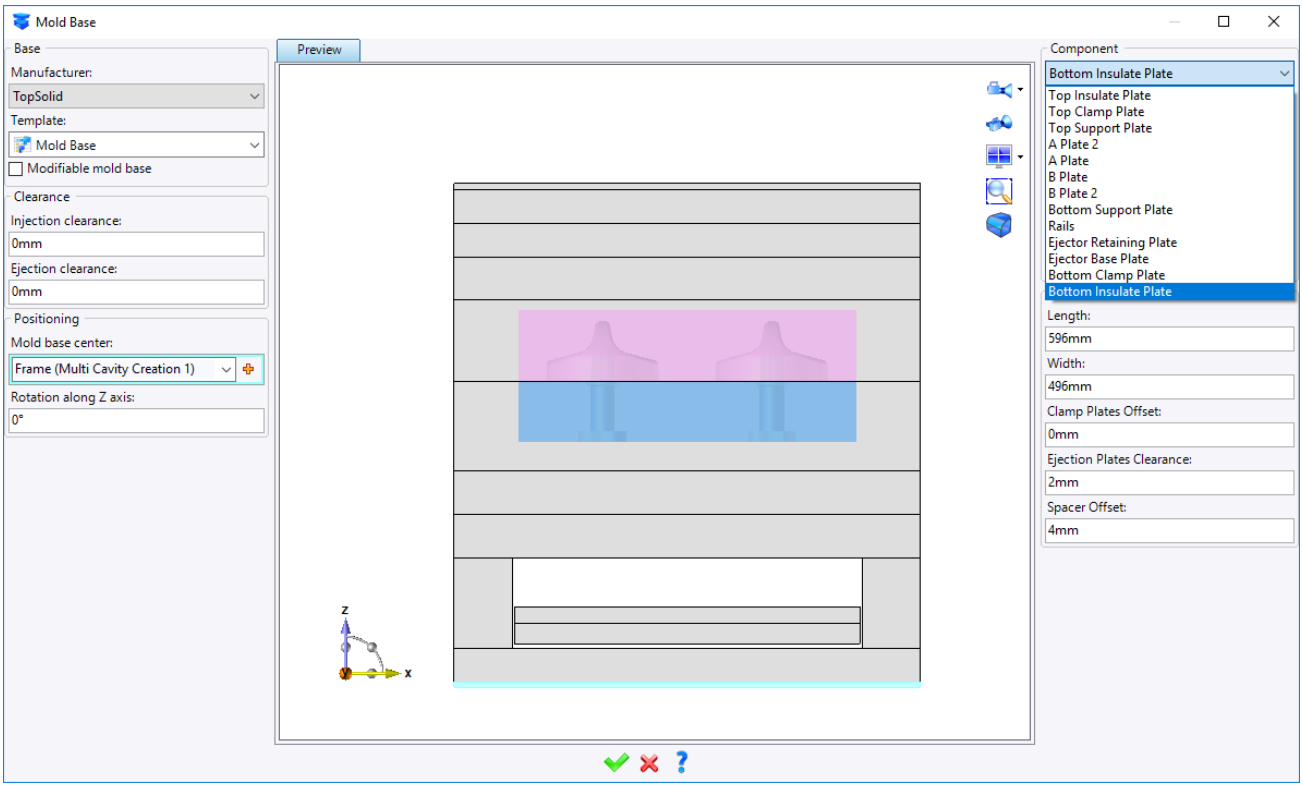
Note: The **Monoblock cavity block** and **Monoblock core block** options allow you to unite the cavity blocks into one single block, and unite the core blocks into another single block.

Including the user mold base

- From the **Mold** tab, select the **Mold Base Inclusion** command.
- Adjust the parameters as shown below.



Reminder: The parameters on the right of the dialog box are the parameters for a plate of the mold base. You can select the plate from the **Component** drop-down list.

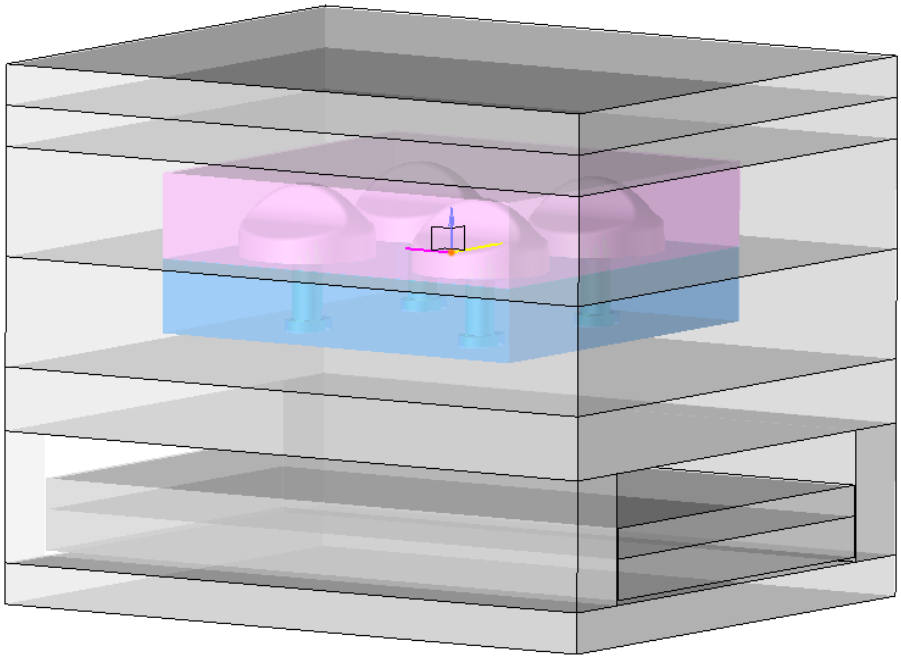


You can also select the plate by clicking directly on the plate in the graphics area.

- Adjust the following parameters for each plate.

Component	Family	Thickness	Width	Material
Top Insulate Plate	Empty Plate			
Top Clamp Plate	Clamp Plate	36mm	-	Steel
Top Support Plate	Support Plate	36mm	-	Steel
A Plate 2	Empty Plate			
A Plate	Core Cavity Plate	96mm	-	Steel
B Plate	Core Cavity Plate	96mm	-	Steel
B Plate 2	Empty Plate			
Bottom Support Plate	Support Plate	56mm	-	Steel
Rails	Rails	115mm	62mm	Steel
Ejector Retaining Plate	Ejector Retaining Plate	27mm	-	Steel
Ejector Base Plate	Ejector Base Plate	36mm	-	Steel
Bottom Clamp Plate	Clamp Plate	36mm	-	Steel
Bottom Insulate Plate	Empty Plate			

- Click on  to **confirm**.

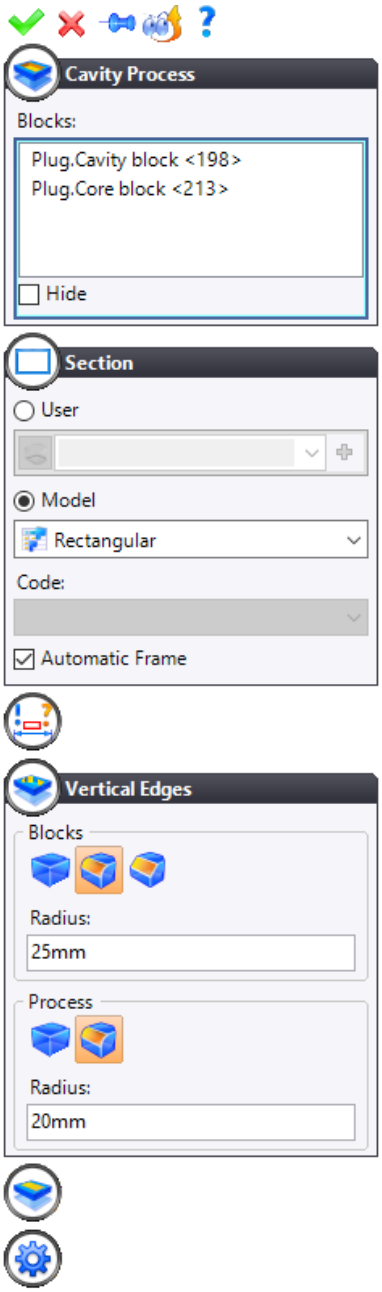


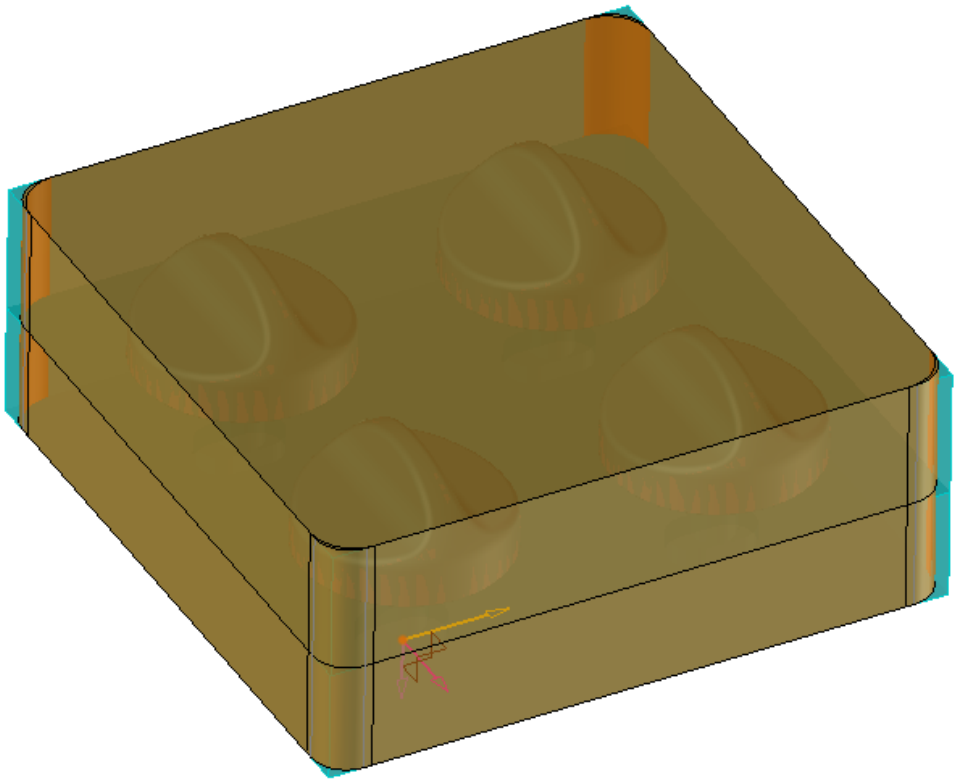
Reminder: At any time you can return to the mold base dialog box by editing the command. From the Operations tree, you only have to right-click on the **Mold Base Inclusion** operation and select the  **Edit** command.

Creating the cavity process

As in the previous exercise, we will apply the **Cavity Process** command only once, for the core and cavity blocks. You can apply the command several times if the values are not the same for both core and cavity blocks.

- Hide the mold base by clicking on the icon in the graphics area.
- Select the **Cavity Process** command. Select the **Rectangular** model, then enter *25mm* for the block radius and *20mm* for the process radius.



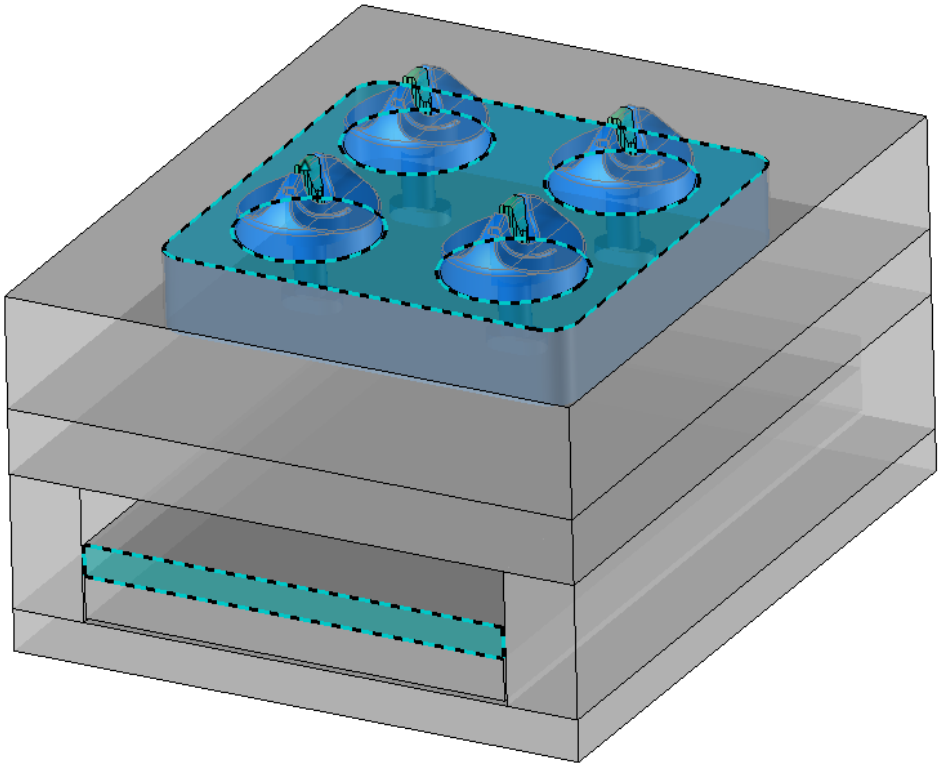


- Click on to **confirm** the operation.
- **Show** the mold base.

Positioning pins

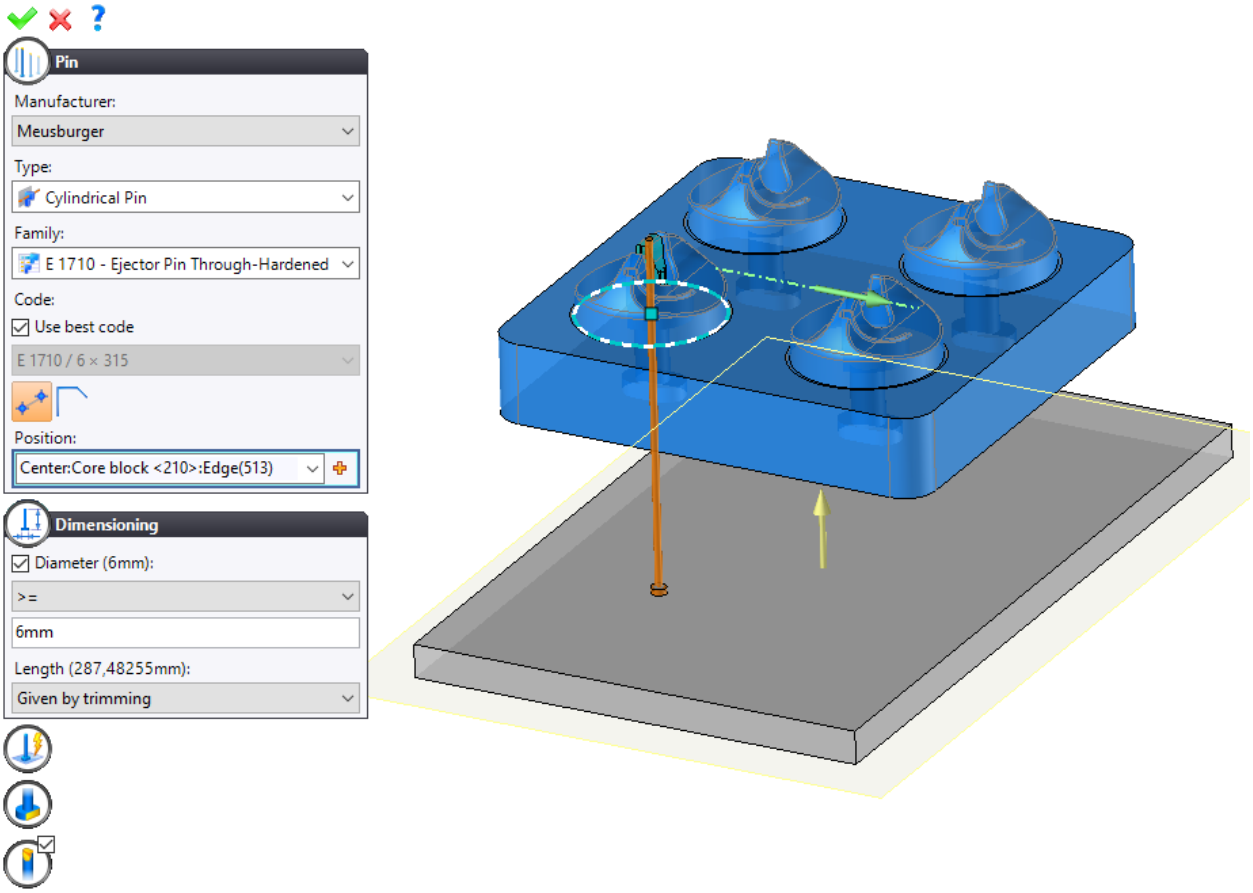
Positioning on point

- To help you position the pin, hide the **A Side** subset by clicking on the icon in the graphics area.
- Select the core block, the first insert and the ejector retaining plate, then right-click and select the **Show Only** command.



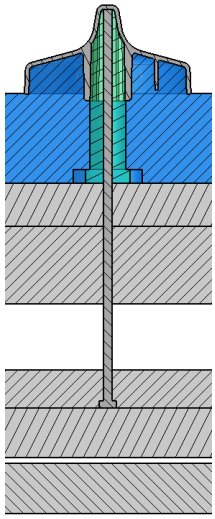
Only three parts remain visible on the screen. In addition, two new icons have appeared at the top right of the graphic area.

- From the **Mold** tab, select the **Pin** command.
- For the positioning, select the **Point** mode, then select the part's circular edge to detect the center of the part.
- Click on the **Dimensioning** icon and enter a **diameter >=** *6mm*. **TopSolid** will select the most appropriate component.

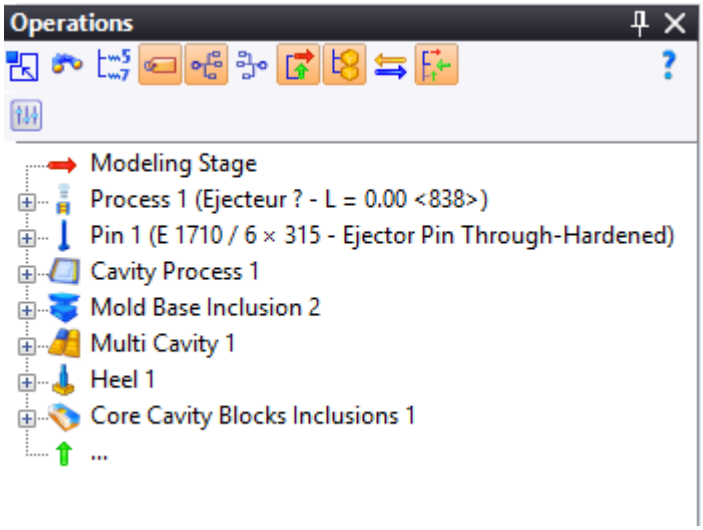


Note: The positioning plane is automatically detected on the top face of the ejector base plate. However, if you want to change the height position, trim with other elements or force a keying on the pin, you can modify the parameters in the **Positioning**, **Keying** or **Trimming** options.

- Once the command has been confirmed, **confirm** the default process.



In the Operations tree, we find the pin creation operation and the process creation operation.

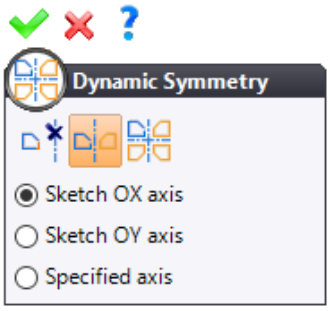


Note: It is recommended to defer the pin processes at the end of the design in order to accelerate the regeneration times due to the synchronization between the different parts of the tooling.

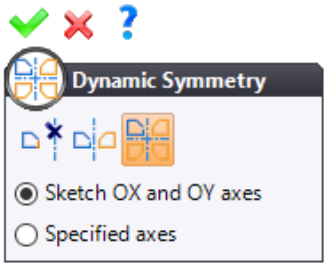
- Delete the pin process.
- From the **Assembly** tab, deactivate the **Automatic Process** mode.

Positioning on sketch

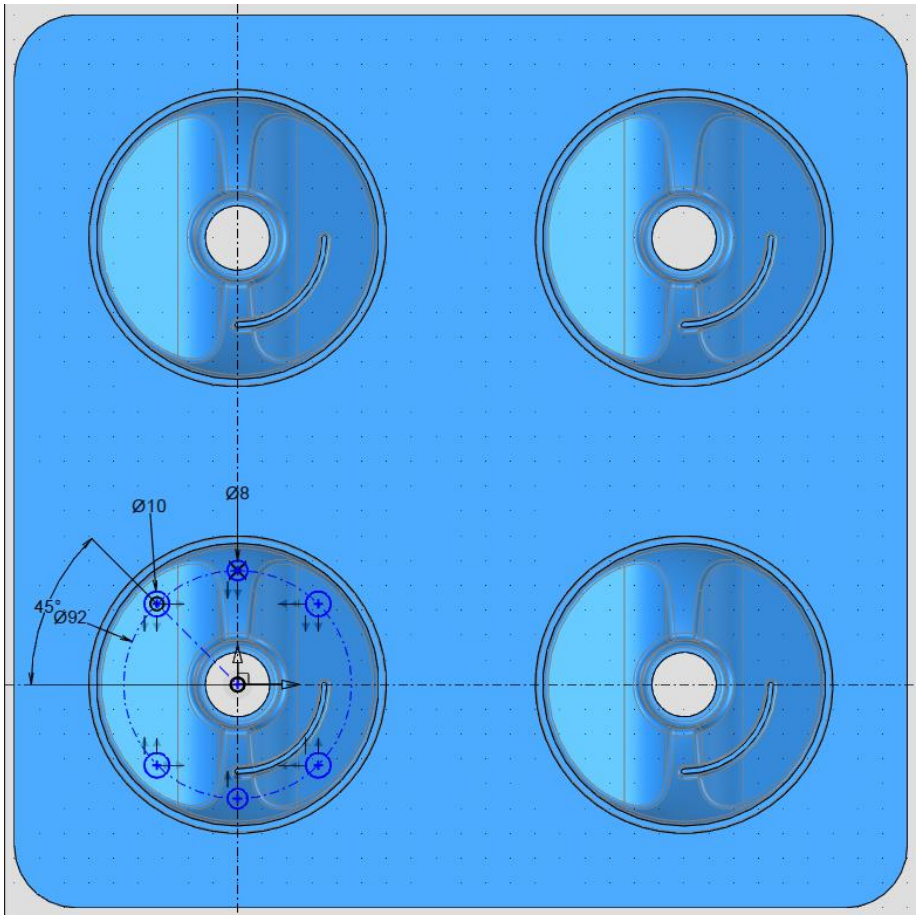
- Create the following sketch on the **absolute XY plane**.
- Reposition the sketch's origin in the center of the pin.
- Enable the **dynamic symmetry** with respect to the X axis of the sketch.



- Create a **circle** of Ø8mm at the intersection of the sketch's Y axis and the Ø92mm circle.
- Enable the **double dynamic symmetry** with respect to the X and Y axes of the sketch.

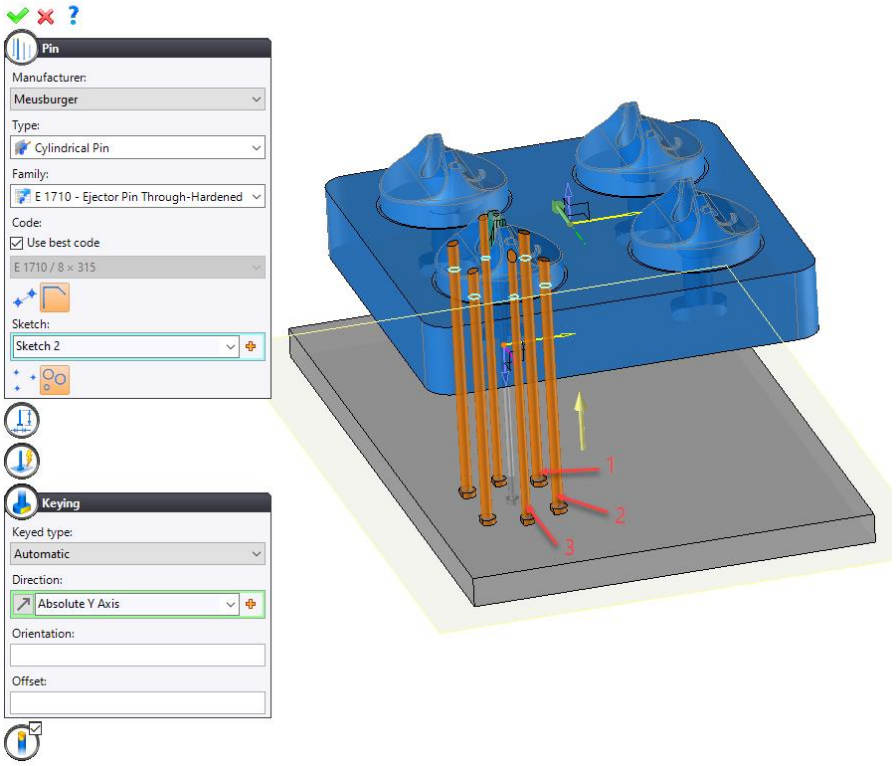


- Create a **circle** of Ø10mm at the intersection of the 45° segment and the Ø92mm circle as shown below.



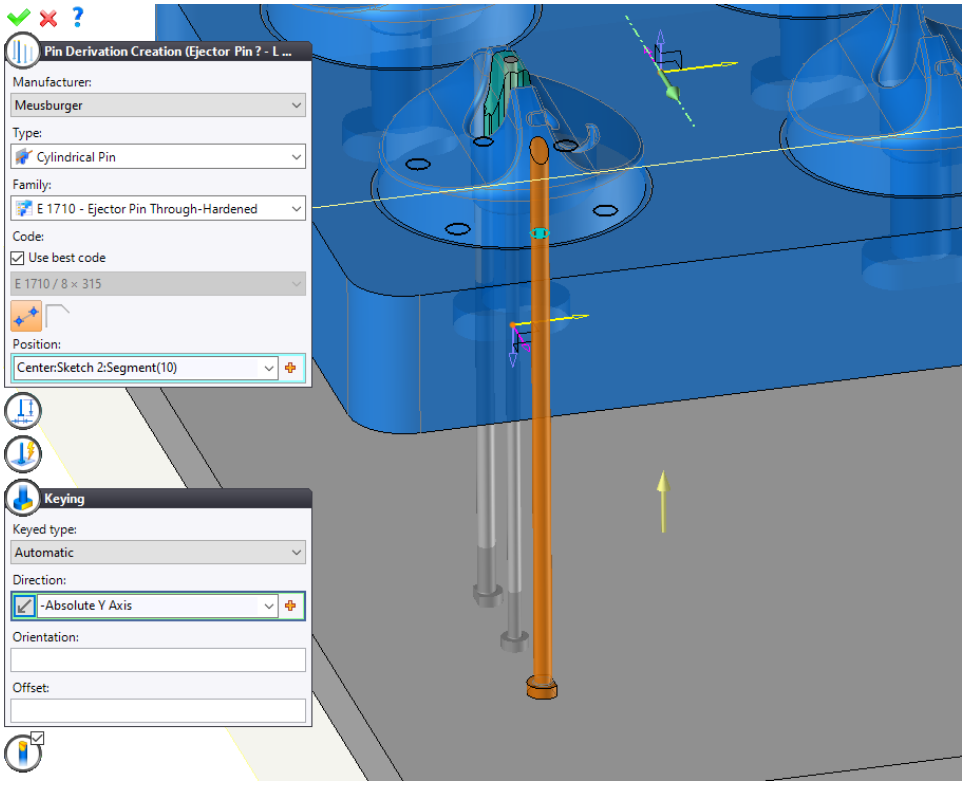
- **Confirm** the sketch.

- From the **Mold** tab, select the **Pin** command.
- For the positioning, select the **Sketch** mode and select the previously created sketch.
- In the **Keying** option, select **Automatic** as the keyed type.



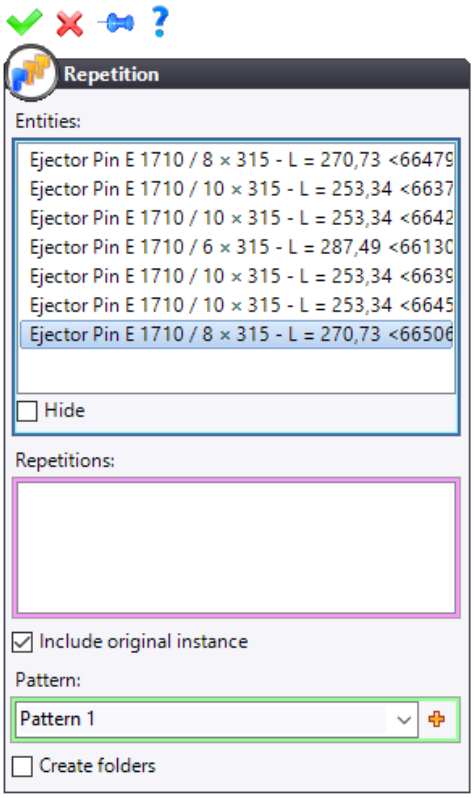
Note: **TopSolid** automatically assigns the pin diameters according to the drawn circles.

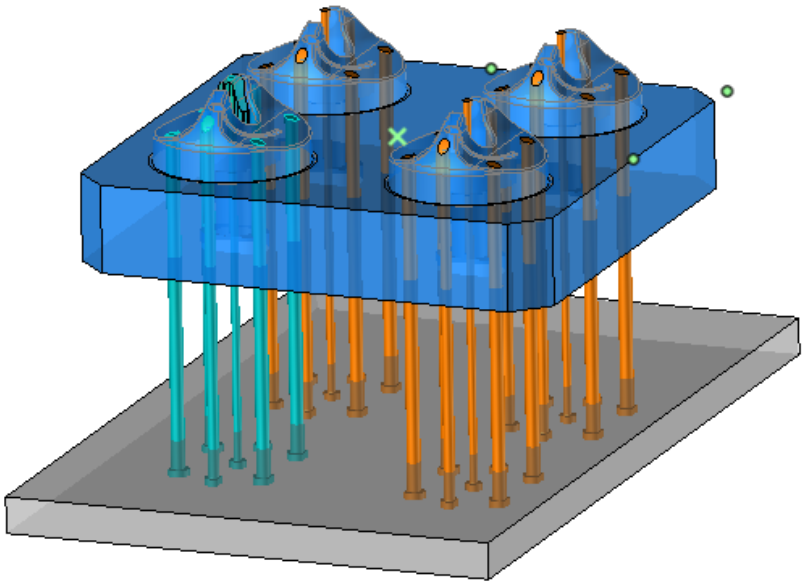
- Click to **confirm**.
- Edit pin 3 and reverse the keying direction to obtain two identical Ø8mm pins.
- Repeat the procedure for pins 1 and 2.



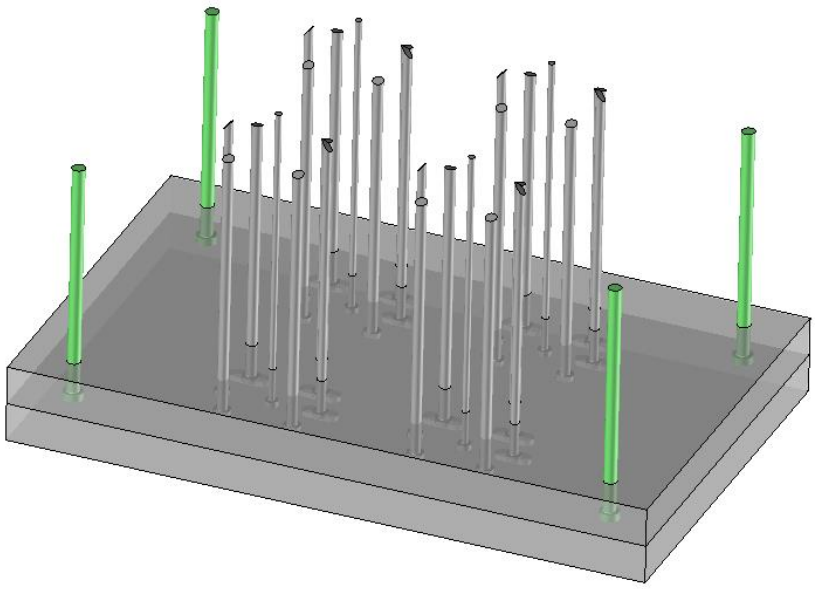
Repeating the pins

- From the **Construction** tab, select the **Repetition** command and select the pins to be repeated.





- For the pattern, reuse **pattern 1** (core cavity block pattern) available in the drop-down list.
- Add four pins as shown below (center distances X=310, Y=550, pins Ø16mm).



Creating the processes

Note: The guiding length of the pins in the blocks is calculated in relation to the maximum point of the pin (by default, 1.5 times the diameter).

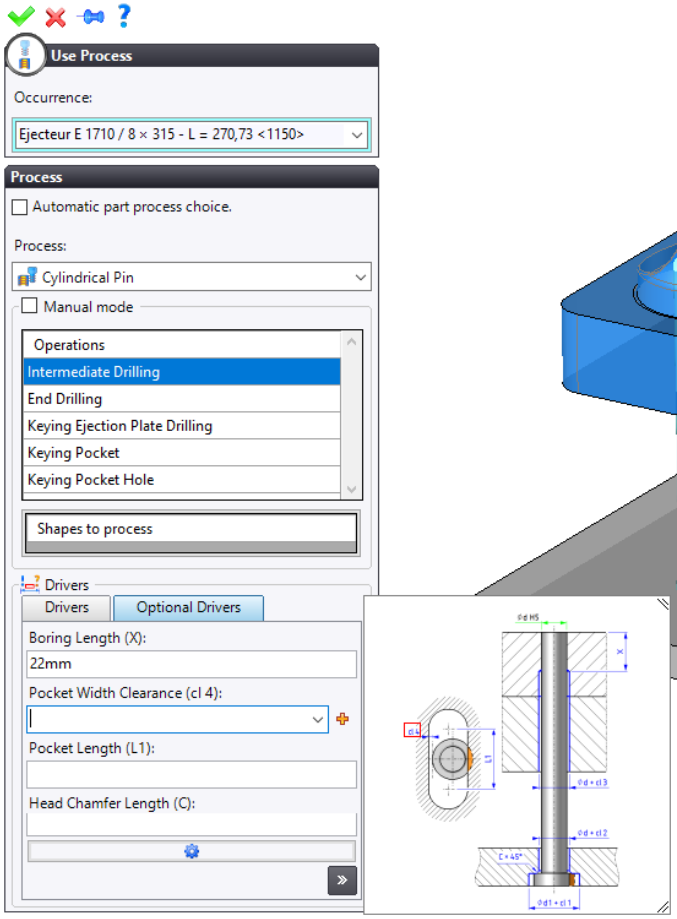
There are three ways to assign processes to the pins.

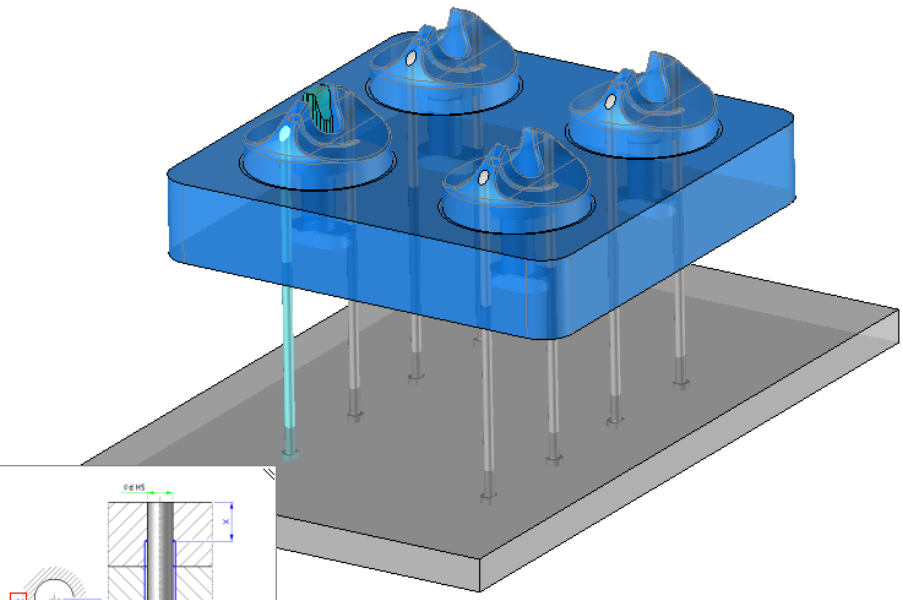
1- Defining a process on a pin and reproducing it on a group

- From the Parts tree, display only the Ø8mm pins.

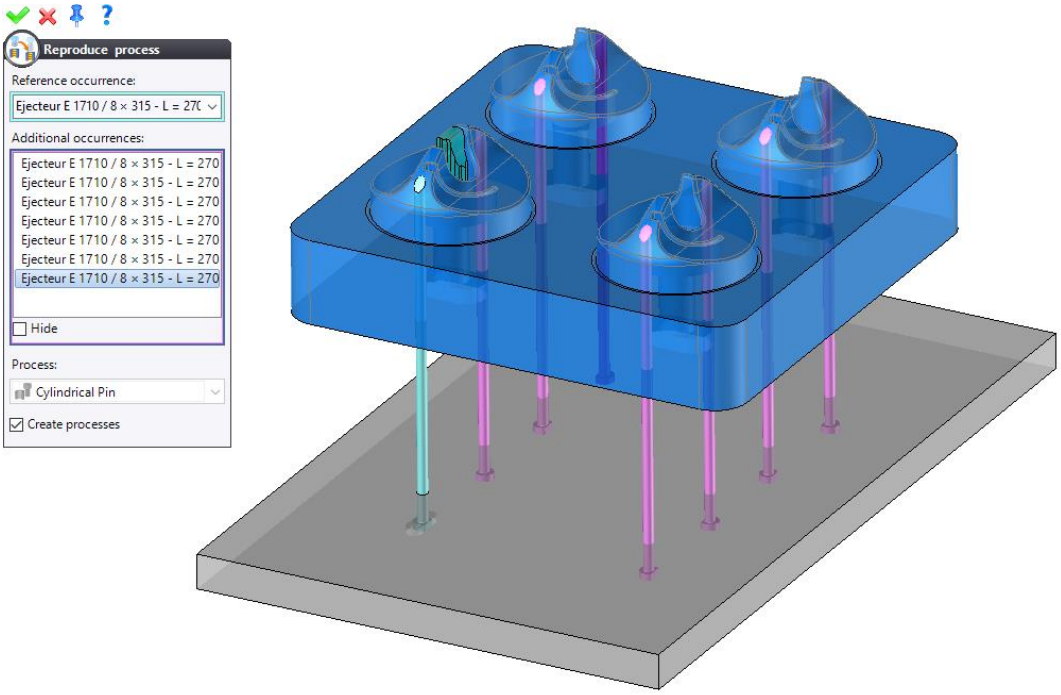
Note: Due to the shape at the end of the pin, the effective boring length should be 22mm.

- From the **Modeling** tab, select the **Use Process** command.
- Select one of the pins.
- In the **Optional Drivers** tab, enter a **boring length (X)** of *22mm*.
- **Confirm** the process.





- From the **Modeling** tab, select the **Reproduce Process** command.
- Check the **Create processes** box.
- Select the previously used Ø8mm pin as the **reference occurrence**, then select the other Ø8mm pins as **additional occurrences**.



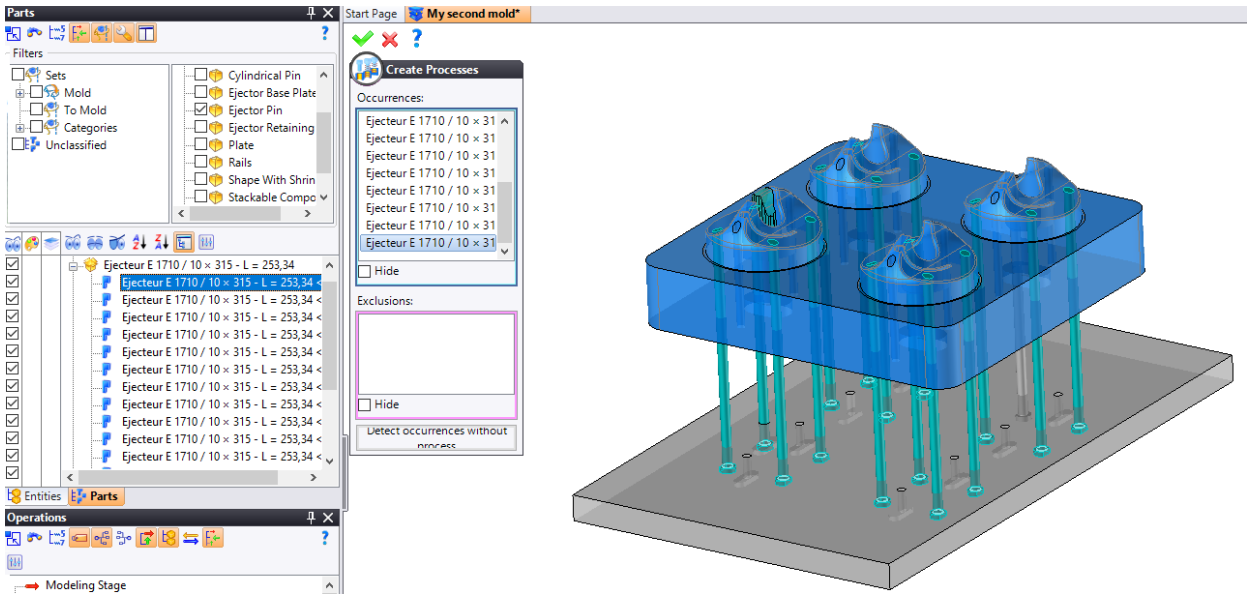
- **Confirm** the process.

2- Creating a process on several pins

- From the Parts tree, display only the Ø10mm pins.

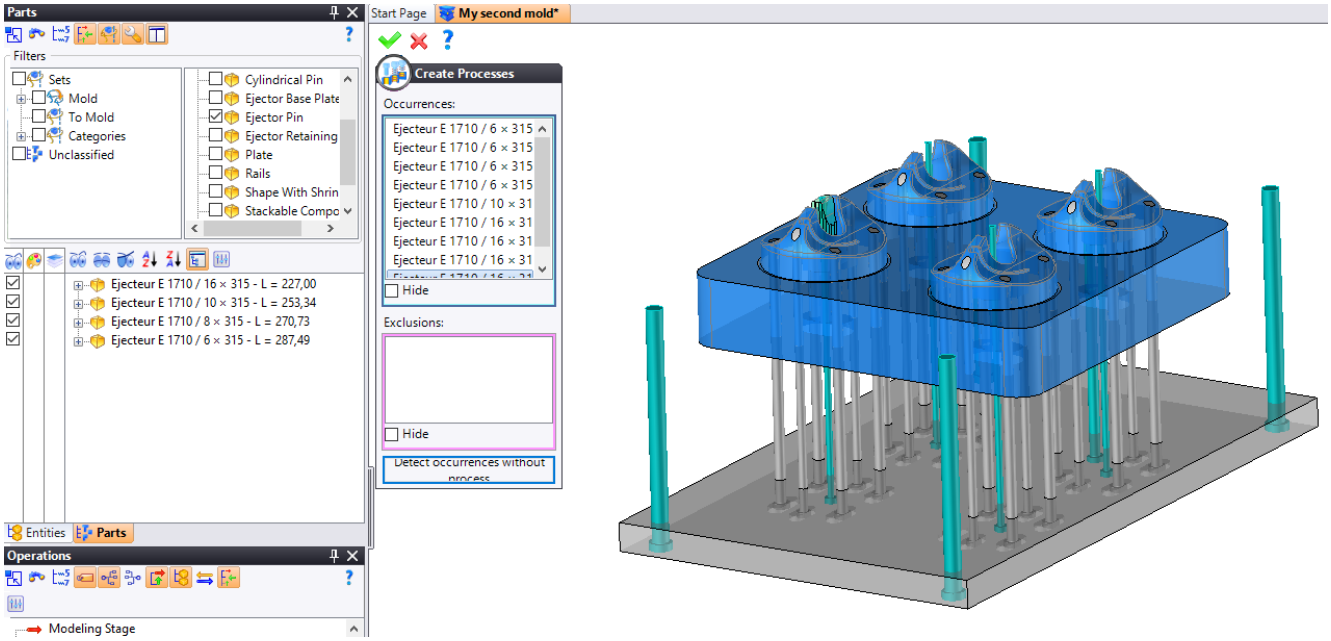
Note: Due to the shape at the end of the pin, the effective boring length should be 18mm.

- From the **Modeling** tab, select the **Create Processes** command.
- From the Parts tree, click on the folder containing the Ø10mm pins and click on to **confirm**.
- In the **Optional Drivers** tab, enter a **boring length (X)** of *18mm*.
- Check the **Apply to all** box.
- **Confirm** the process.

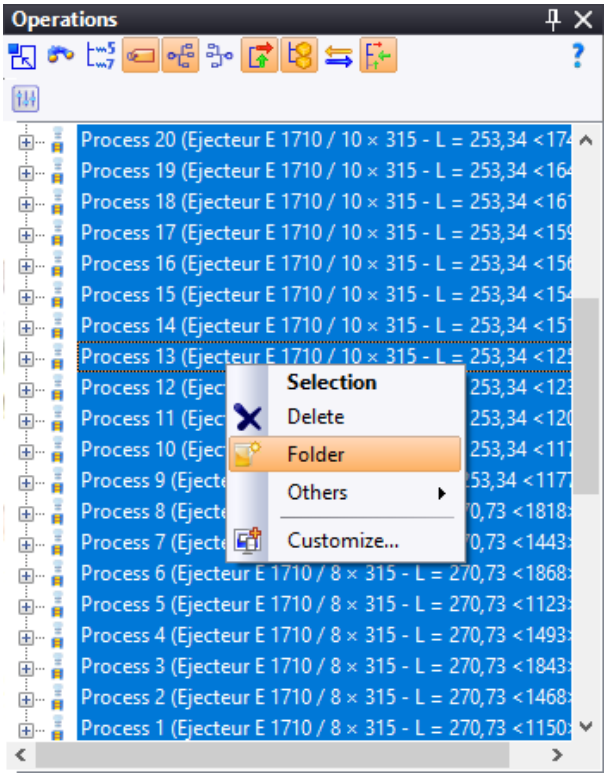


3- Creating a process on the pins without process

- From the Parts tree, display all pins.
- From the **Modeling** tab, select the **Create Processes** command.
- In the dialog box, click on the **Detect occurrences without process** button and click on to **confirm**.
- **Confirm** the process.



- From the Operations tree, select all the pin processes and create a folder named *Pin processes*.



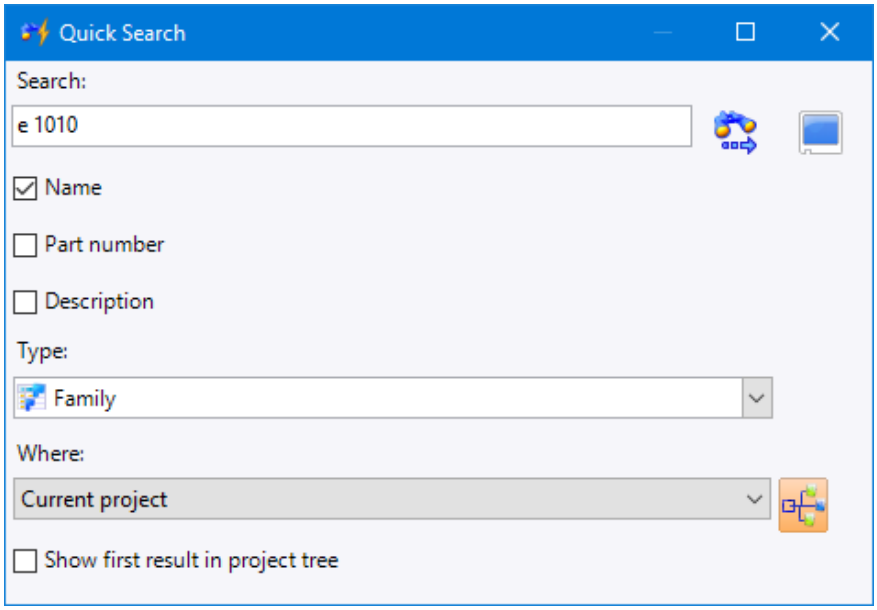
Including standard components from TopSolid libraries

Search and customization

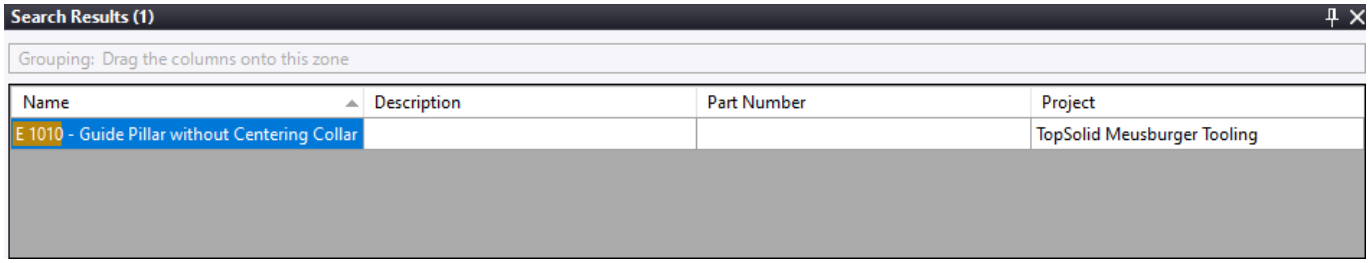
Reminder: The library in which you are searching for your components must be referenced in your project.

Method 1: Creating a quick search

- Select the quick search command by clicking on the icon at the top right of the screen.
- Adjust the following criteria to search for a standard guide pin, then click on the button to start the search.



The **Search Results** dialog box opens.



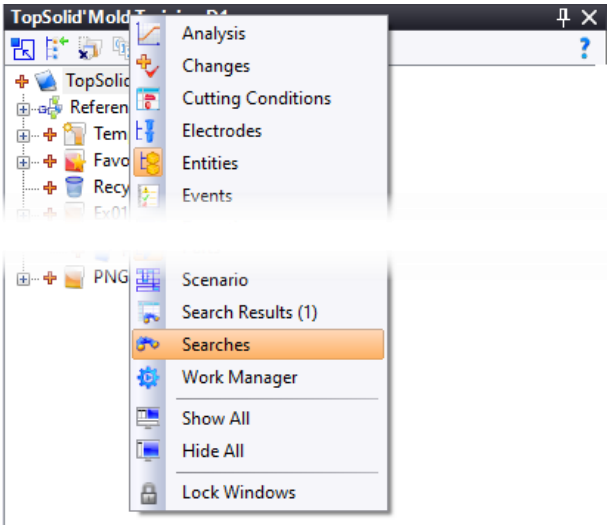
This method is fast but requires you to fill in the search criteria each time you change components.

Method 2: Creating a search document

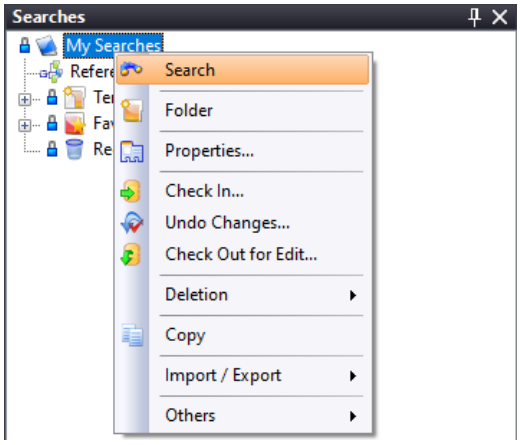
This method is useful if you are often searching for the same components.

We will create a search document for the guiding elements: Meusburger components E 1010, E 1100, E 1110 and E 1160.

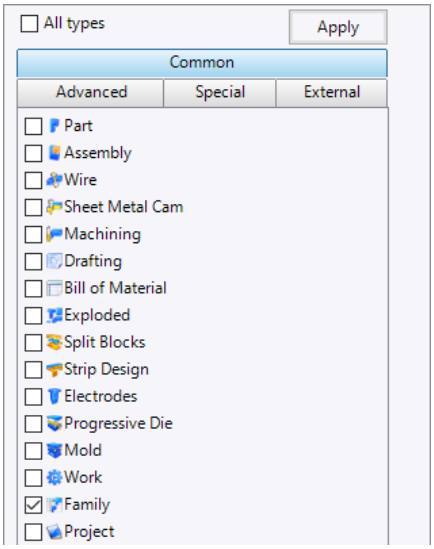
- Open the Searches tree by right-clicking on the Project tree's black title bar and selecting **Searches**.



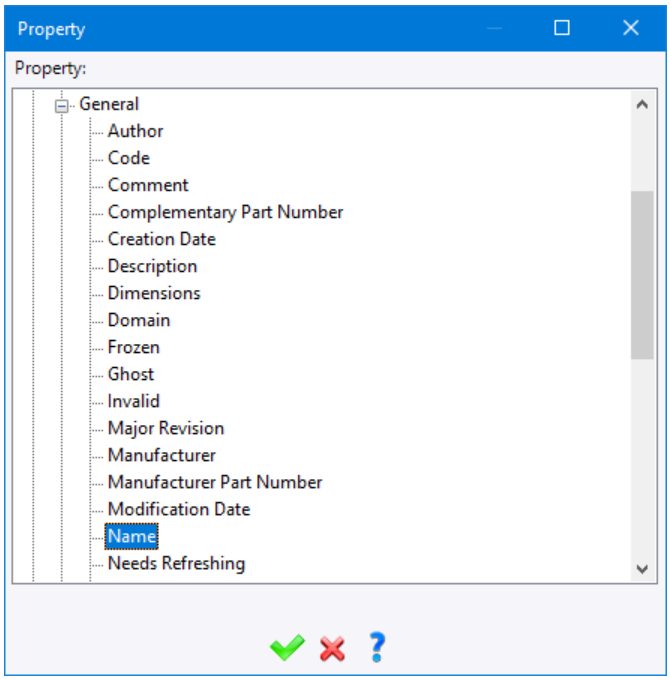
- From the Searches tree, right-click on **My Searches** to create a new search.



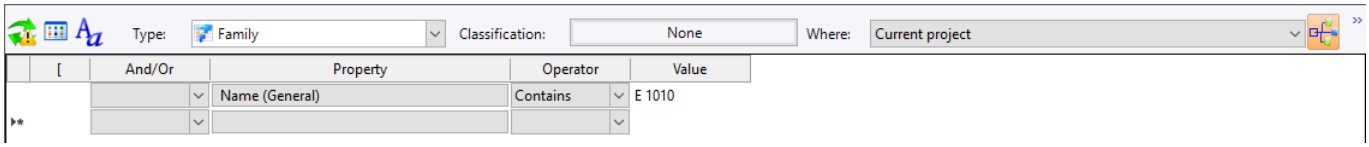
- Select **Blank Template**.
- Select only the **Family** type from the **Type** drop-down list, then click on the **Apply** button.



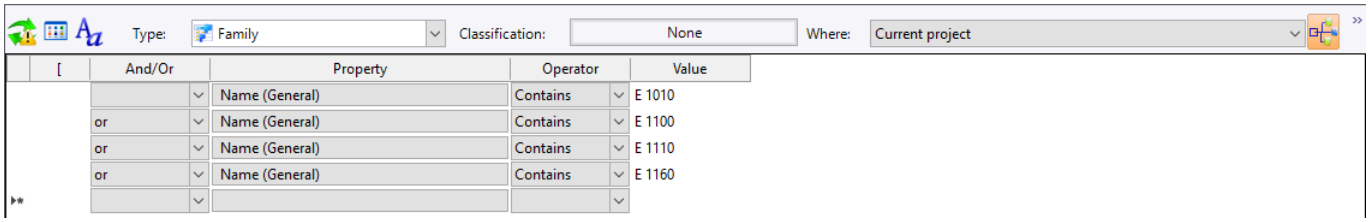
- Search in **Current project**.
- Click on the **Search in project references** icon.
- Click on the area below the **Property** field and select **General** > **Name**.



- Click on to **confirm**.
- In the **Operator** field, select **Contains**.
- In the **Value** field, enter *E 1010*.

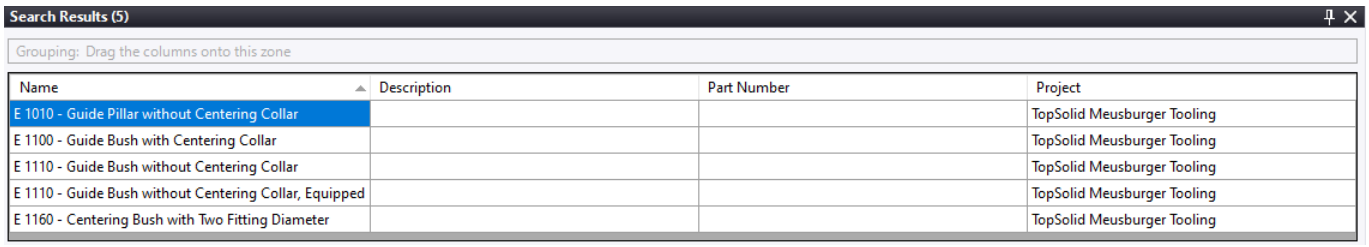


- Add three new search lines for the **E 1100**, **E 1110** and **E 1160** components.



- Click on the **Refresh** icon to test.

The result is displayed as a list.



- Rename the search document *Meusburger guides*.
- **Save** and **close** the document.

- To create a new **Meusburger screws (E 1200, E 1240)** search, copy and paste the *Meusburger guides* search and rename it *Meusburger screws*.
- Open the *Meusburger screws* search. Replace the values in the first two lines and delete the unnecessary lines as shown below.

[	And/Or	Property	Operator	Value
▶		Name (General)	Contains	E 1200
	or	Name (General)	Contains	E 1240
*				

Search Results (2)

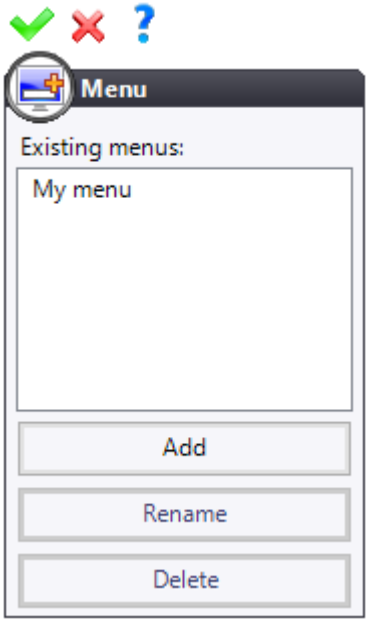
Grouping: Drag the columns onto this zone

Name	Description	Part Number	Project
E 1200 - Cylinder Head Screw with Hexagon Socket			TopSolid Meusburger Tooling
E 1240 - Shoulder Screw			TopSolid Meusburger Tooling

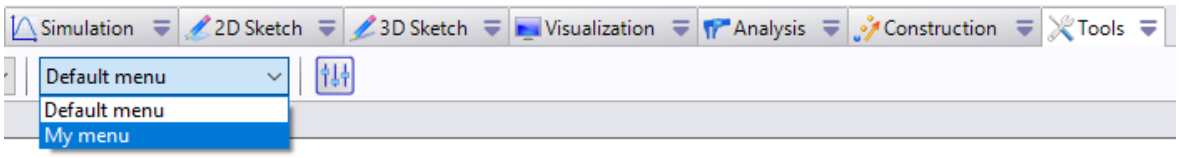
- **Refresh**, then **save** and **close** the document.
- Once all the tests have been performed, **check** the two searches into the vault and **validate** them.

To facilitate the insertion of these components without using the Searches tree, we will create two icons corresponding to each search that will appear in the icon bar at the bottom of the graphics area. To do this, you have to create and enable a custom menu to modify the content of the icon bars.

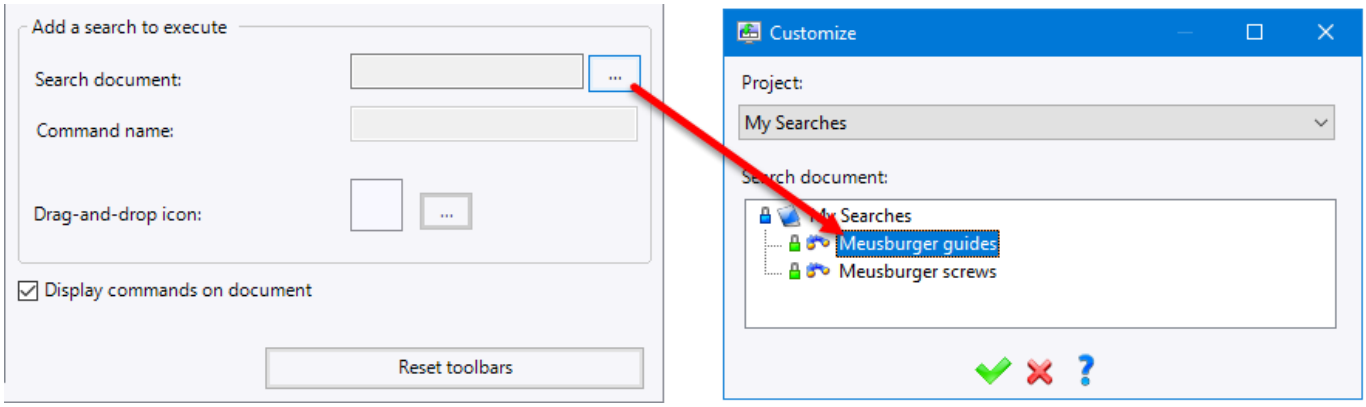
- From the **Tools** tab, select the **Menu** command.
- In the dialog box, click on the **Add** button, enter a **name** for the menu, then **confirm** twice.



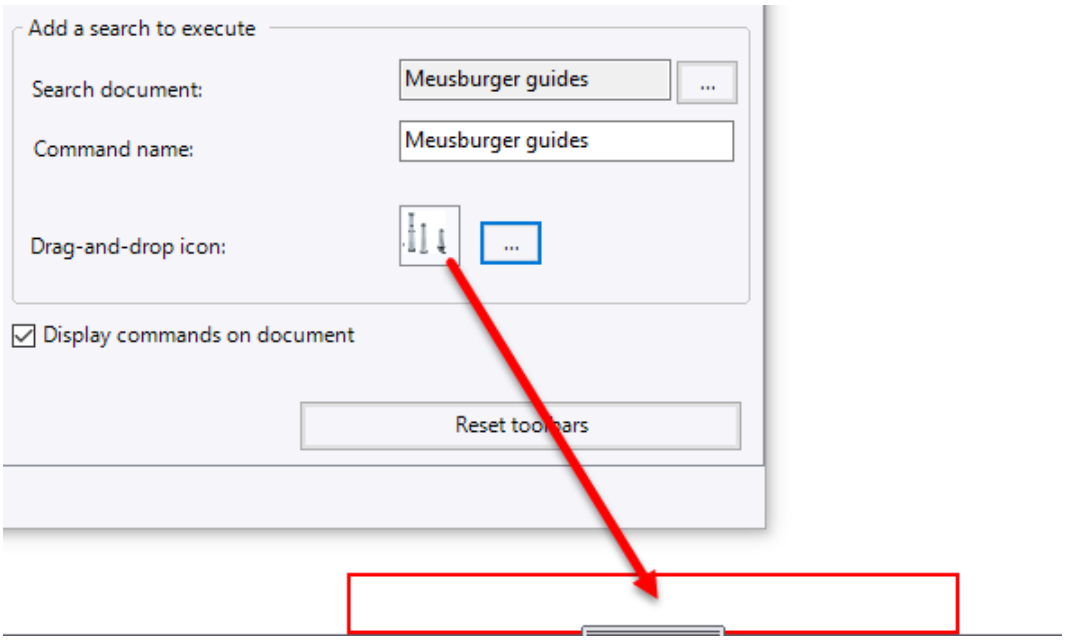
- From the **Tools** tab's icon bar, select the previously created menu.



- From the **Tools** tab, select the **Customize** command.
- In the **Add a search to execute** area of the dialog box, click on the ... button next to the **Search document** field and select the **Meusburger guides** document from the list.



- Click on the ... button next to the **Drag-and-drop icon** field and assign an icon to your search. You can select an icon from the list or include your own icons in PNG format. Icons in PNG format are provided with the project.
- Drag and drop the icon at the bottom center of the graphics area.



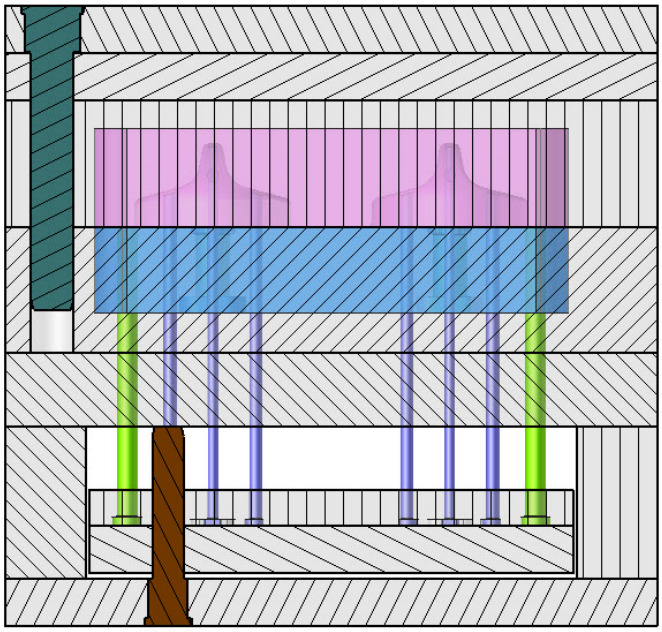
- Repeat the operation for the second search, then click on to **confirm**.

This customization will only be assigned to the **mold** documents. You can repeat this operation for other types of documents (part, assembly, etc.) if necessary.

We will now use the previously created searches to include these components in the mold.

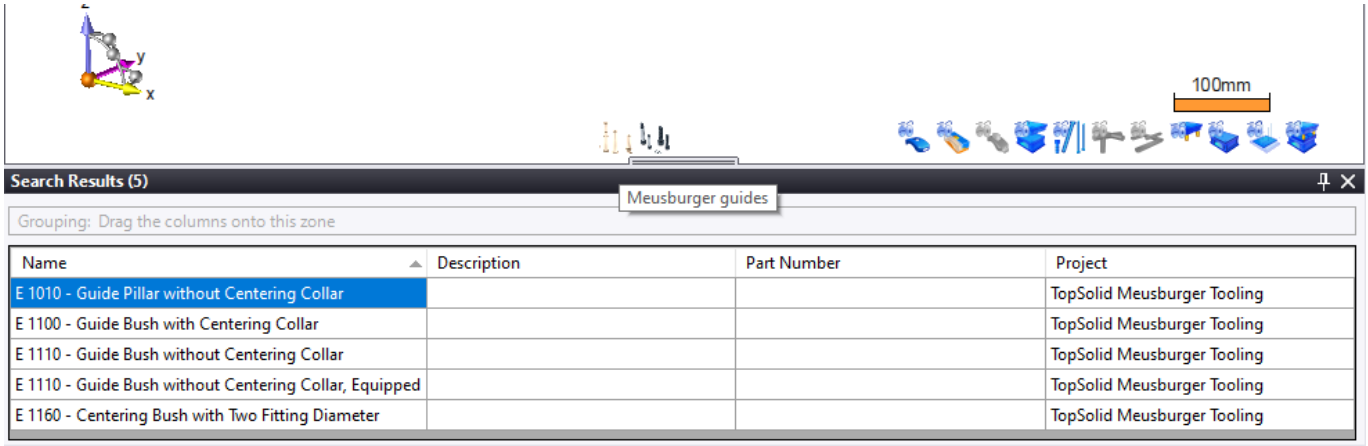
Guide pins

We will insert two guide pins using the **Guide Pin** wizard.

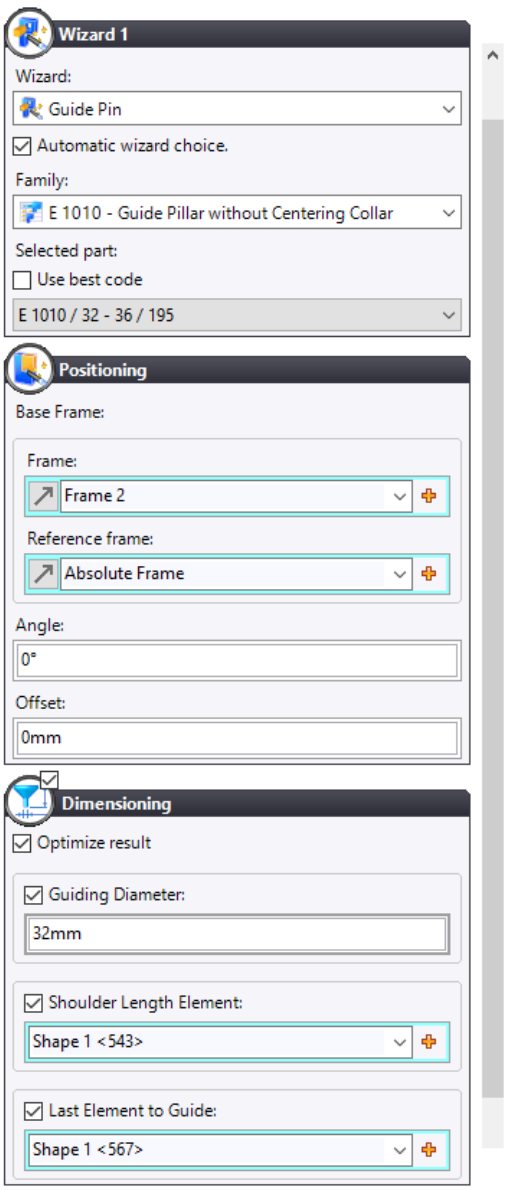


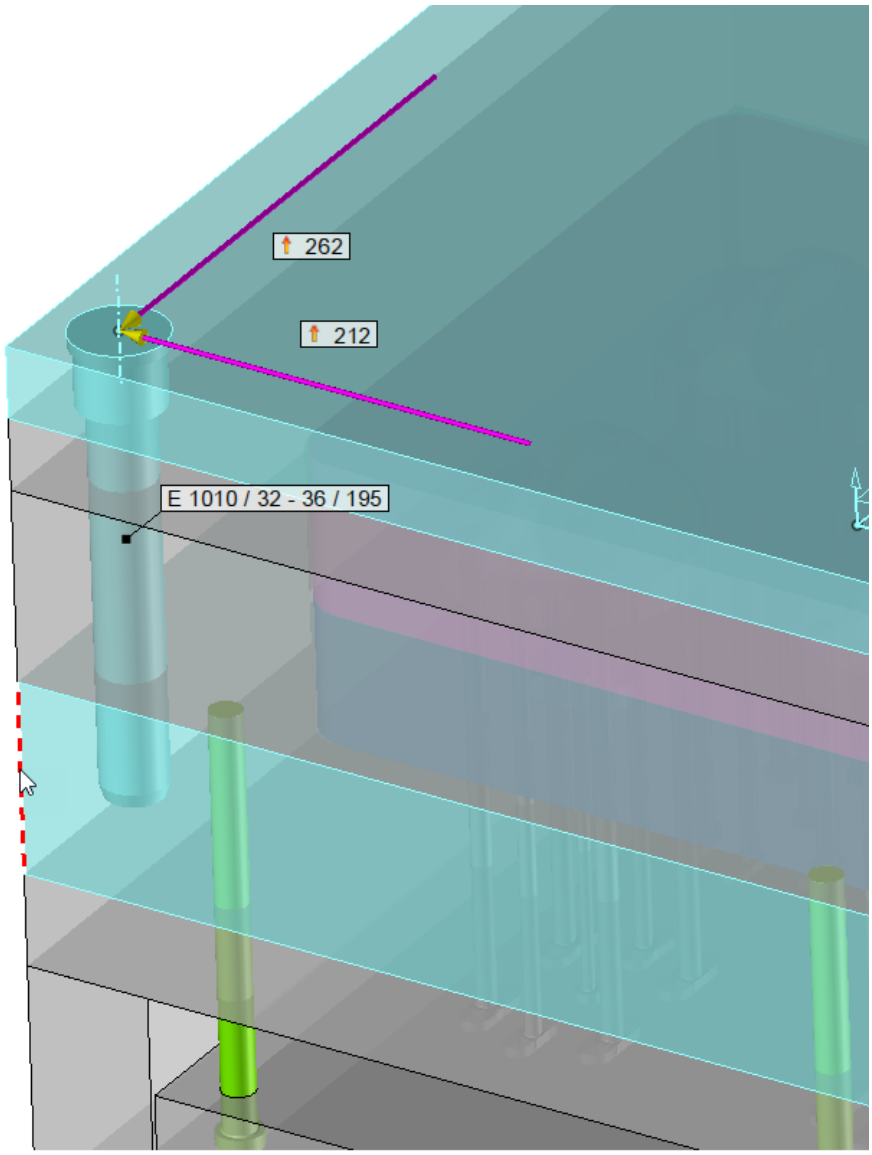
- Click on the **Meusburger guides** search icon.

The **Search Results** dialog box opens.

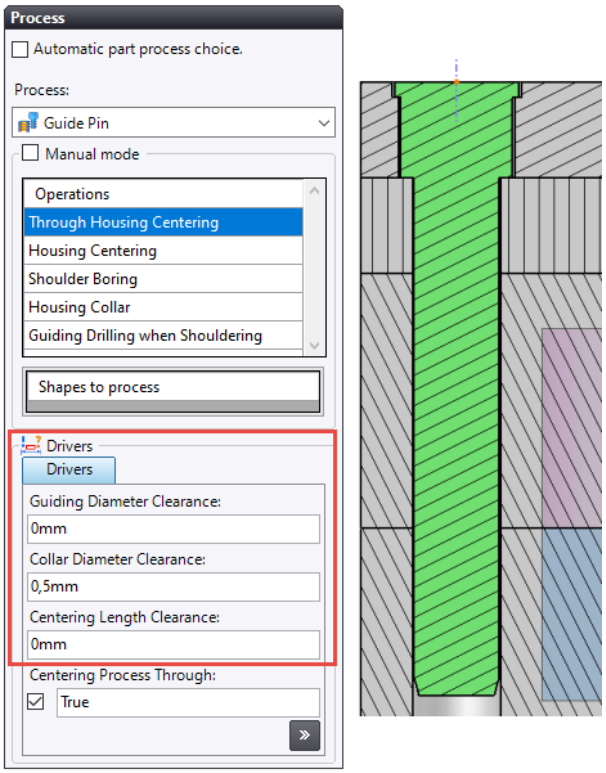


- Drag and drop the **E 1010** component into the graphics area and adjust the following parameters to position the first guide pin.
- Select the **Guide Pin** wizard. The positioning face will be the top face of the top plate.
- Select the **absolute frame** as the **reference frame** and enter the coordinates as shown below.
- In the **Dimensioning** option, fill in the fields as shown below, then click on to **confirm**.

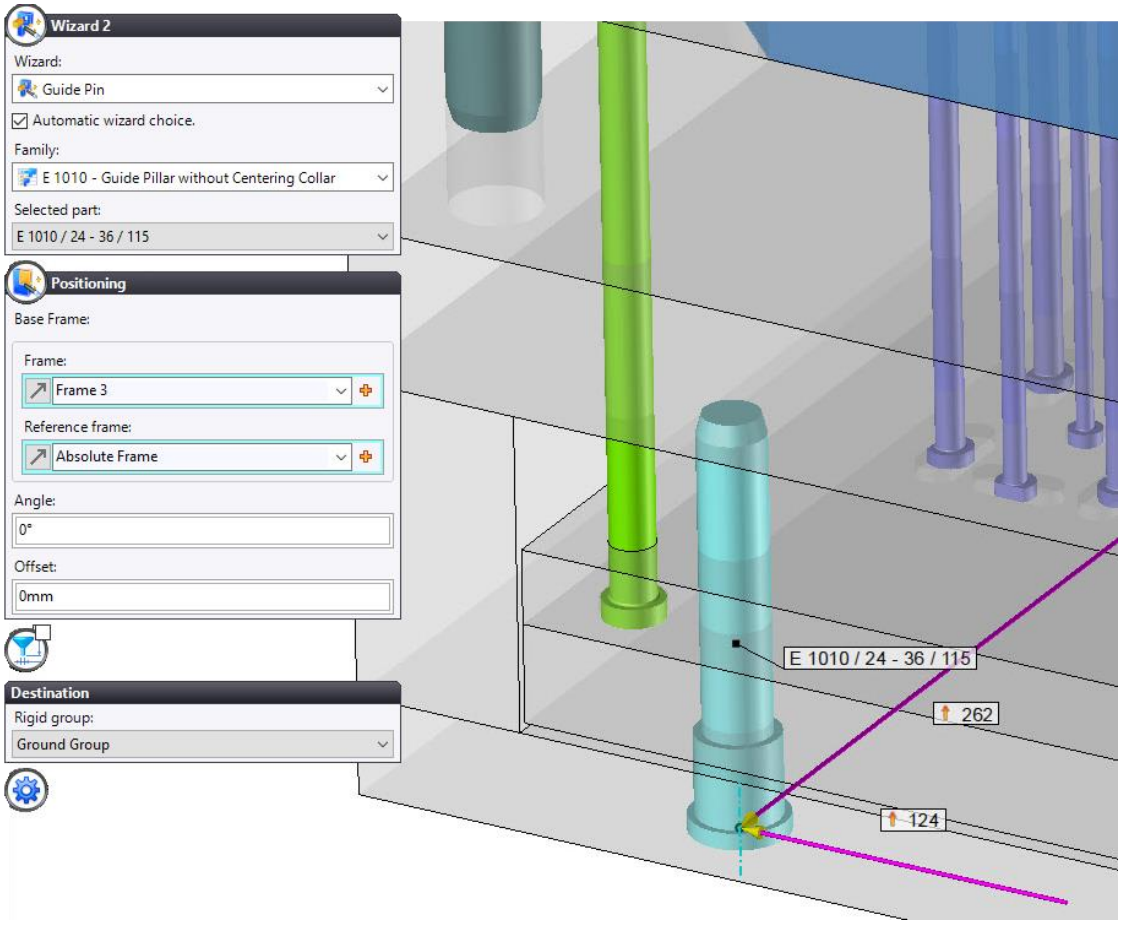




Note: If you want to change the diameter and length clearances, edit the **Process** operation in the Operations tree and modify the desired values in the **Drivers** section.

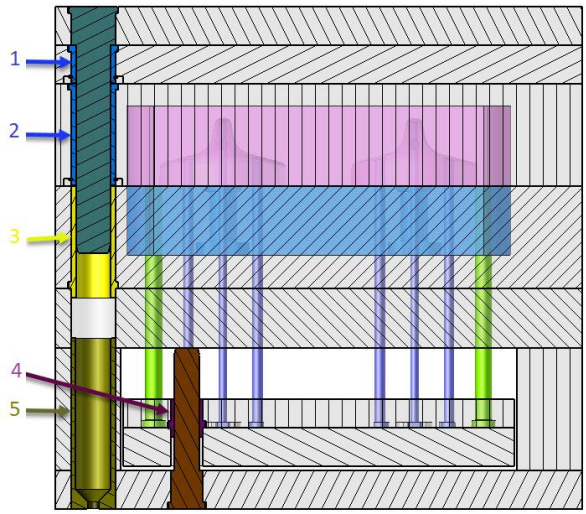


- Include the same component using the **Guide Pin** wizard and the **E 1010/24 - 36/115** code, then position the guide pin on the top face of the lower support plate. Reverse the frame direction if necessary. Select the **absolute frame** as the **reference frame** and enter the coordinates as shown below.



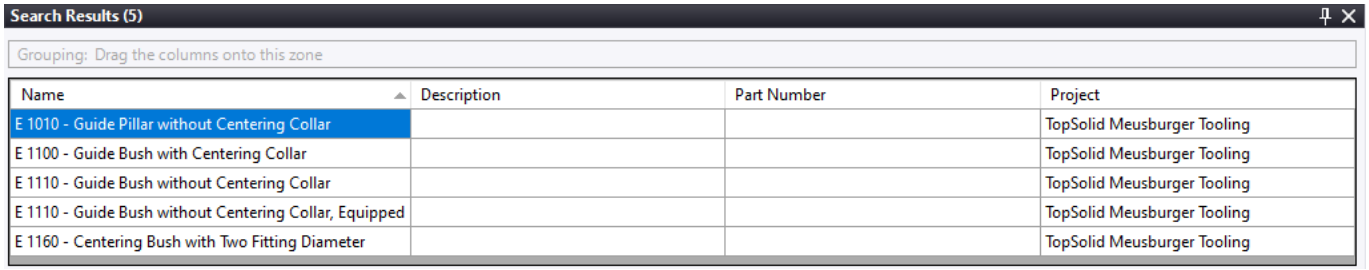
Guide bushes

We will now insert four guide bushes and a centering bush using the **Guide Bush** wizard.

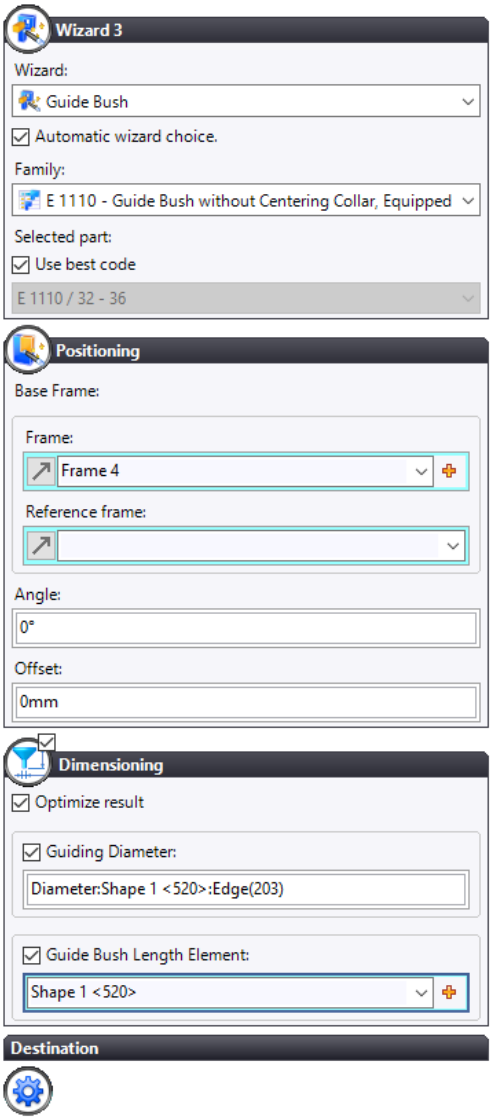


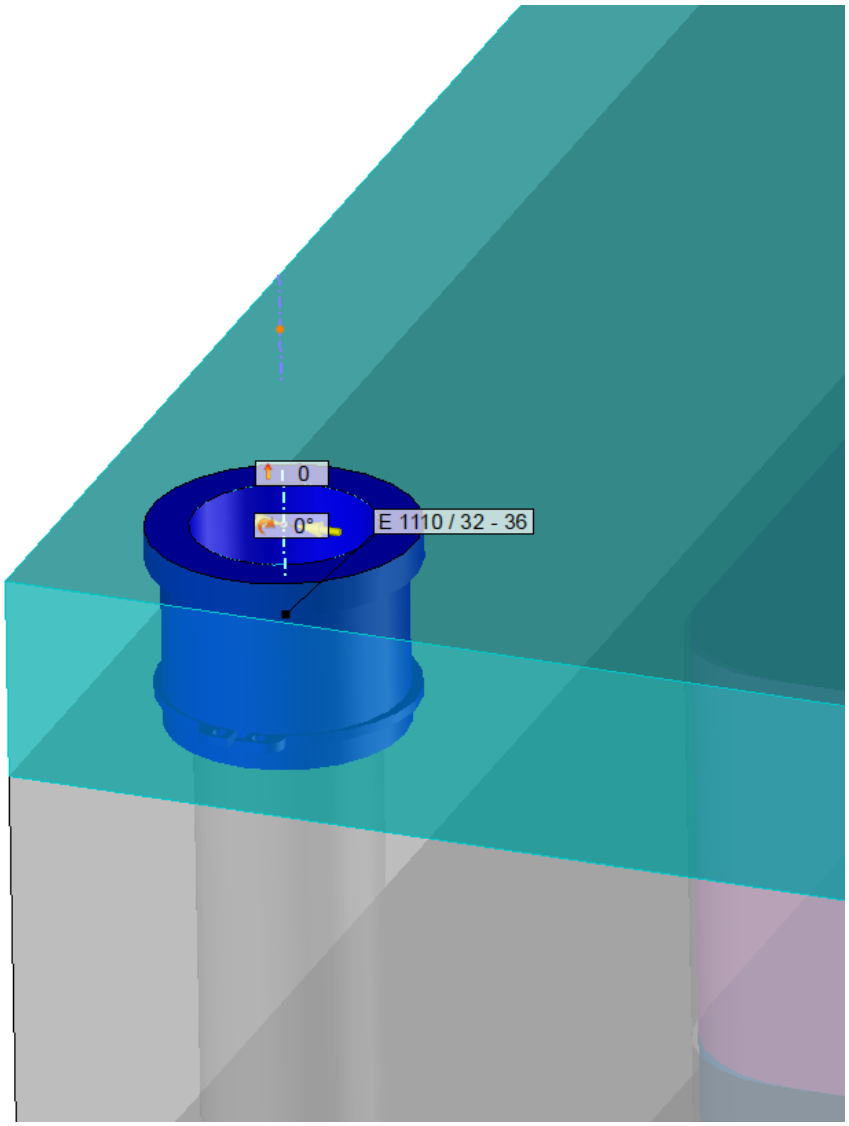
1: **E 1110 32-36**
2: **E 1110 32-96**
3: **E 1100 32-96**
4: **E 1100 24-27**
5: **E 1160 42-160**

- To help you position the component, hide the first guide pin and the top clamp plate or **show only** the support plate.
- Relaunch the **Meusburger guides** search if necessary.

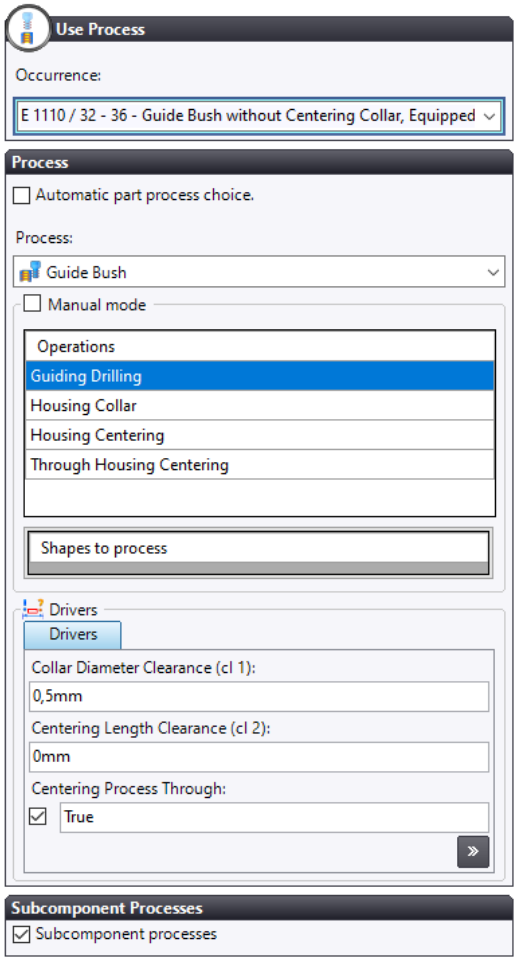


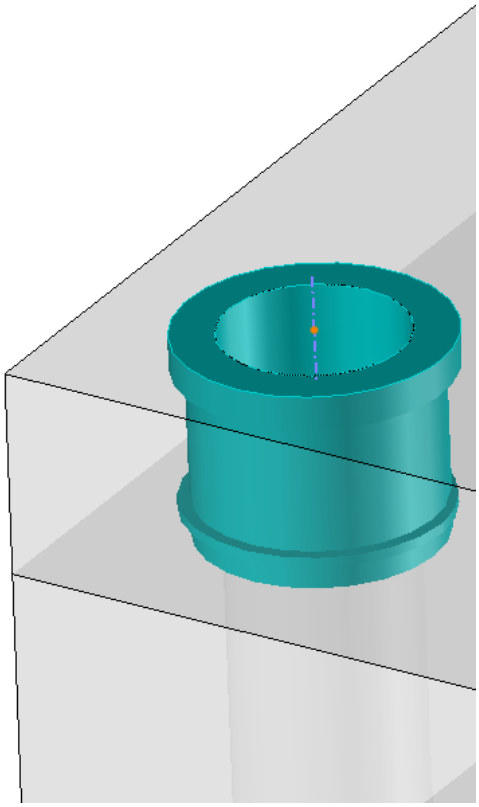
- Drag and drop the **E 1110** component into the graphics area and adjust the following parameters to position the first guide pin.
- Select the **Guide bush** wizard.
- In the **Dimensioning** option, select the hole in the plate as the **guiding diameter** and the plate as the **guide bush length element**. Hover the mouse cursor over the guide pin's drilling so that **TopSolid** automatically detects the hole center.

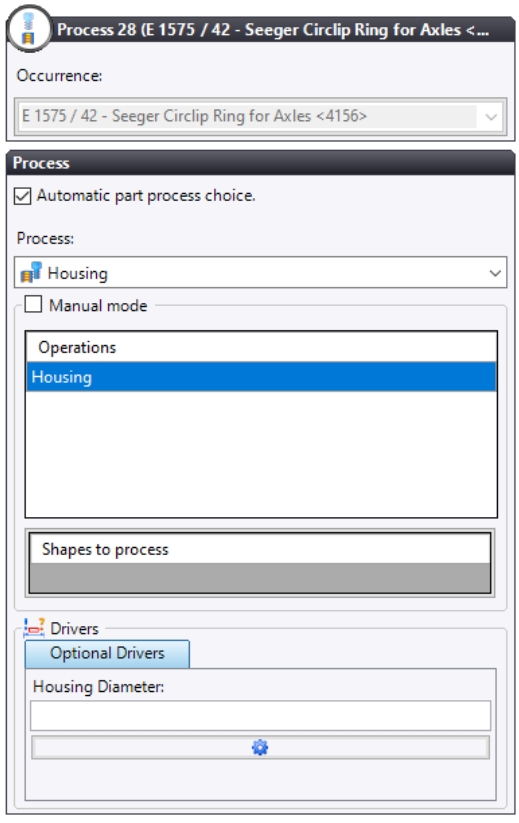


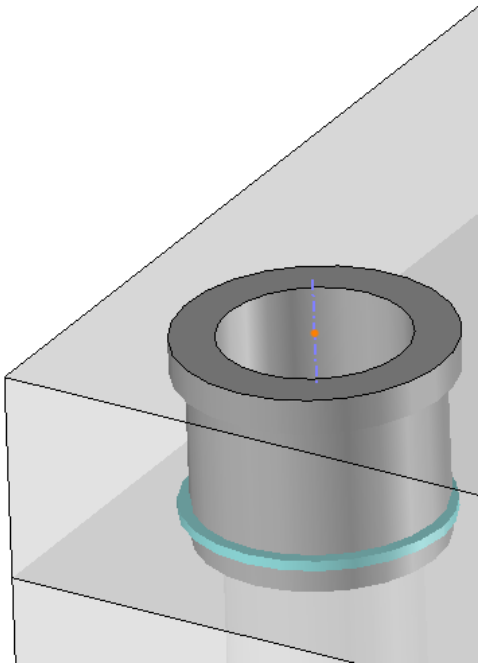


- After confirming the wizard, **confirm** the default process, then select the **Housing** process for the circlip in the second dialog box.

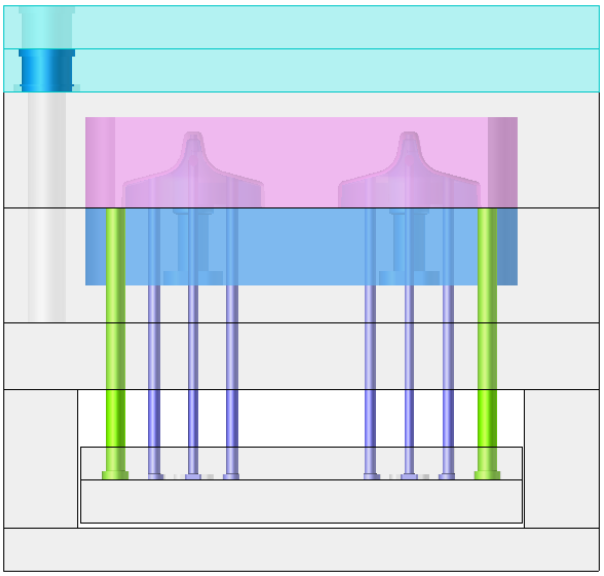




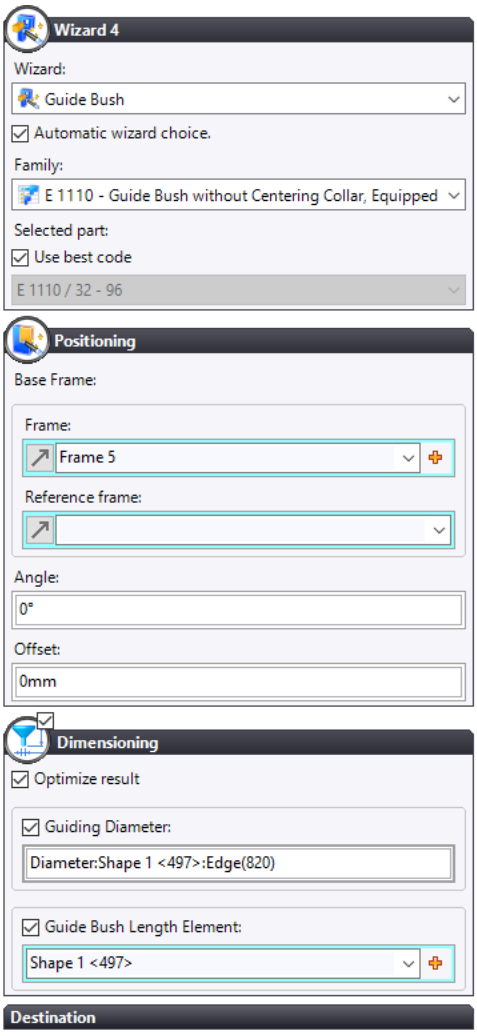


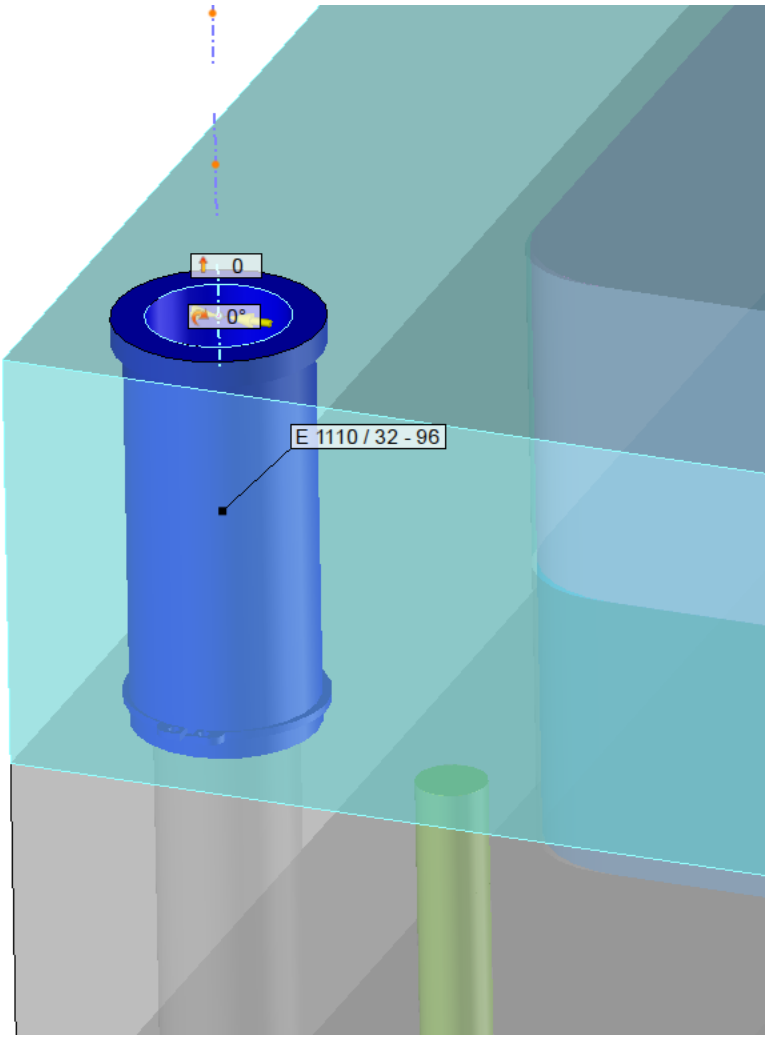


- To help you position the next component, leave the top clamp plate hidden and hide the second support plate as well as the created bush.

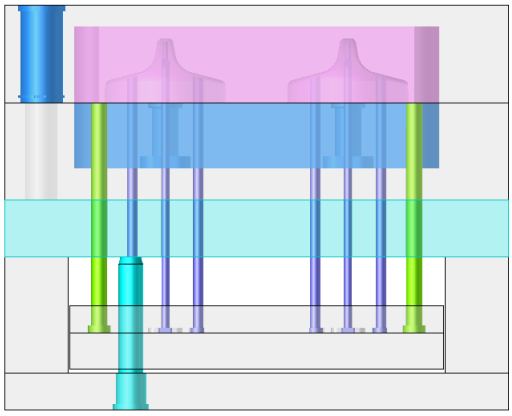


- Drag and drop the **E 1110** component into the graphics area. Select the **Guide bush** wizard.
- Repeat the same operations as for the previous bush.

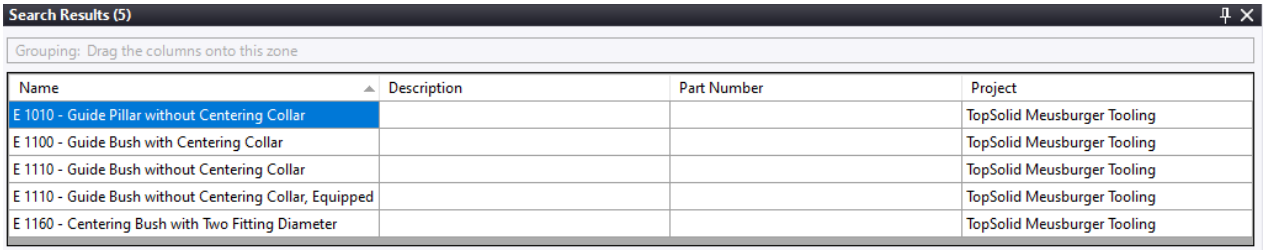




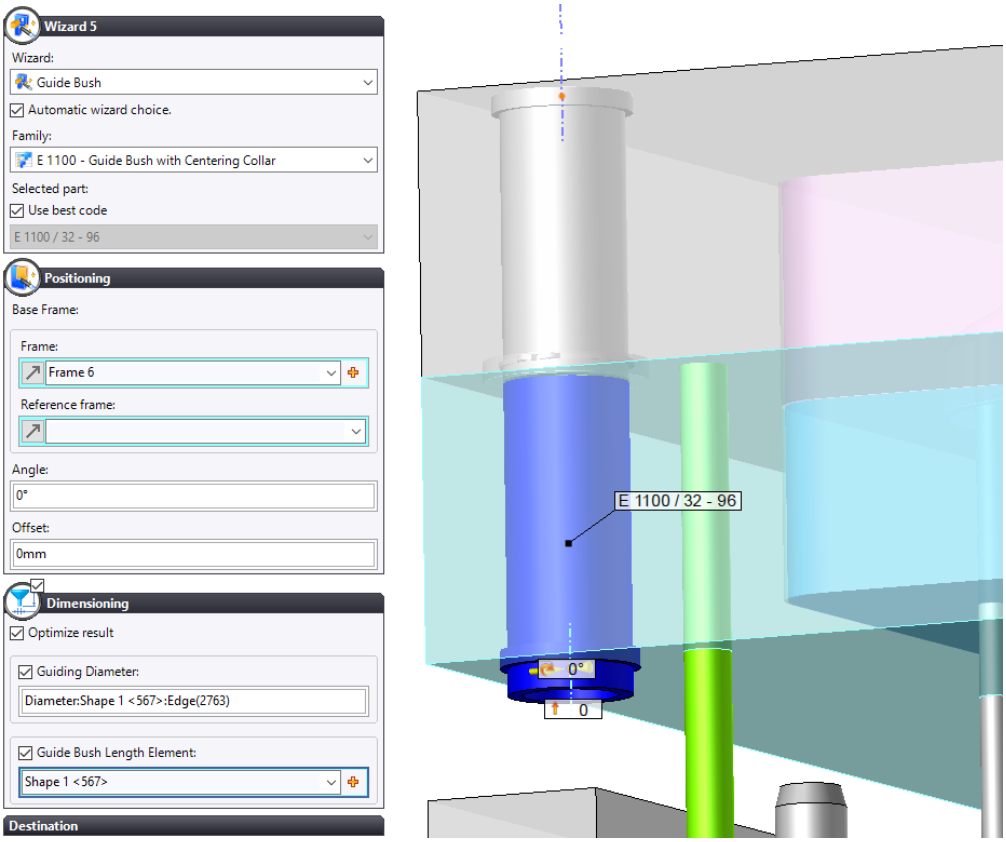
- To help you position the next component, hide the bottom support plate.



- Relaunch the **Meusburger guides** search if necessary.

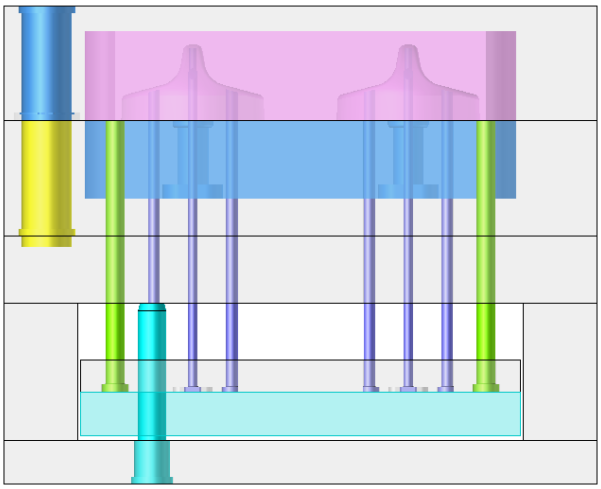


- Drag and drop the **E 1100** component into the graphics area and adjust the following parameters to position the third guide pin.
- Select the **Guide bush** wizard.
- In the **Dimensioning** option, select the hole in the plate as the **guiding diameter** and the plate as the **guide bush length element**.
- For the positioning, hover the mouse cursor over the drilling to detect the hole center.

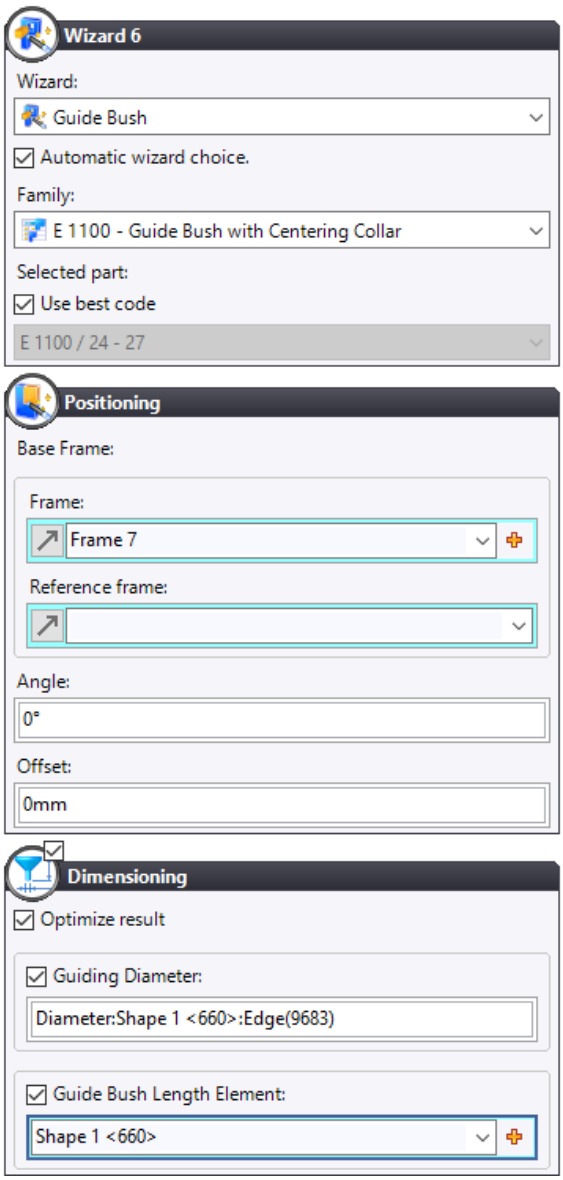


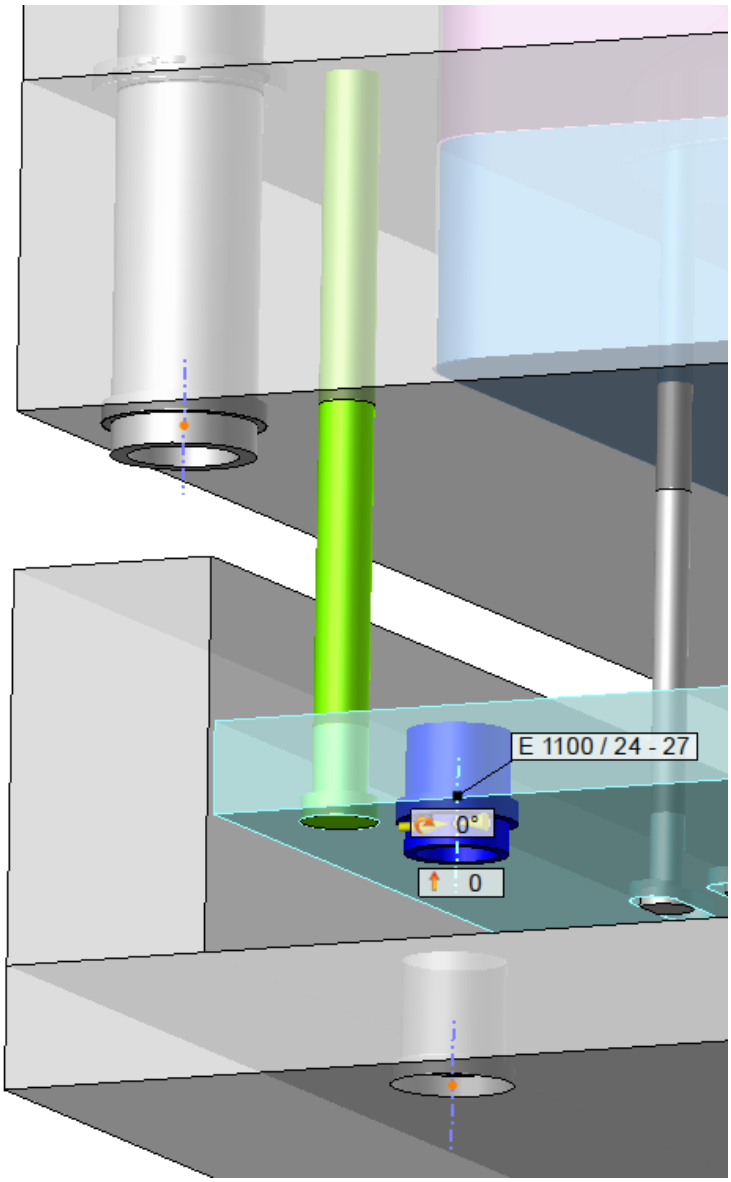
- After confirming the wizard, check the **Centering Process Through** box in the **Use Process** dialog box.

- To help you position the fourth screw, hide the ejector base plate and the guide pillar as shown below.

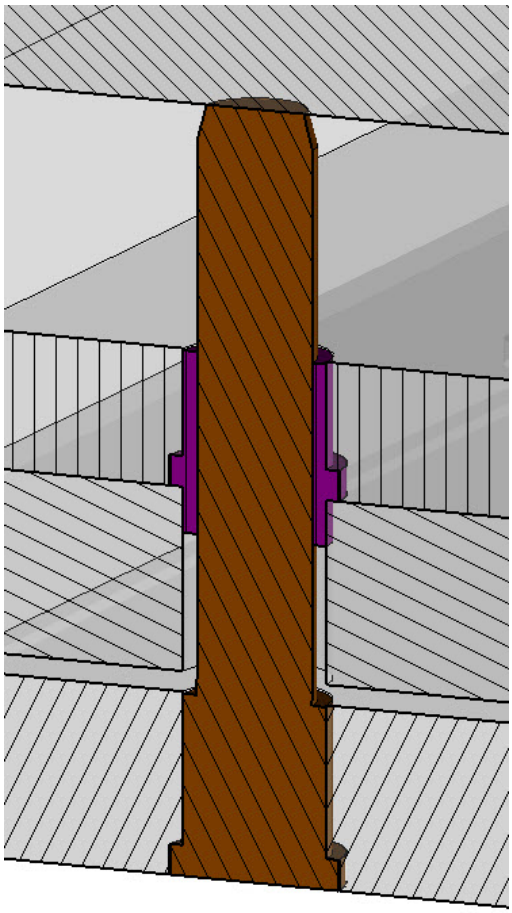


- Insert the same component using the **Guide bush** wizard.
- In the **Dimensioning** option, select the hole in the ejector retaining plate as the **guiding diameter** and the ejector retaining plate as the **guide bush length element**. Hover the mouse cursor over the drilling to detect the hole center.

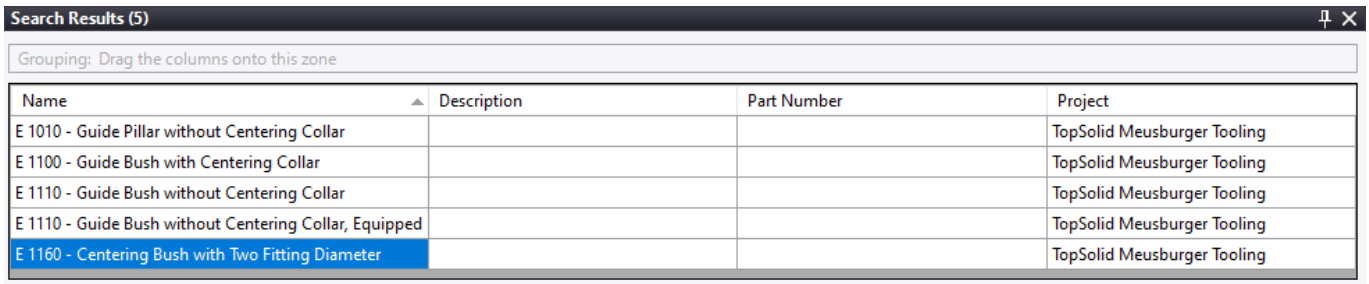




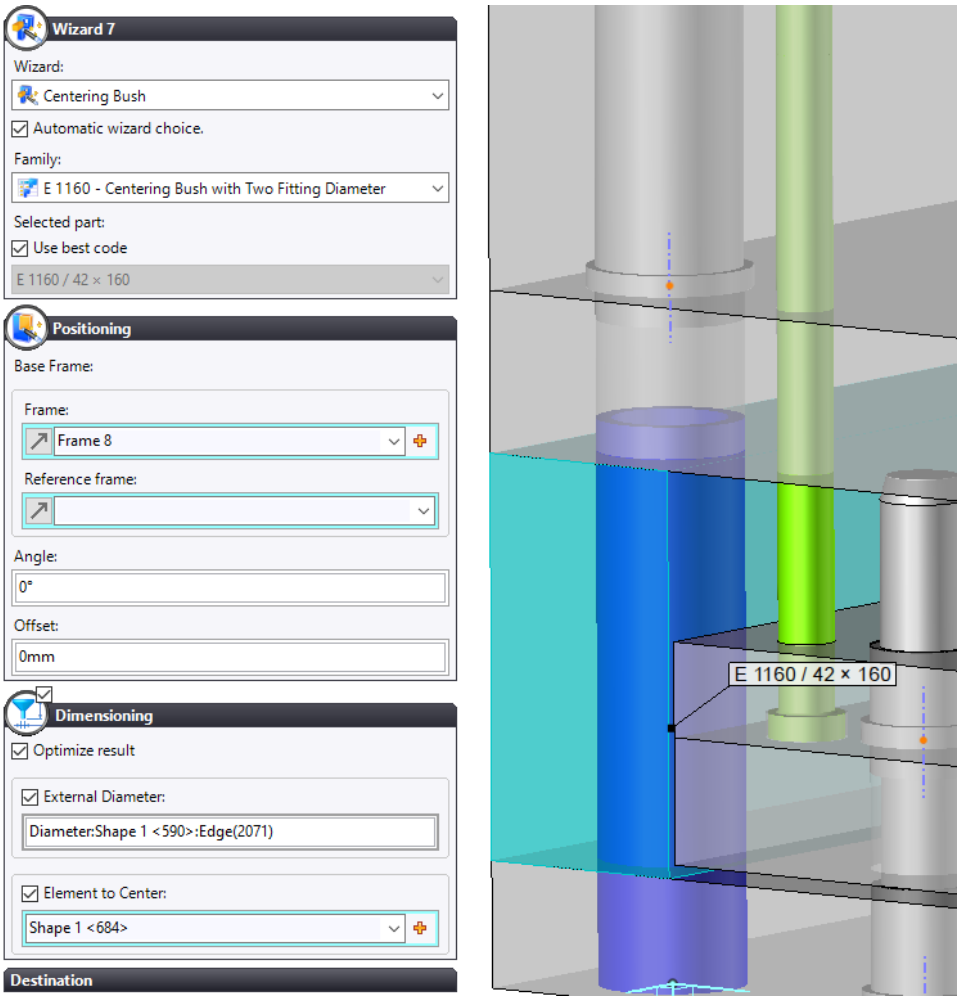
- After confirming the wizard, check the **Centering Process Through** box in the **Use Process** dialog box.



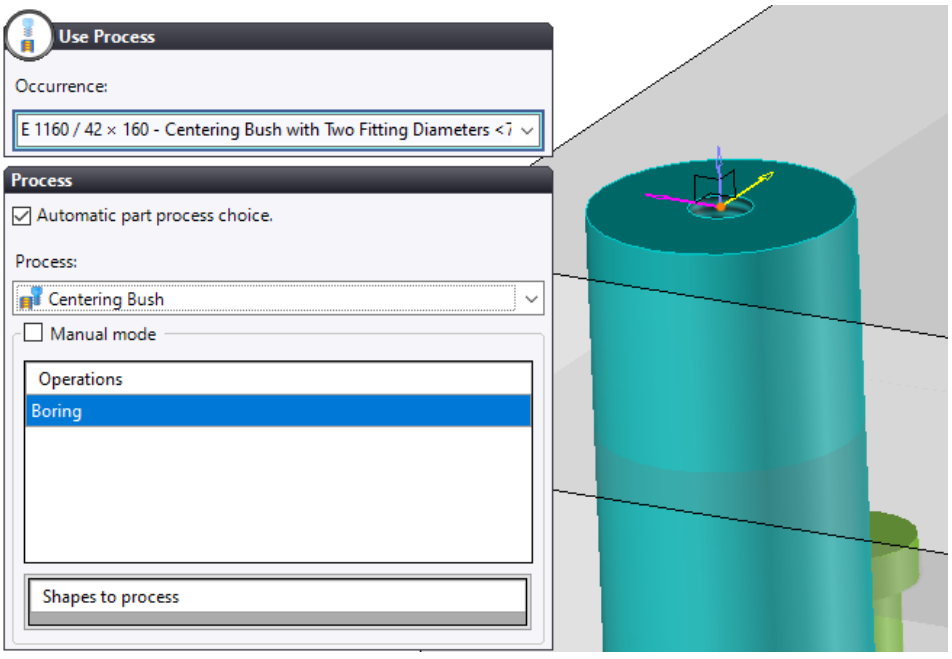
- Relaunch the **Meusburger guides** search if necessary.



- Drag and drop the **E 1160** component into the graphics area and adjust the following parameters to position the centering bush.
- Select the **Guide bush** wizard. In the **Dimensioning** option, select the hole in the bottom backing plate as the **external diameter** and the rail as the **element to center**. Create a frame on plane as the positioning frame using the external plane of the clamp plate and a center point on one element of the guide pillar as the reference.

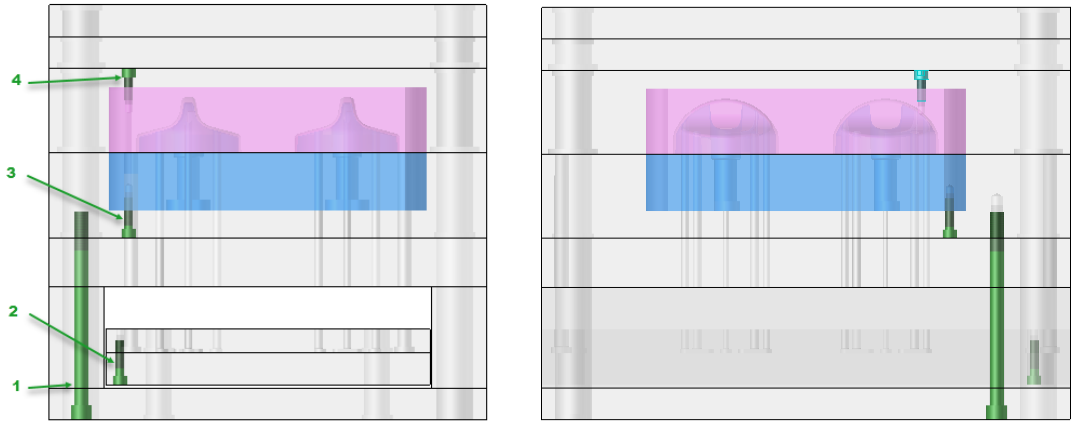


- In the **Use Process** dialog box, adjust the following parameters.

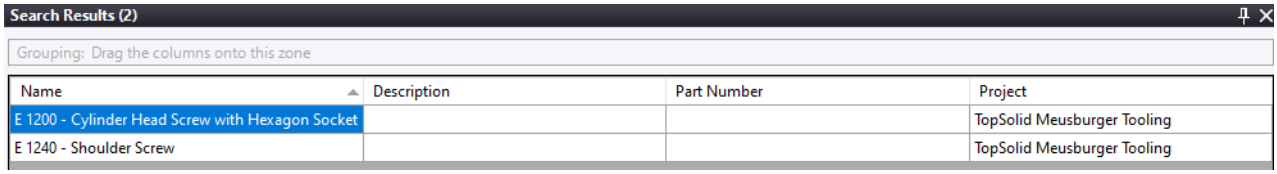


Fixing screws

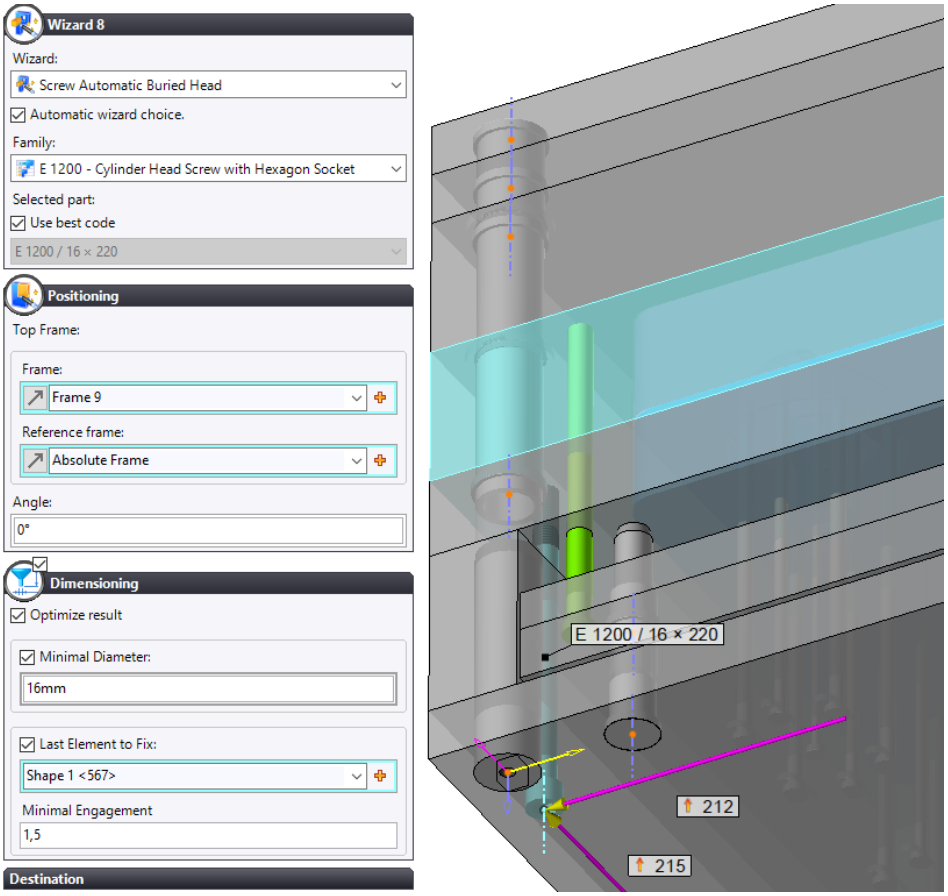
We will now insert four standard screws using the **Screw Automatic Buried Head** wizard. They will then be repeated using a double symmetry.



- Click on the **Meusburger screws** search icon.

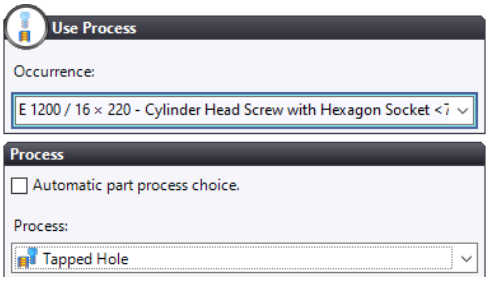


- Drag and drop the **E 1200** component into the graphics area and adjust the following parameters to position the first screw.
- Select the **Screw Automatic Buried Head** wizard. In the **Dimensioning** option, adjust the **minimum diameter** to *16mm* and select the B plate as the **last element to fix**. The positioning face will be the bottom face of the B plate. Select the **absolute frame** as the **reference frame** and enter the coordinates as shown below.

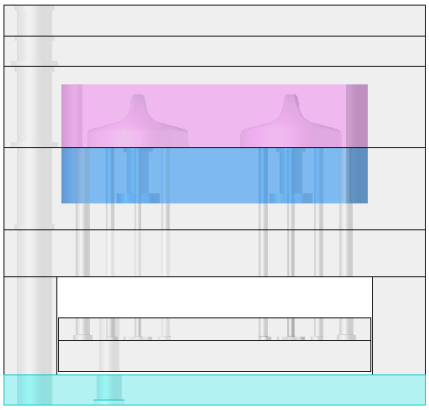


After confirming the wizard, the process dialog box appears. Various processes are possible for this type of component.

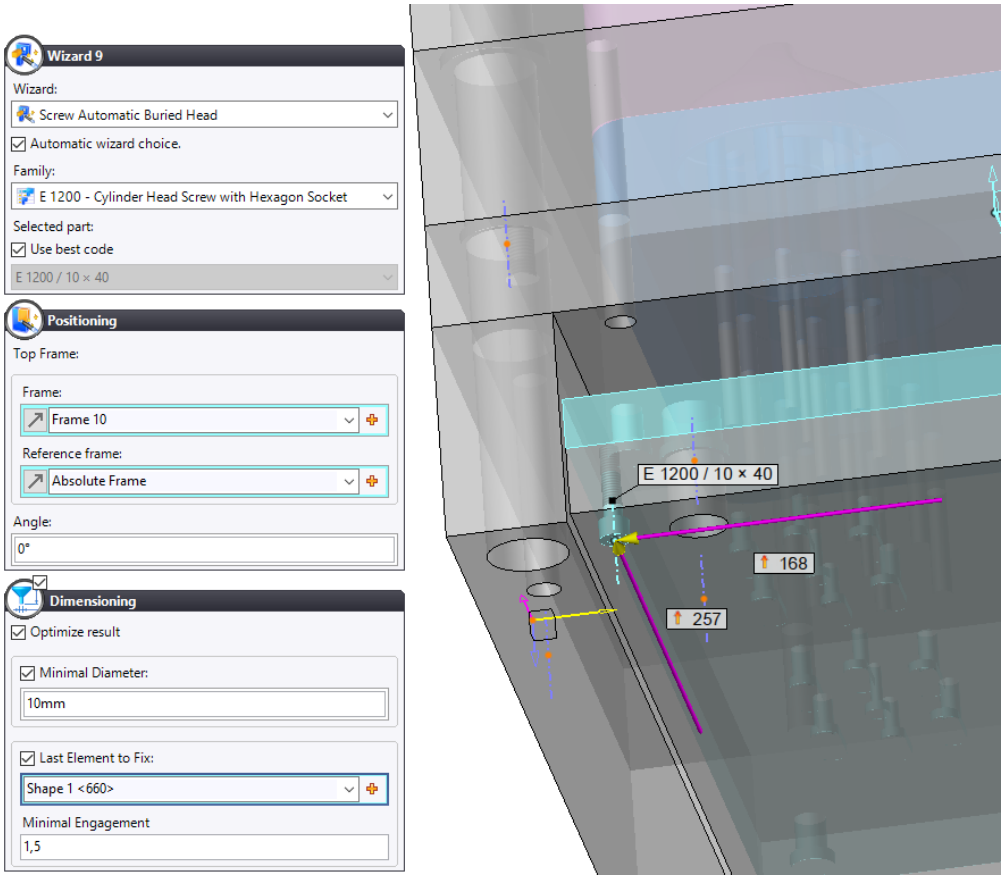
- Select **Tapped hole**.



- To help you position the next component, hide the bottom clamp plate.

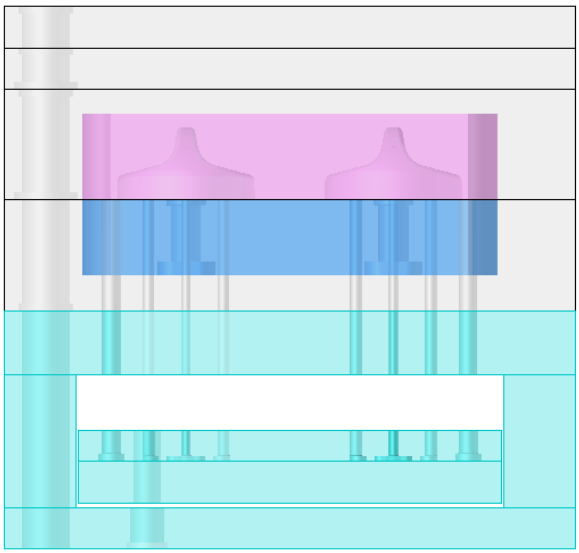


- For the second screw, drag and drop the **E 1200** component into the graphics area to fix the ejector retaining plate and the ejector base plate. Enter the following values for the dimensioning and positioning.

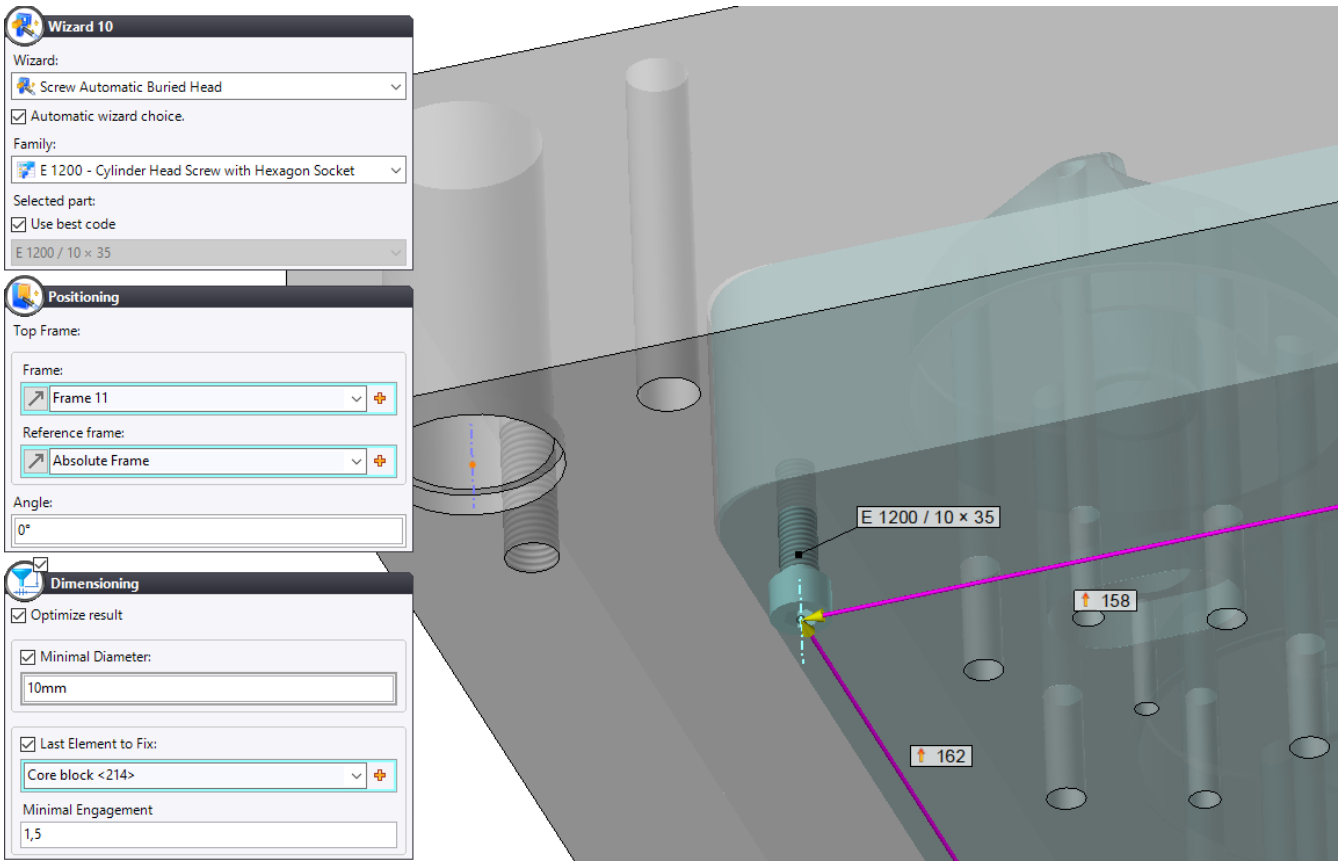


- After confirming the wizard, select the **Tapped hole** process in the **Use Process** dialog box.

- To help you position the next component, hide the following mold base elements.

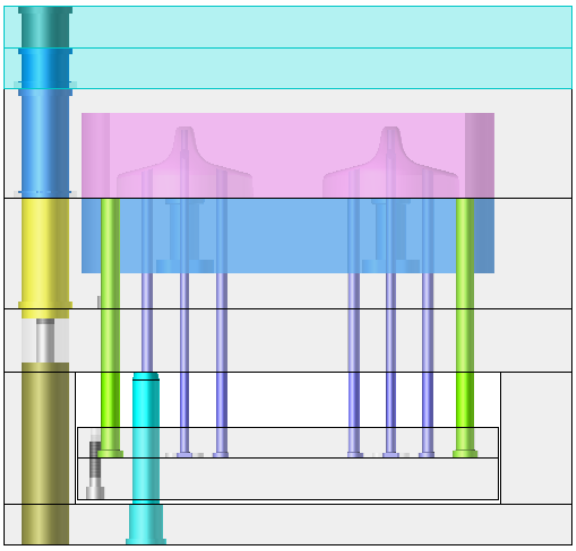


- For the third screw, drag and drop the **E 1200** component to fix the core block and the B plate. Enter the following values for the dimensioning and positioning.

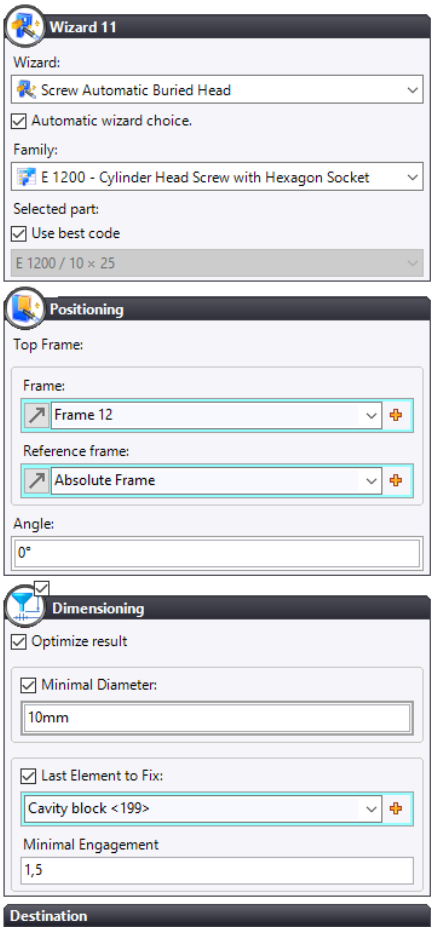


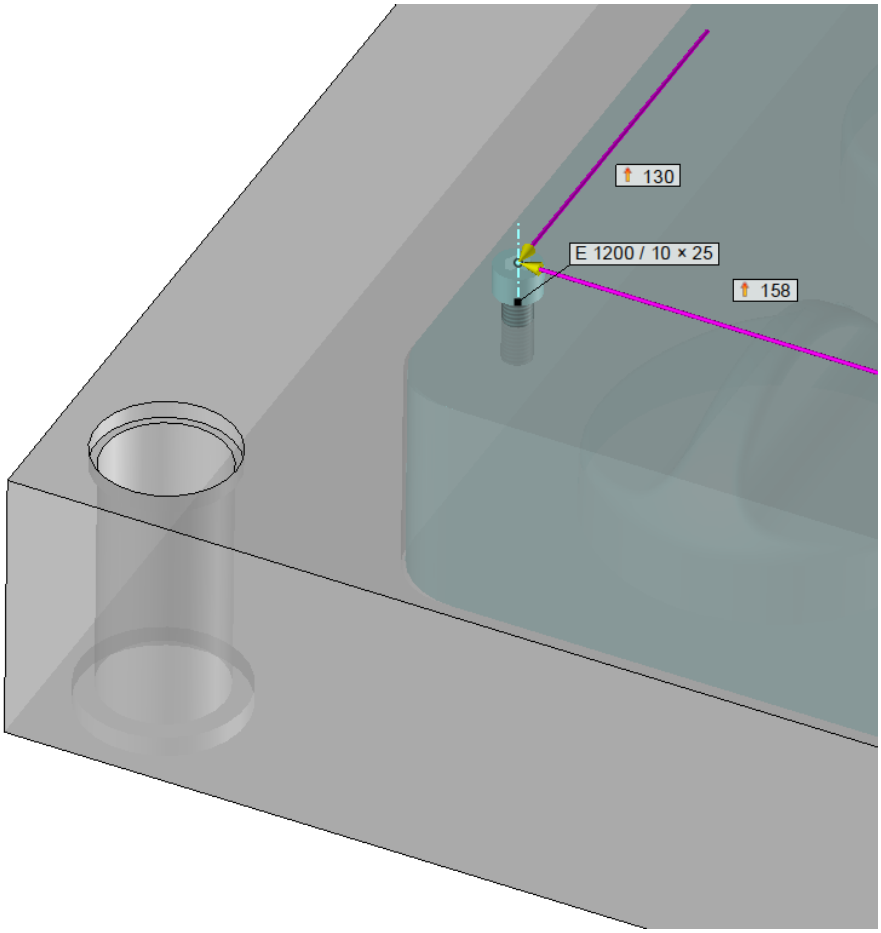
- After confirming the wizard, select the **Tapped hole** process in the **Use Process** dialog box.

- To help you position the last screw, hide the first two plates on the A side.



- For the fourth screw, drag and drop the **E 1200** component into the graphics area to fix the cavity block and the A plate. Enter the following values for the dimensioning and positioning.

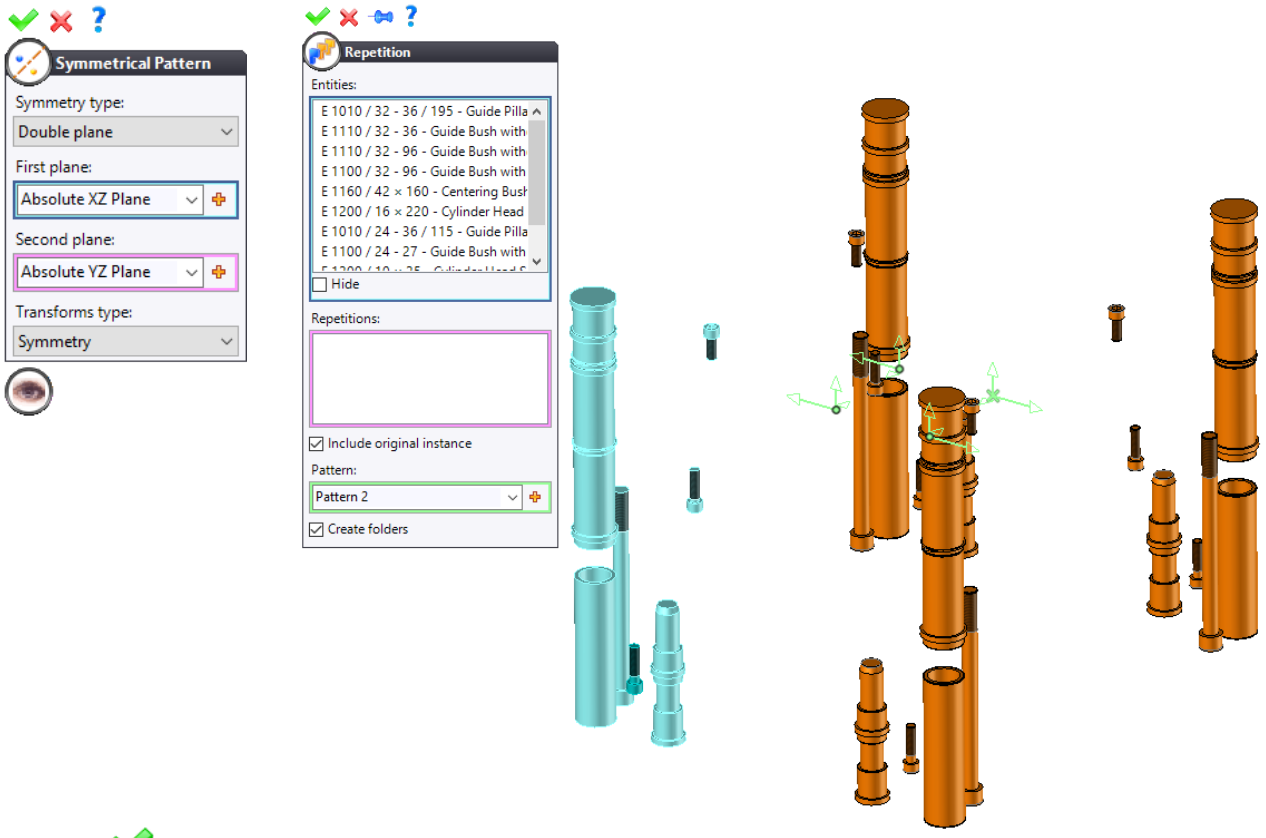




- After confirming the wizard, select the **Tapped hole** process in the **Use Process** dialog box.

Repeating the components

- For the next manipulation, hide all the elements except the last inserted components: guide pins, guide bushes and screws.
- From the **Construction** tab, select the **Repetition** command and select the components to be repeated by symmetry: the screws, the guide pins and the guide bushes.
- For the pattern, select **Symmetrical Pattern** using the special inputs. Adjust the following parameters for the **double symmetry**.



- Click to **confirm**.

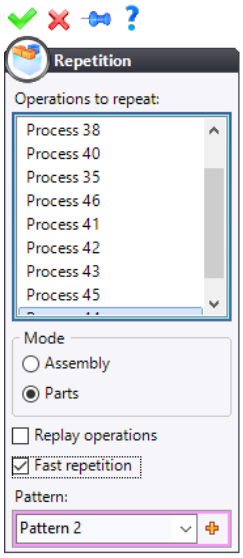
Difference between the process repetition modes

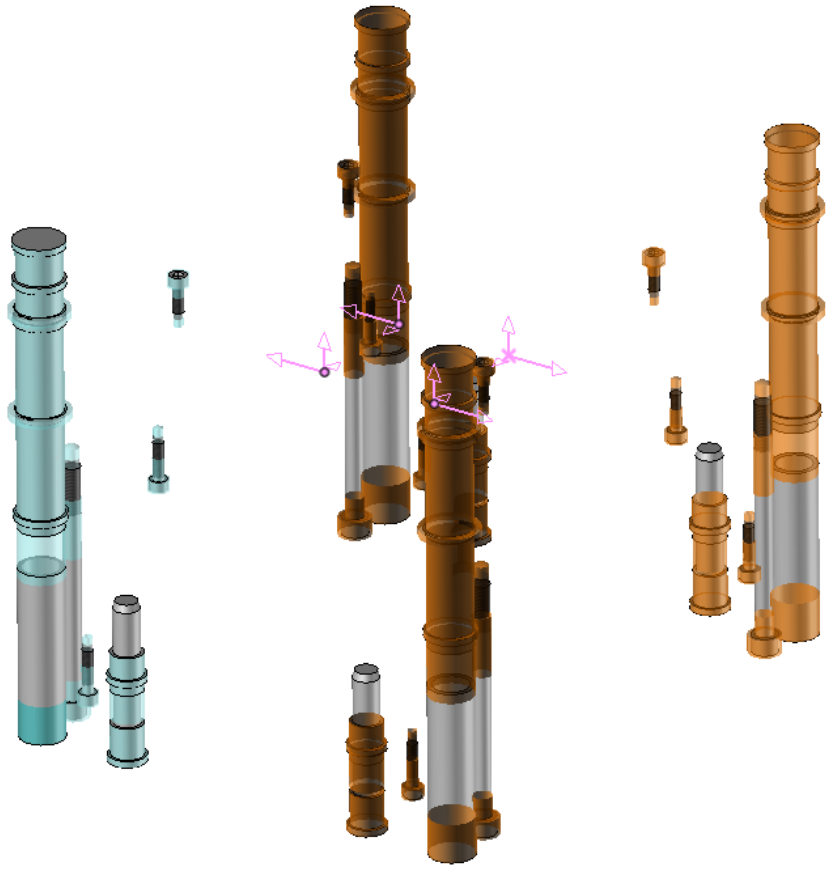
The **Assembly** mode is safer but slower and the part contains as many basic operations (no repetition in each impacted part). This mode is recommended when the repetition impacts new parts.

The **Parts** mode has two options (repetition in each impacted part):

- **Replay operations**: This option is useful when the faces to be operated are not the same as the first component.
- **Fast repetition**: This option is useful when the faces to be operated are the same.

You must first define the option in the **Tools** > **Options** > **Assembly** > **Process Repetition** > **Show Dialog** command.



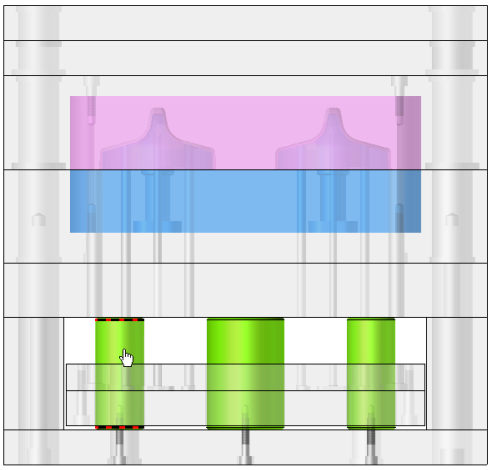


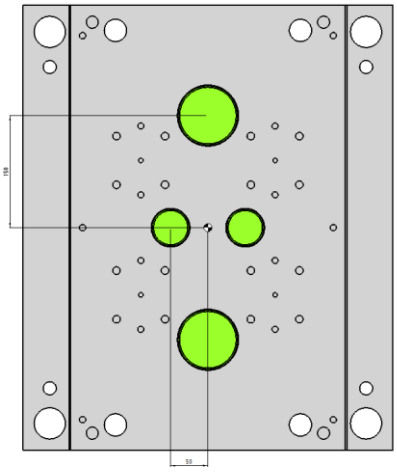
- From the Entities tree, open the **Patterns** folder and rename pattern 2 *Double Sym XZ - YZ.*

Including user components

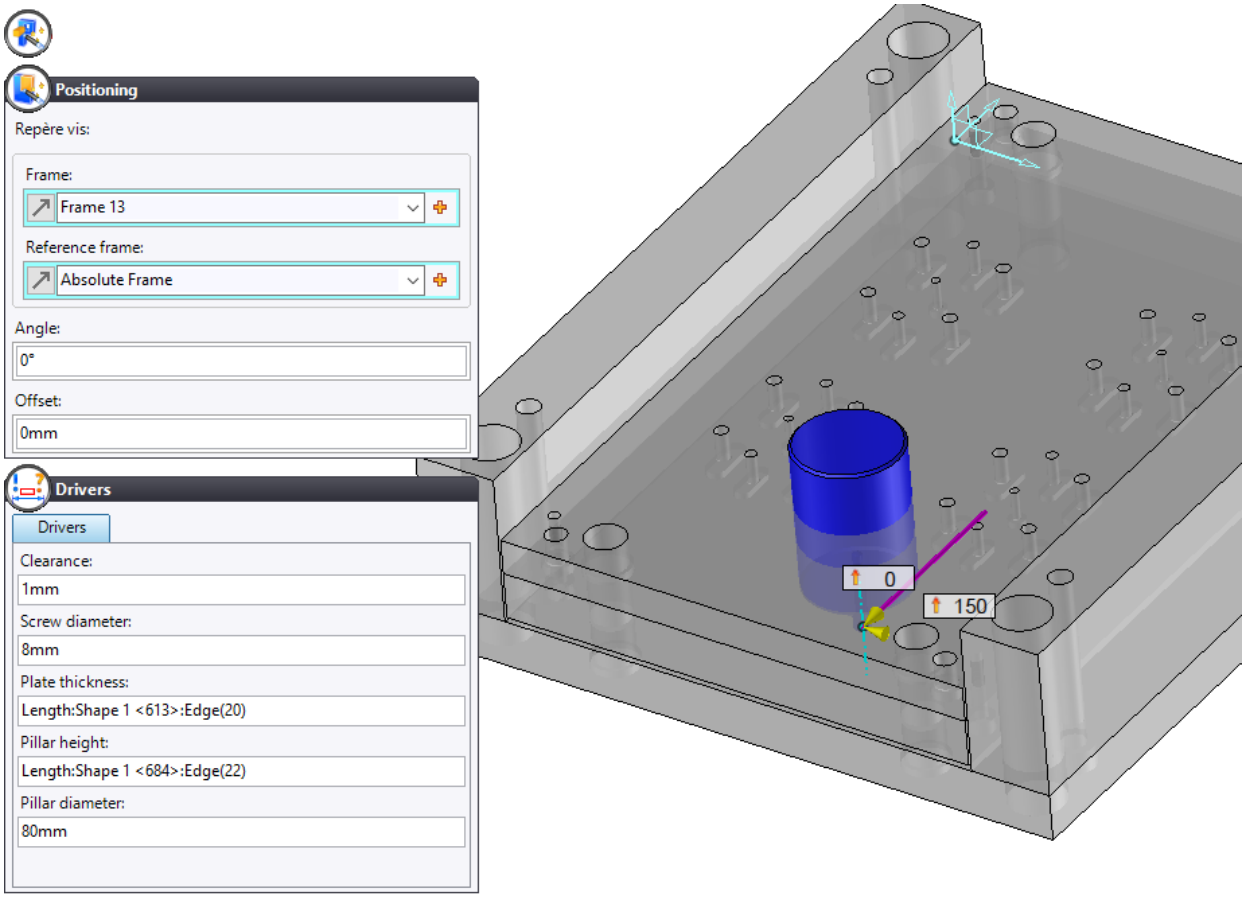
Support pillars

We will insert four support pillars on the ejector set whose positioning face will be the bottom face of the clamp plate.

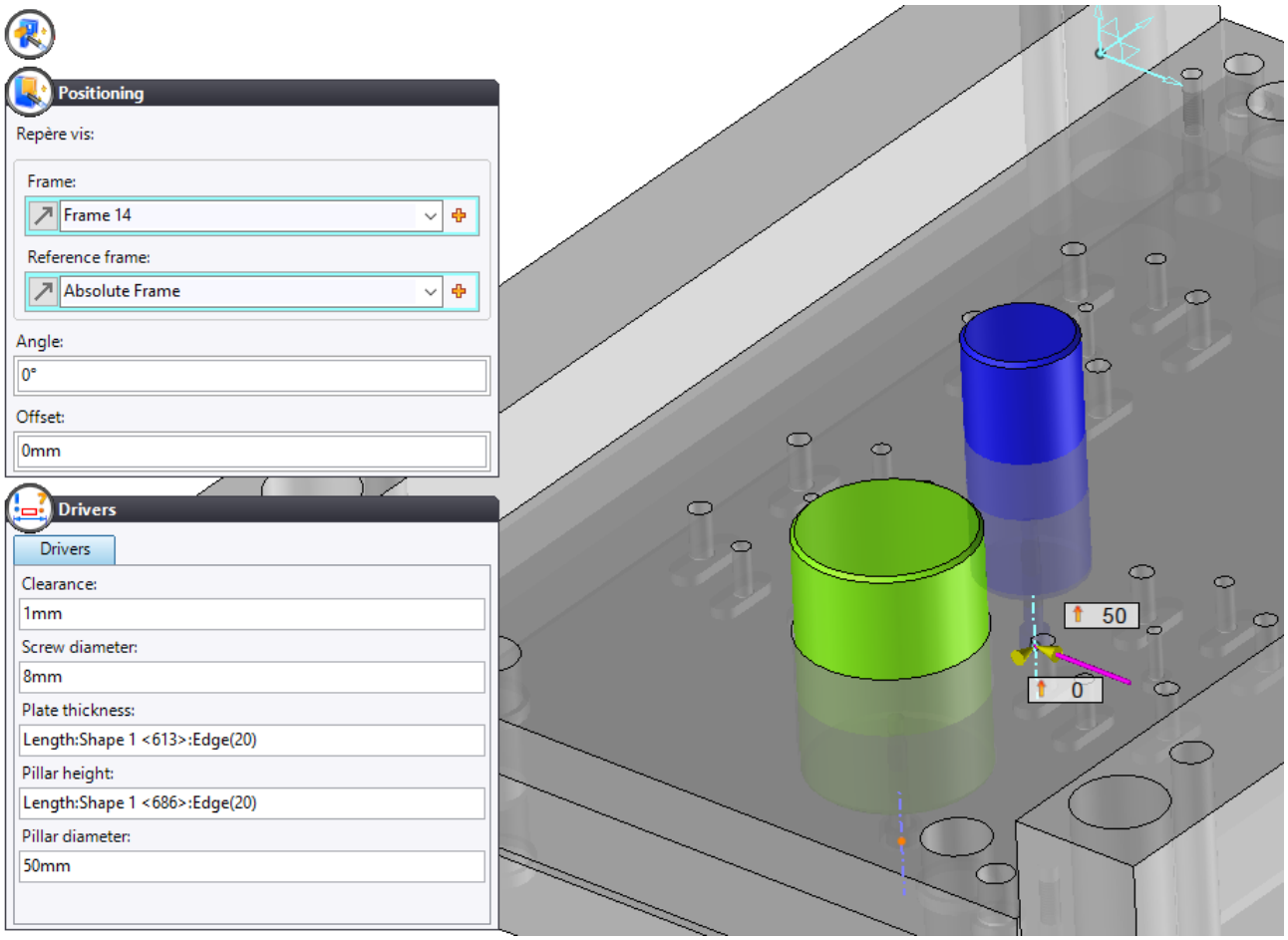




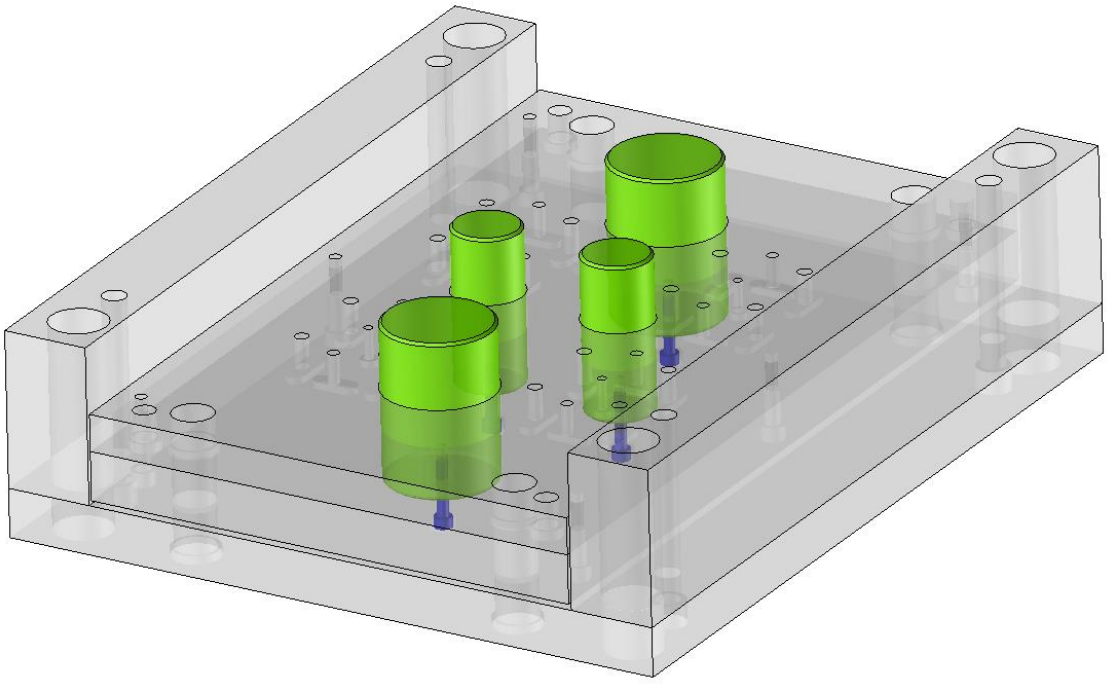
- From the Project tree, open the *Ex02 - Multi-cavity mold > A - User library > 1 - Pillar support* folders, select the *Pillar support* family document and drag and drop it into the graphics area.
- Select the **absolute frame** as the **reference frame**, position the first support pillar on the clamp plate's bottom face, then enter the following coordinates.
- For the **plate thickness**, click on the icon, select the **Associative value** option and click on an edge defining the thickness of the clamp plate. For the **pillar height**, proceed in the same way, this time clicking on an edge defining the height of the rails.



- **Confirm** the proposed processes (pillar+screw).
- Include the same component for the second support pillar by adjusting the **diameter** to *50mm*, then repeat the previous operation for the **plate thickness** and the **pillar height**.

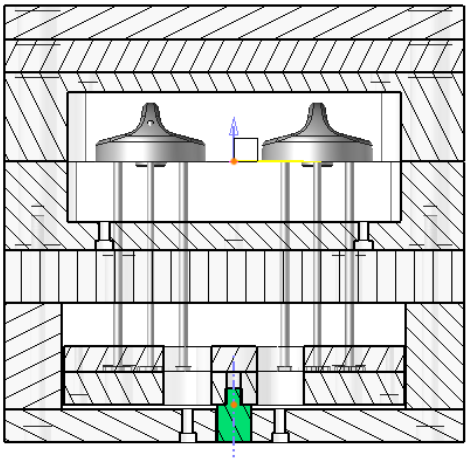


- Using a **circular pattern**, perform a **repetition** of the two support pillars around the Z axis by adjusting the total number to *2*.

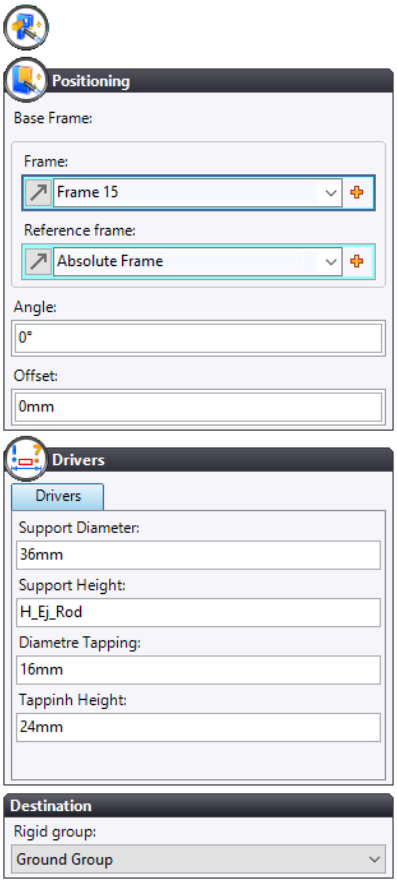


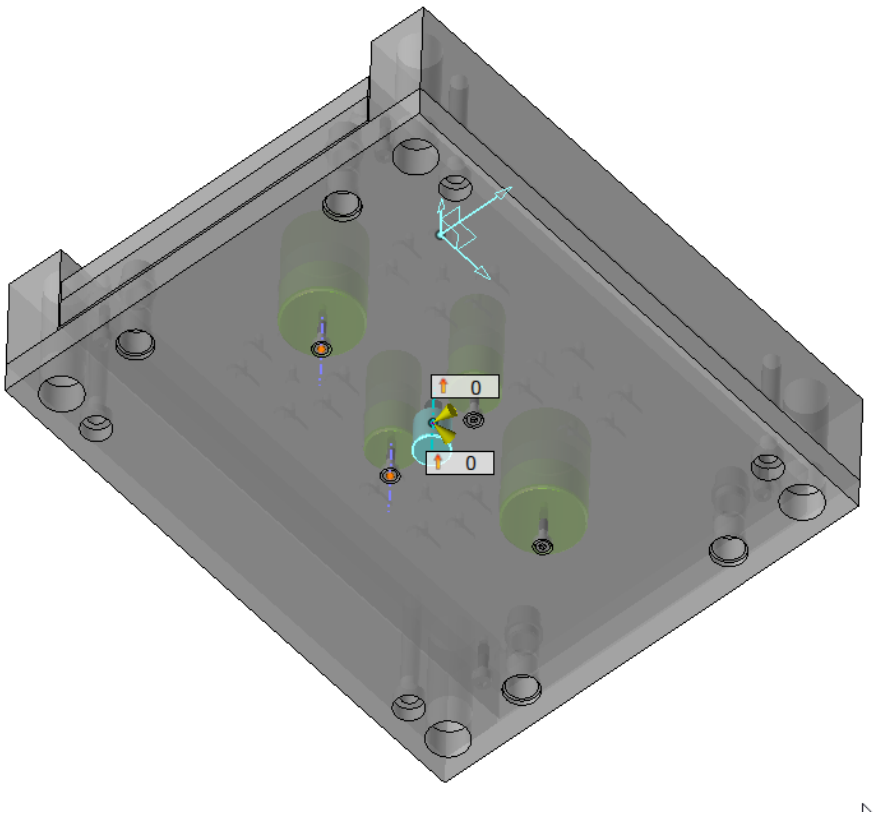
Ejector rod

The positioning face of the ejector rod will be the ejector plate's bottom face.



- To help you position the next component, only display the necessary elements to position this component.
- From the Project tree, open the *Ex02 - Multi-cavity mold > A - User library > 2 - Ejector rod* folders, select the *Ejector rod* family document and drag and drop it into the graphics area.
- Select **Absolute frame** as the **reference frame**, then position the component in the center of the mold and against the bottom face of the ejector plate. Reverse the frame direction if necessary. Create a **distance parameter** for the rod length.





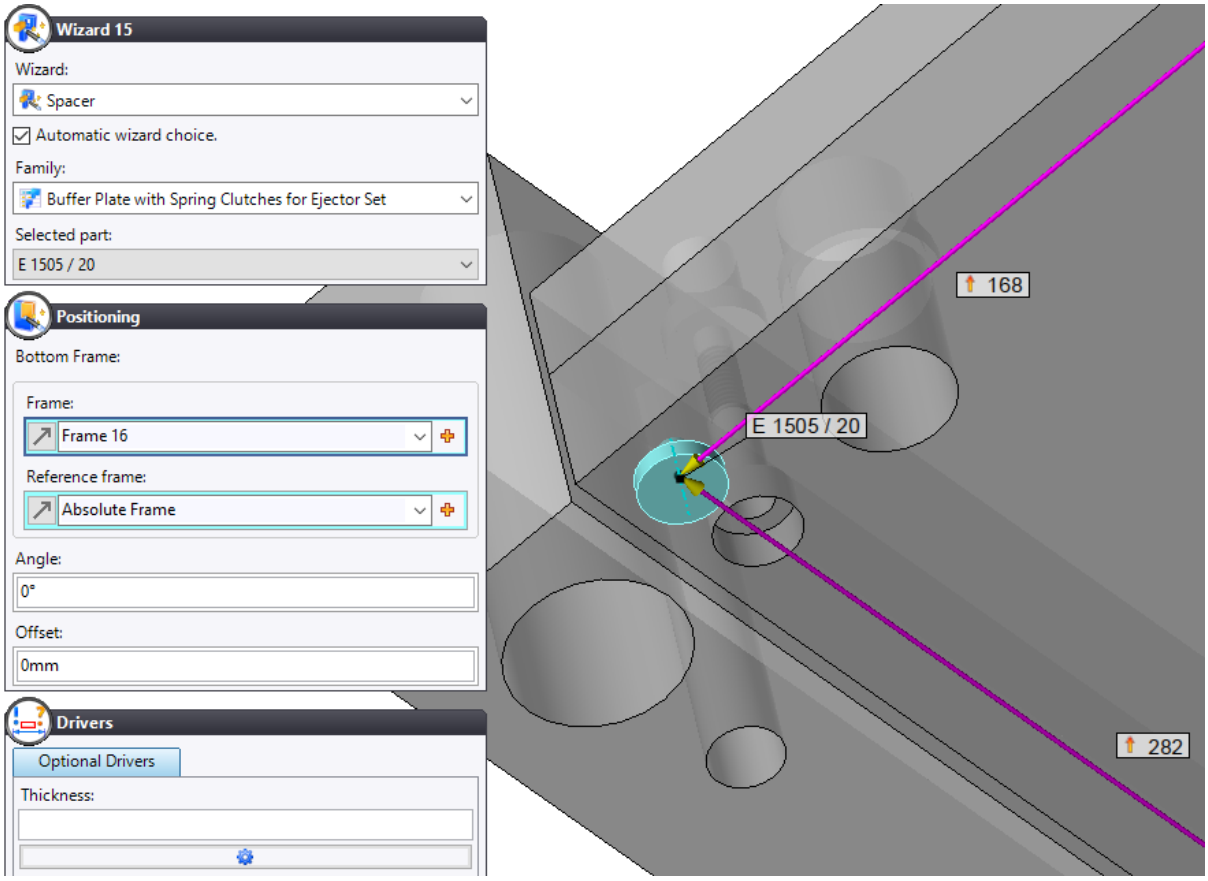
After confirming the wizard, the process dialog box appears since several choices are possible.

- Select the **Ejector rod** process.

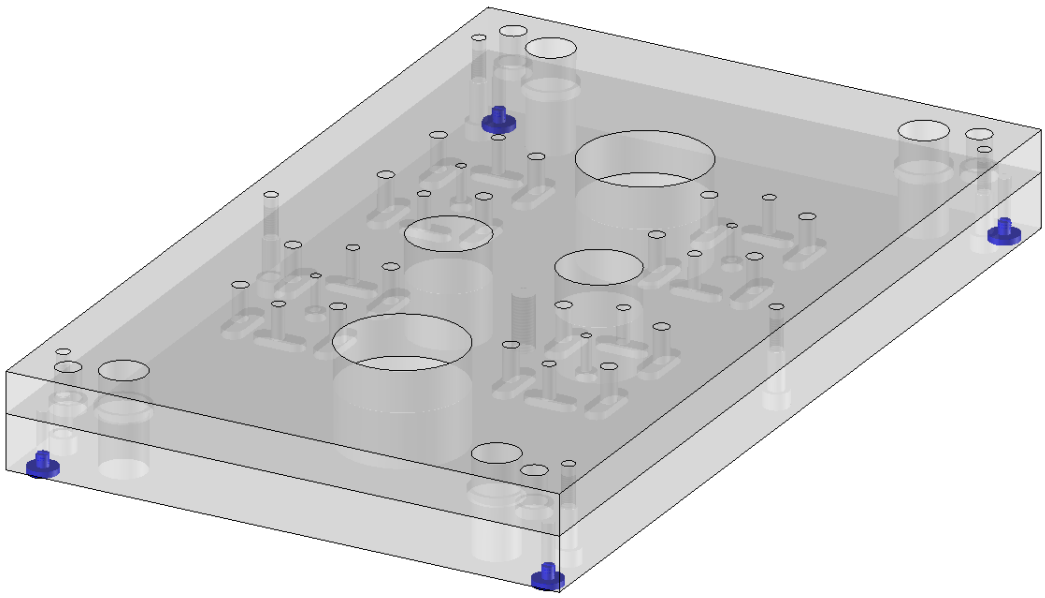
Note: You can assign this default process to this component by moving it to the project's **Favorites** folder.

Spacer

- Hide the clamp plate to make it easier to position the component.
- From the Project tree, open the *Ex02 - Multi-cavity mold > A - User library > 3 - Buffer plate* folders, select the *Buffer plate* family document and drag and drop it into the graphics area.
- Select the **E 1505/20** code and select **Absolute frame** as the **reference frame**.
- Position the component against the ejector plate's bottom face and adjust the following parameters. Reverse the frame direction if necessary.

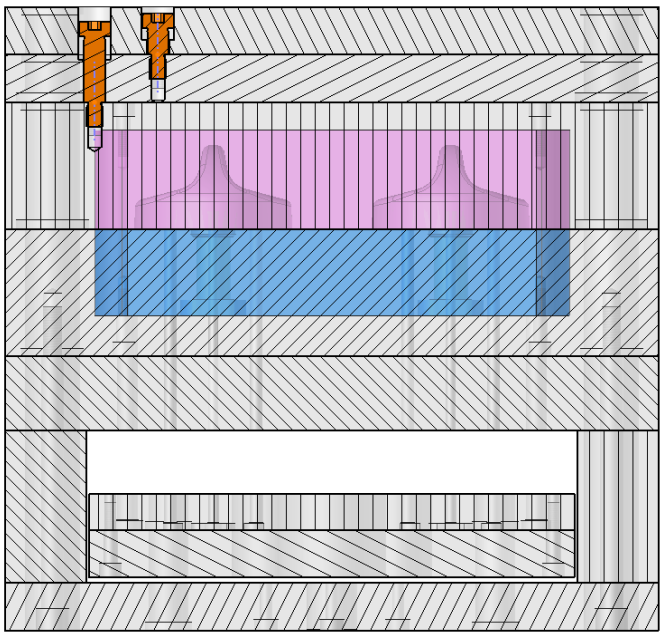


- **Confirm** the default process.
- Perform a repetition by double symmetry by reusing the **Double Sym XZ - YZ** pattern.

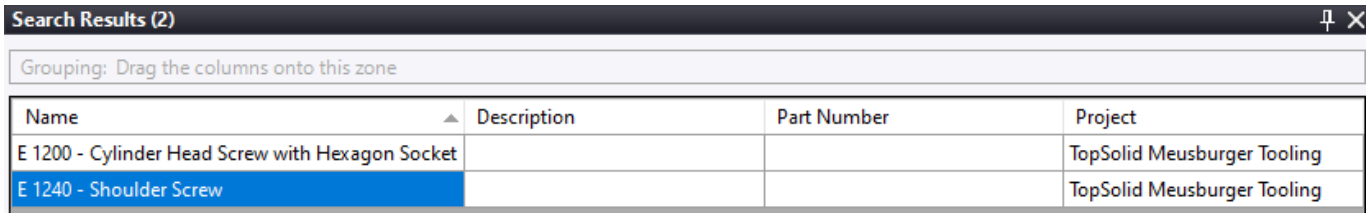


Ground puller bolts

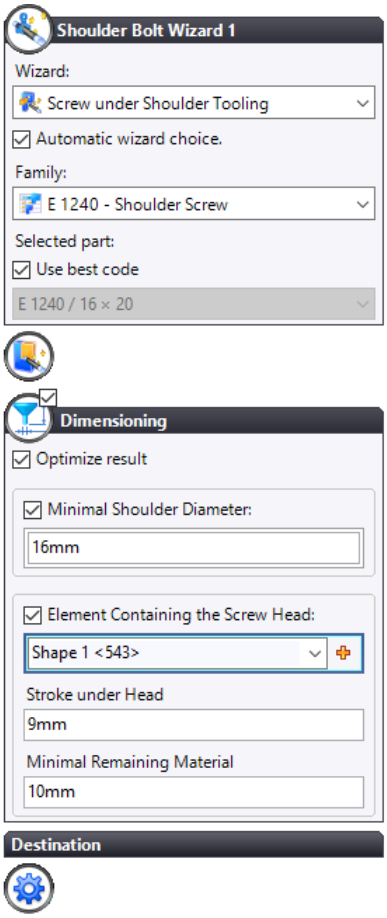
We will insert two shoulder bolts that will be used to create several collinear movements with delay in order to break the sprue as the mold opens. These bolts will be repeated by double symmetry.

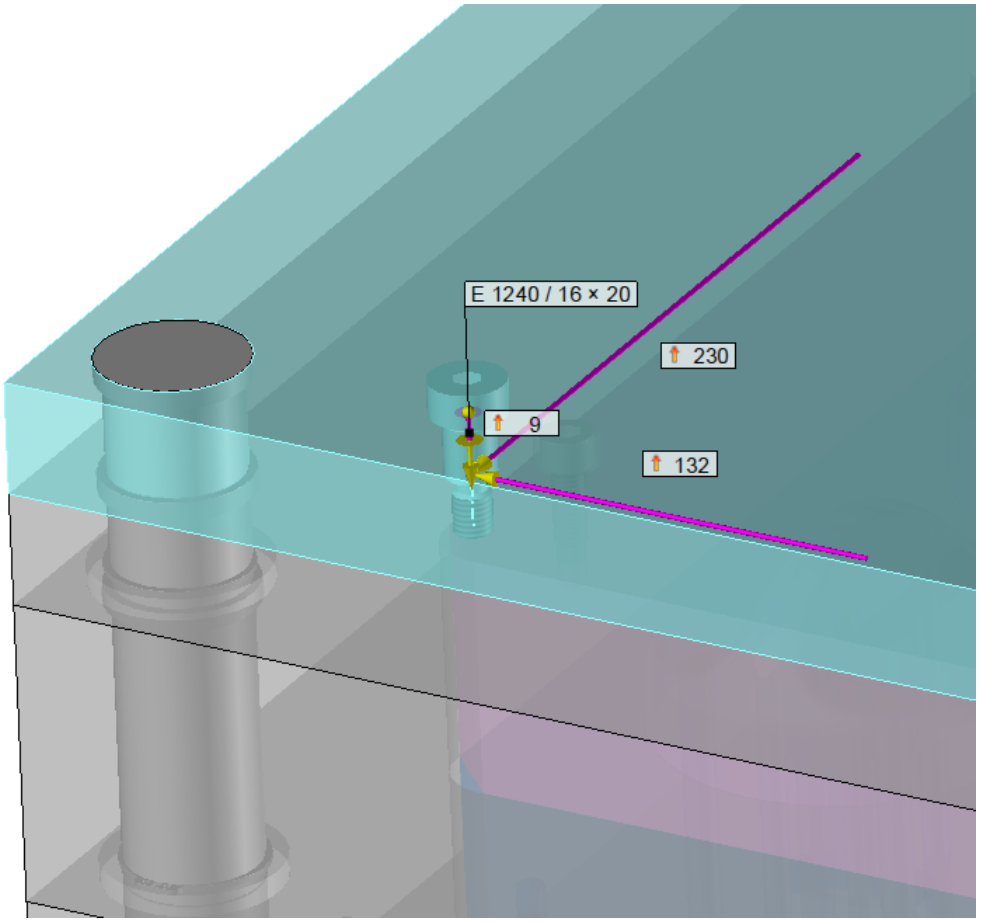


- Click on the **Meusburger screws** search icon.

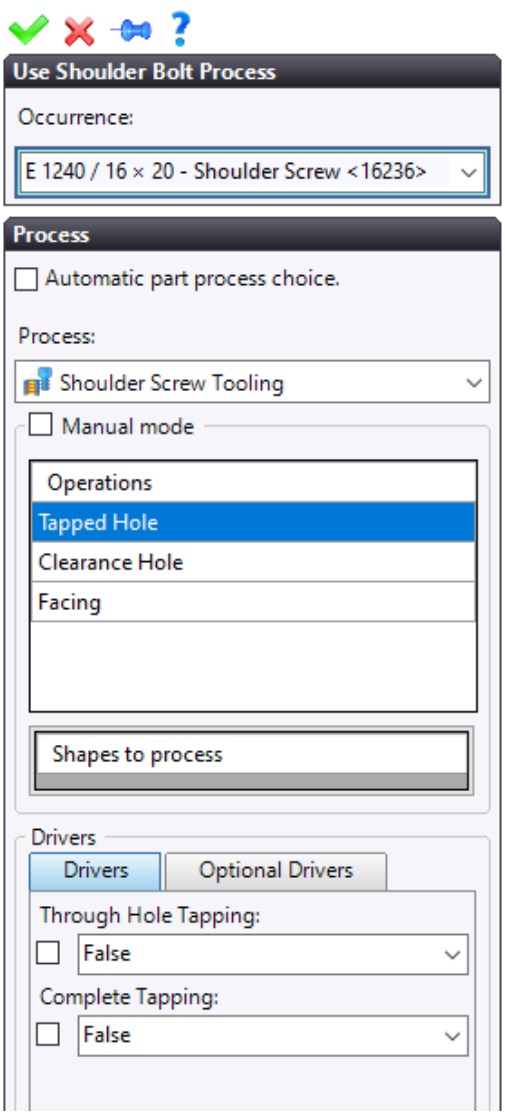


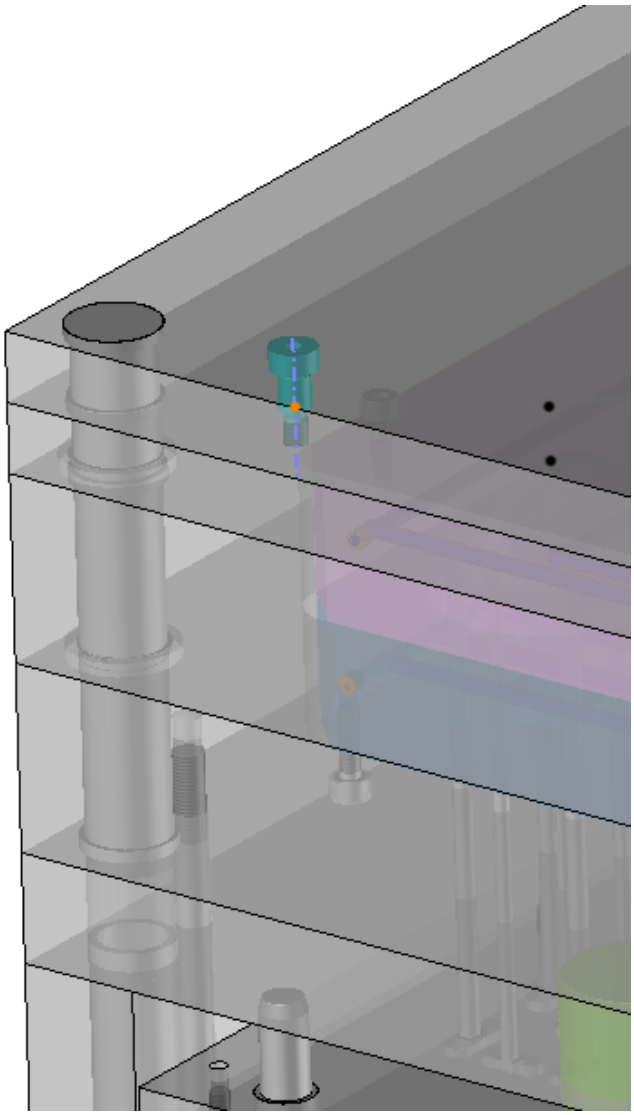
- Drag and drop the **E 1240** component into the graphics area, then position it against the external face of the support plate and adjust the following parameters. Select the clamp plate as the **element containing the screw head**.



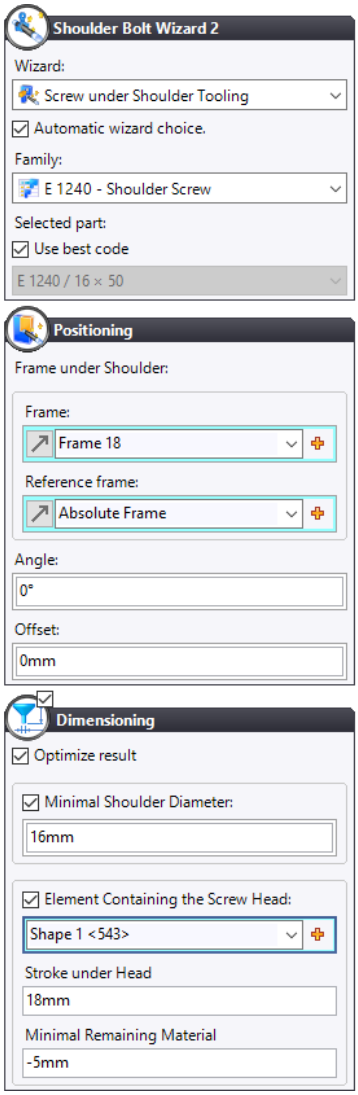


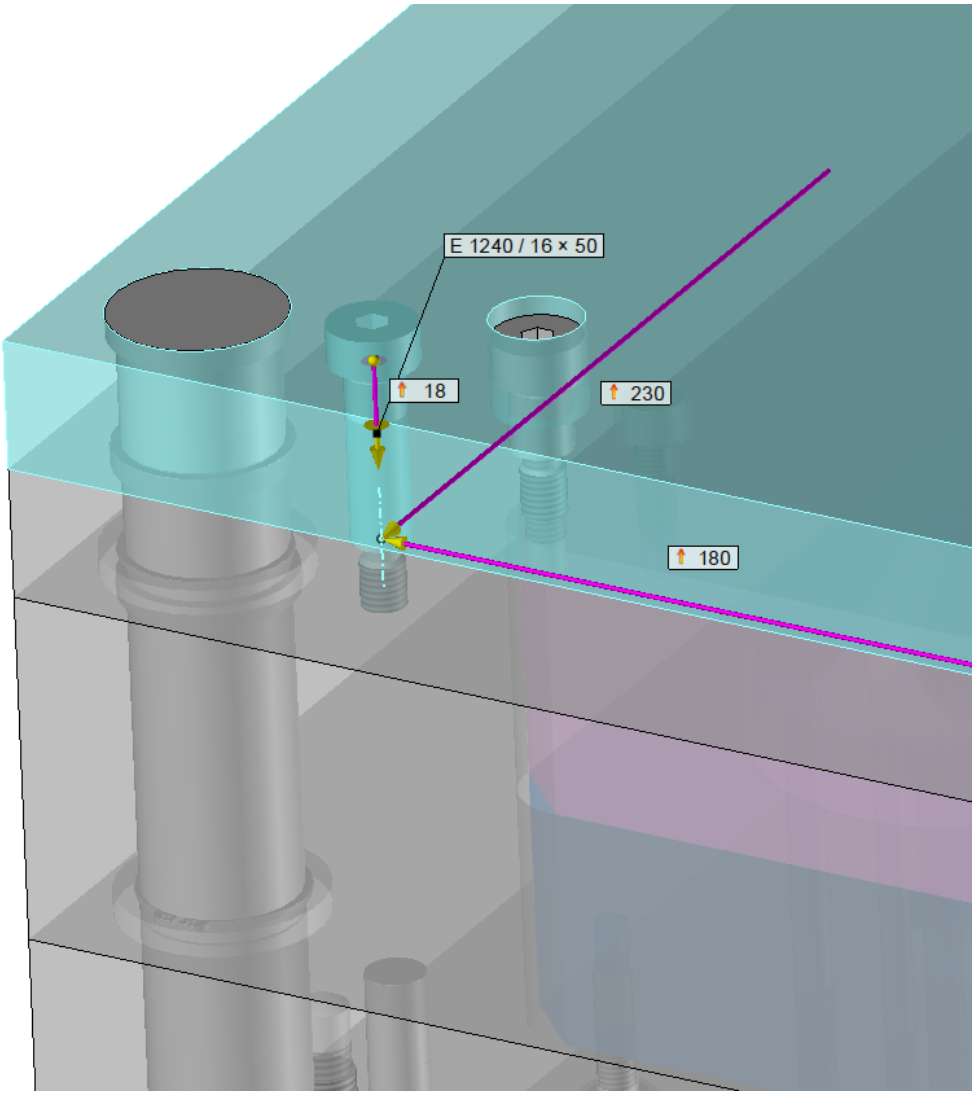
- In the process dialog box, adjust the following parameters.



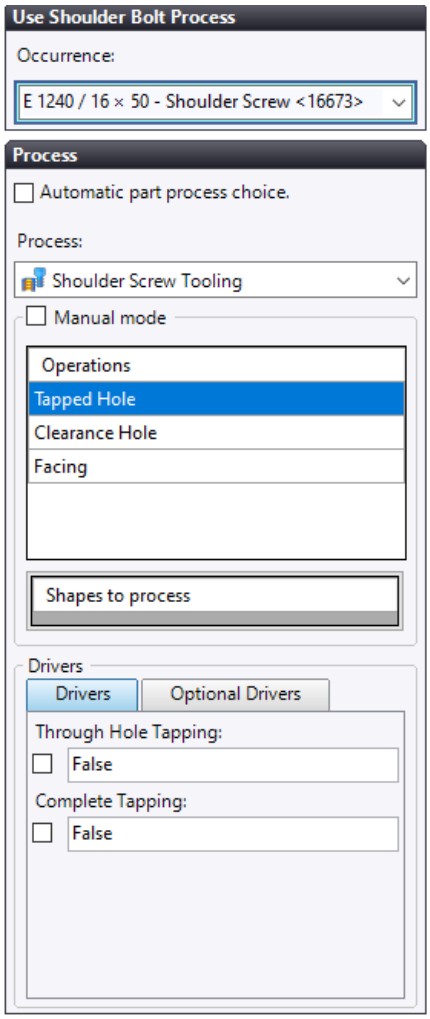


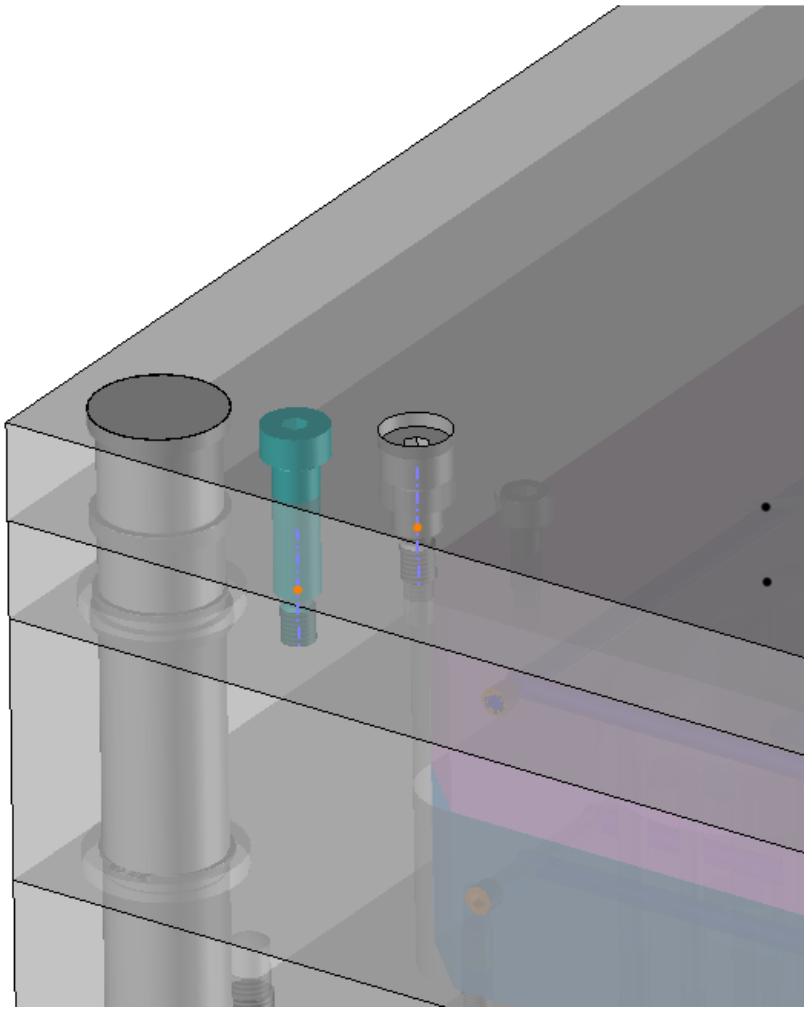
- Include the same component against the top face of the A plate, then adjust the following parameters. Select the clamp plate as the **element containing the screw head**.





- In the process dialog box, adjust the following parameters.

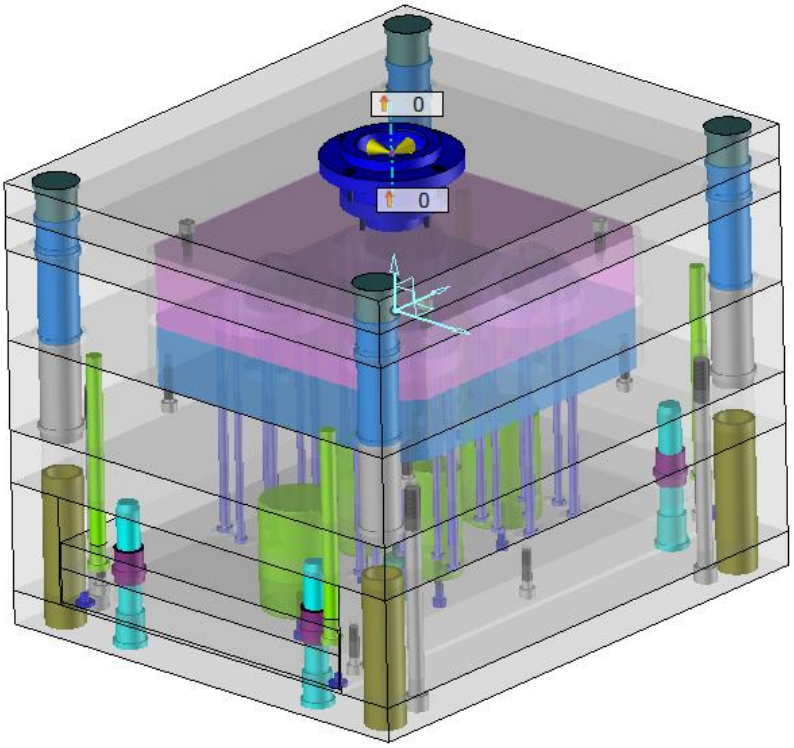




- From the **Construction** tab, select **Repetition**, then select the shoulder bolts and perform a double symmetry by reusing the **Double Sym XZ - YZ** pattern.
- Modify the color of the components via the Parts tree.

Locating ring

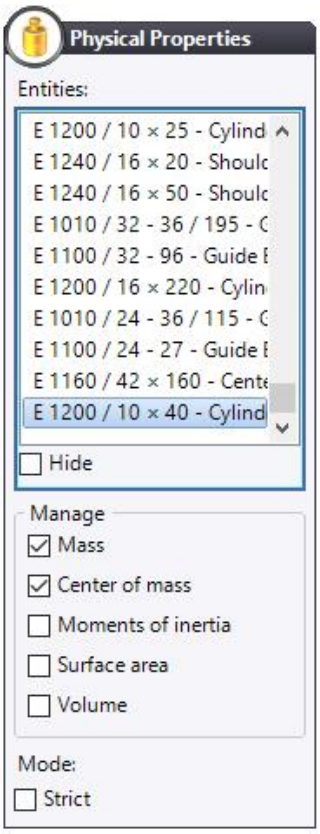
- Display the top clamp plate again.
- From the Project tree, open the *Ex02 - Multi-cavity mold > A - User library > 5 - Locating ring* folders, select the *Locating ring* assembly document and drag and drop it into the graphics area.
- Position the component against the top face of the clamp plate and centered on the mold.

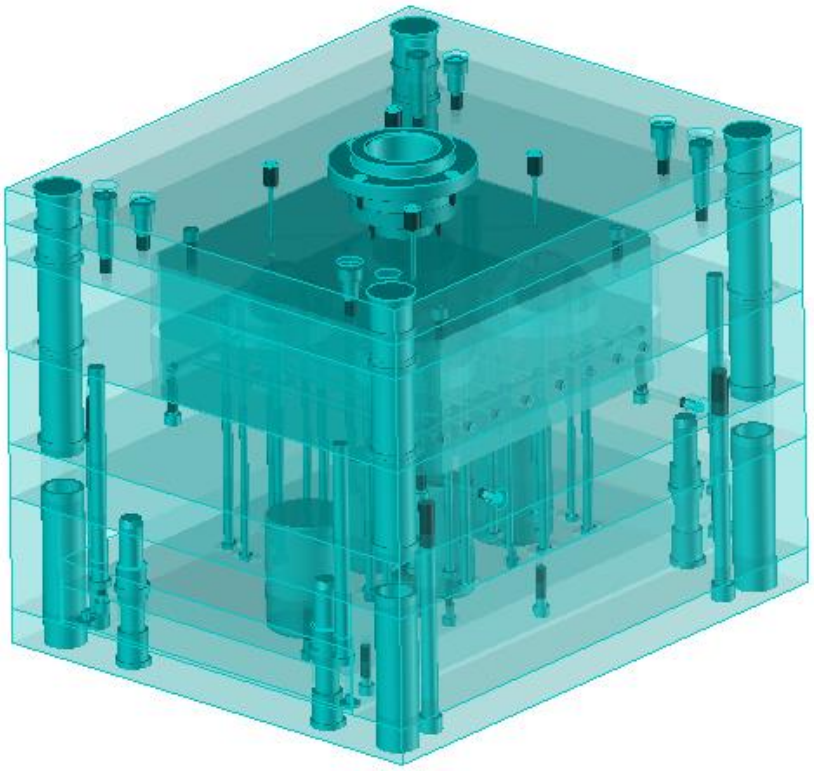


- After confirming the wizard, select the default processes.

Lifting eye bolt

- From the Project tree, open the *Ex02 - Multi-cavity mold > A - User library > 6 - Eye bolt* folders and drag and drop the *Eye bolt M16x27* part document into the graphics area.
- Position the component against the side face of the B plate as shown below. You can calculate the center of mass using our tool for analyzing physical properties that can be accessed from the **Analysis** tab > **Physical Properties**. This point must be calculated at the end of the mold design.



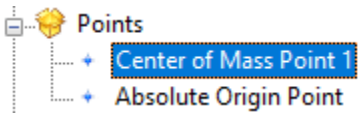


If you want to position a lifting eye bolt on the A side and B side, you have to select the parts of the B side set, and then select those of the A side set. You will then have three centers of mass: one for the whole mold and two for the fixed and movable parts of the mold.

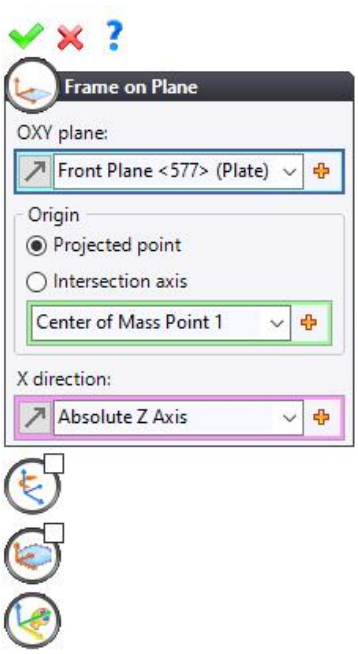
A new operation is created in the Operations tree.

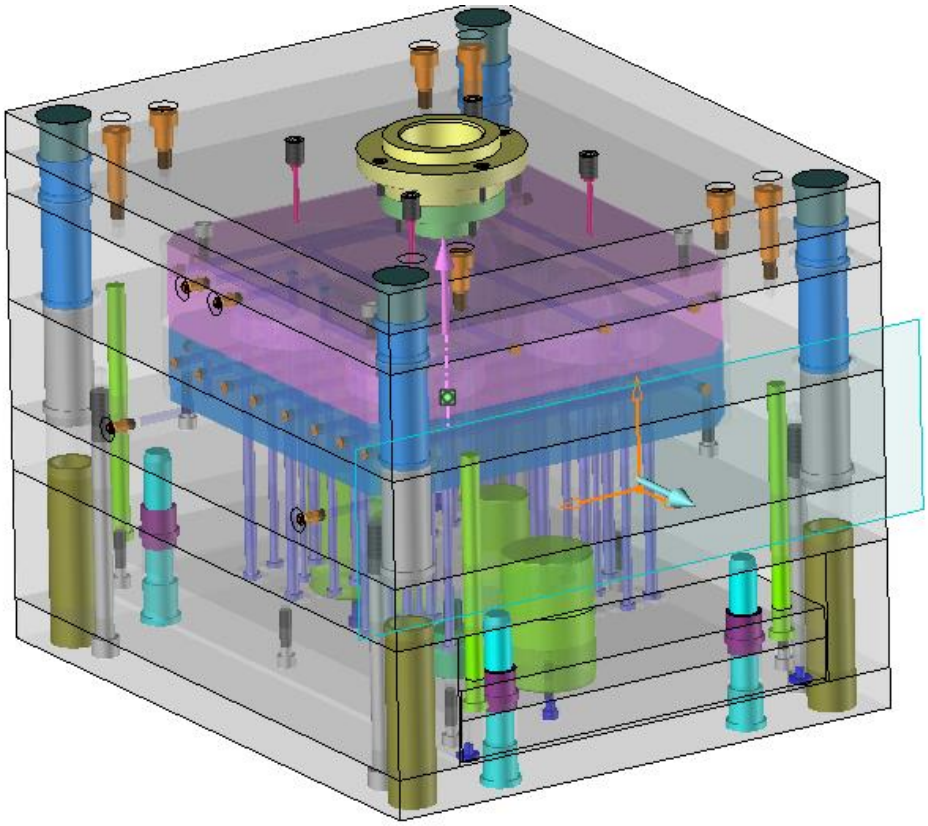
Note: If you add a new part, you need to move this operation above this new inclusion in order to recalculate the new center of mass.

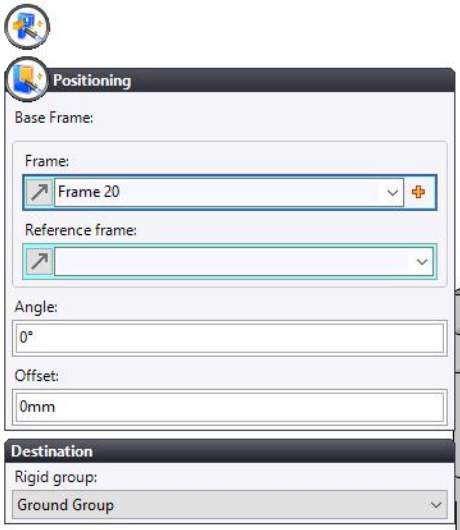
A **center of mass point** is created in the Entities tree.

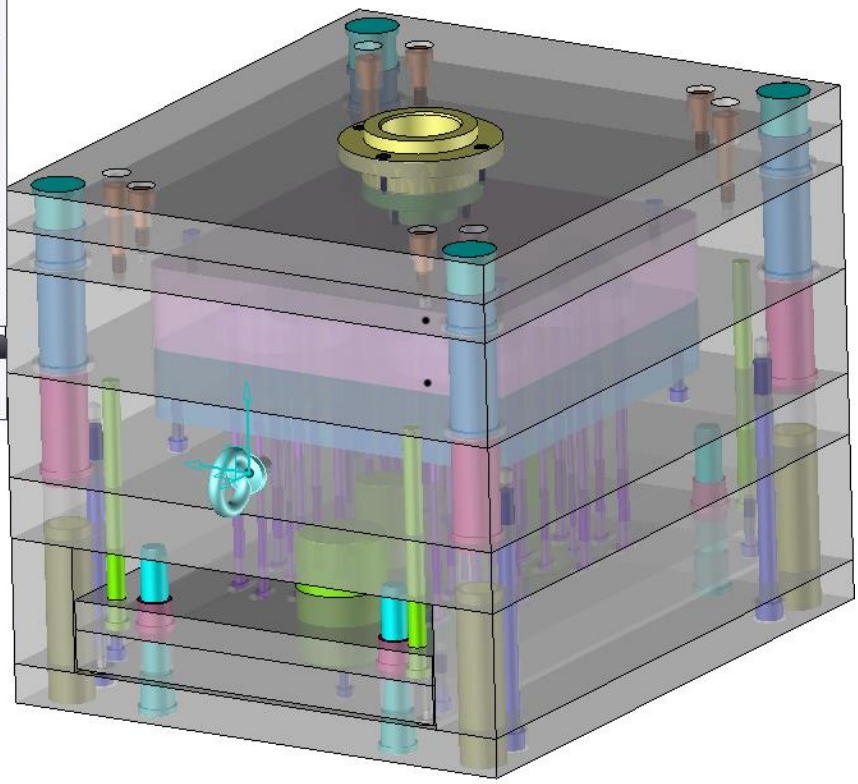


- To position the lifting eye bolt, create a frame on face and projected point on this face.









- **Confirm** the default process.
- **Save** the document.

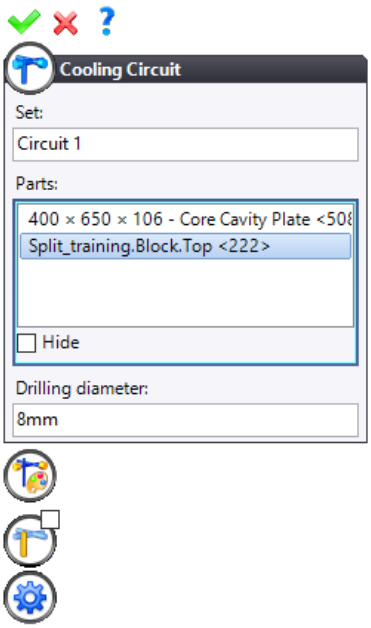
Creating the cooling circuit

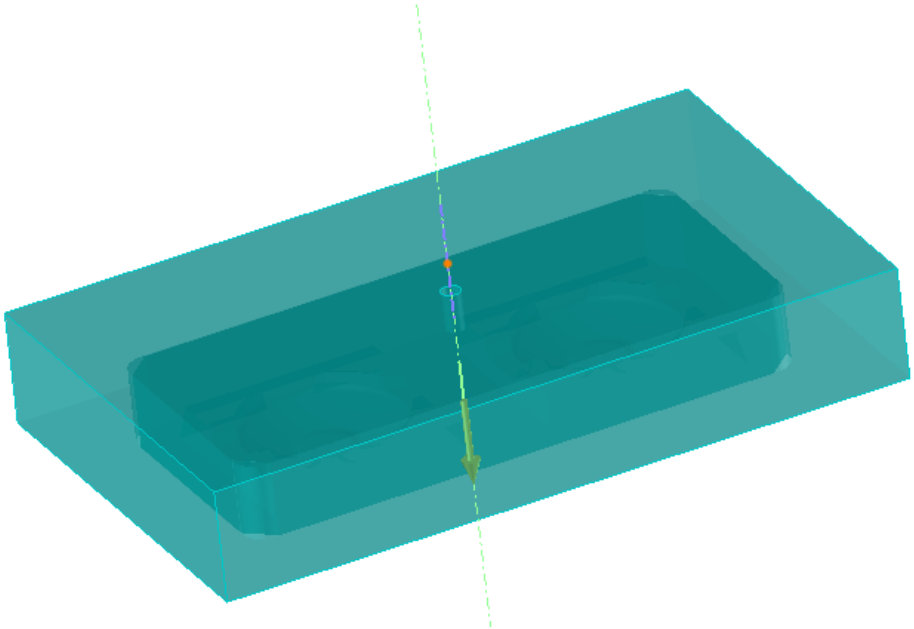
The cooling circuit is based on a 3D sketch. Before continuing with the current mold, we will create a cooling circuit using a simple example.

- Open the *3D Sketch* mold document from the *Ex02 - Multi-cavity mold > B - 3D sketch for cooling and runner circuits* folders.

Configuring the cooling circuit

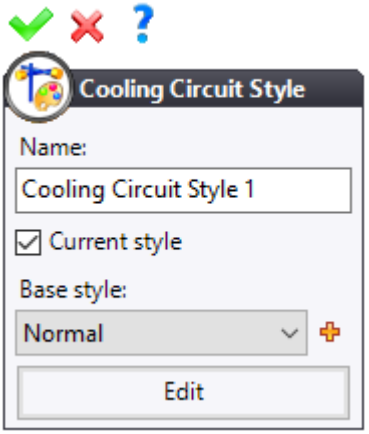
- Hide all parts except the cavity block and the A plate.
- To better view the circuit, adjust the transparency of both parts to *80%*.
- From the **Mold** tab, select the **Cooling Circuit** command and fill in the dialog box as shown below. The parts to be selected will be those affected by the cooling circuit. Enter a **drilling diameter** of *8mm.*



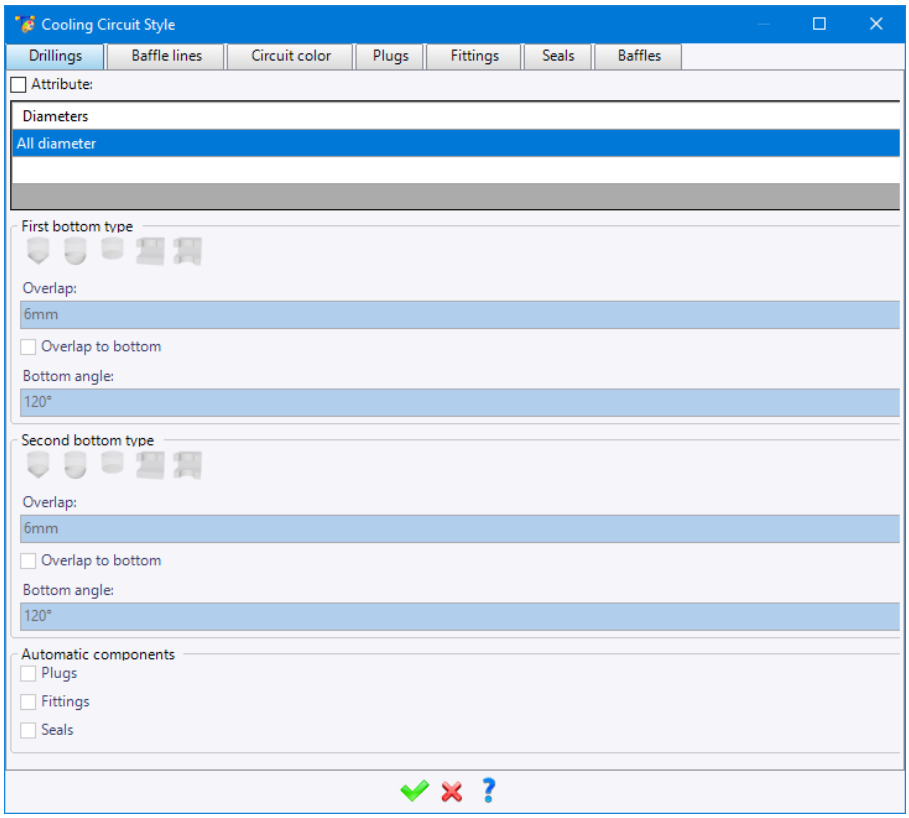


The **Style** option allows you to manage the different settings of the cooling circuit using a style that can be reused at any time.

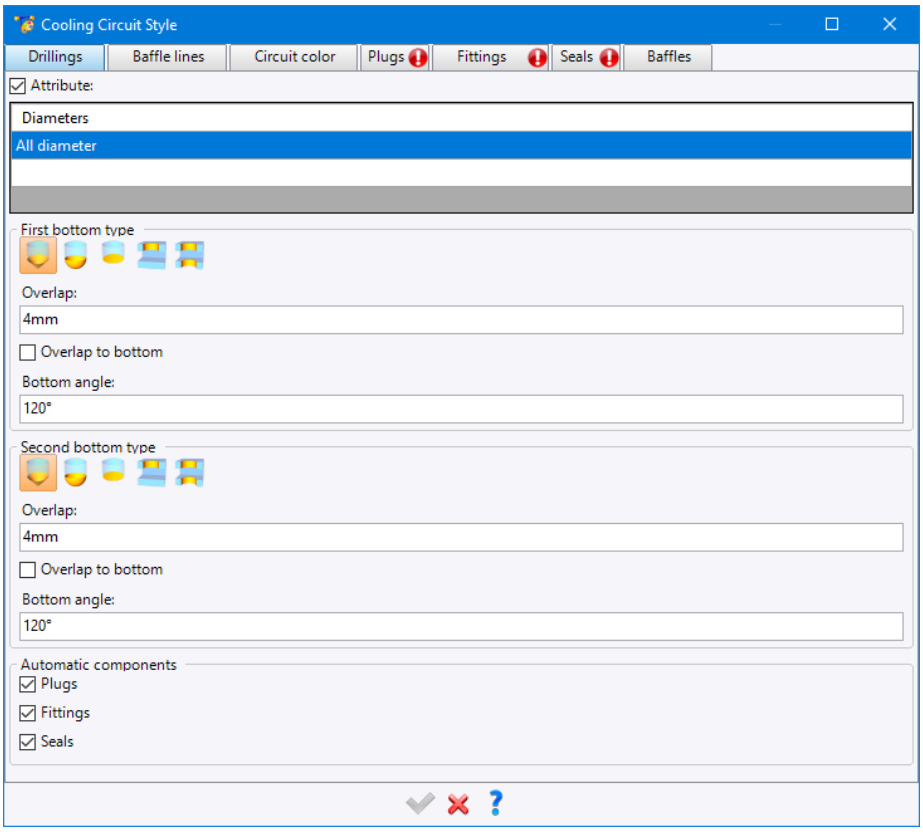
- Click on the **Style** icon, click on the icon and select **Cooling Circuit Style**.
- Rename the style if necessary, then click on the **Edit** button.



The **Cooling Circuit Style** dialog box opens.

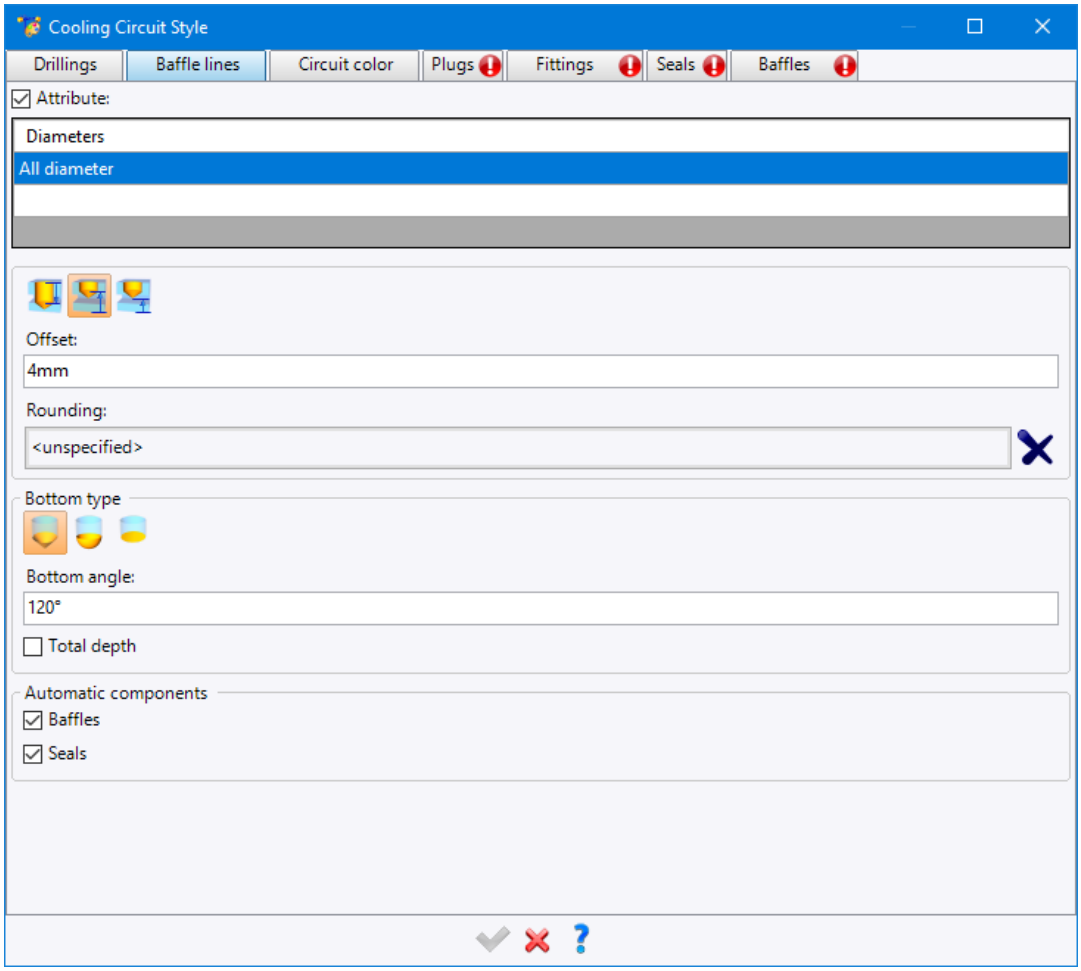


- From the **Drillings** tab, check the **Attribute** box to enable the different settings and adjust the following parameters.

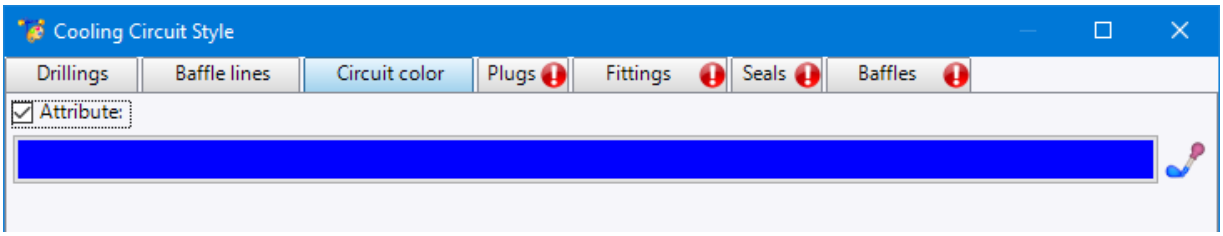


Note: The cooling components (plugs, fittings, seals, etc.) can be placed automatically by checking the corresponding boxes in the **Automatic components** field.

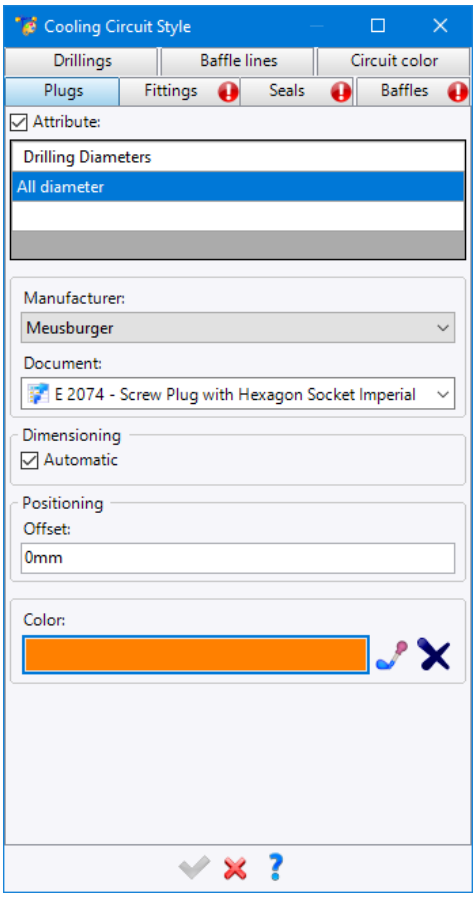
- From the **Baffle lines** tab, check the **Attribute** box to enable the different settings and adjust the following parameters.

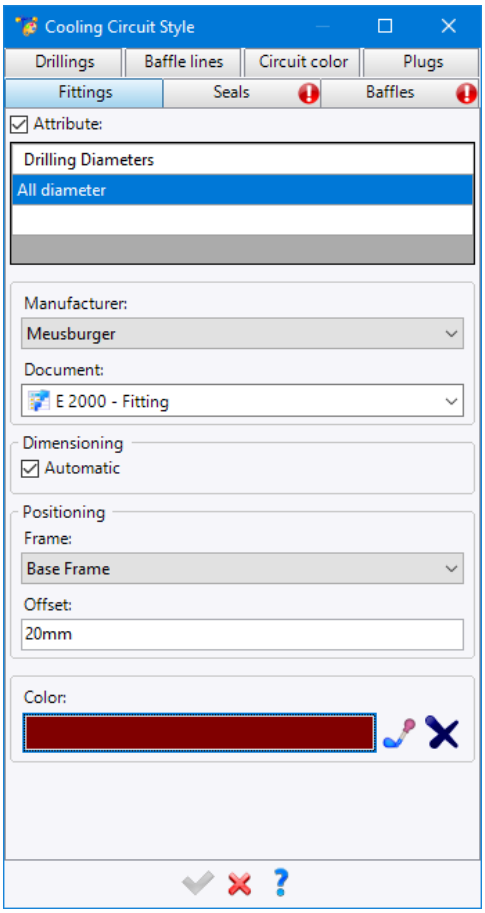


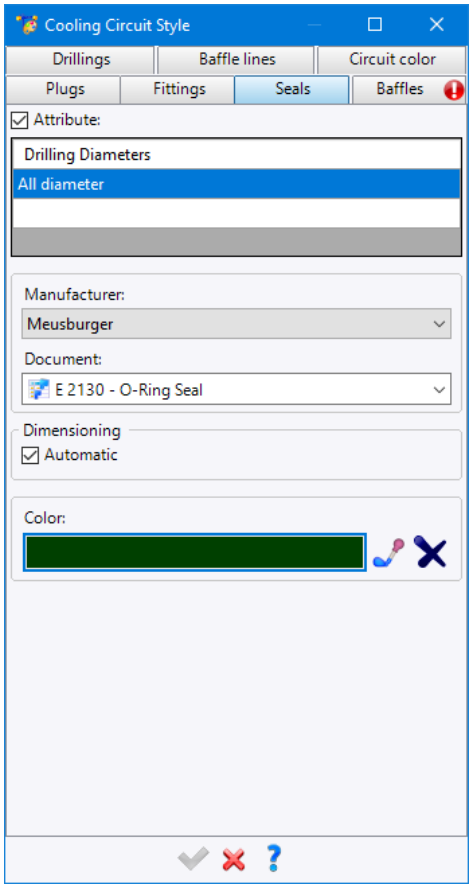
- From the **Circuit color** tab, adjust the color of the cooling circuit if necessary.

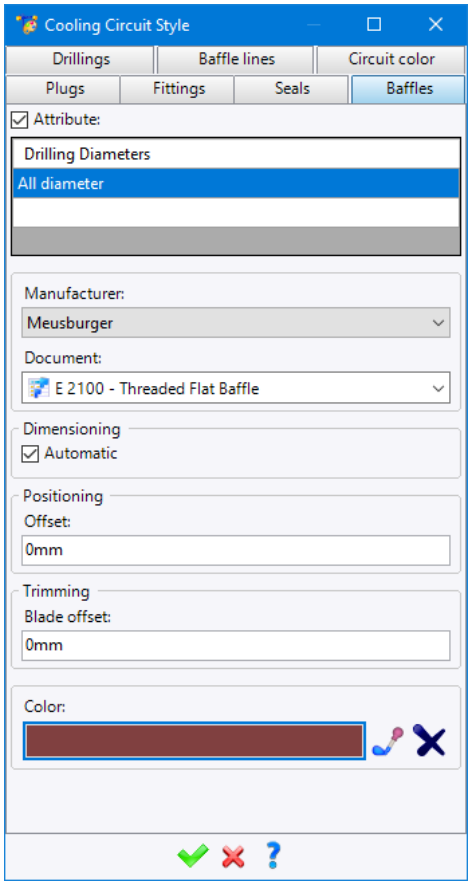


- From the **Plugs**, **Fittings**, **Seals** and **Baffles** tabs, adjust the following parameters.





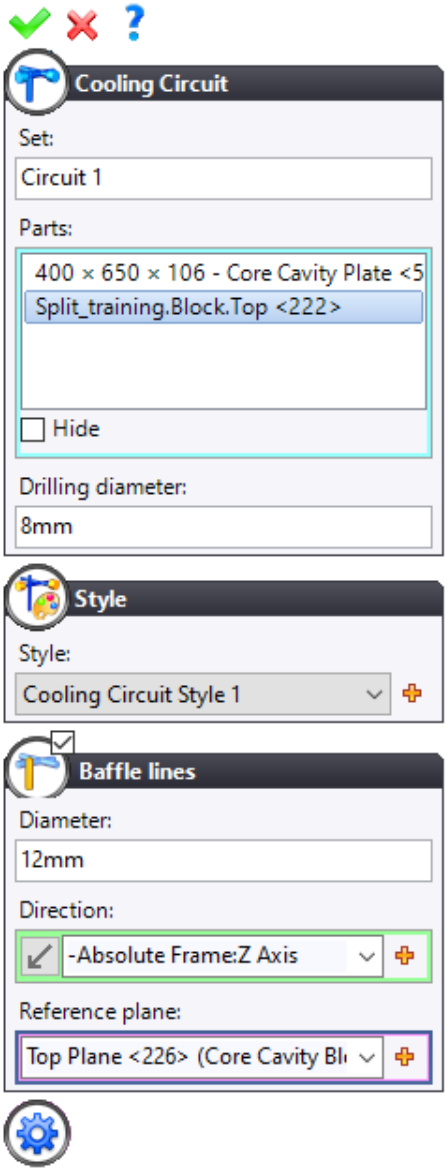


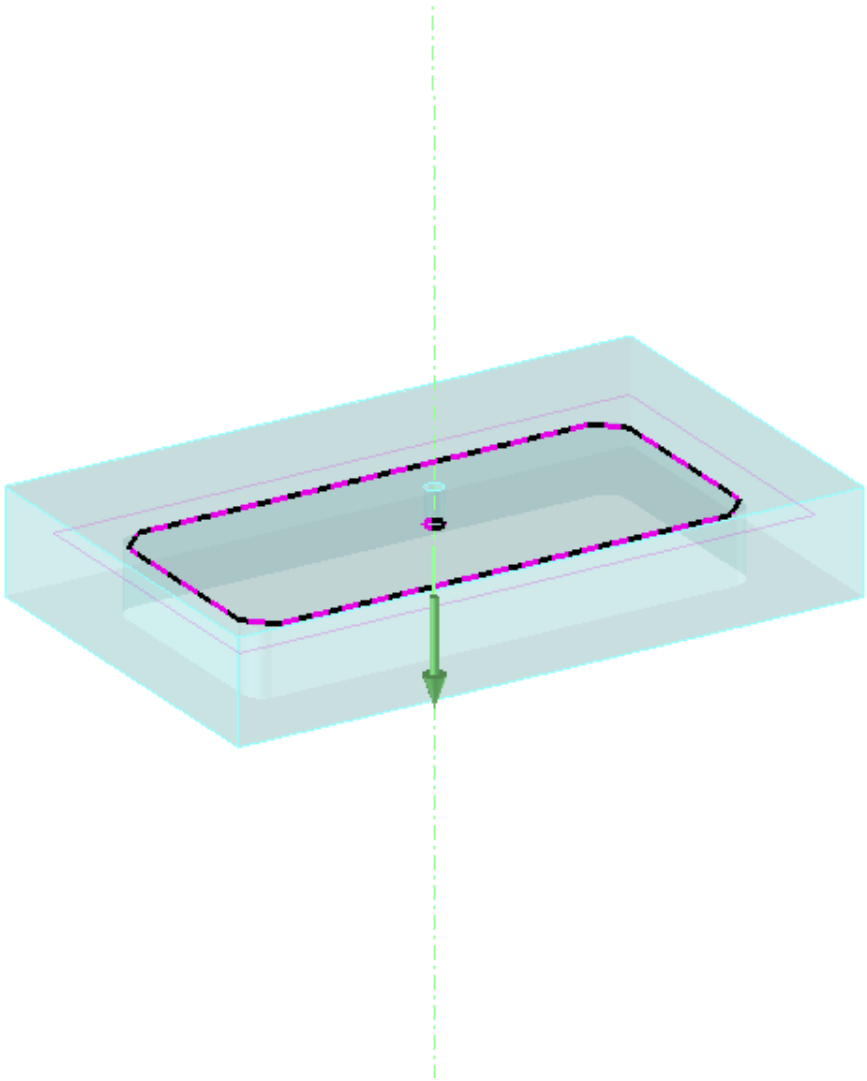


- **Confirm** all the cooling circuit style's dialog boxes. The style is created in the **Styles** > **Cooling Circuit Style** folders of the Entities tree and can be edited at any time.

Note: It may make sense to define your cooling circuit styles in a mold document template to save you from recreating them each time.

- In the **Cooling Circuit** command, enable the **Baffle lines** option and adjust the **diameter** to *12mm*. The **reference plane** is the starting face for the drillings. If you select the block first, the face in contact with the plate will be automatically selected. If necessary, you can change the face by hiding the core cavity plate for easy selection.

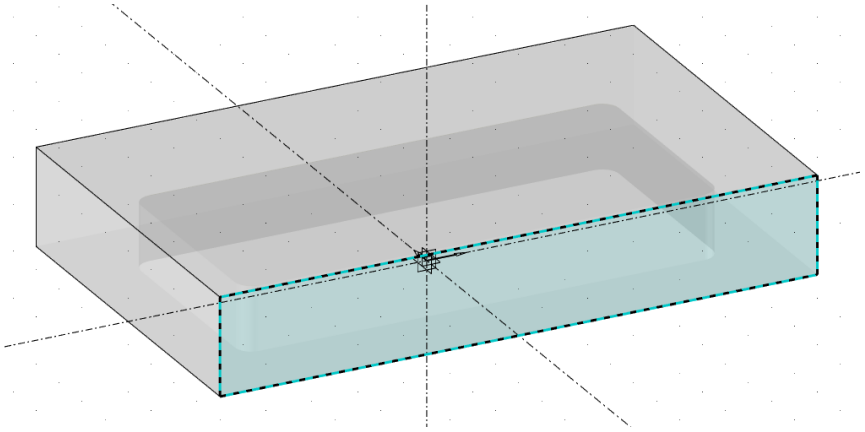




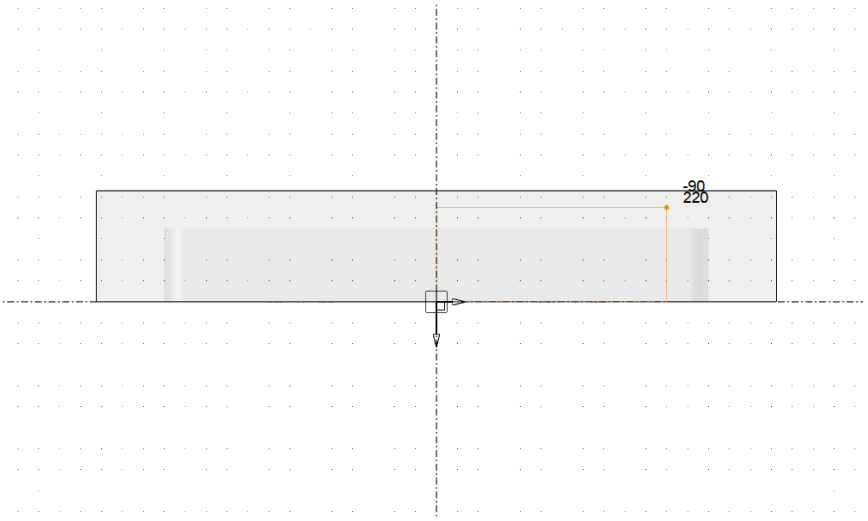
- **Confirm** the cooling circuit.

Creating the 3D sketch

- Right-click on the face as shown below and select the **View Along Normal** command or press **N** on your keyboard.

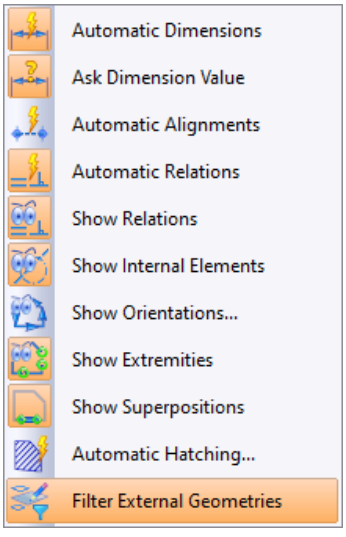


- Leave the face selected, right-click in the graphics area and select the **Set Input Plane** command.
- From the **3D Sketch** tab, select the **Contour** command and click on the face as shown below.

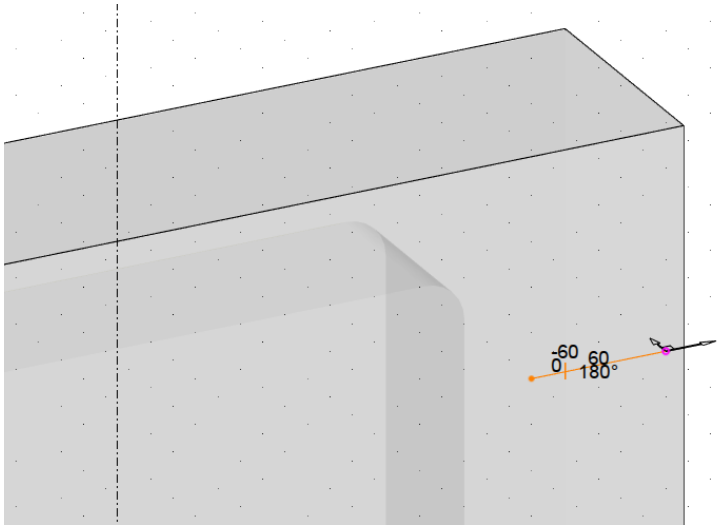


Note: You should avoid hooking an existing geometry when drawing the lines.

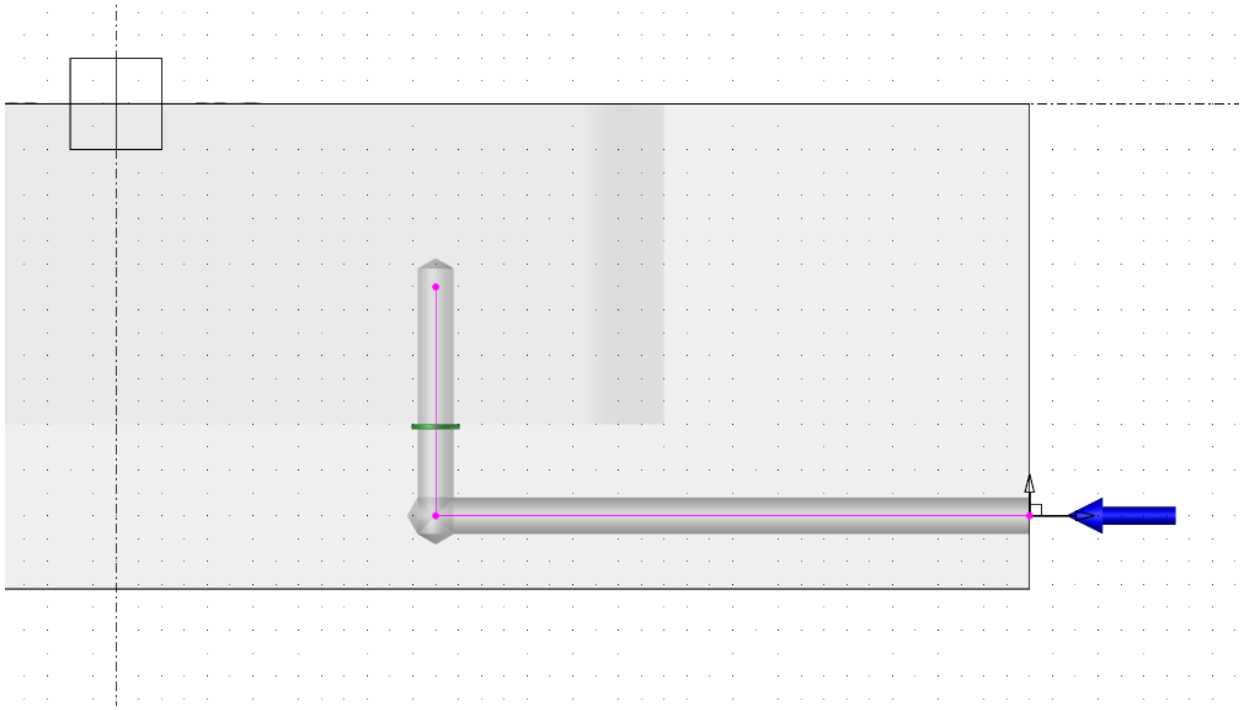
- In the sketch settings, select the **Filter External Geometries** mode.



- Put the view into perspective by clicking on the icon and change the orientation of the frame by pressing **Ctrl** + **Space bar** as shown below.

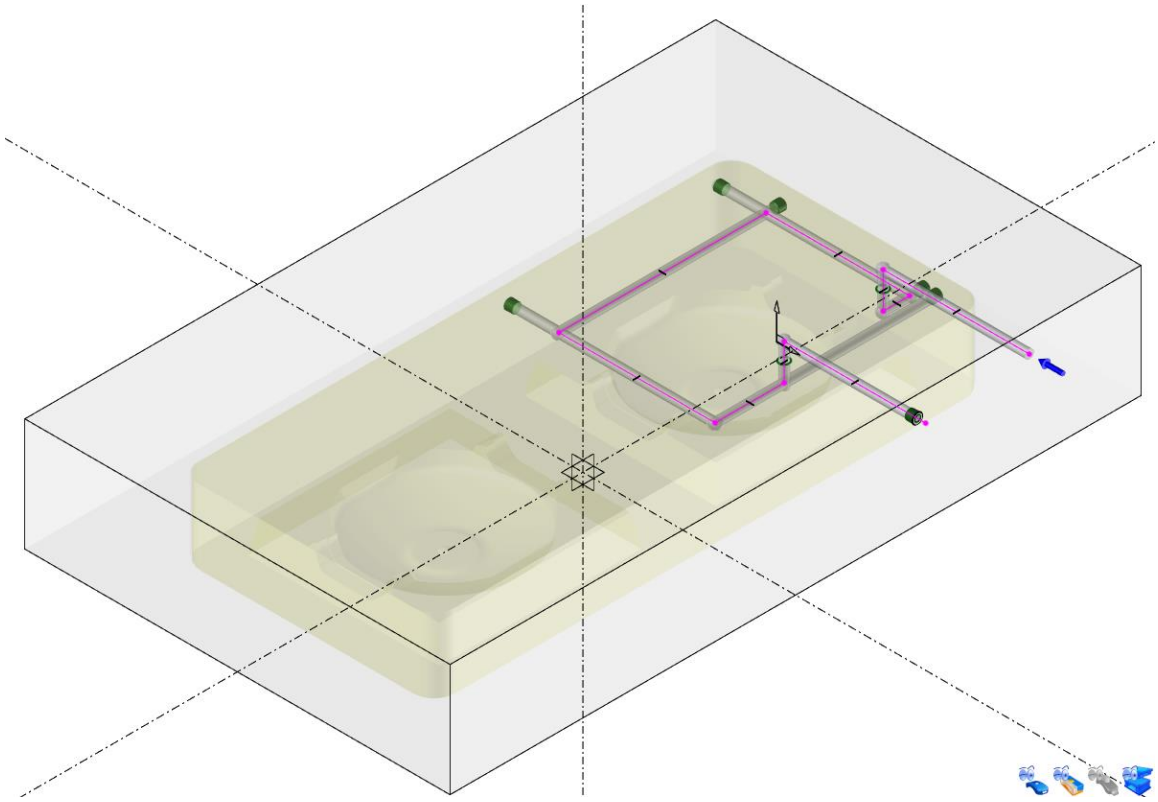


- Once the orientation is selected, select **View Sketch From Top** and continue the sketch by drawing two lines as shown below.



Note: Put the view into perspective and change the orientation of the frame again by pressing **Ctrl** + **Space bar**.

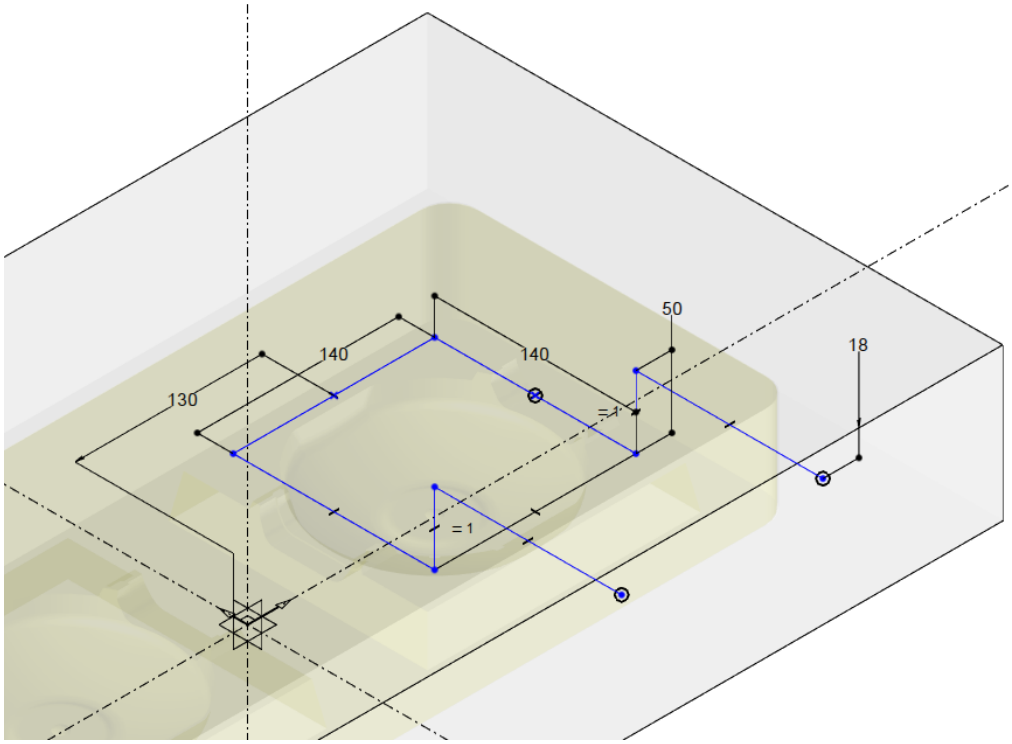
- Once the orientation is selected, select **View Sketch From Top** to continue the sketch. If necessary, rotate the view using the arrow keys on the keyboard.



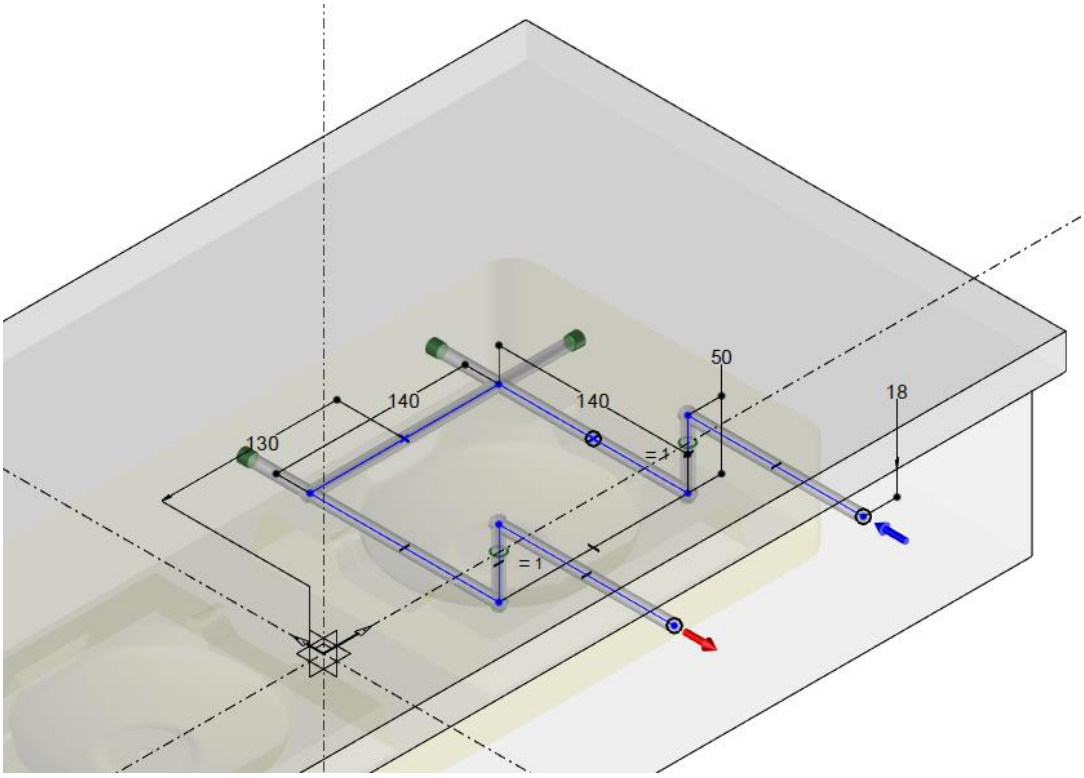
- Add a **coincidence** constraint between the start point and end point with the starting face.

Note: To only display the cooling sketch, you only have to click on the **Hide Cooling** icon at the top right of the graphics area.

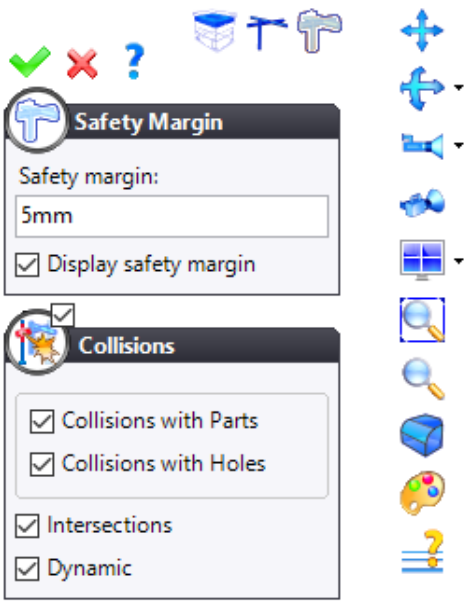
- Add the **alignment**, **equality** and **dimensioning** constraints as shown below. For the dimensions, use the faces and planes with the elements of the 3D sketch.



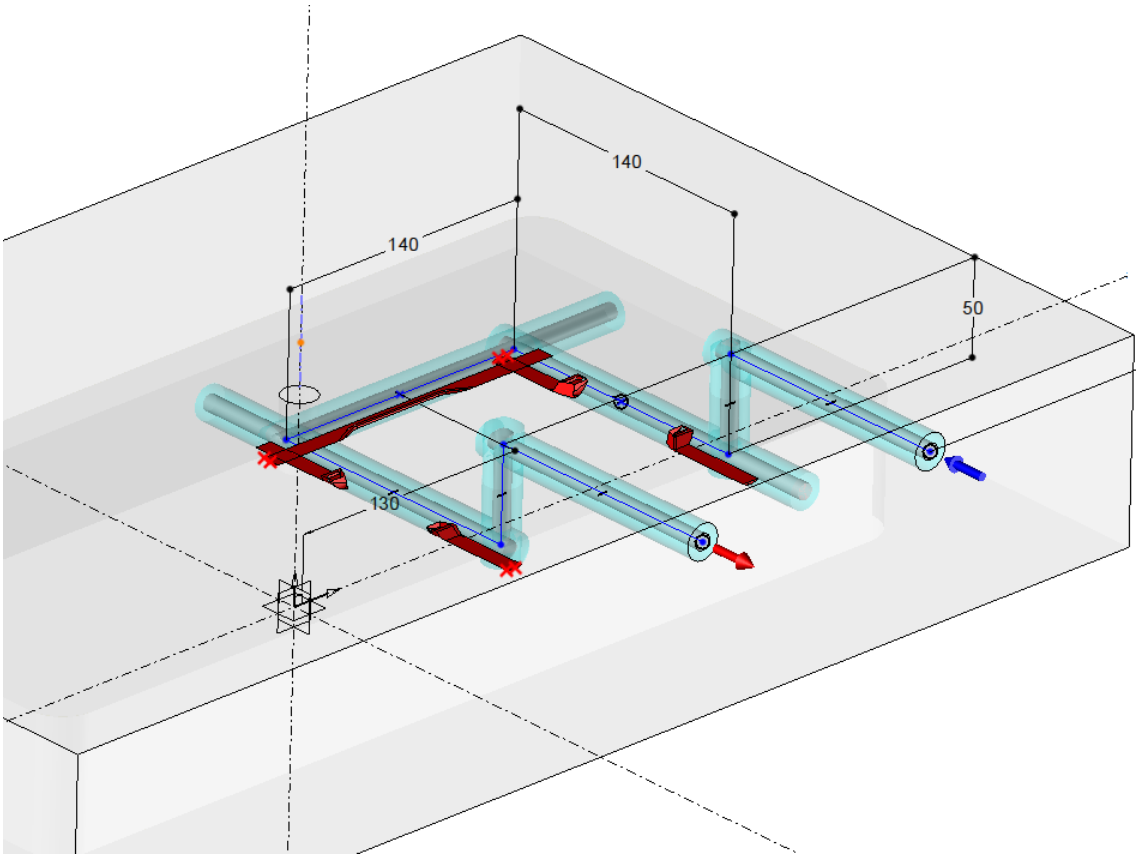
- To define the exit, right-click on the segment or cylinder of the exit drilling and select the **Set As Exit** command.



- At the top right of the graphics area, click on the **Safety Margin** icon to enable the dynamic collision detection with a margin of safety.



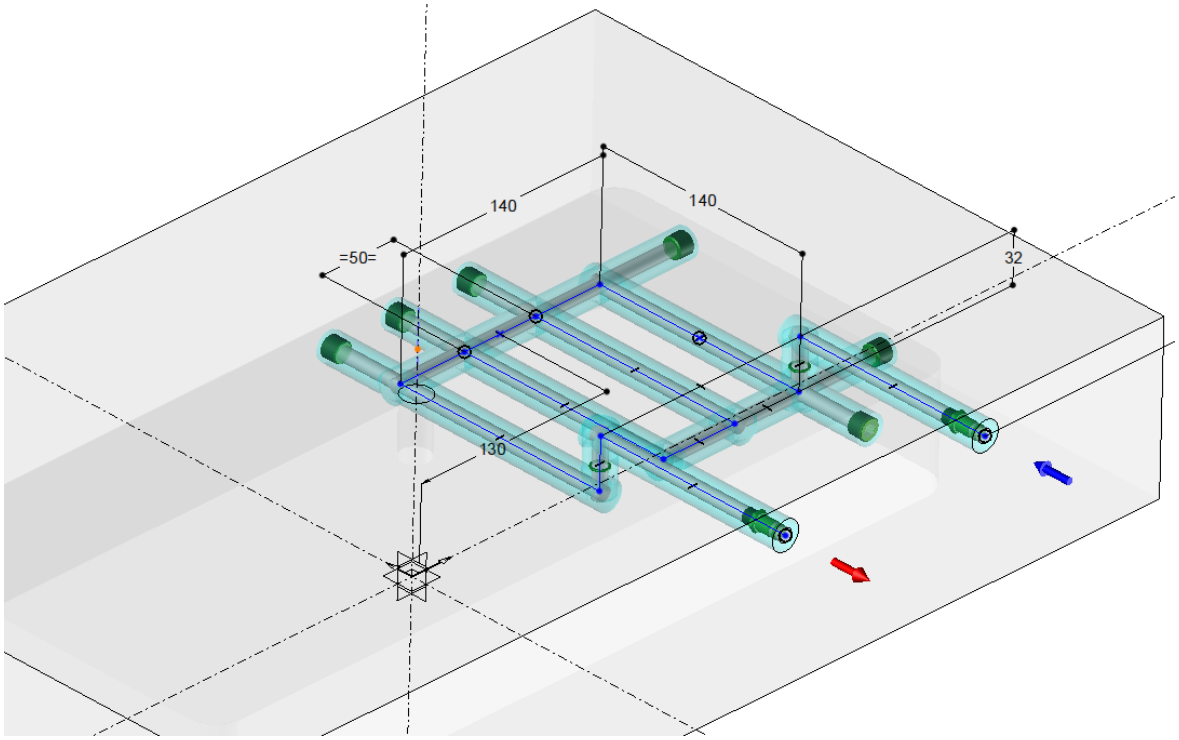
- Adjust the dimension to *50mm* to remove the invalid areas.



Modifying the cooling circuit and adding baffle line

- Right-click on the desired starting point or segment and select **Set Input Plane**.
- Draw the two lines shown below.

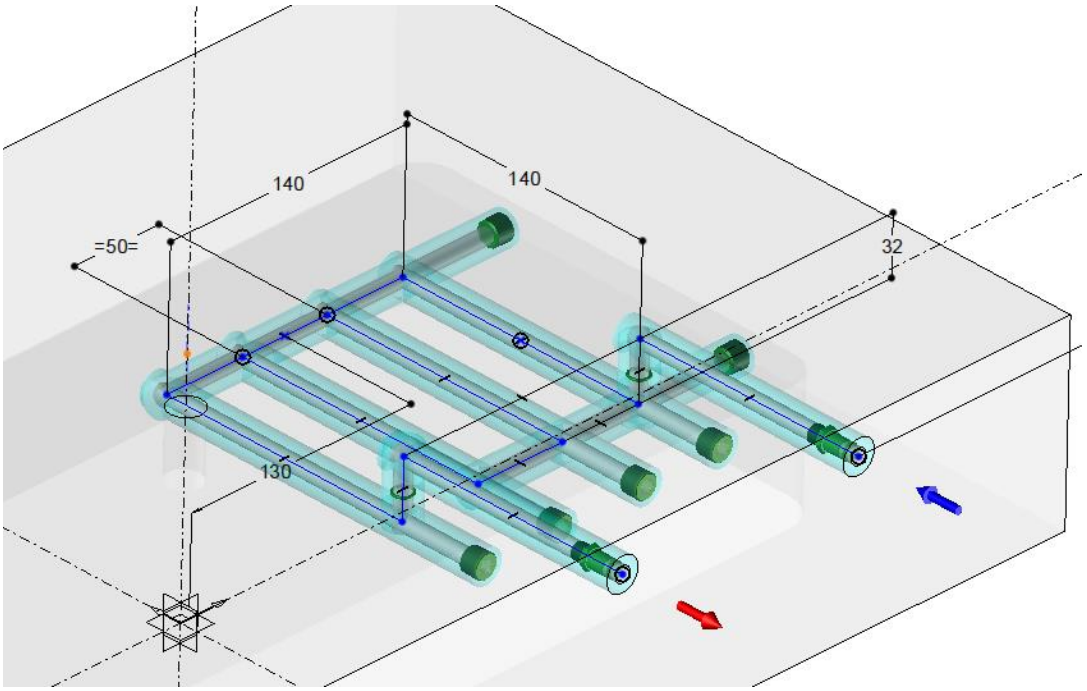
Note: The first point gives the direction of the drilling.



Modifying the drillings

If some through holes are facing the wrong direction, you need to edit the drilling by double-clicking on the cylinder of the drilling to be modified, and then double-click on the direction arrow.

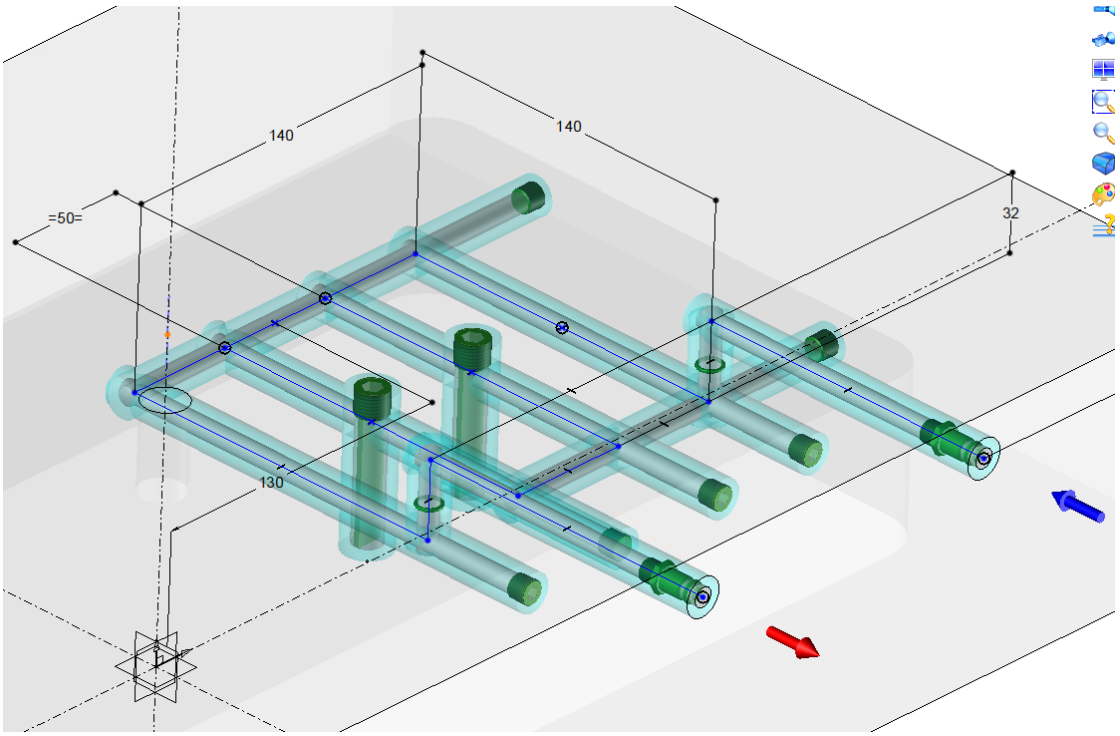
Here we want all drillings to be on the same face of the block, i.e. the face on the entry/exit side.



Adding baffle line

- Create the middle point of the last two segments.
- Right-click on the previously created point and select the **Create Baffle Line** command.

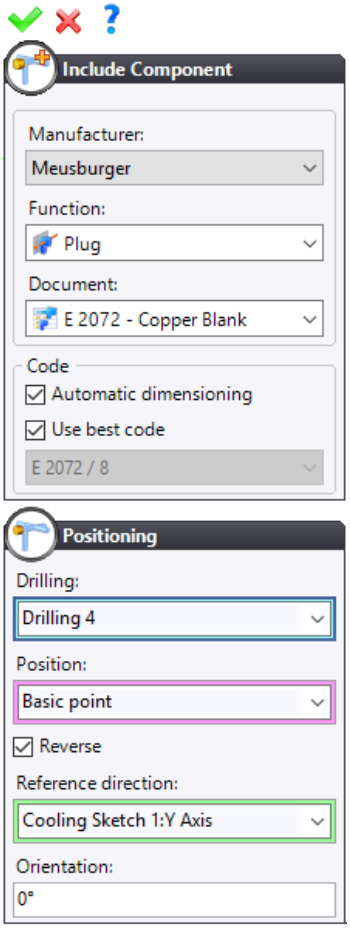
The drilling is automatically added with the values you previously entered, and the blade and plug are positioned.



Inserting components in the cooling circuit

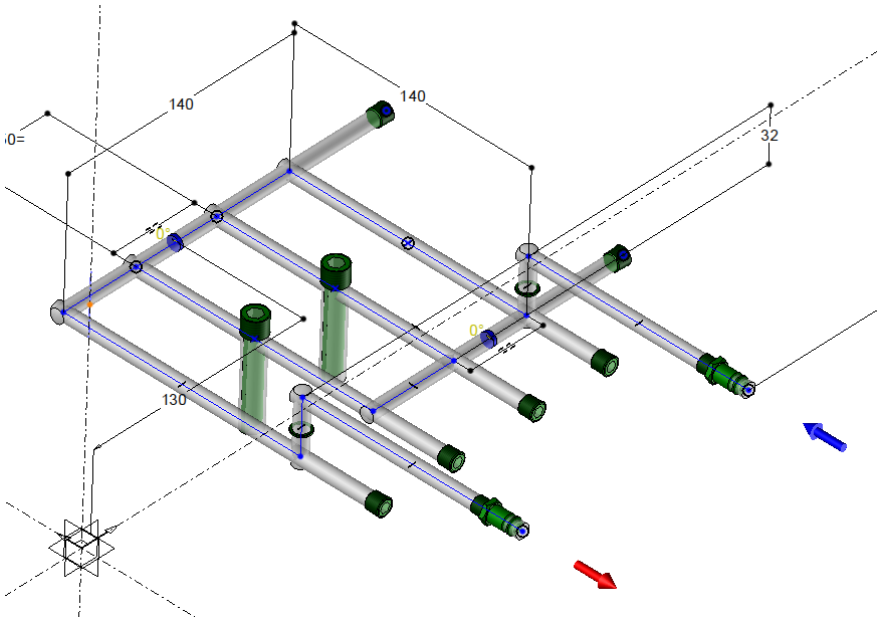
We will now insert plugs in the cooling circuit.

- Right-click on the line where you want to position the plug and select the **Include Component** command.
- Select a **Meusburger E 2072 plug**.



- Position the plug on the segment and adjust the dimension by adding a **centering** constraint for example.

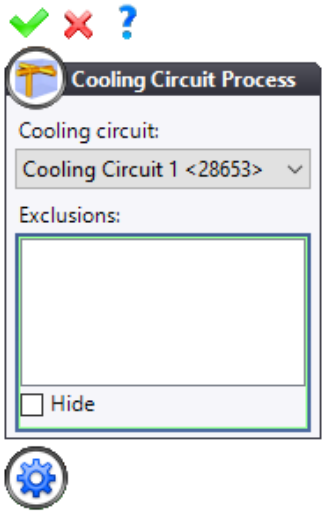
In the example below, the plugs are centered between the two segments.



Creating the cooling process

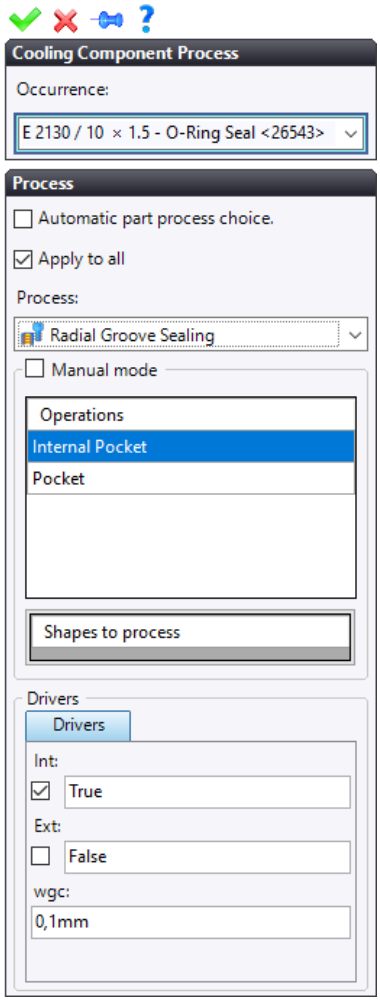
To ensure performance, the cooling circuit actually creates cylindrical "positive" shapes. The drillings are not calculated. You can then start this calculation when you want using the **Cooling Circuit Process** command.

- From the **Mold** tab, select the **Cooling Circuit Process** command.

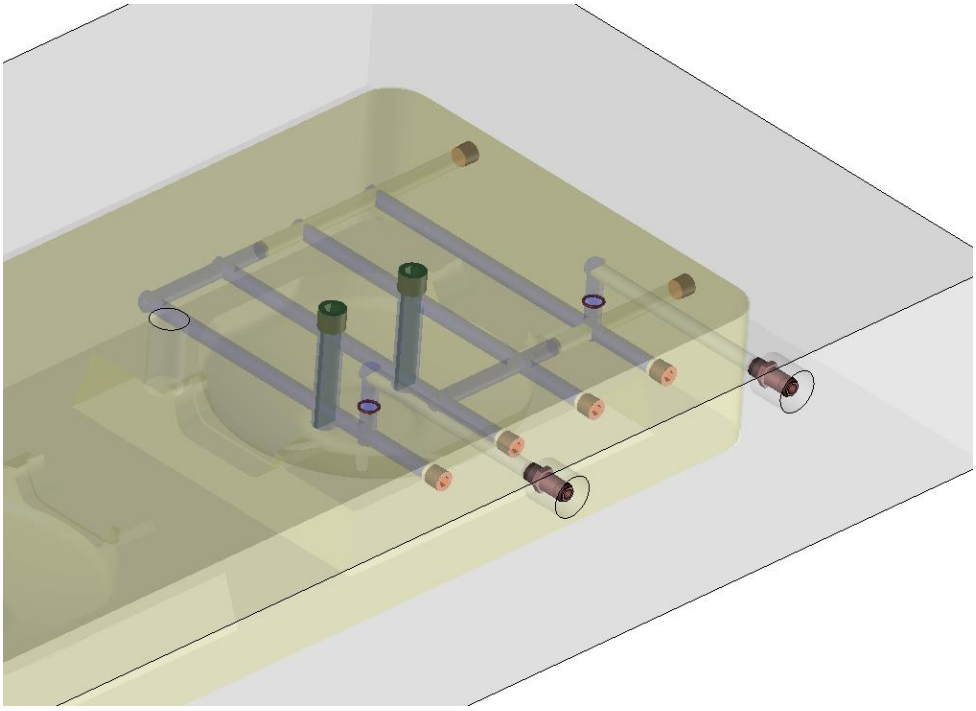


- **Confirm** the default circuit. If there are several cooling circuits, select the one you want in the drop-down list.

The process dialog box appears for each type of component.

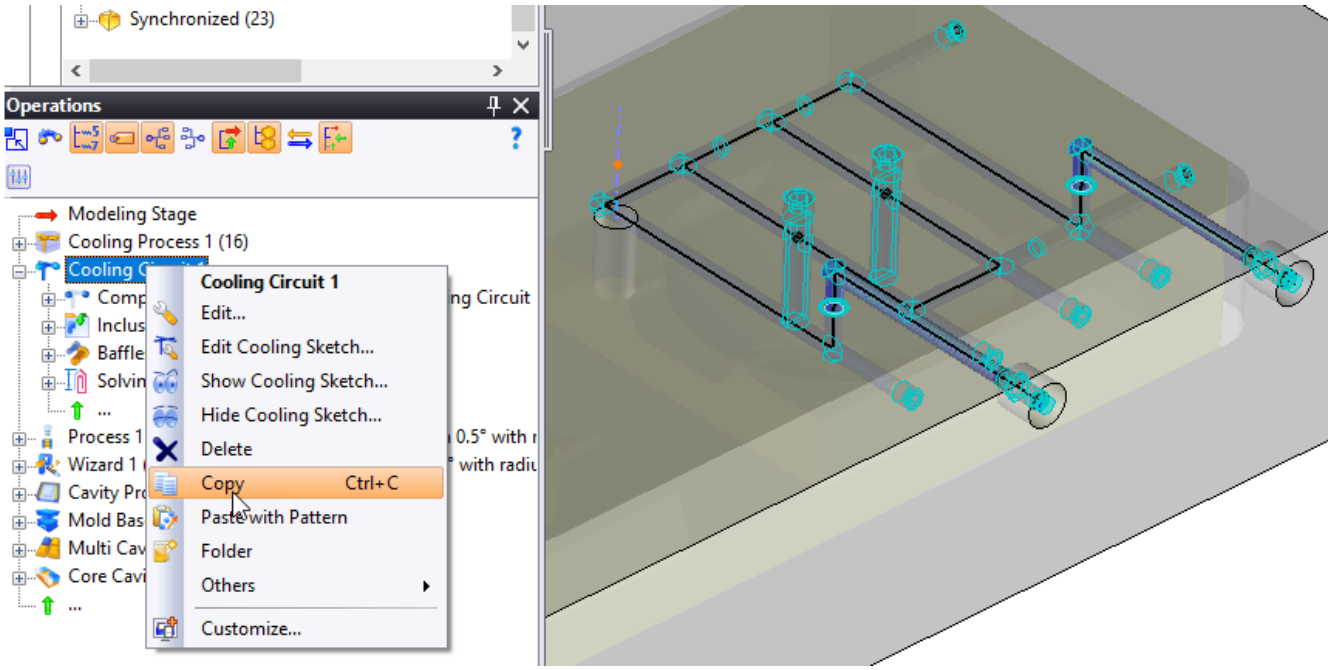


- **Confirm** the default process.

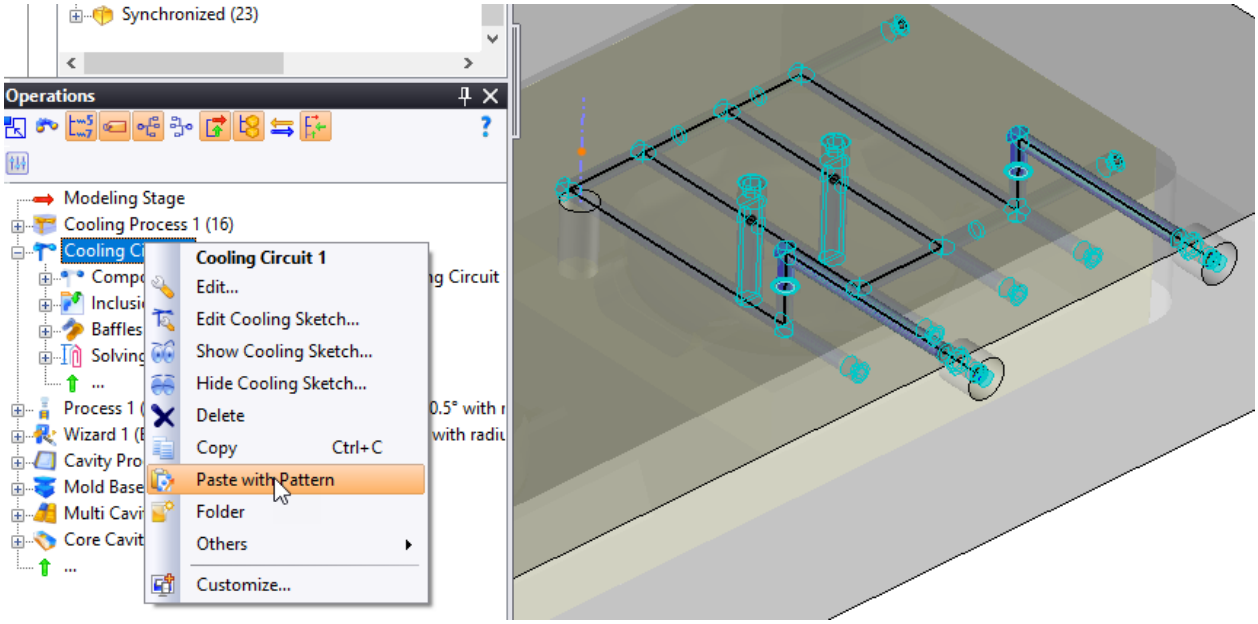


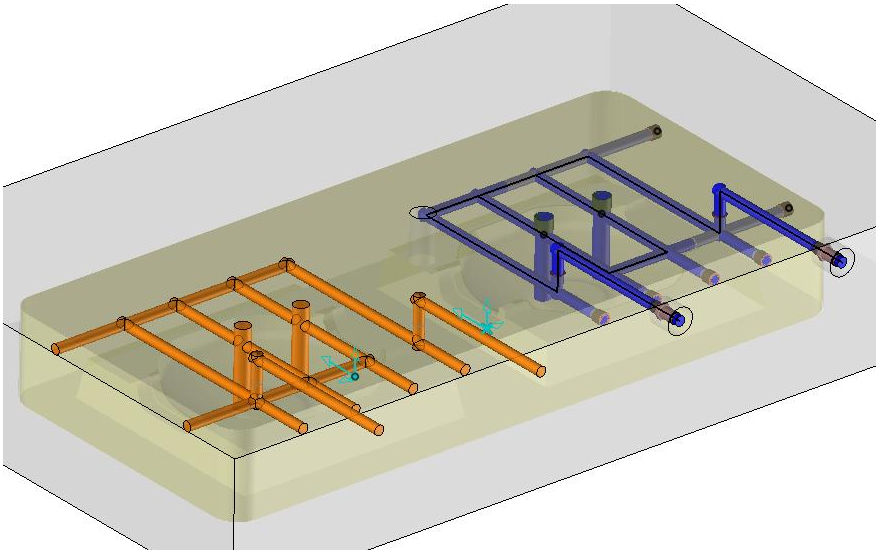
Cooling symmetry

- Right-click on the existing circuit and select the **Copy** command.

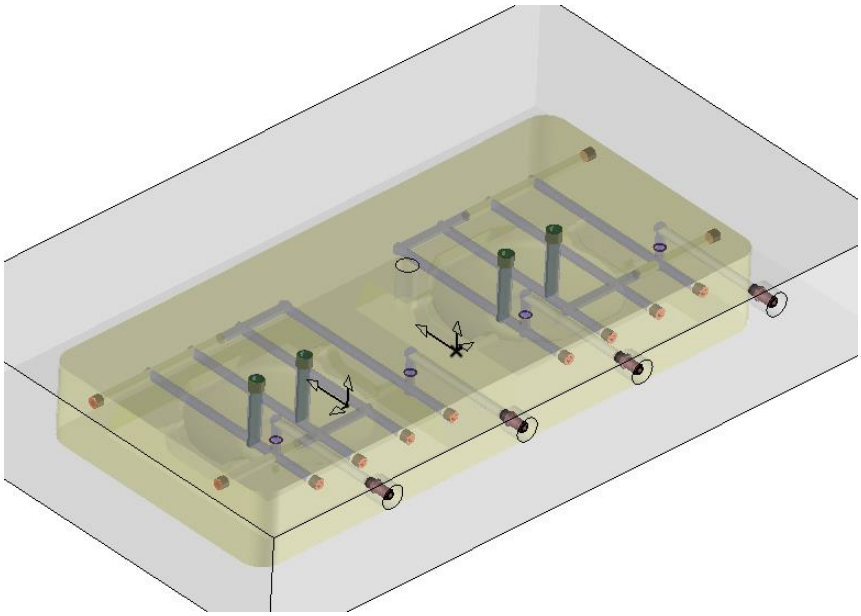


- Then select the **Paste with Pattern** command.





- Edit the cooling sketch 2 and add the appropriate constraints.



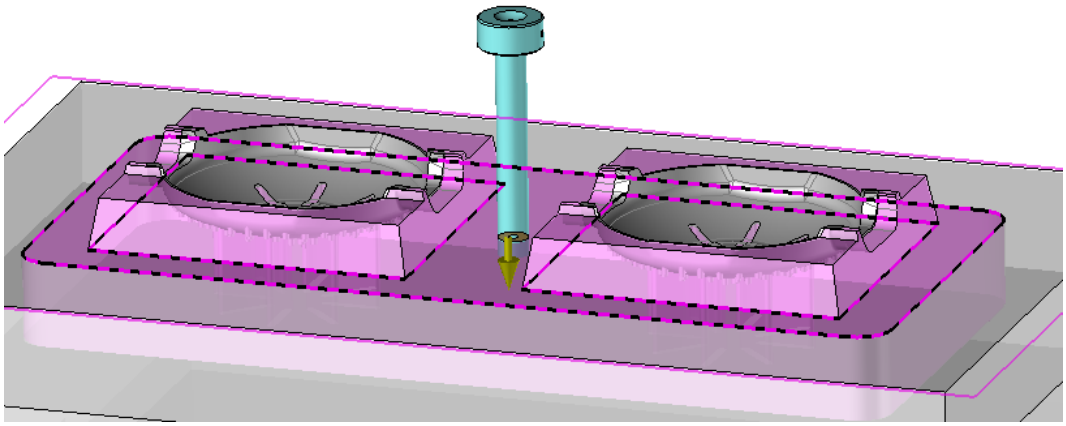
Derivation for modification

When using supplier's standard components, some of these may be modified with drilling or pocket operations, or may be shortened. Therefore, they must be declared as derived parts before any modification is made.

- Display the **E 1625 / 18 x 124 / 4 - Sprue Bush 0.5° with radius 15.5** component, then right-click on this component and select the **Others** > **Derive Part for Modification** in the component section.

Note: A new **Replacement** operation is created in the Operations tree.

- From the **Modeling** tab, select the **Trim** command. Perform a trimming operation in relation to a plane whose part is the component to be trimmed and the plane is the face shown below.



Note: If the derivation step is not performed by the user, **TopSolid** will do this automatically and will notify about it through a message.

Creating the runner circuit

The runner circuit is based on a 2D sketch. **TopSolid** automatically projects the casting shape on the part.

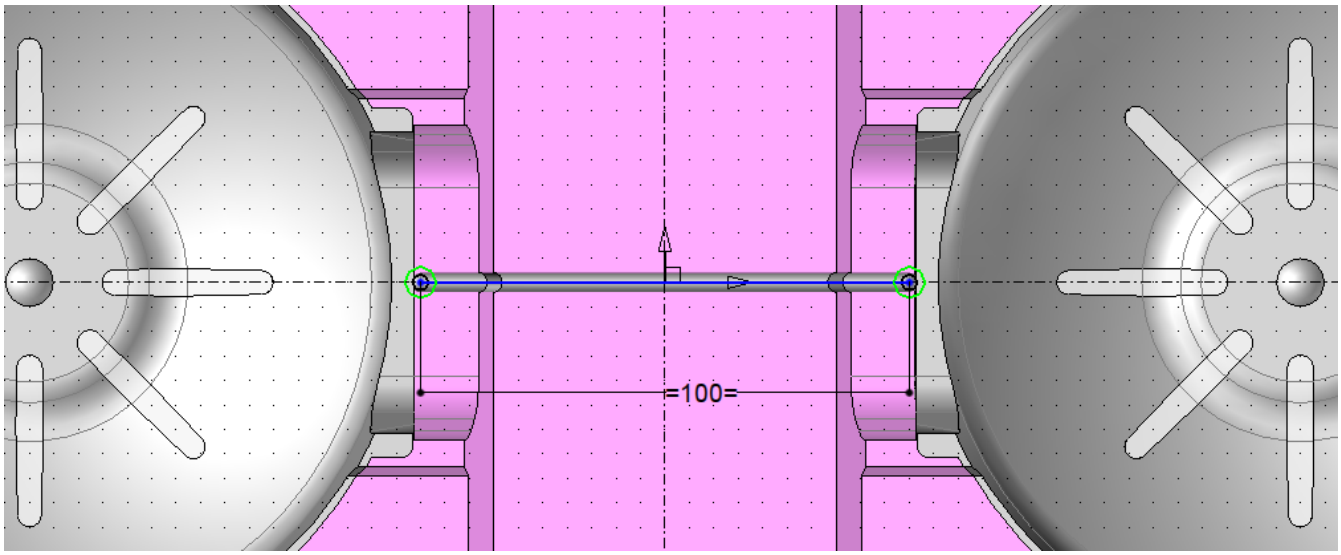
- Click on the icon in the graphics area to hide the **A side**.

Configuring the runner circuit

- From the **Mold** tab, select the **Runner Circuit** command.
- Select the part that will be used as a support to project the runner circuit.
- Click on the **Channels** option, select the **Full Round** profile in the **Section** field, then adjust the **overlap** to *0mm*.
- In the **Drivers** option, enter a **diameter** of *4mm.*
- Click on to **confirm**.

Creating the 2D runner sketch

- In the runner sketch, draw a line as shown below.

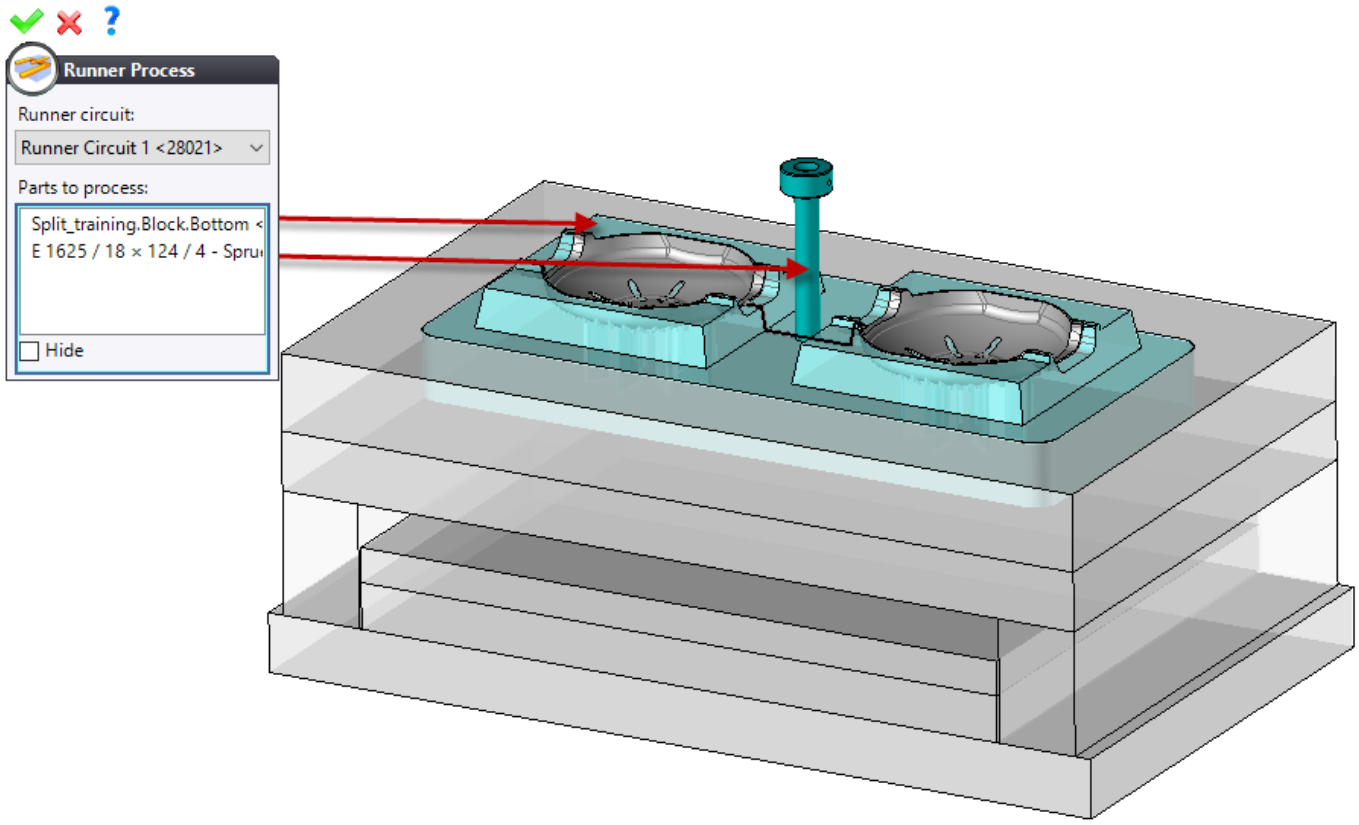


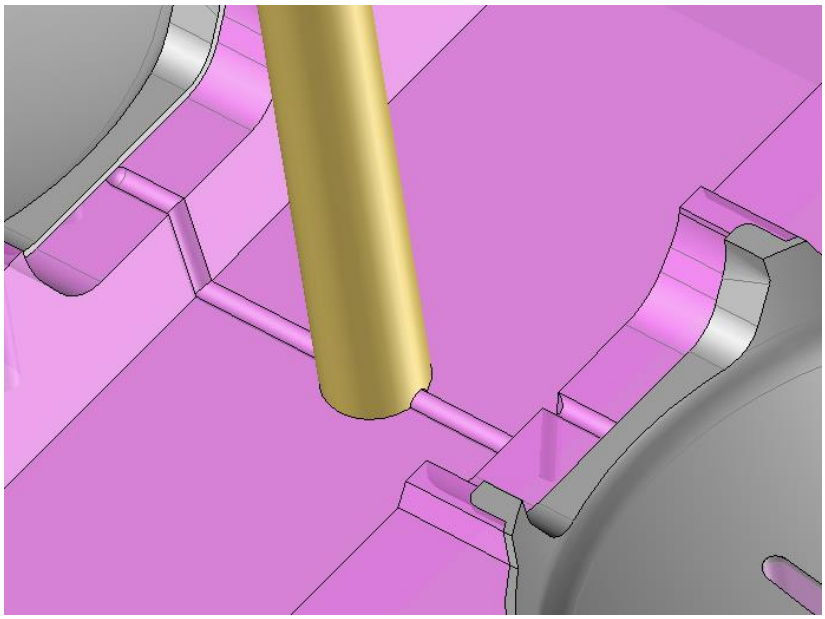
- Confirm the runner sketch.

Creating the runner process

To ensure performance, the runner circuit actually creates cylindrical "positive" shapes. The drillings are not calculated. You can then start this calculation when you want using the **Runner Process** command.

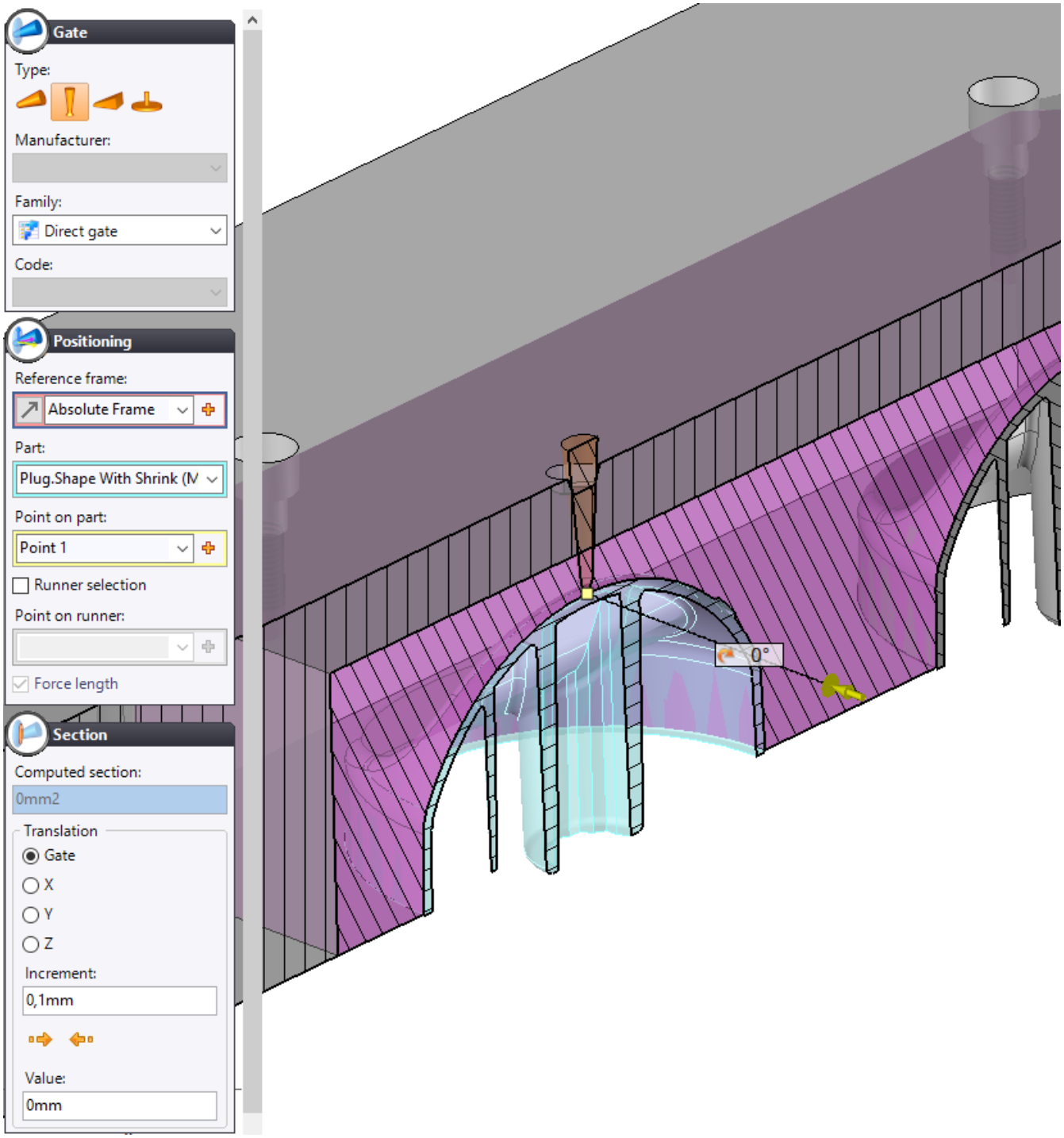
- Display the sprue bush component **E 1625 / 18 x 124 / 4 - Sprue Bush 0.5° with radius 15.5**.
- From the **Mold** tab, select the **Runner Process** command.
- Leave the default circuit. If there are several runner circuits, select the one you want in the drop-down list.
- Select the parts that will be affected by the material removal.





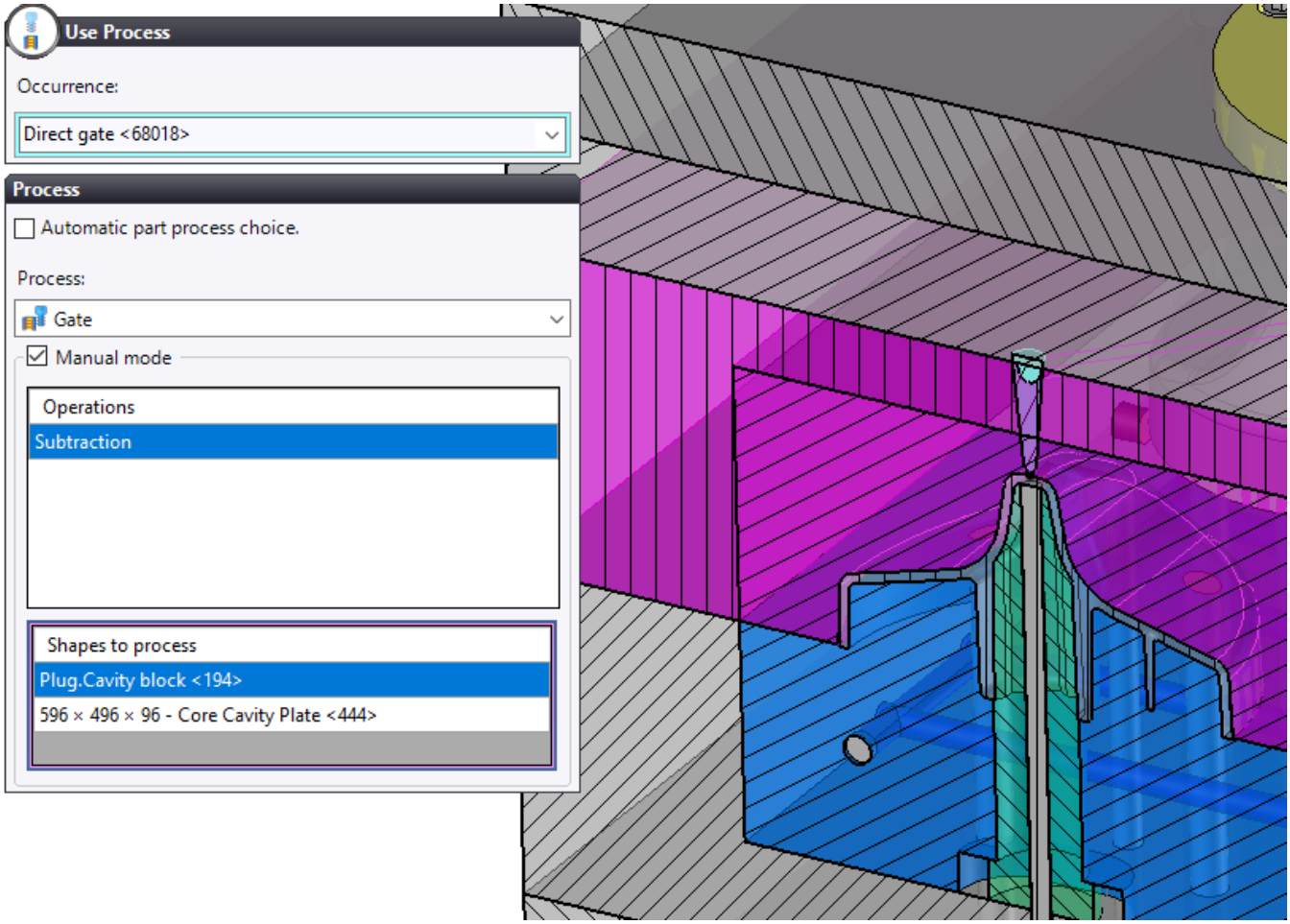
Inserting a user component: Gate

- From the Project tree, open the *Ex02 - Multi-cavity mold > C - Mold after cooling and runner circuits* folders, and then open the *My 2nd mold after cooling and runner circuits* mold document.
- Display the part to be injected, the A plate, the cavity block, as well as the points.
- From the **Visualization** tab, select the **Cut by planes** command. Select the **absolute YZ plane**, select the point on the part as the **passing point**, then click on to **confirm**.
- From the **Mold** tab, select the **Gate** command.
- Fill in the fields as shown below.



- Click to **confirm**.

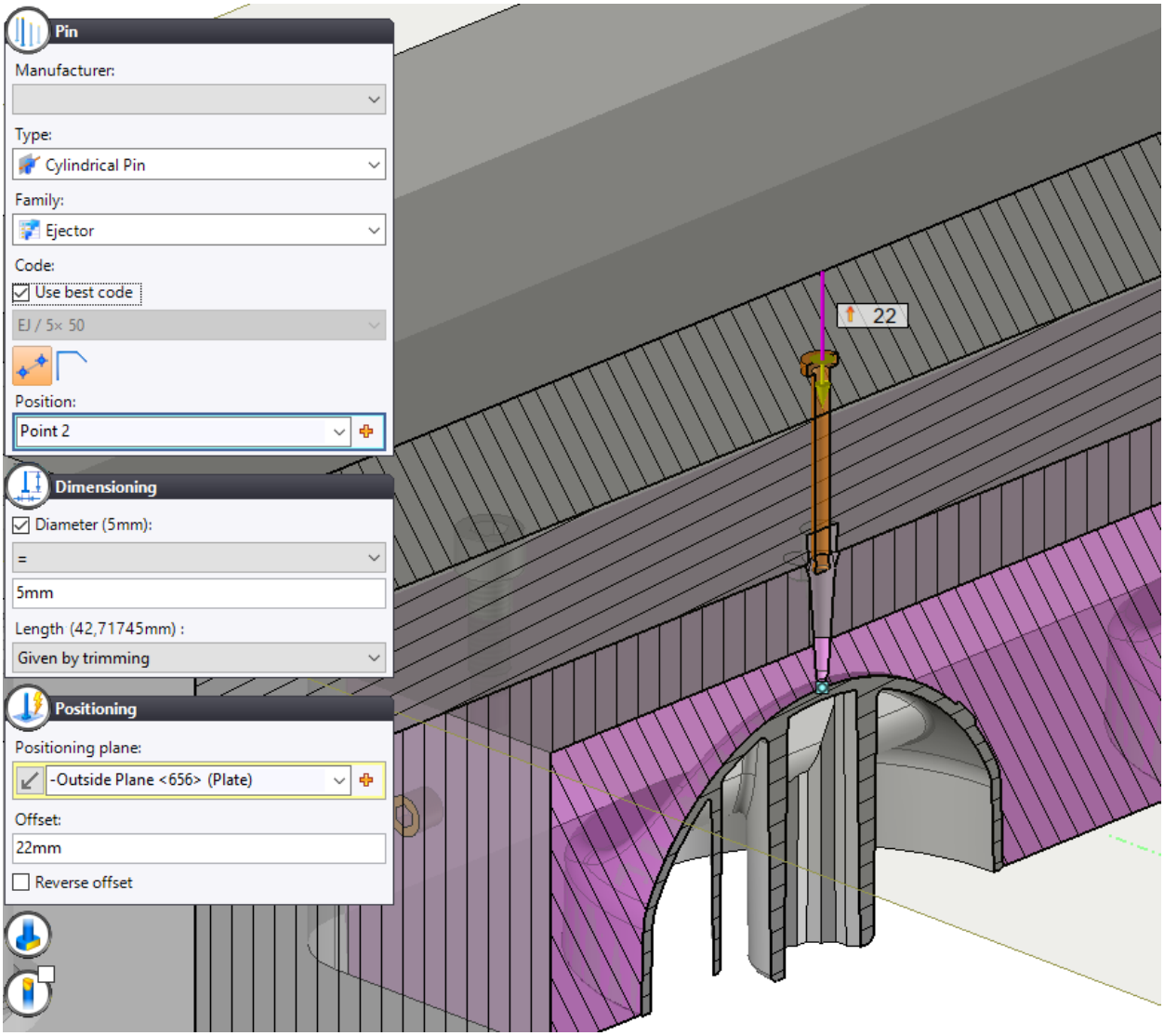
- **Confirm** the process.



- Hide the gate.

Inserting a user component: Sprue puller

- Show the plates on the **A side**.
- From the **Mold** tab, select the **Pin** command and fill in the dialog box as shown below.

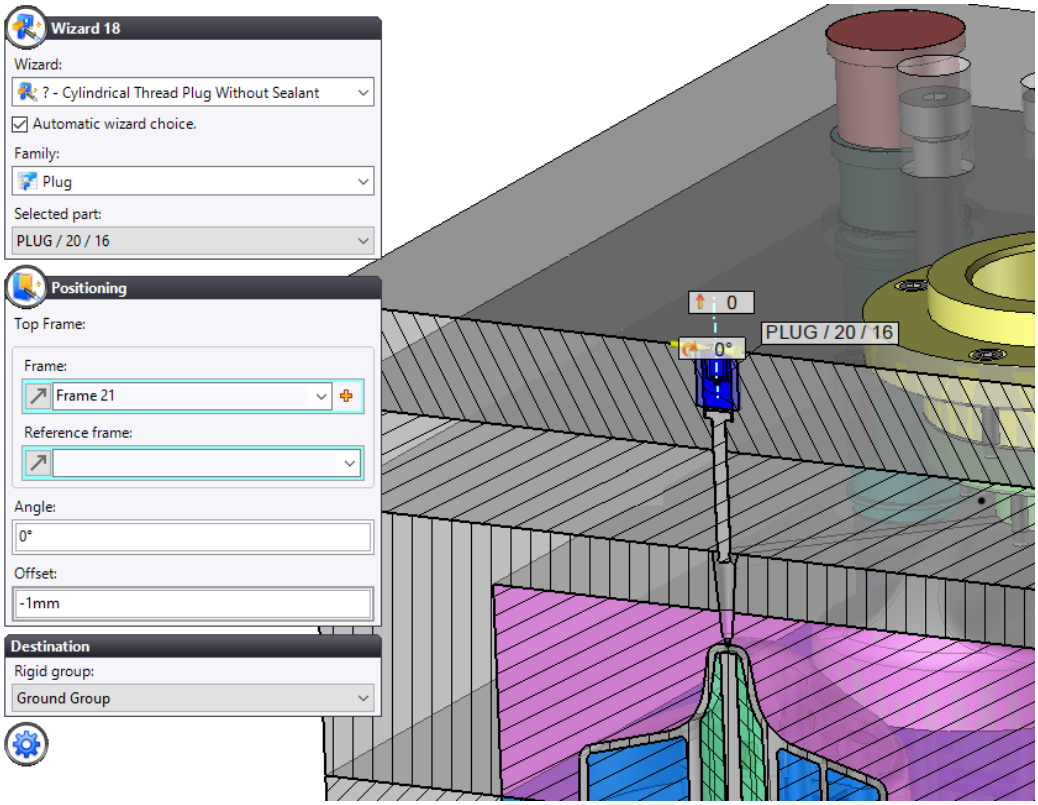


After confirming the wizard, the process dialog box appears.

- Select the **Cylindrical Pin** process and disable the **Manual** mode.

Inserting a user component: Plug

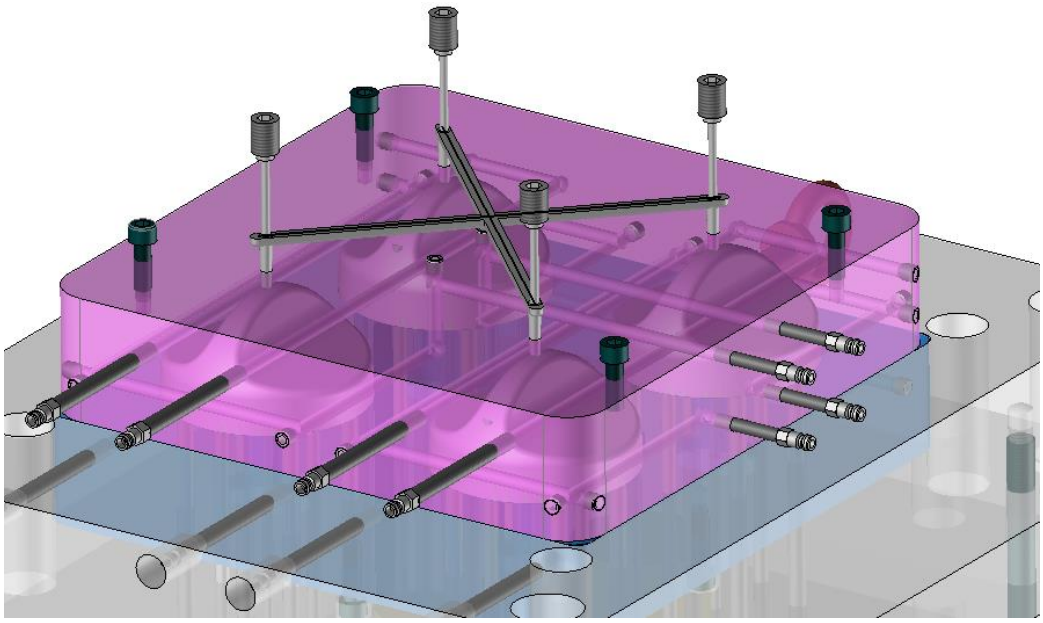
- From the Project tree, open the *Ex02 - Multi-cavity mold > A - User library > 10 - Plug* folders, select the *Plug* family document and drag and drop it into the graphics area.
- Adjust the following parameters.



- After confirming the wizard, select the **Cylindrical Thread Plug** process and click on to **confirm**.

Repeating the components

- From the **Construction** tab, select the **Repetition** command and select the components to be repeated by symmetry: the sprue puller, the plug and the gate.
- For the pattern, reuse the core cavity block pattern available in the drop-down list.

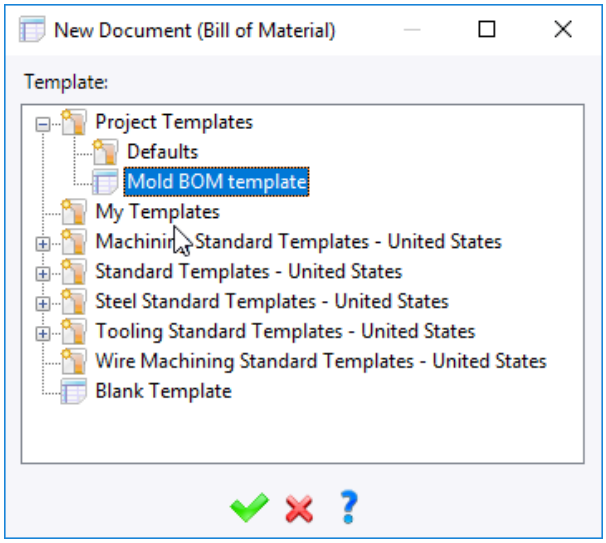


Creating the bills of materials

We will create two bills of materials: the first will list the mold set and the second will list the parts for machining.

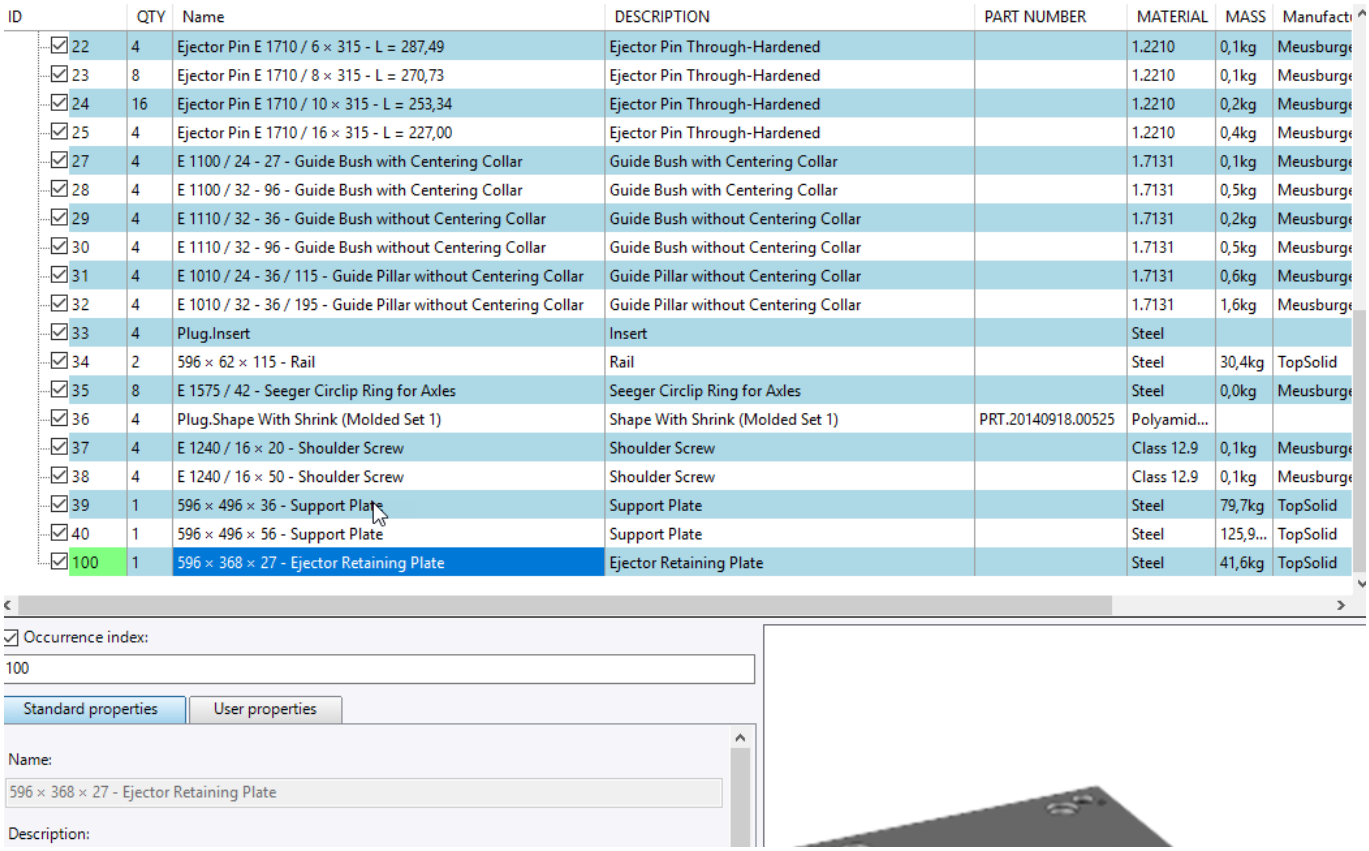
First bill of materials

- Right-click on the *My 2nd mold after cooling and runner circuits* mold document from the Project tree or on the document's upper tab and select the **Bill of Material** command.
- From **Project Templates**, select **Mold BOM template**.



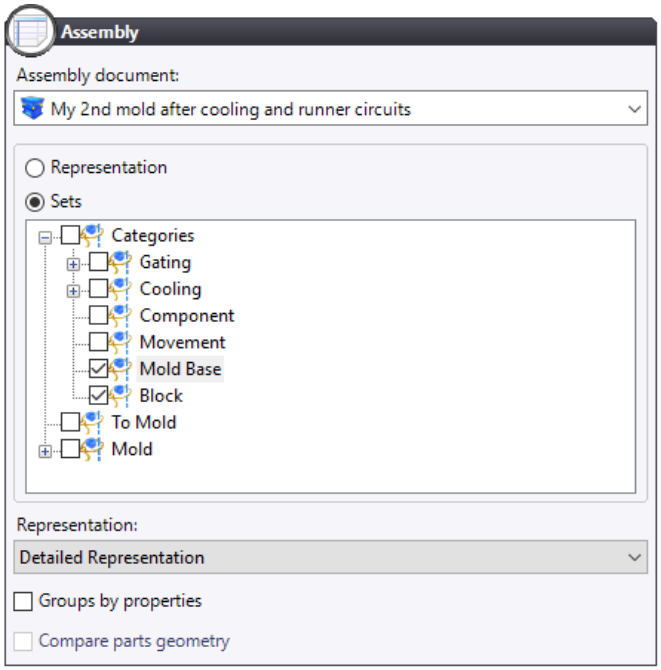
- **Confirm** the default assembly using detailed representation.
- From the Project tree, rename the document *Complete Mold.*

Note: You can manually force the indexing of the parts by checking the **Occurrence index** box and entering the index value as shown below. The forced index then appears with a green background in the bill of materials.



Second bill of materials

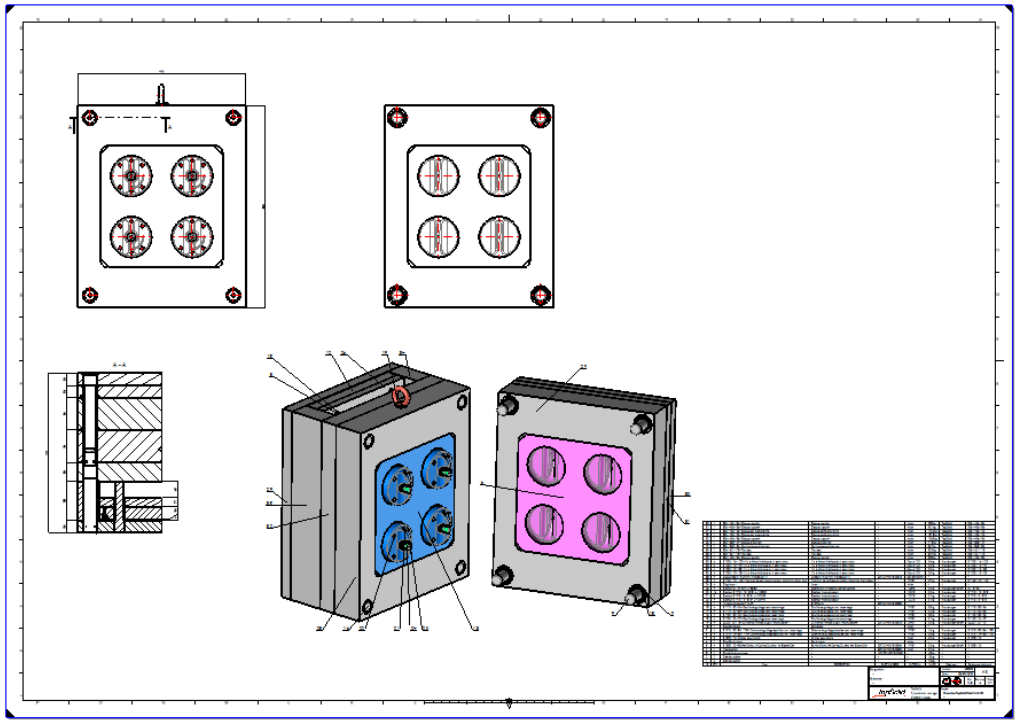
- Right-click on the *My 2nd mold after cooling and runner circuits* mold document from the Project tree or on the document's upper tab and select the **Bill of Material** command. Select **Mold BOM template**.
- For the proposed assembly, select the **Sets** option and check the **Mold Base** and **Block** categories.



- Click to **confirm**.
- From the Project tree, rename the document *Workshop Parts.*

Drafting the complete mold

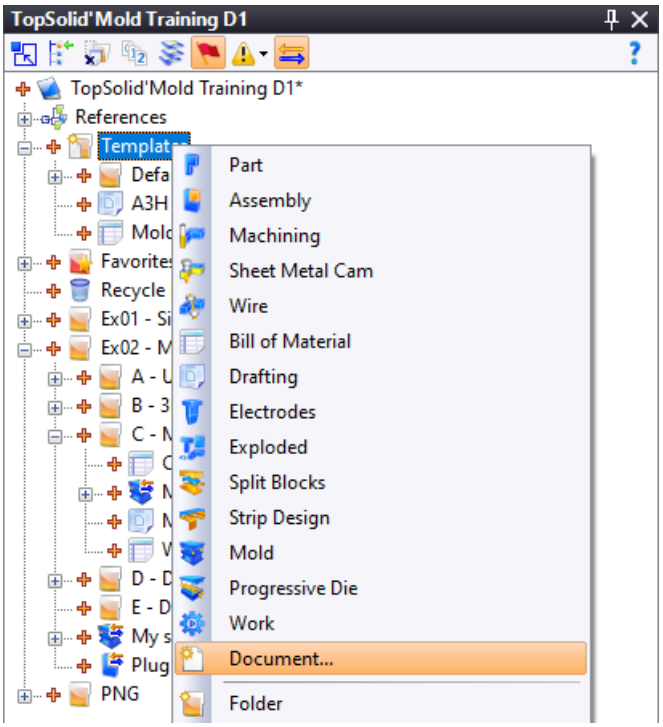
- Create a **Drafting** document for the whole mold by adding the **A Side** and **B Side** views, the bill of materials of the complete mold, a cross section view, as well as the customized views with indexes.



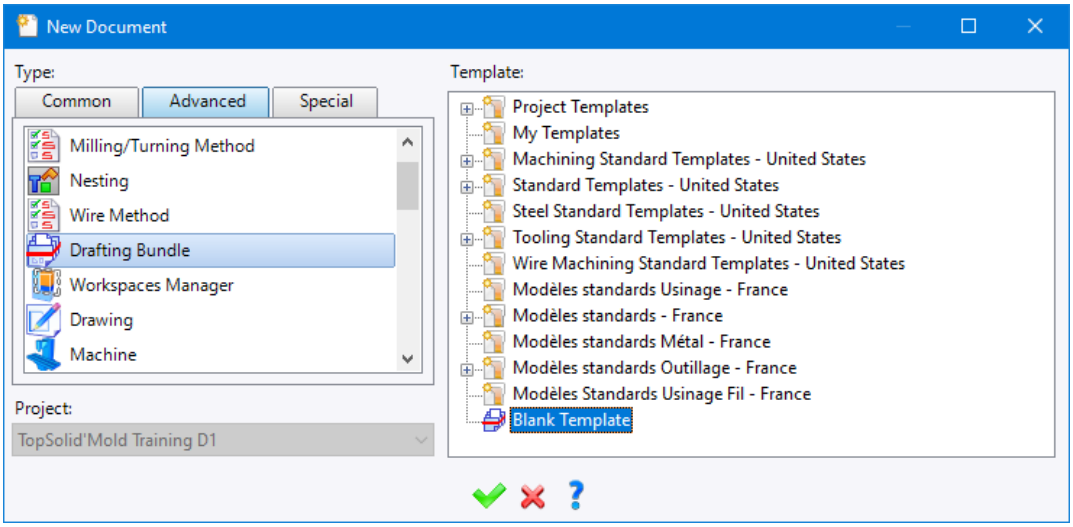
Multiple draftings of the parts and drafting bundle

The multi-drafting operation will allow you to draft a set of parts from a bill of materials, using one or more drawing templates. In our case, we will create multiple draftings of all the parts available in the *Workshop Parts* bill of materials. We will also generate a drafting bundle simultaneously that will group all the drawings into a single document.

- Create a new **document** in the **Templates** folder of the *TopSolid'Mold Training D1* Project tree.

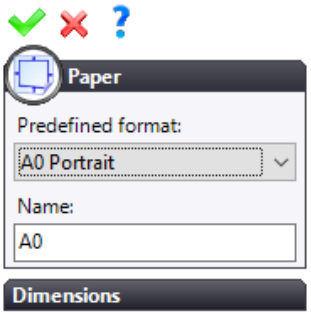


- From the **Advanced** tab, select the **Drafting Bundle** document type and select **Blank Template**.

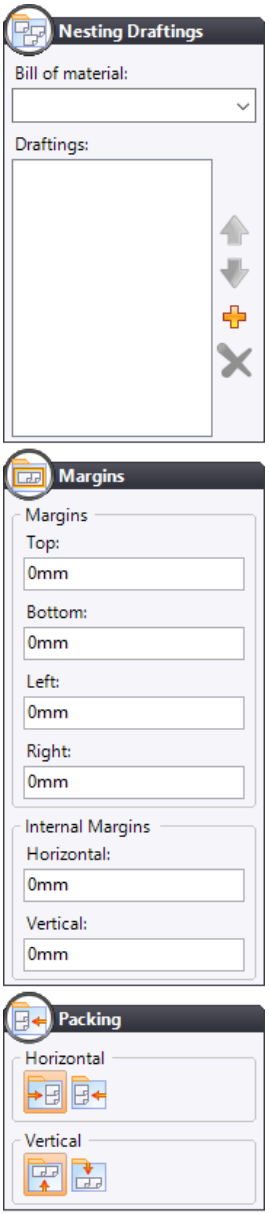


- Click on to **confirm**.
- Close the inclusion dialog box.

- In the graphics area, right-click on the frame and select the **Edit** command.
- Select the **A0 portrait** format.

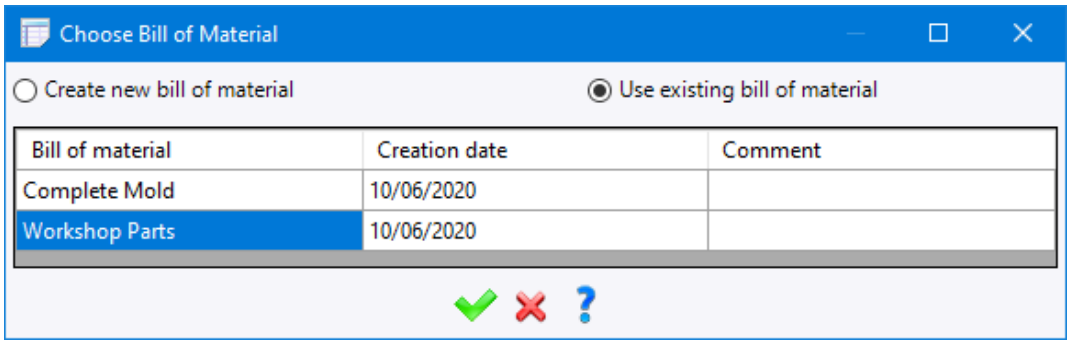


- Click on to **confirm**.
- From the **Bundle** tab, select the **Nesting Draftings** command.
- If necessary, adjust the **margin** values and the desired **packing** direction.
- Leave the **Draftings** field empty.

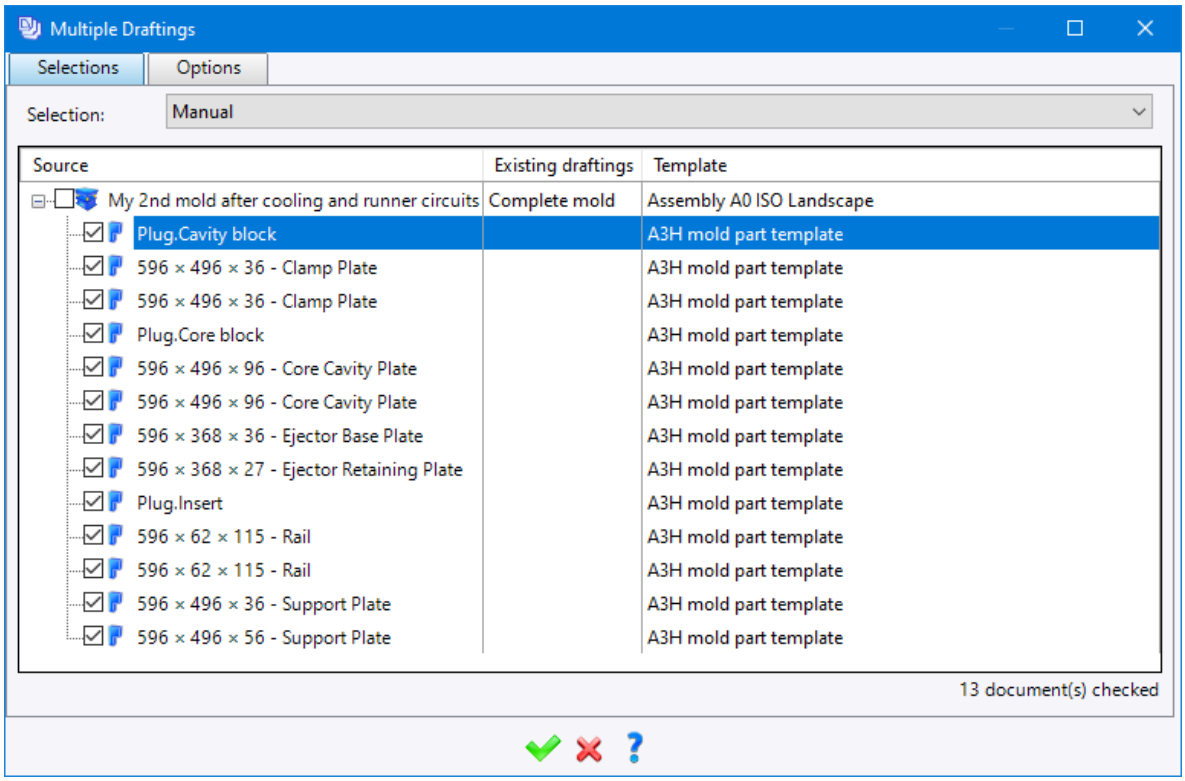


- Click on to **confirm**.
- **Save** and **close** the document.

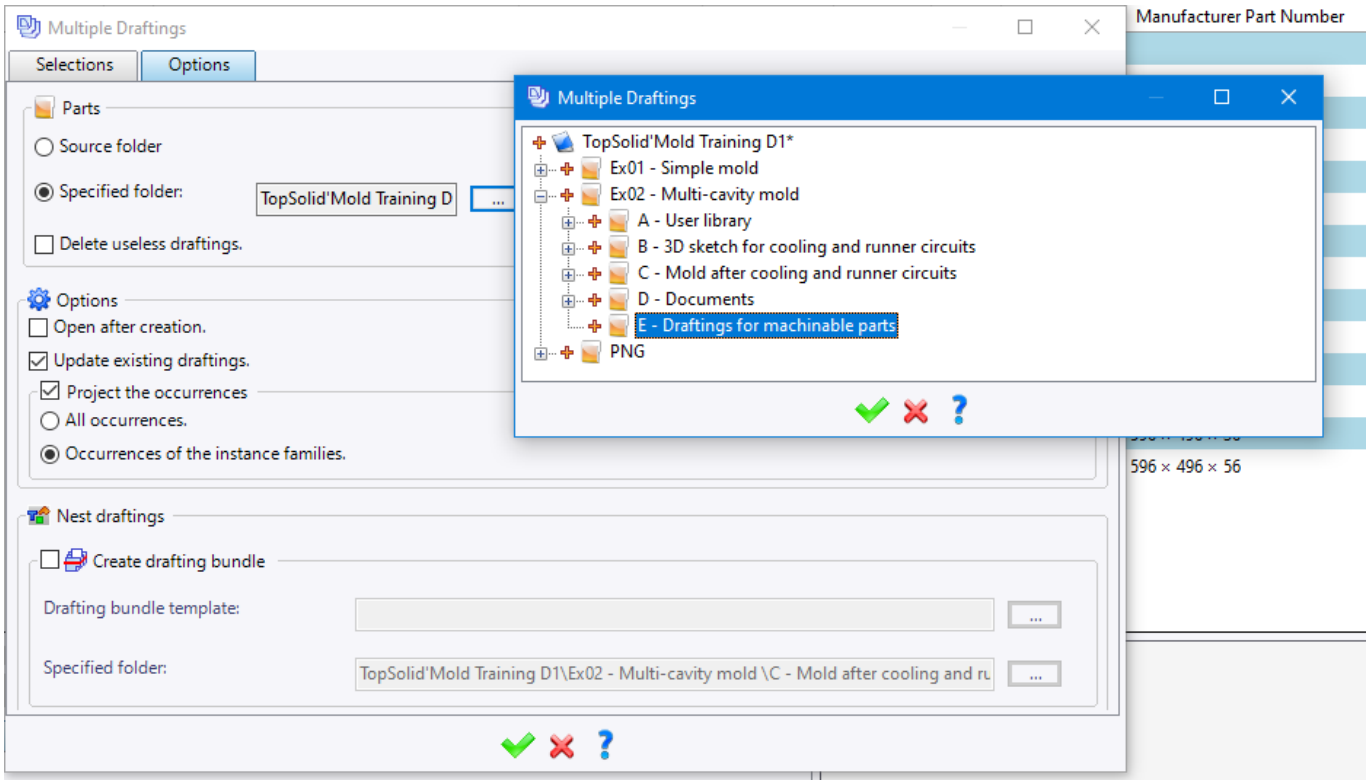
- Right-click on the *My 2nd mold after cooling and runner circuits* mold document from the Project tree or on the document's upper tab and select the **Multiple Draftings** command.
- Select the existing **Workshop Parts** bill of materials.



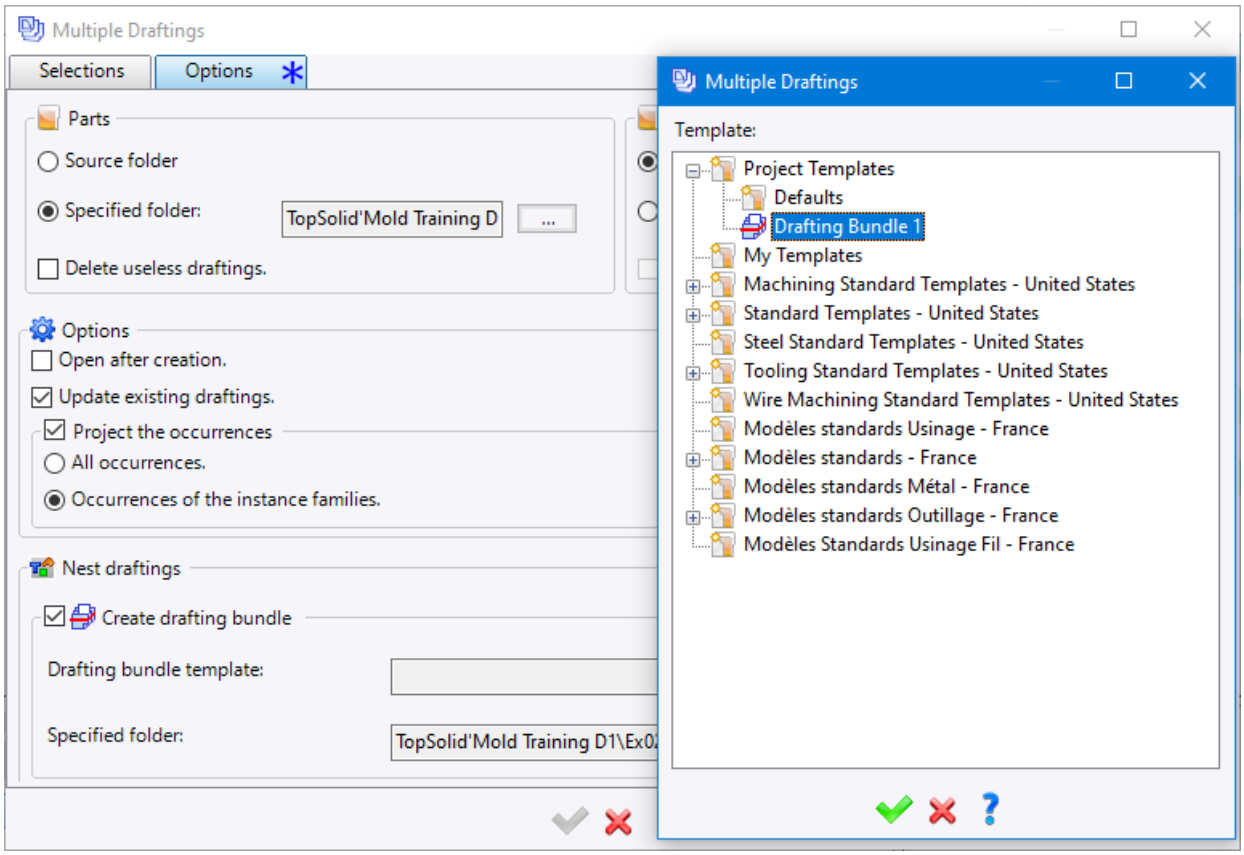
- Click on to **confirm**.
- Adjust the following parameters.
- From the **Selections** tab, for the first part, double-click in the **Template** column and select the **A3H mold part template** from the project templates.
- Right-click in the same column and select **Apply the template to this type**.



- Click on the **Options** tab. Select the **Specified folder** option for the parts, then indicate the path as shown below.



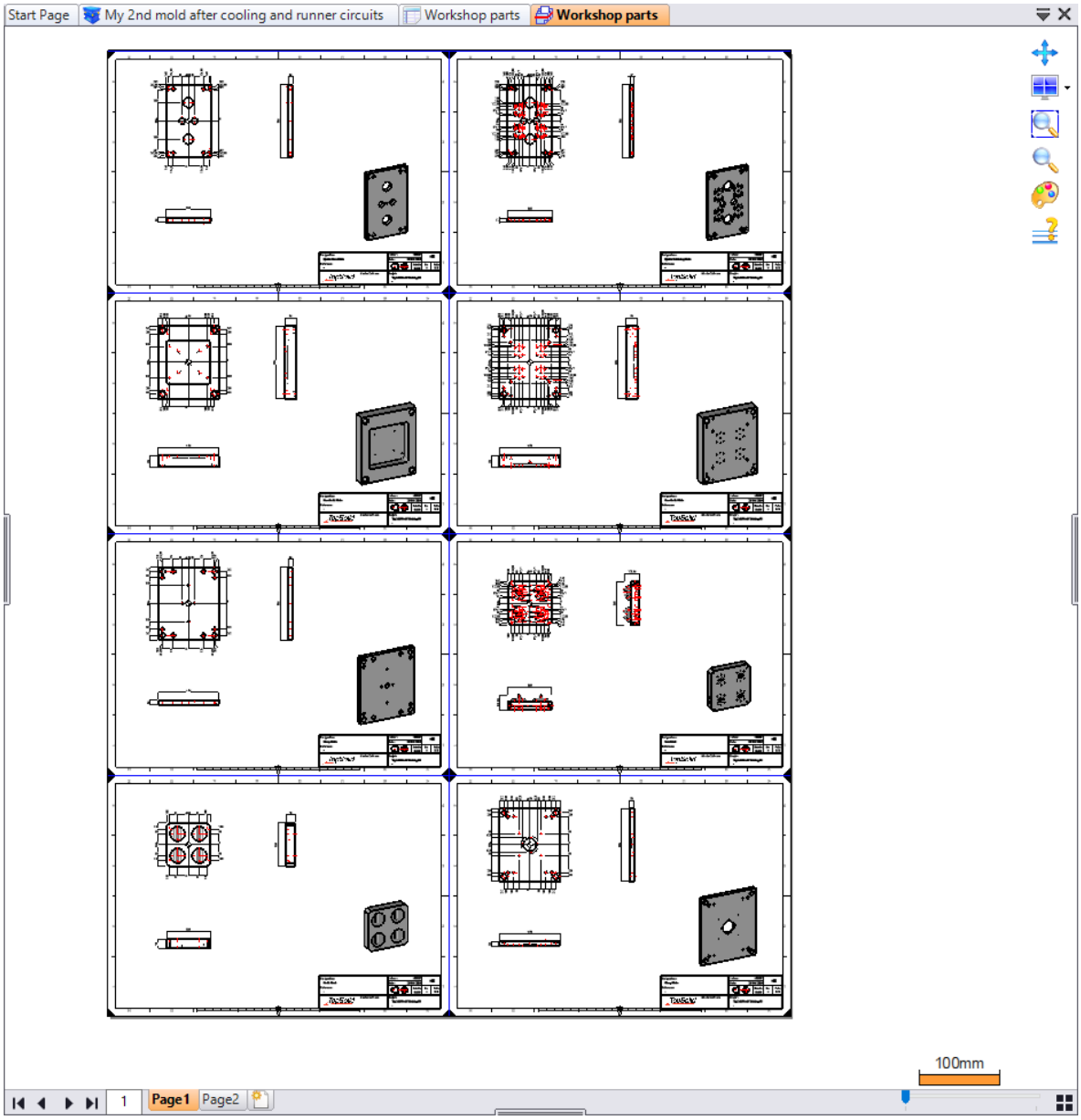
- Check the **Create drafting bundle** box. In the **Drafting bundle template** field, select the previously created bundle.



- **Confirm** the bundle selection.
- **Confirm** the creation of the multiple draftings.

The bundle document contains all the drawings on two pages. You can also create a drafting bundle manually. The procedure will be detailed in a future exercise.

- Rename the bundle document *Workshop parts*.



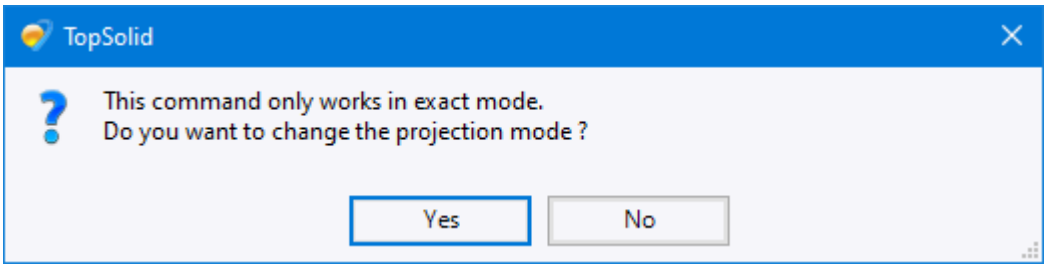
Drawing Detailing

Cooling circuit

- Open the drawing for the A plate.
- From the **Mold** tab, select the **Cooling Circuit Attributes** command.

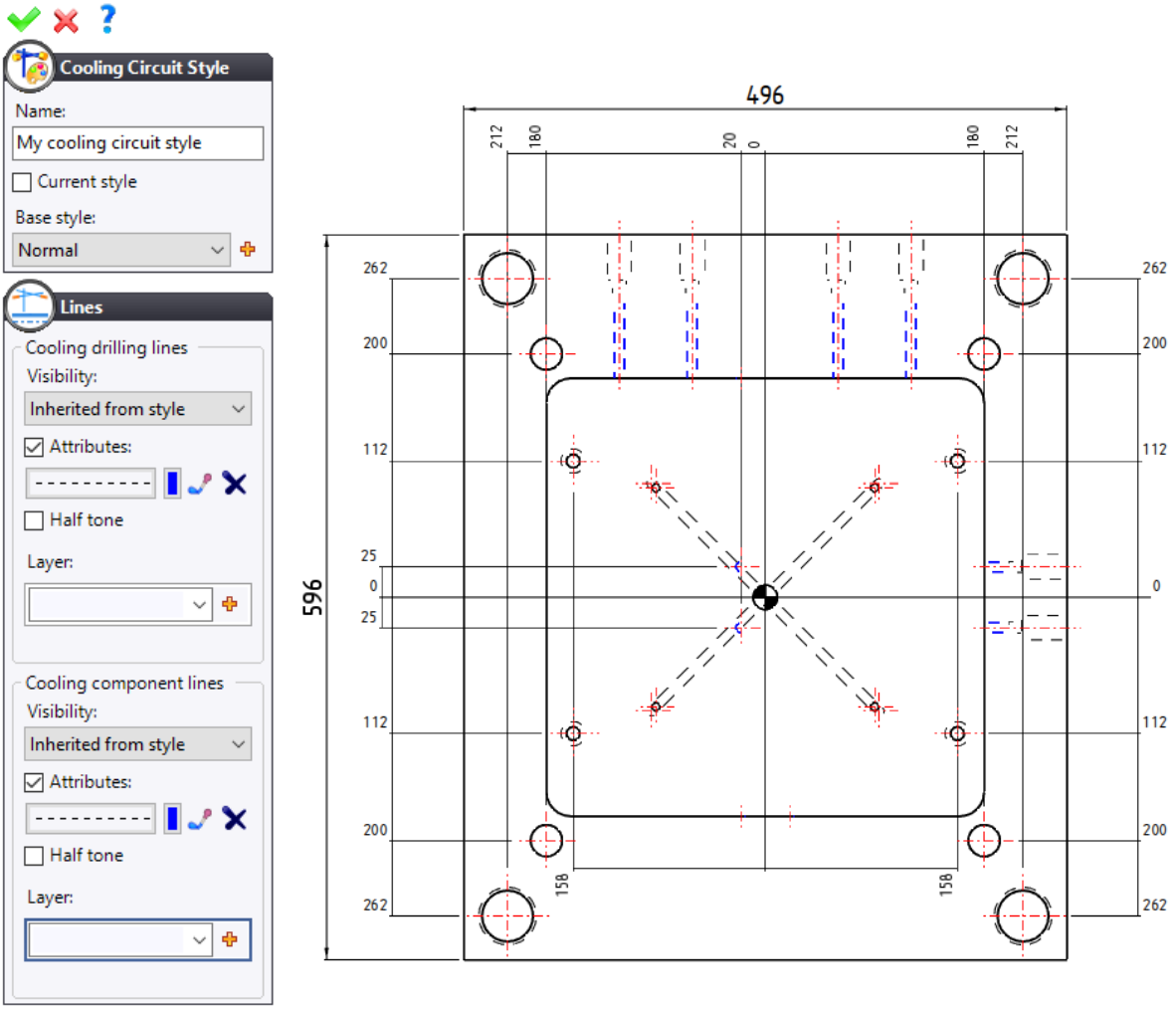
If you are not in the exact mode, the following message appears.

- Click on the **Yes** button.



- Select the view to be modified.

Note: You can create your own cooling circuit style.

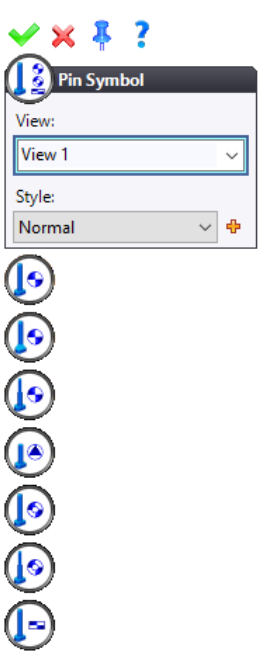


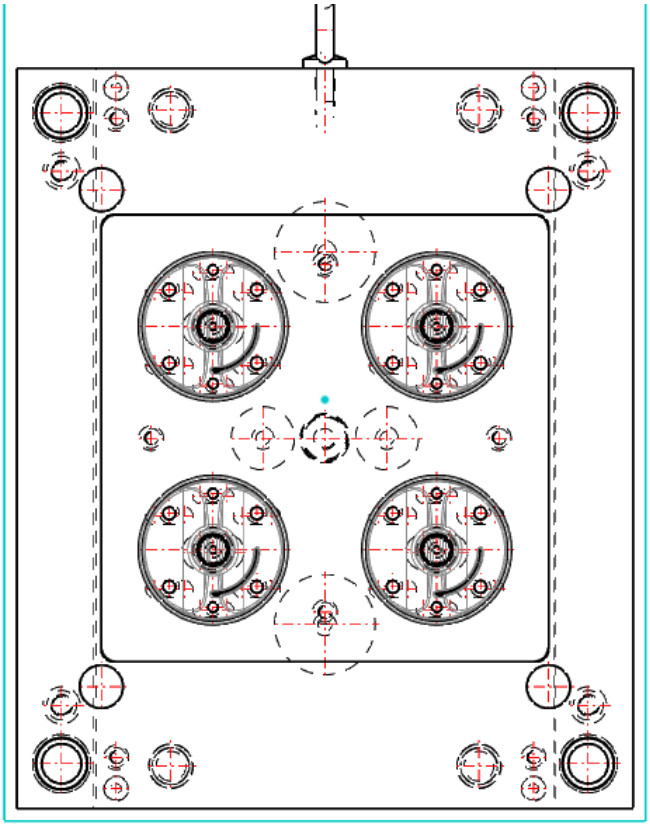
- Click on to **confirm**.

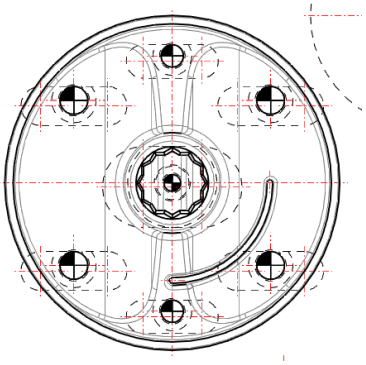
Pin symbols

- Open the mold drawing.
- From the **Mold** tab, select the **Pin Symbol** command.
- Click on the desired view.

Note: You can create your own symbol style.







- Click on to **confirm**.

Customizing the Processes

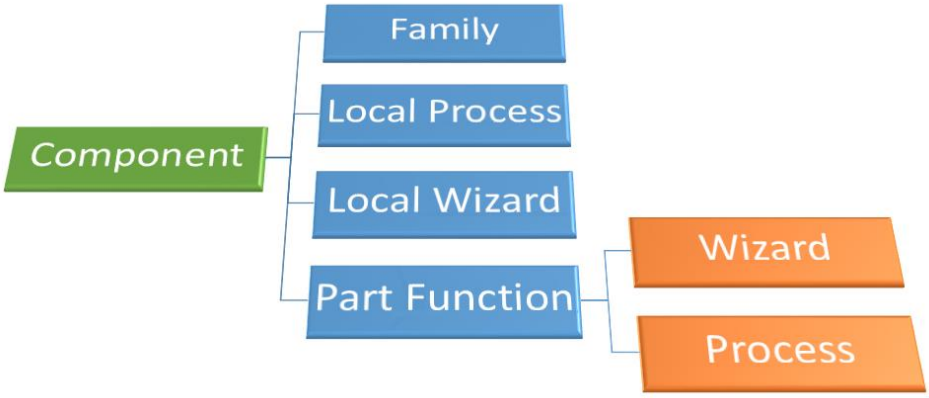
In this exercise, we will customize the standard processes of **TopSolid** to better align them with your design habits.

The purpose of this exercise is to manage the parameters of the tapped holes and the screw insertion.

Summary of component's documents

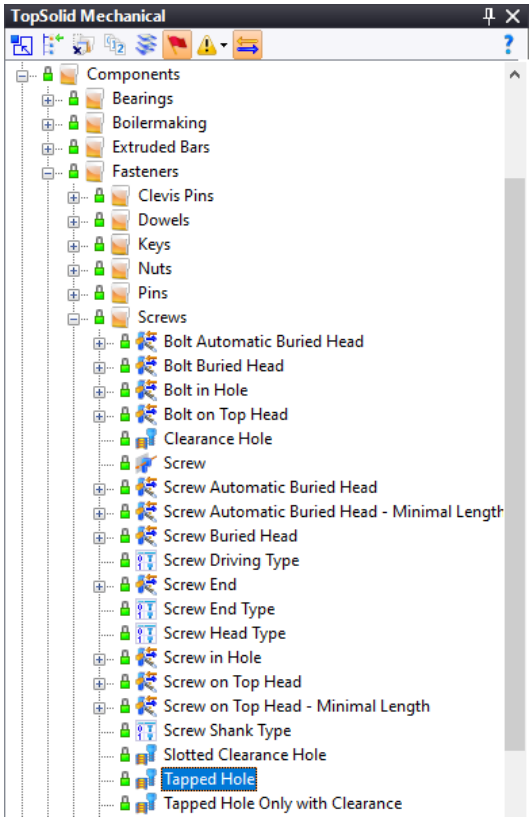
A component can have a local process and wizard for a single use.

If this component is to be replaced by another similar component, it is strongly recommended that you use a global function, wizard and process.
TopSolid provides a number of global functions, processes and wizards, the most commonly used being the **Screw** function.

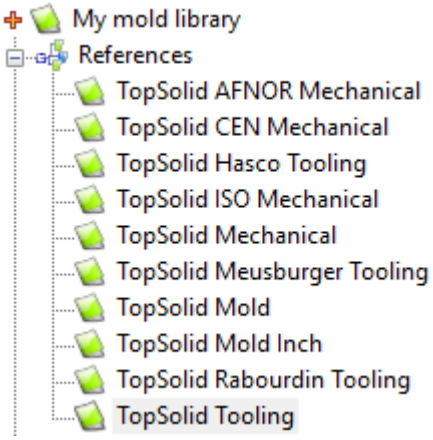


Copying a standard process

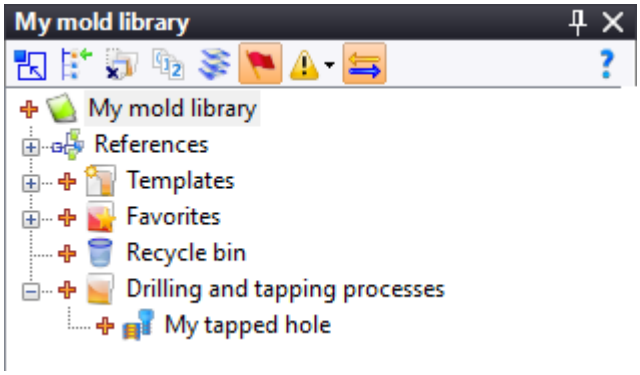
- Import the package named *TopSolid'Mold Training D2.TopPkg*.
- Open the **TopSolid Mechanical** library.
- Open the **Components** > **Fasteners** > **Screws** folders and **copy** the **Tapped Hole** part process document.



- Create a **new library** using a **blank template** and rename it *My mold library*.
- **Reference** the following libraries.



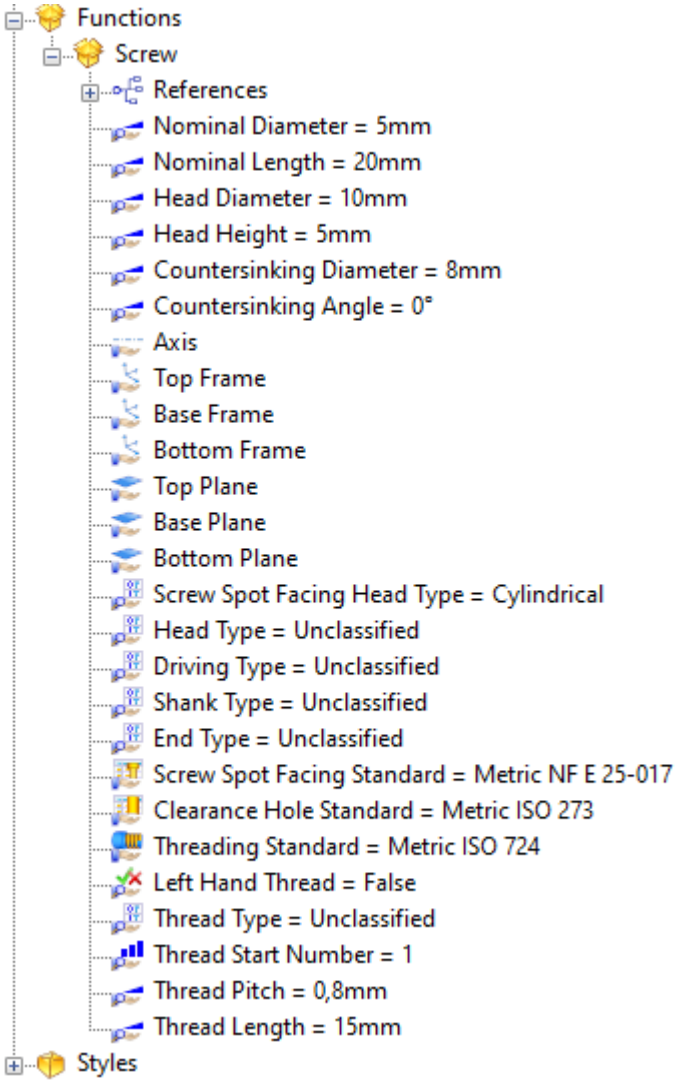
- In the new library, create a folder named *Drilling and tapping processes*.
- **Paste** the part process document into the new folder.
- Rename the document as you wish, *My tapped hole* for example.



Customizing the processes

The goal is to manage the clearances on the spot facing diameter and the clearance hole of screws.

- Open the *My tapped hole* process document.
- From the Entities tree, open the **Functions** > **Screw** folders to display the list of the parameters required for a screw.

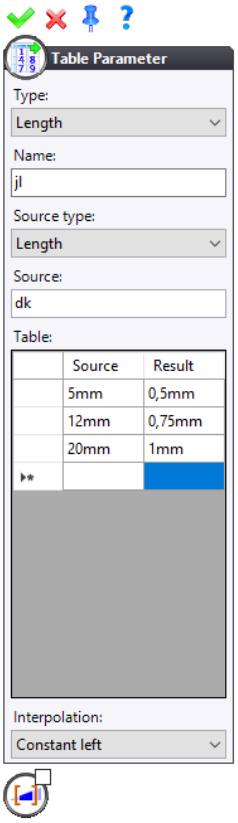


Note: If you hover the mouse cursor over a description, the name of the parameter is displayed.
In this exercise, **dk** is the head diameter parameter, **k** is the head height parameter and **d** is the nominal screw parameter.

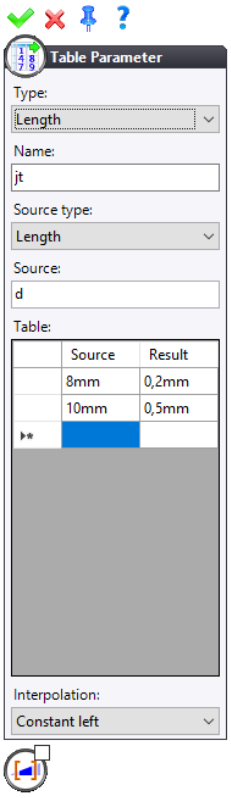
- From the **Construction** tab, select the **Parameters** > **Table Parameter** command and create the following two table parameters:

  - The **jl** parameter manages the clearance on the spot-facing diameter:
    - *0.50mm* for the *5mm* diameter;
    - *0.75mm* for the *12mm* diameter;
    - *1mm* for the *20mm* diameter.

  The **interpolation** type is used to handle the cases where the source value is different from the indicated values. With a left interpolation, the **jl** value will be:
    - *0.5mm* if **dk** < *12mm*;
    - *0.75mm* if **dk** >= *12mm* and < *20mm*;
    - *1mm* if **dk** >= *20mm.*

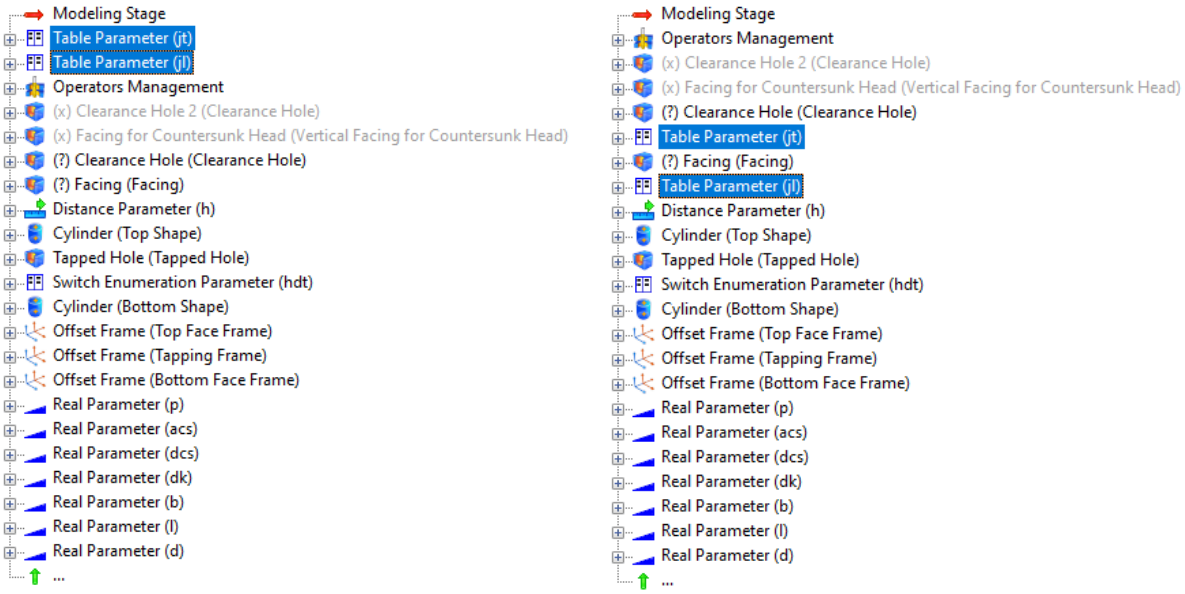


  - The **jt** parameter manages the clearance on the clearance hole diameter:
    - *0.2mm* for the *8mm* diameter;
    - *0.5mm* for the *10mm* diameter.

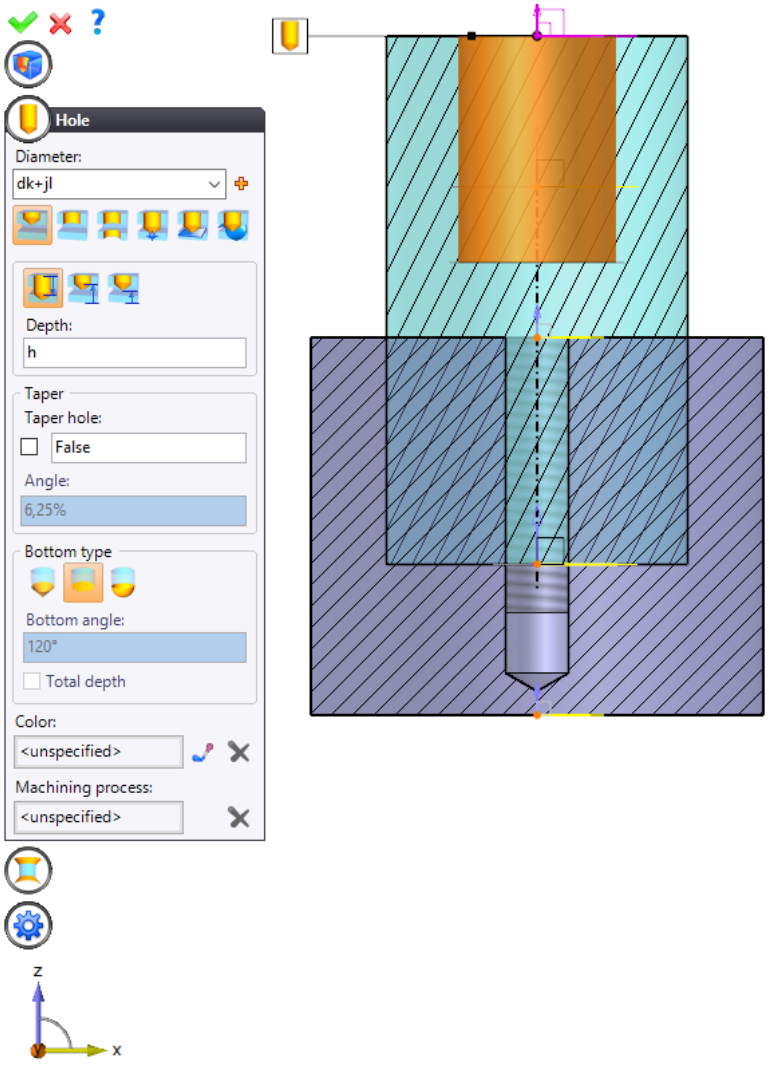


These two new parameters are available at the top of the Operations tree.

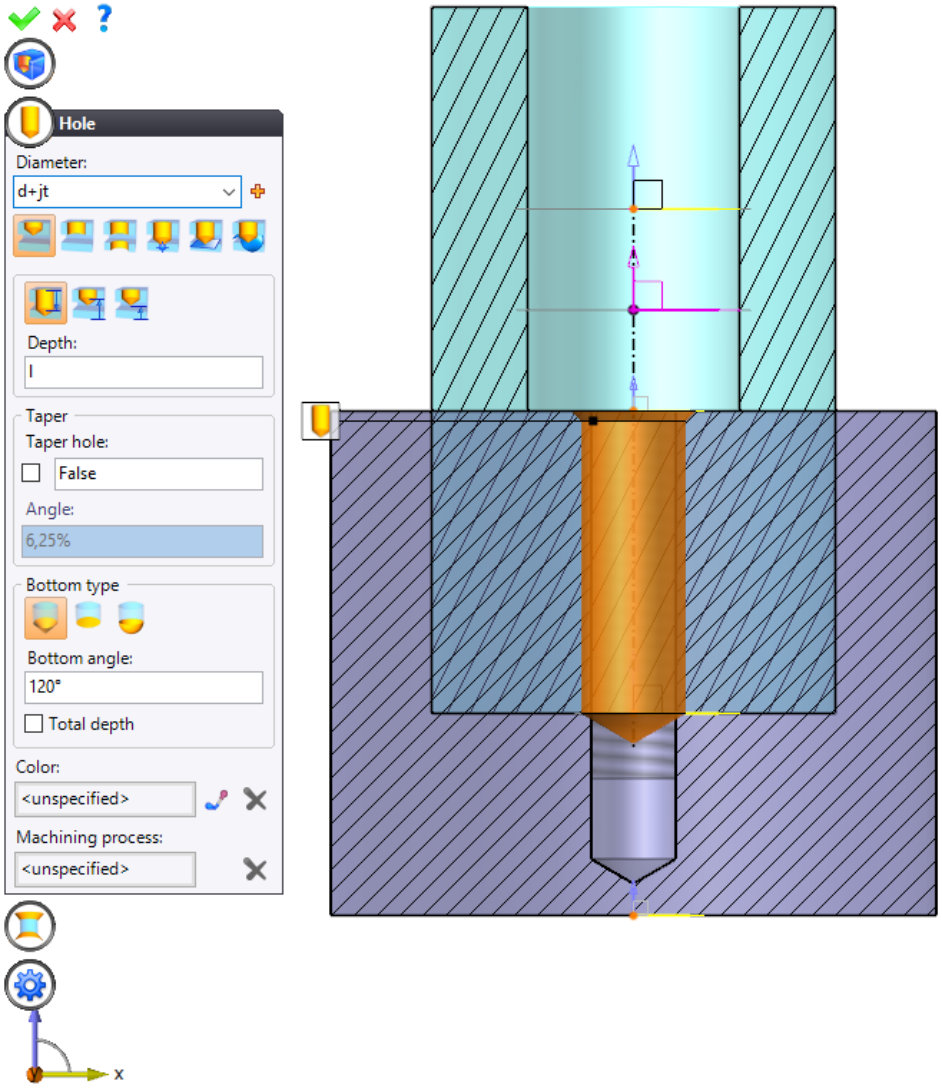
- From the Operations tree, move the **jl** parameter below the **Facing** operation which corresponds to the spot facing operation, then move the **jt** parameter below the **Clearance Hole** operation which corresponds to the clearance hole.



- Always from the Operations tree, edit the **Facing** operation and replace the screw spot facing operation by a flat-bottomed blind hole with a diameter equal to *dk+jl* and a depth equal to *h*.



- Edit the **Clearance Hole** operation and replace the clearance hole operation by a blind hole with a diameter equal to *d+jt* and a depth equal to *l*.

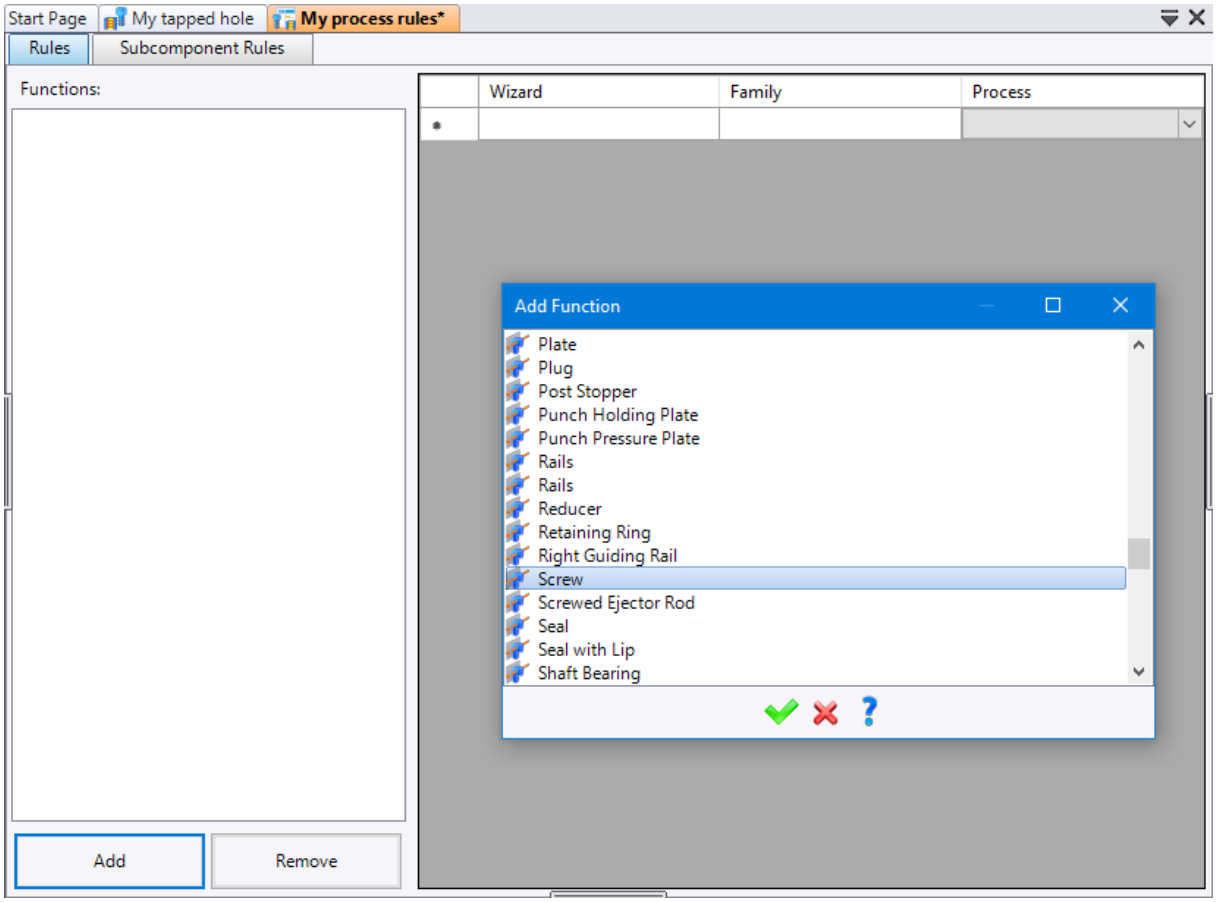


- **Save** the *My tapped hole* document and **check** it into the vault.

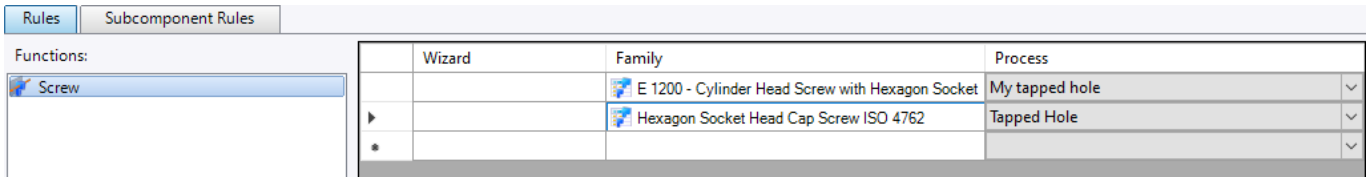
Defining a process rule according to your components

The process rules are used to define the default process for each of your library's components.

- Right-click on the *My mold library* name and create a new **Process Rules** document from the **Special** tab. Rename the document *My process rules*.
- Click on the **Add** button, select the **Screw** function from the list, and click on to **confirm**.



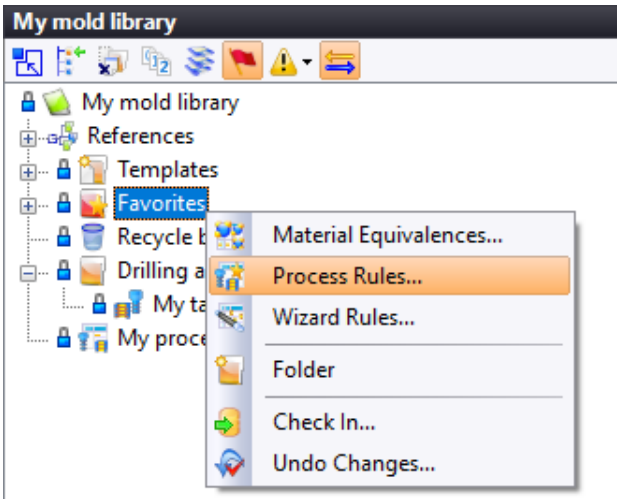
- In the **Process** column, select the **My tapped hole** and **Tapped Hole** processes.
- In the **Family** column, select the **E 1200** screw family for your process and the **ISO 4762** screw family for the standard process.



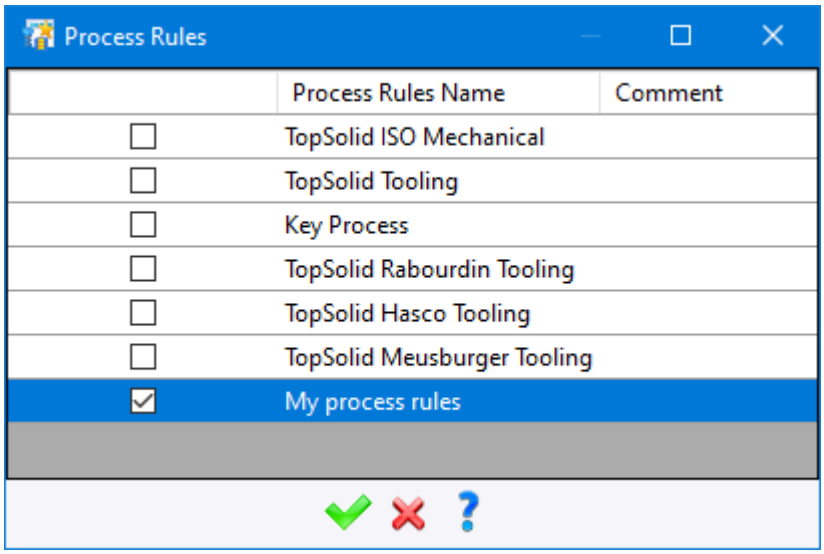
- **Save** the document and **check** it into the vault.

Testing the process rule

- Right-click on the **Favorites** folder in your project and select the **Process Rules** command.



- Select your process rules named **My process rules**.

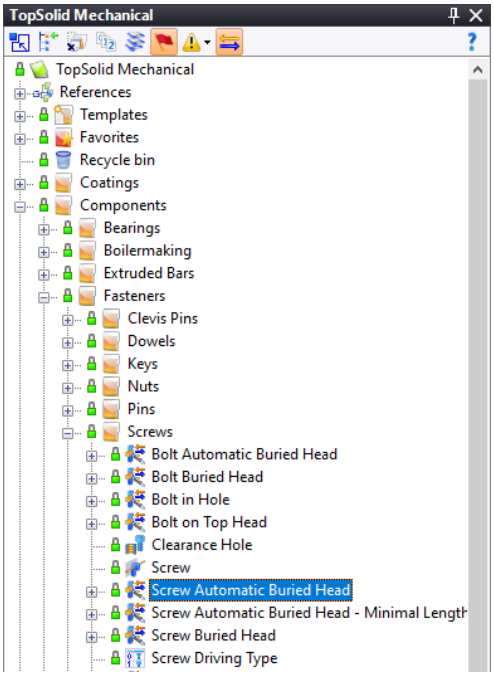


Note: This must be done in your project template so that all new projects will benefit.

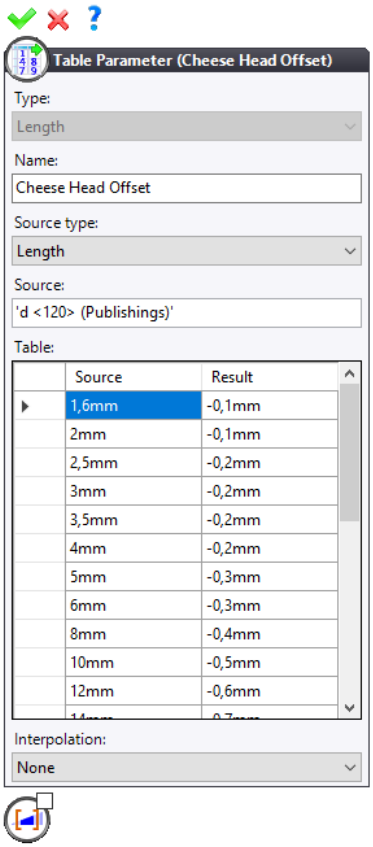
Customizing the screw insertion

Sometimes, it is also useful to manage the screw insertion depth. When positioning screws, the **Screw Automatic Buried Head** wizard allows you to manage this value.

- If you want to customize the insertion standard values to suit your habits, copy the wizard document from the **TopSolid Mechanical** > **Components** > **Fasteners** > **Screws** > **Screw Automatic Buried Head** library and paste it into your library.



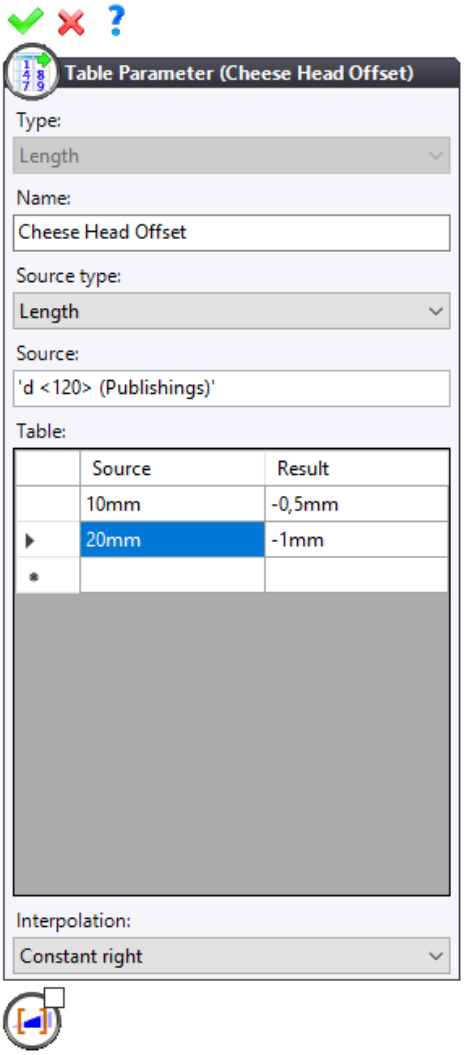
- Rename the wizard document *My screw automatic buried head.*
- Open the wizard document and edit the **Cheese Head Offset** parameter.



Here, we want to drive this value based on diameter ranges:

- from Ø3mm to Ø10mm, the clearance must be 0.5mm;
- from Ø10mm to Ø20mm, the clearance must be 1mm.

- Delete all values except 10mm and 20mm and select the **Constant right** interpolation.



Note: You can also rename the parameters according to your terminology.

- **Save** the *My screw automatic buried head* document and **check** it into the vault.

Defining a wizard as a favorite

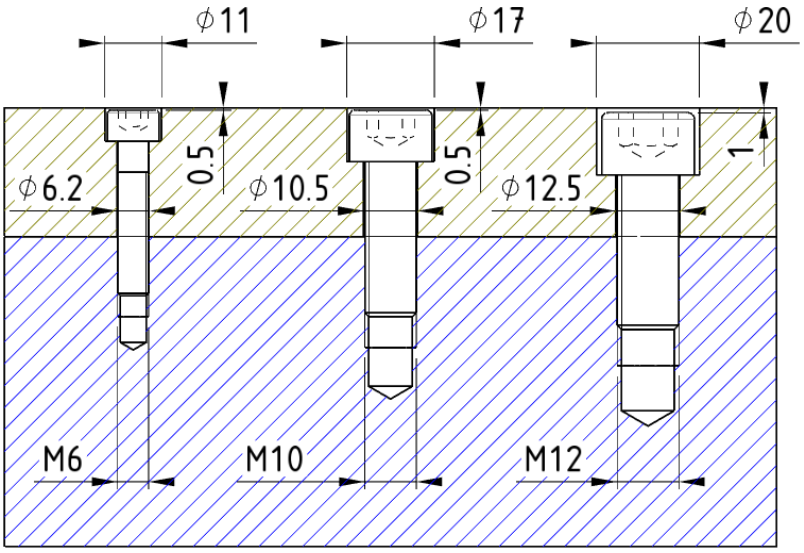
If you always use the same wizard, you can set it as a favorite so that it can be used by default when inserting your component.

- Right-click on the *My Screw Automatic Buried Head* wizard and select the **Others** > **Add to Favorites** command.
- Select the destination project.

Warning: This must be done in your project template.

Test

- Open a mold and position some screws.
- Check the values.



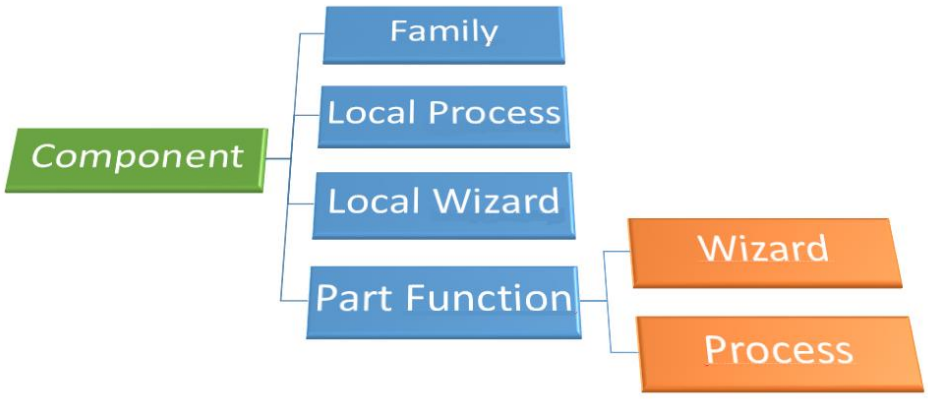
- Once all the tests and modifications are done, **validate** the life cycle of all documents.

Importing and Creating Components

In this exercise, we will import or create components and associate custom processes to them.

Note: All documents in this exercise must be created in your own library.

Summary of component's documents



A simple component is made up of a generic document, a local process and a local wizard.

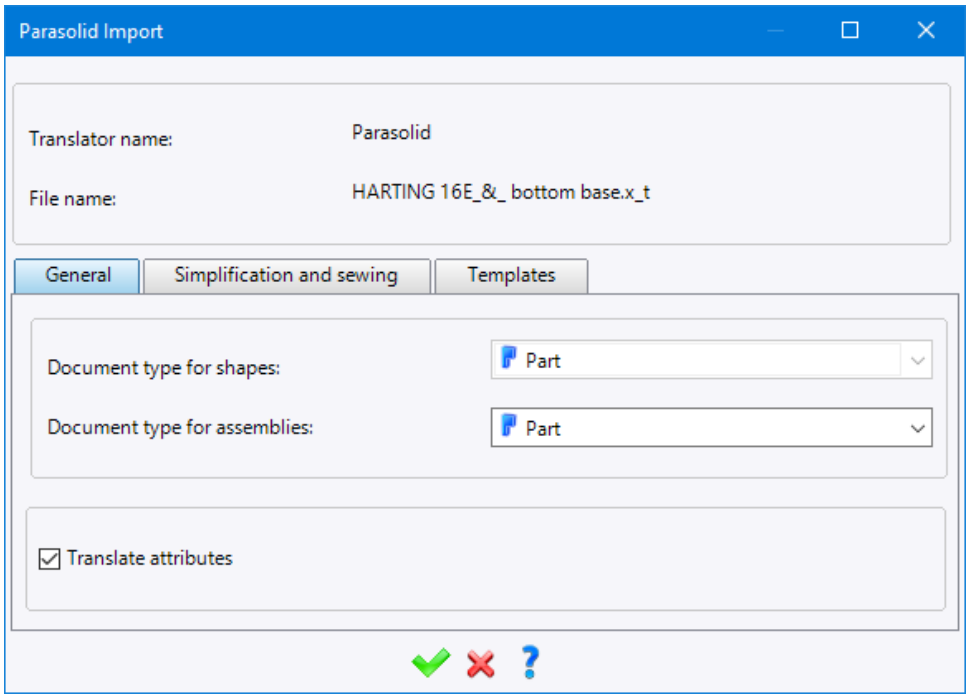
A parameterized component is made up of a generic document, a family, a local process and a local wizard.

A global parameterized component is made up of a generic document, a function and a family.

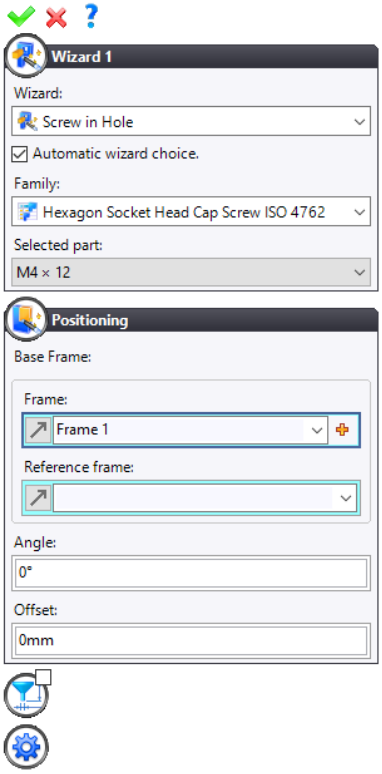
The wizard and the process are then driven by the function. Consequently, all the components that inherit the function also inherit the process and the wizard.

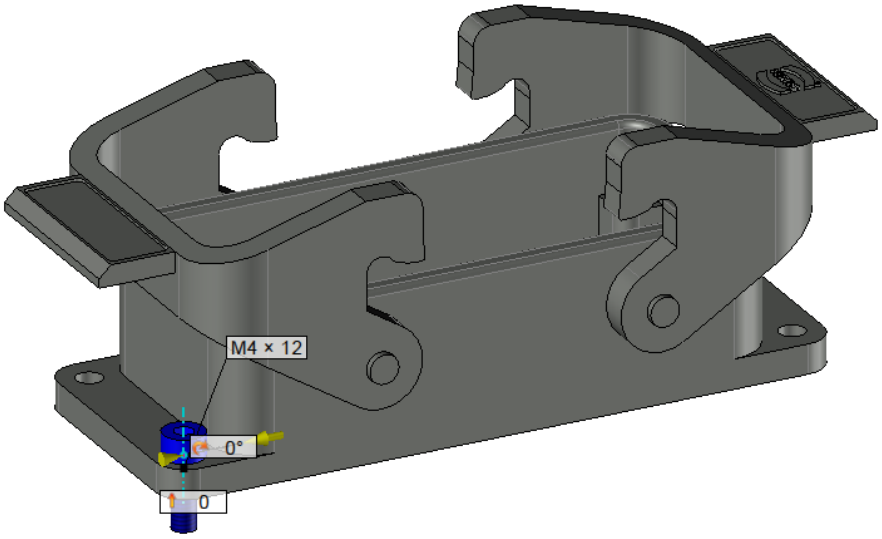
Connector

- From the *TopSolid'Mold Training D2* project, open the *Ex01 - Import and creation of components > Connector* folders, then right-click on the *HARTING 16E Bottom Base* file and select the **Convert Document** command.
- In the **Document type for assemblies** field, select **Part** since, in this example, we do not want to create an assembly document.

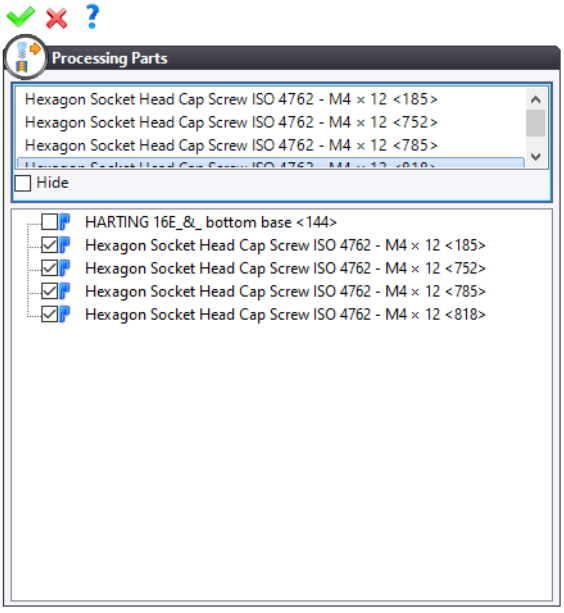


- Create a new **Assembly** document and rename it *HARTING 16E*.
- Drag and drop the *HARTING 16E Bottom Base* part document into the assembly document you just created.
- Select the **Quick Search** command and enter *screw%4762*.
- Drag and drop the **Hexagon Socket Head Cap Screw ISO 4762** family into the assembly document.
- Select the **Screw in Hole** wizard and the **M4 x 12** code.
- Position the screw in one of the holes as shown below.





- Press the **Esc** key when the process dialog box appears.
- Repeat this operation for the four holes or create a repetition by double plane symmetry.
- From the **Tools** tab's drop-down menu, select the **Processing Parts** command.
- Select the four screws.

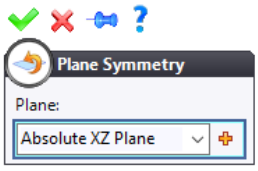


- **Save** the *HARTING 16E* assembly document.

Defining the symmetry plane

To define this part as identical in a repetition by symmetry, we need to identify the symmetry plane both in the part document and the assembly document. Here, the symmetry plane is the XZ plane.

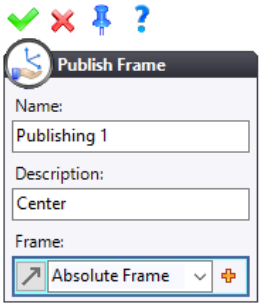
- In the part document, open the **Tools** tab's drop-down menu and select the **Symmetries** > **Plane Symmetry** command. Select **Absolute XZ plane**.



- Repeat the previous operation in the assembly document.

Defining the positioning wizard

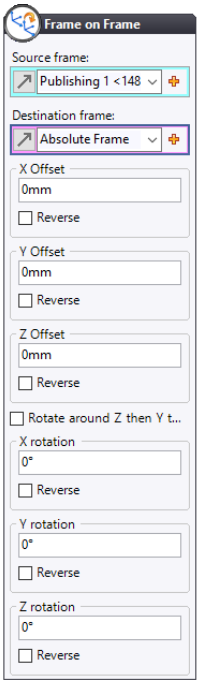
We want the connector to be positioned always in the same way, i.e. in the middle of the support face. Here, the absolute frame is the appropriate choice.

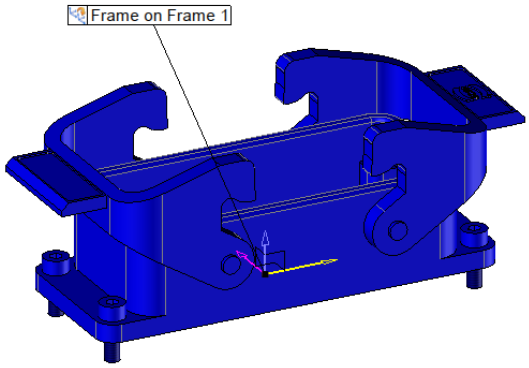


- From the assembly document's Entities tree, open the **Frames** folder, right-click on **Absolute Frame** and select the **Others** > **Publish Frame** command.
- Enter a description for this publishing.
- Adjust the properties for the bill of materials, then **save** the document.
- From the **Tools** tab, select the **Create Wizard** command.

TopSolid creates a new **Wizard** document in the Project tree.

- Rename this document *HARTING 16E - Center*.
- From the **Assembly** tab, select the **Frame on Frame** command.
- Select the frame on the connector as the **source frame** and select **Absolute Frame** from the drop-down list as the **destination frame**.





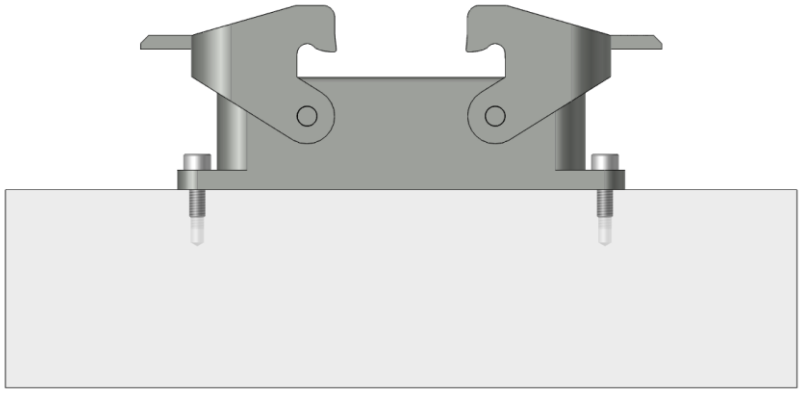
- **Save** the document.

Testing the component

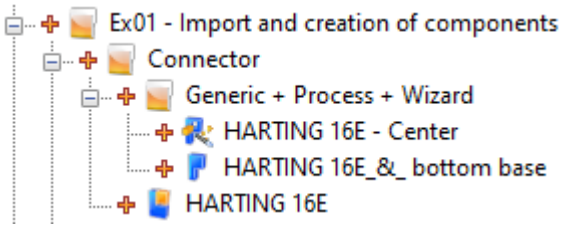
- Create a new **Assembly** document.
- Create an **in-place part** and a **block** of *200* x *200* x *50mm.*
- Drag and drop the connector's assembly document into the assembly you just created.

It is like when you position a screw; the connector will hook the nearest edges of the part.

You should end up with the following result.



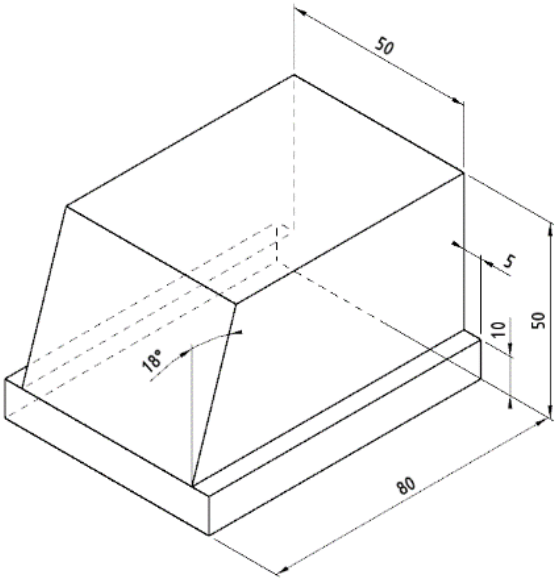
- Drag and drop the wizard and the generic (part) of the connector into the *Generic + Process + Wizard* folder.
- Delete the *HARTING 16E Bottom Base* source file.



- Move the *Connector* folder to your library.
- **Check** the component into the vault, then **validate** it.

Part component

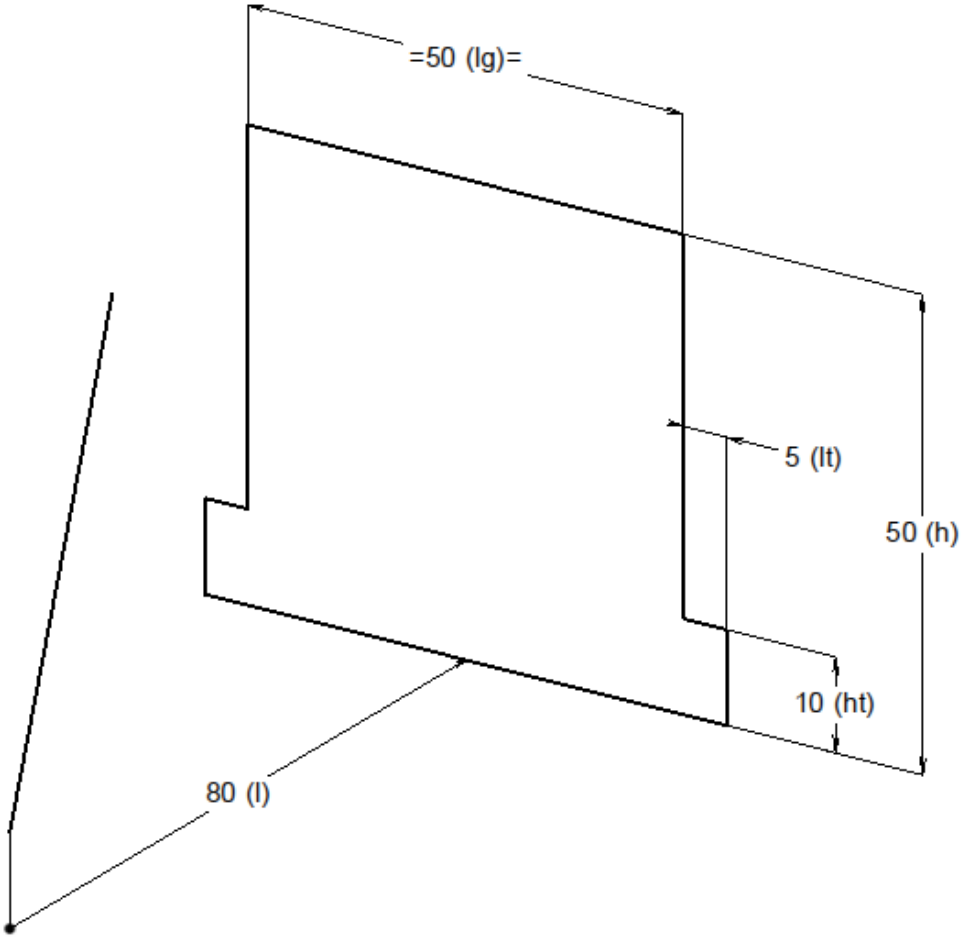
In this exercise, we want to create a slide. This part is driven as shown below.



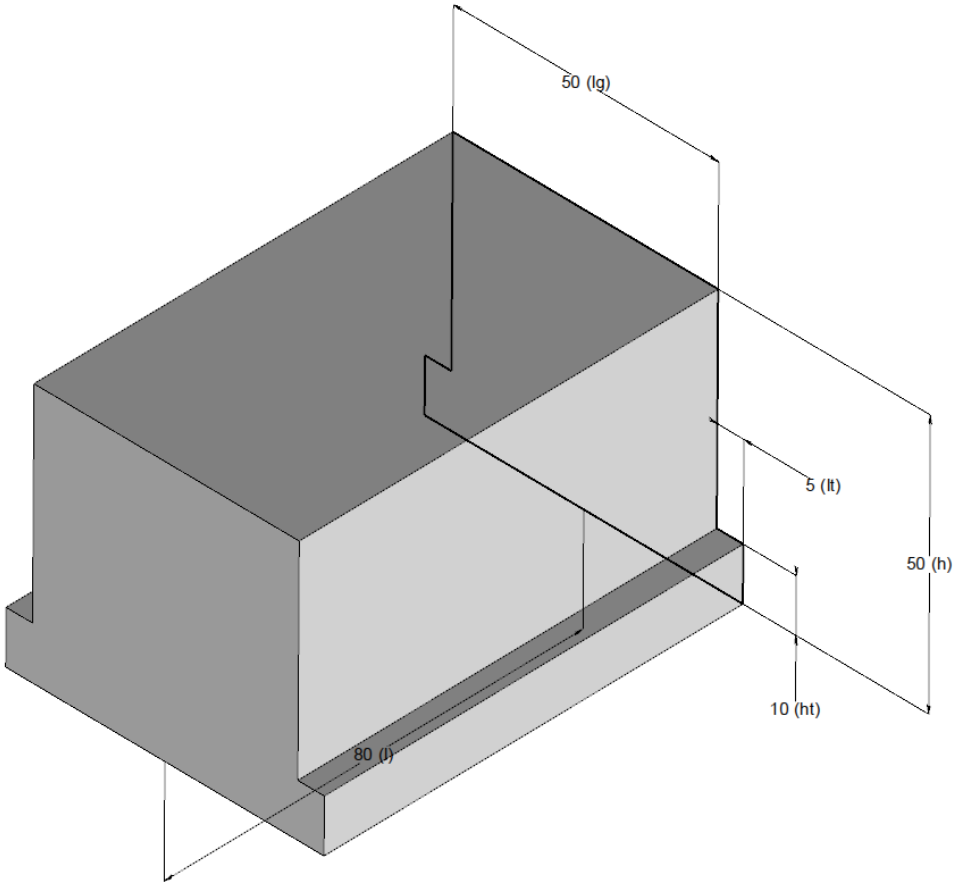
Designing the part

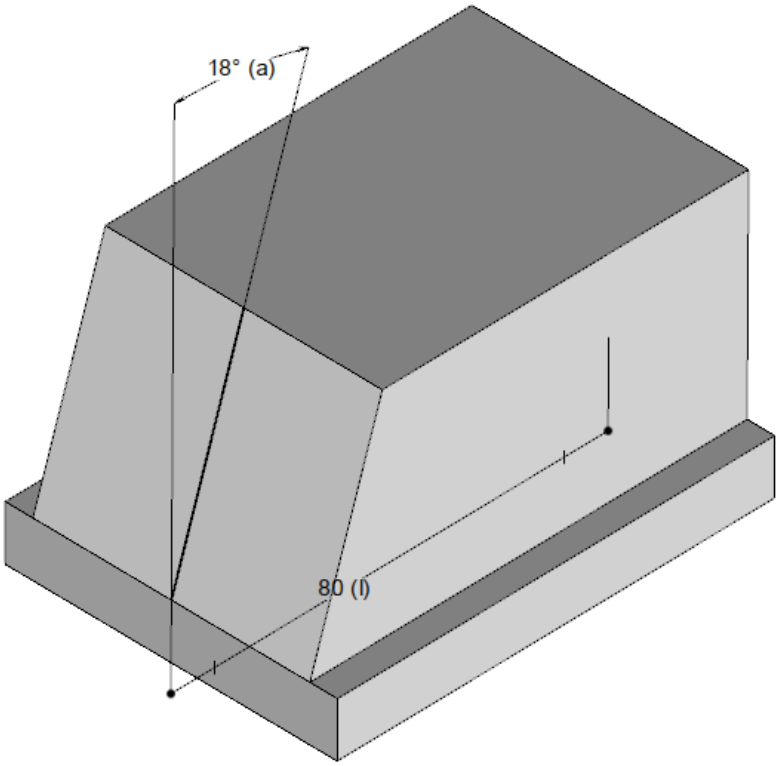
- Open the *Ex01 - Import and creation of components > Movement* folders, copy the *Slide* part document and rename it *My slide.*

The document contains the construction sketches and parameters of the part.



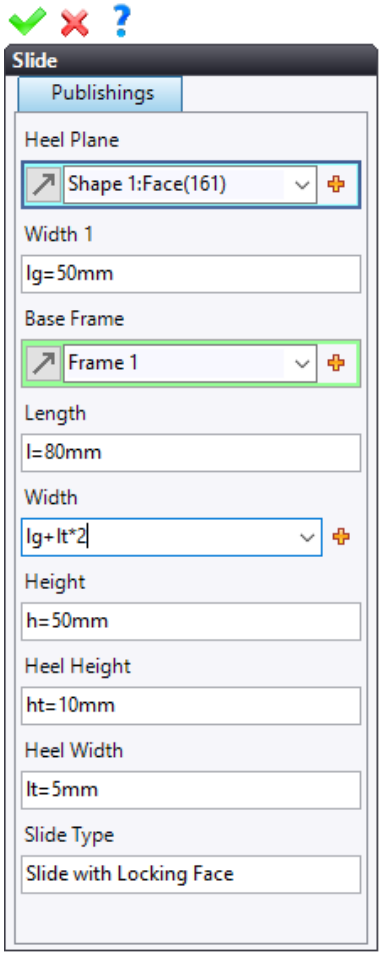
- Create an **extruded** shape from sketch 1 (slide section).
- Perform a **trim by profile** operation using sketch 2.

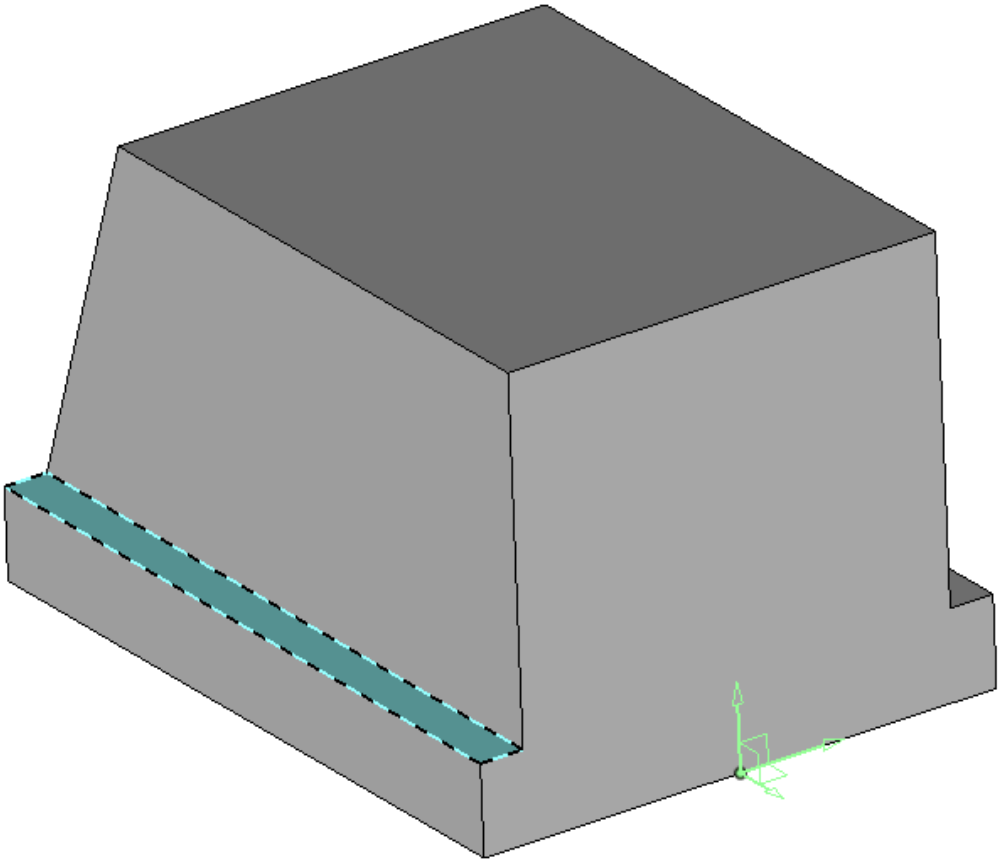




- From the **Tools** tab's drop-down menu, select the **Functions** > **Provide Function** command.
- Select the **Slide** function and fill in the fields with the part parameters.

Note: The heel plane allows you to position the slide on the receiving plate. The basic frame (frame by point and 2 directions) allows you to position the angle pin. The Z axis must be normal to the support face of the slide.





- **Parameterize** the part description as follows: *Slide [lg]x[l]x[h]x[lt]x[ht]x[a]*.
- **Save** the document.

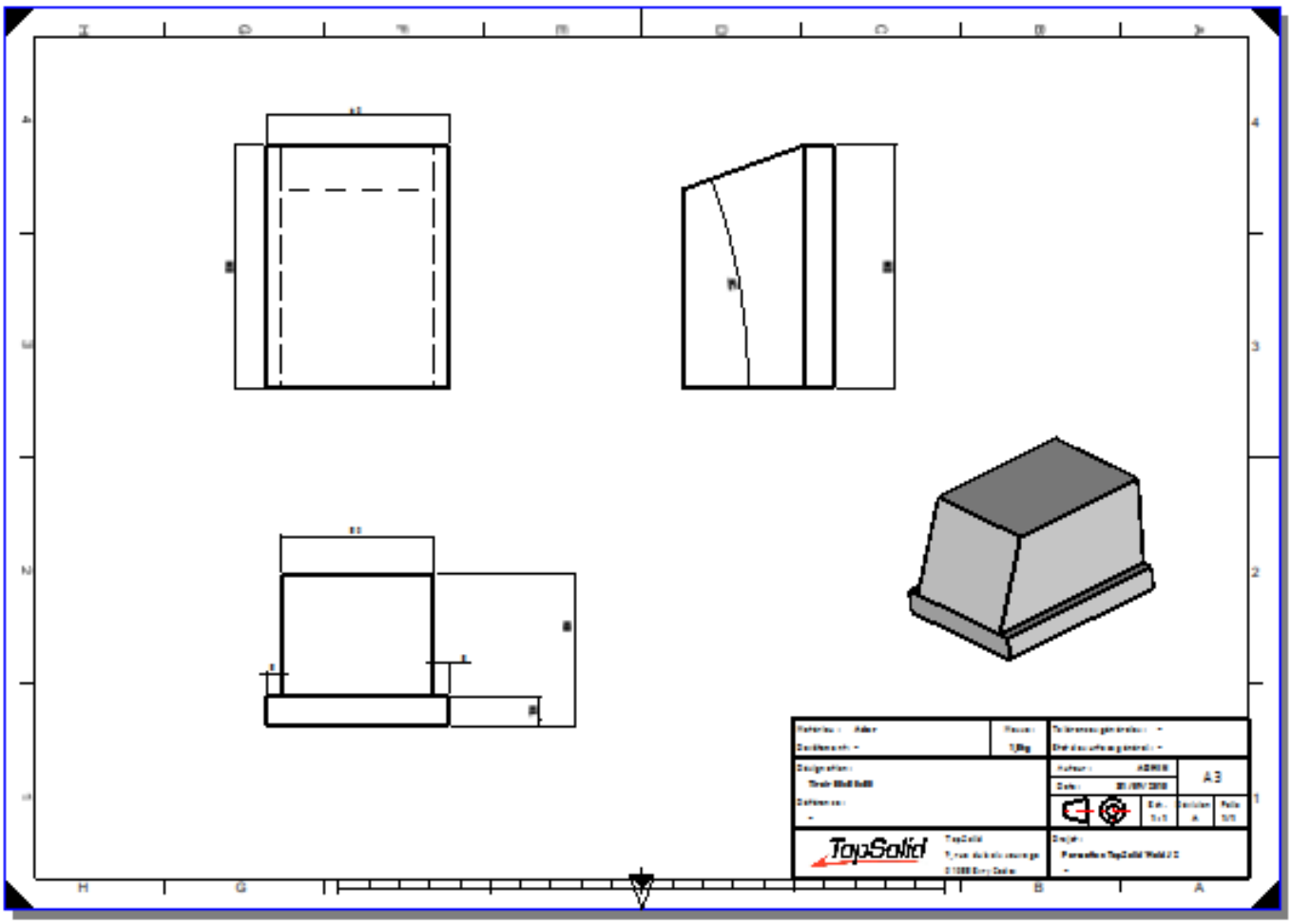
Creating the family

- Create a **Family** document from the *My slide* part document.
- From the Entities tree's **Generics** folder, drag and drop the **l**, **a**, **lg**, **h**, **lt** and **ht** parameters into the **Drivers** folder.
- **Save** the family document.

Predefined drafting

The final step in creating an intuitive component is to generate the related drafting. The goal is to save the user of the component from having to systematically recreate the drafting even if the dimensions/codes change.

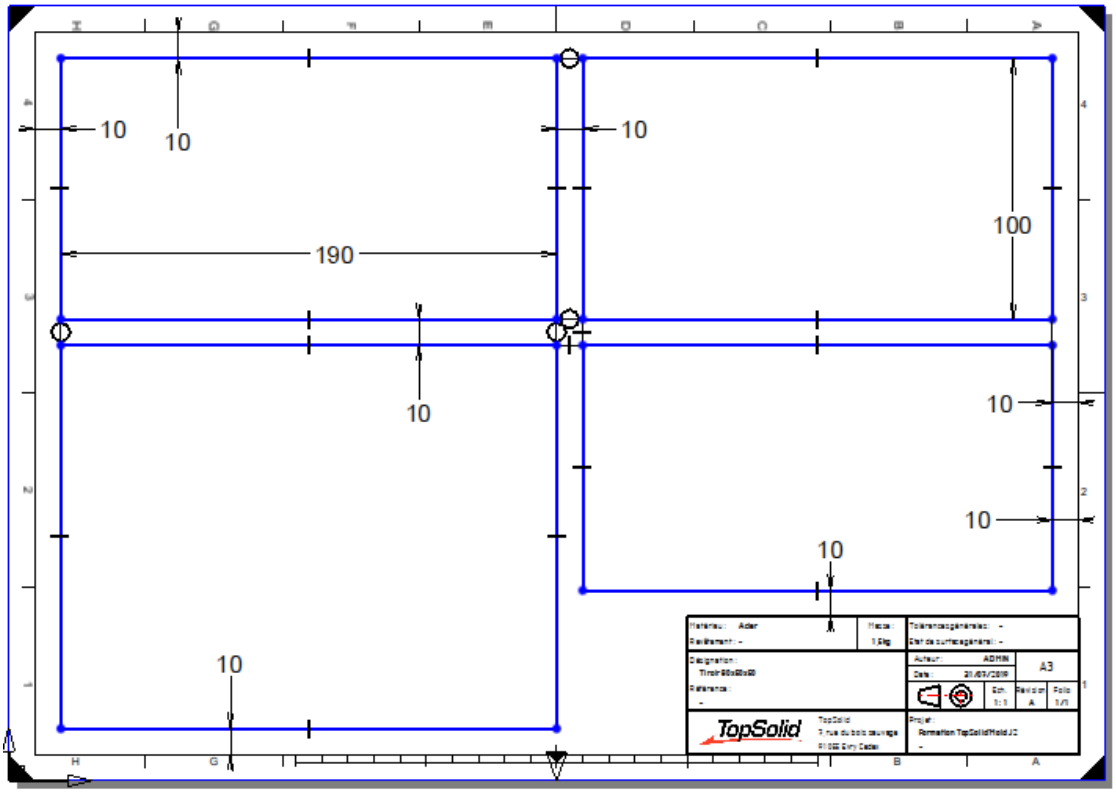
- Create a new **Drafting** document from the *My slide* generic document.
- Create a drafting similar to the one below.



- In order to save time when setting the dimensions, display the annotations of the front view. To do this, right-click on the front view and select the **Projected Annotations** command.

Note: As the component is parameterized, you have to define areas on each view. When resizing the slide, **TopSolid** will adapt the scale and recenter each view.

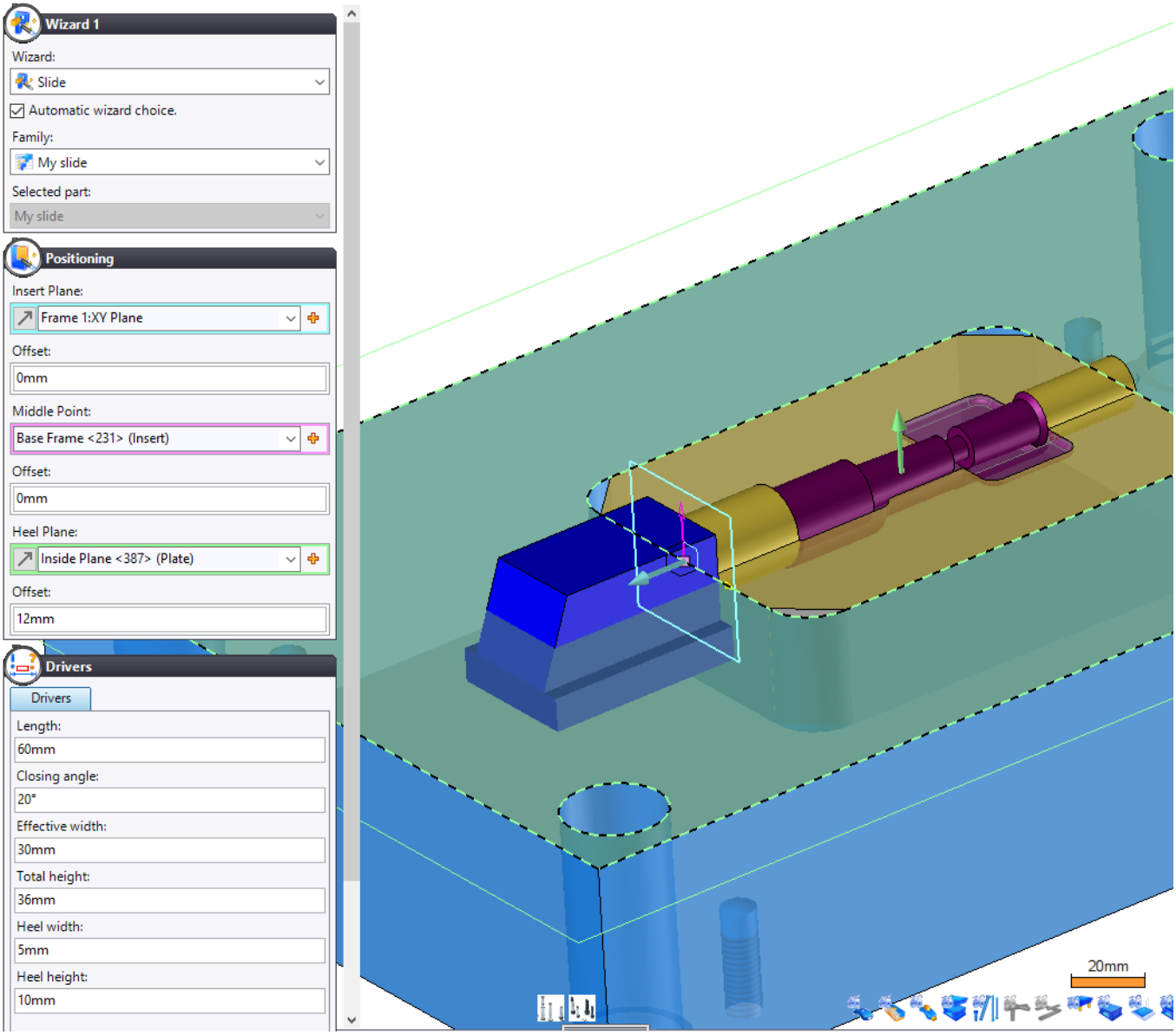
- Select the **View** > **Layout Sketch** command and dimension the sketch as shown below.



- Confirm the layout sketch.
- Optimize the layout of the views using the **View** > **Optimize View Layout** command. The views are replaced according to the layout sketch.
- Declare the drawing as a predefined drawing using the **Tools** > **Predefined Drafting** command.
- **Save** the drafting document.

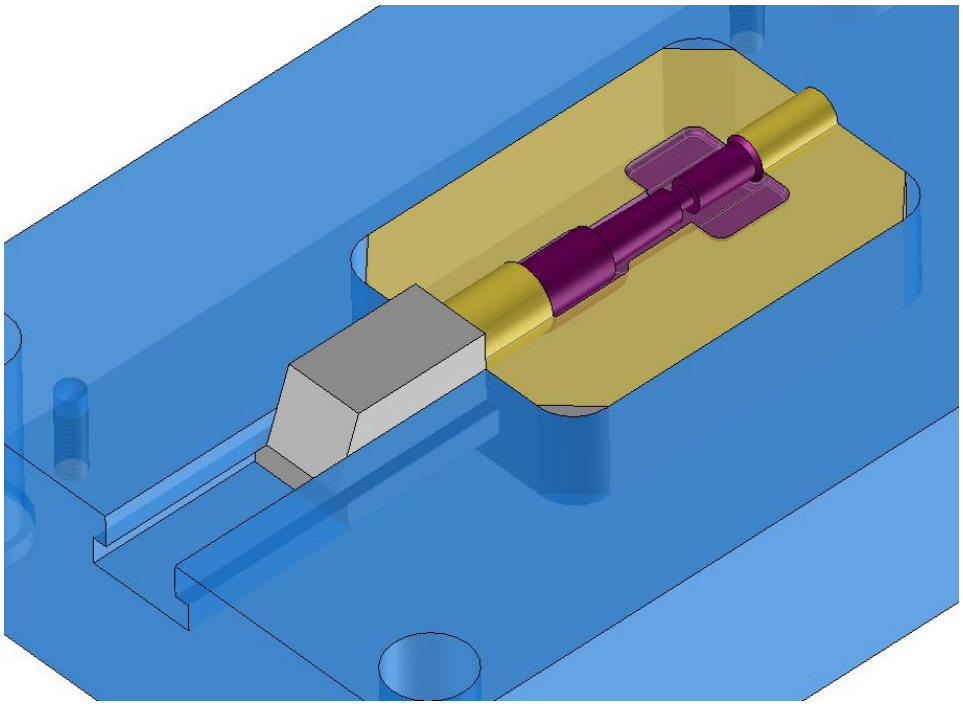
Testing the component

- From the *Mold* folder, open the *Test* mold document.
- From the **Construction** tab, create a **frame on plane** on the face as shown below.
- Drag and drop the slide's family document, then assemble the slide as indicated below:
  - **Insert plane**: Select the support face of the insert.
  - **Middle point**: Select the point that will be used to center the slide.
  - **Heel plane**: Select the plane of the plate that will be used as a guide.



- Enter the values of the slide, then click on to **confirm**.
- **Confirm** the process.

You should end up with the following result.



- Once all the tests are done, move the documents related to the slide to your library, **check in**, then **validate** the life cycle of the different documents of the component.

Assembly component with screws and processes

In this exercise, we want to create two rails equipped with fixing screws. This assembly must be driven as indicated below:

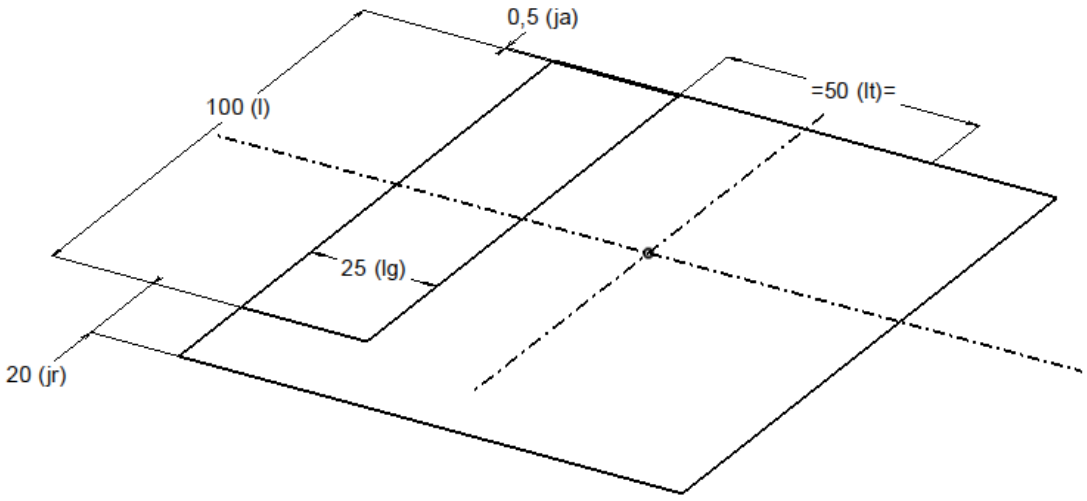
- Slide width = **lt**
- Rail width = **lg**
- Thickness = **h**
- Length = **l**
- Chamfer = **Ch**
- Front clearance = **ja**
- Rear clearance = **jr**
- Diameter of the fixing screws = **dv**

Creating the assembly

- From the *Ex01 - Import and creation of components > Movement* folder, open the assembly document named *Rail*.

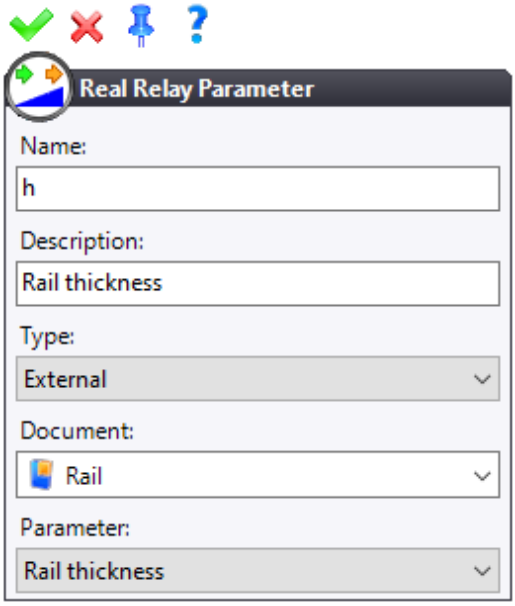
The document contains the following two sketches:

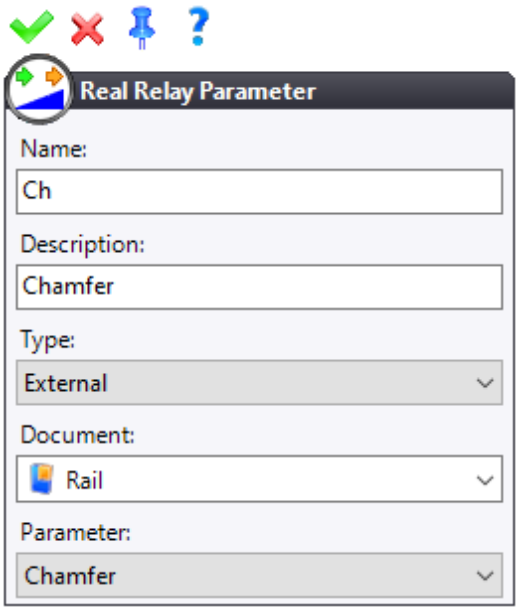
- the sketch of the rail driven by **l**, **lg** and **lt**;
- the sketch of the housing driven by **ja** and **jr**.



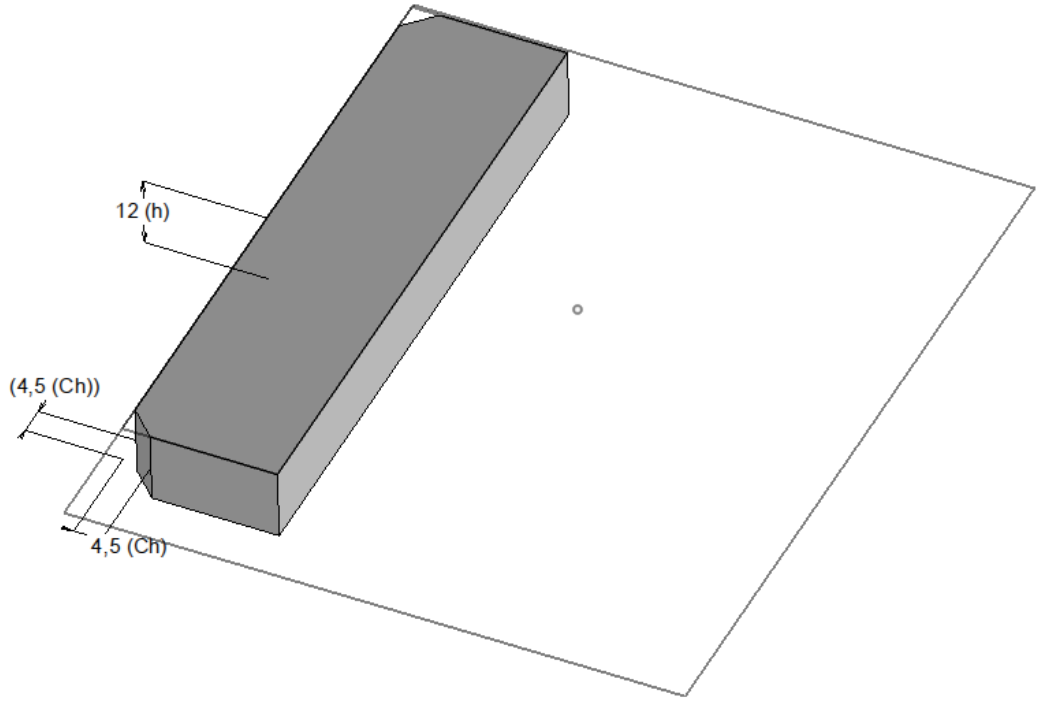
- Create a new **in-place part** using the **Steel Part** template. Rename this part *Rail.*

- Create the following two **real relay parameters**, one being the rail thickness parameter and the other the chamfer parameter.





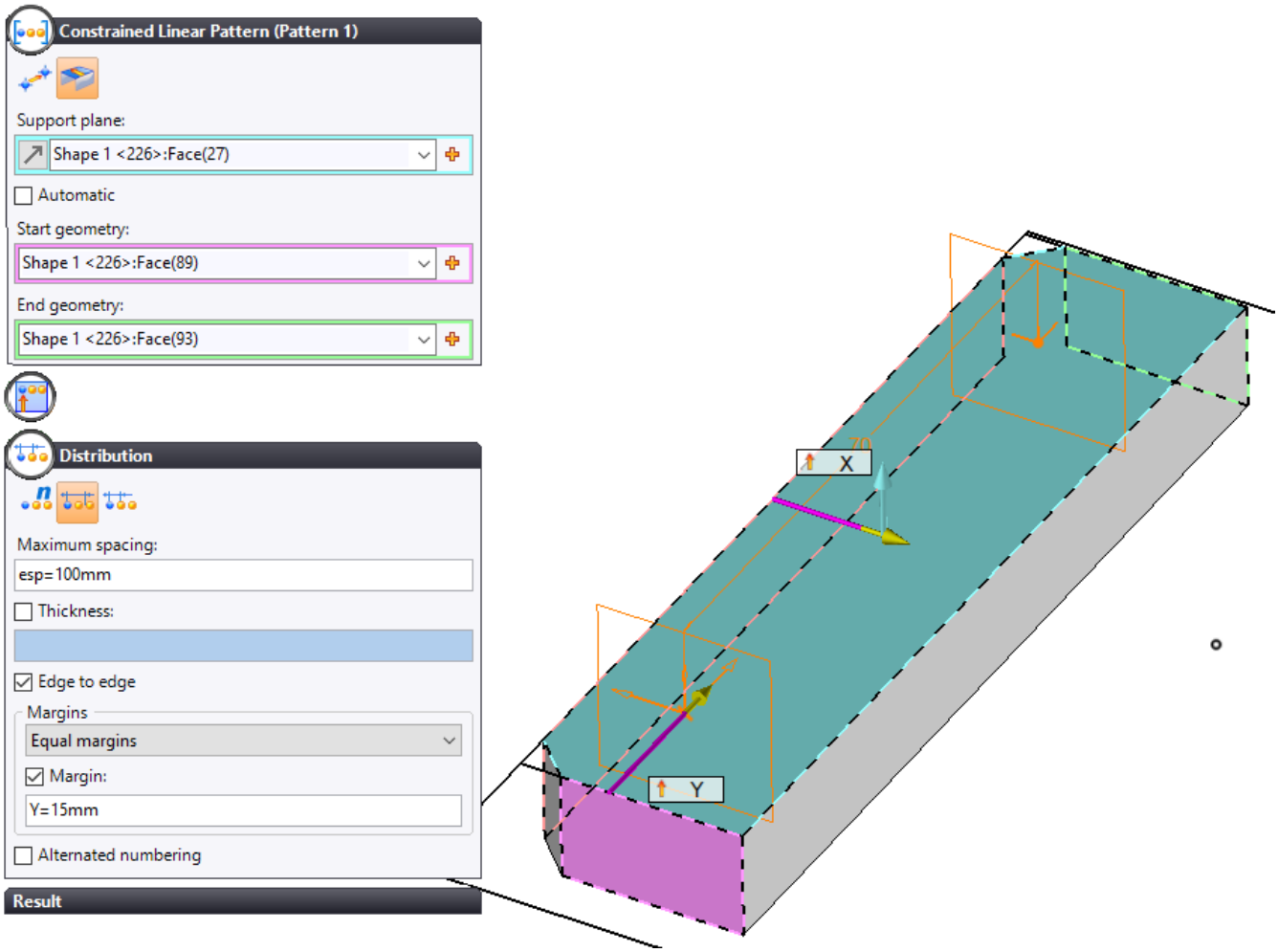
- From the rail sketch, create an **extruded** shape with a length = **h**.
- Create a **chamfer** using the **Ch** parameter on the external vertical edges.



- From the **Tools** tab, select the **Symmetries** > **Plane Symmetry** command.
- Select the **absolute XZ plane**, then click on to **confirm**.
- Hide the symmetry plane.
- **Confirm** the in-place editing.

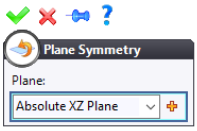
We will now create a pattern to distribute a number of fixing screws that will adapt to the length of the rail.

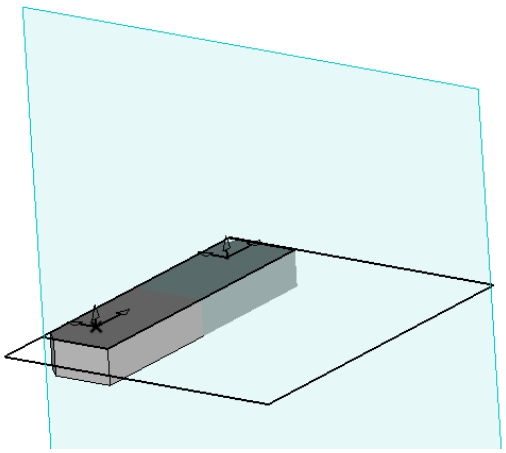
- From the **Construction** tab, select the **Constrained Linear Pattern** command.
- Fill in the fields as shown below (**X**=*10mm*, **Y**=*15mm*).



- Rename pattern 1 *Constrained linear pattern*.

- From the **Tools** tab, select the **Symmetries** > **Plane Symmetry** command.
- Select the **absolute XZ plane**, then click on to **confirm**.

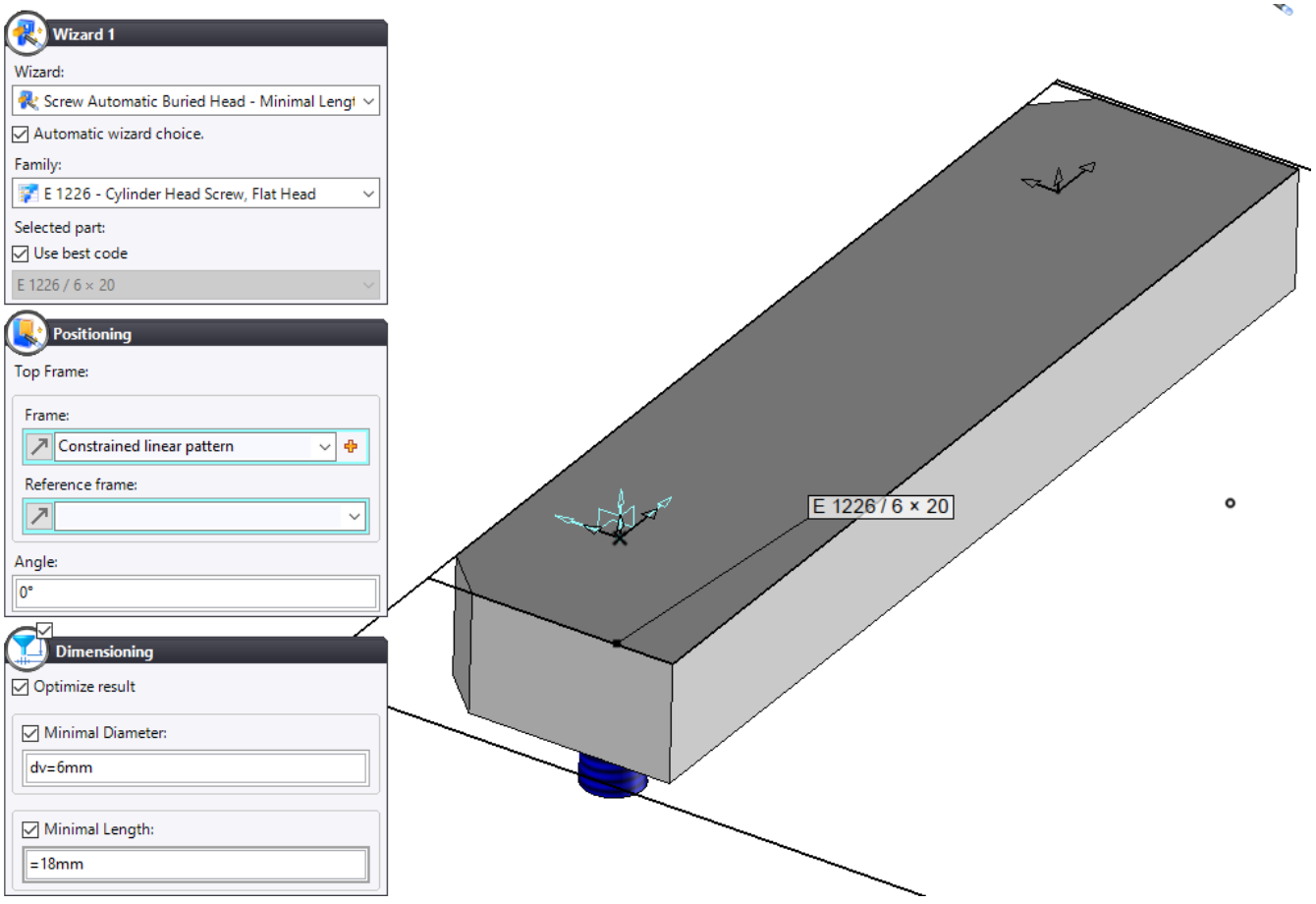




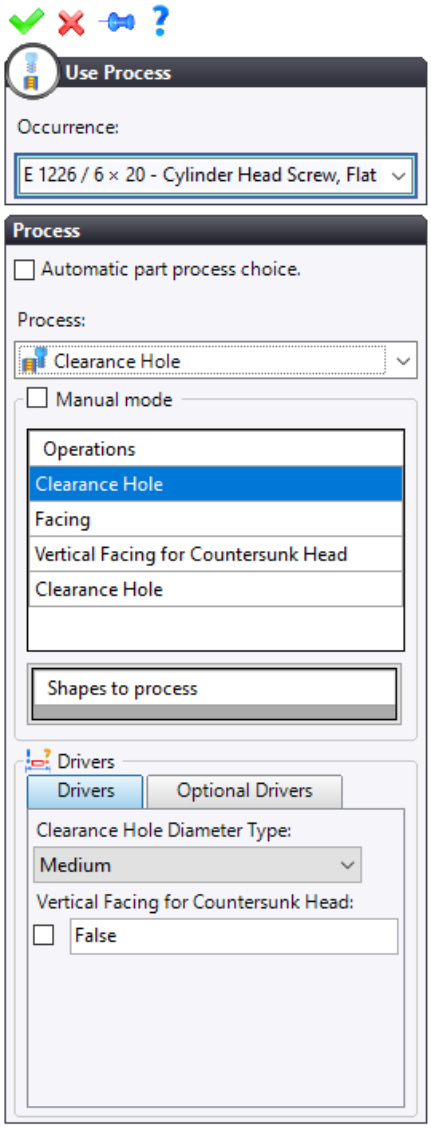
- Hide the symmetry plane.

Reminder: The definition of the symmetry plane on the rail and in the assembly makes it possible to have two identical copies of the rail.

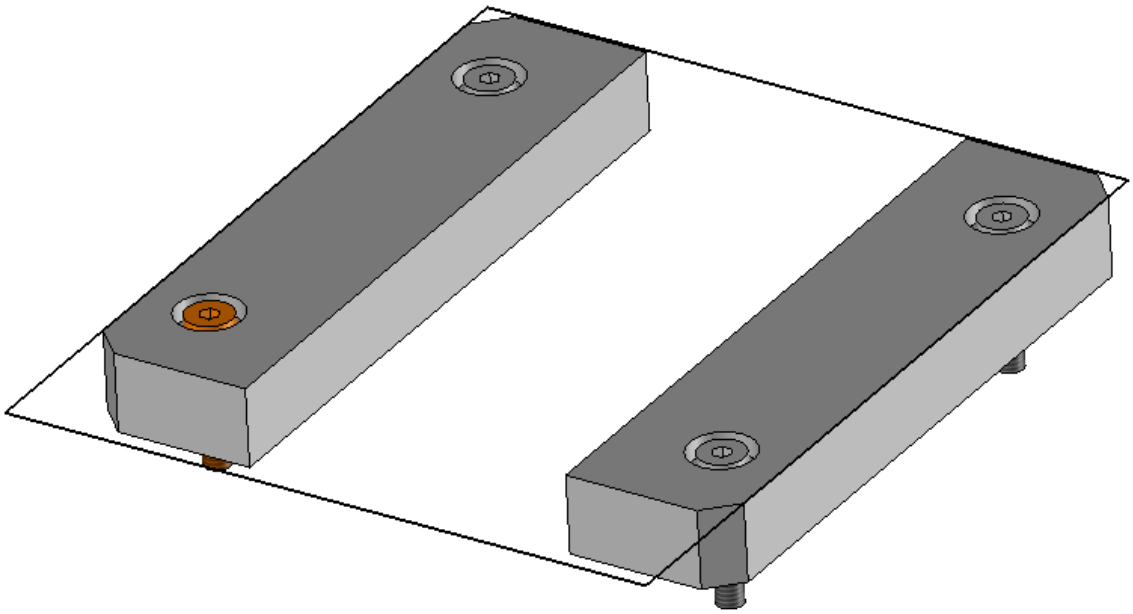
- Select the **Quick Search** command and enter *E 1226*.
- Drag and drop the **E 1226 Cylinder Head Screw, Flat Head** component into the assembly document. Select the **Screw Automatic Buried Head - Minimal length** wizard. For the dimensioning, adjust the **minimum diameter** to *dv=6mm* and the **minimum length** to *(h+dv*1.5)-dv/2*.
- Position the screw on the first frame of the constrained linear pattern as shown below.



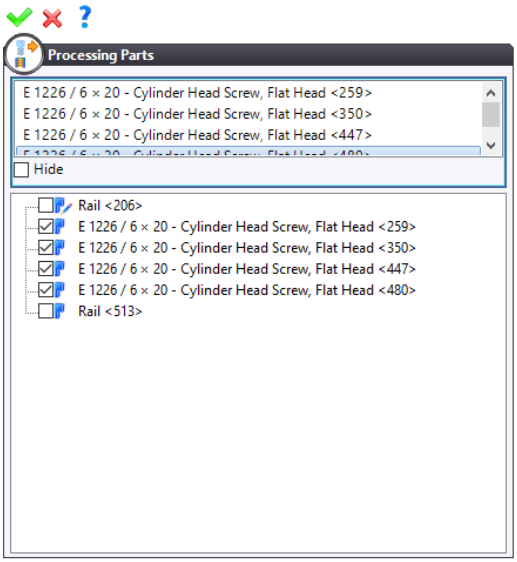
- Adjust the following parameters for the process.

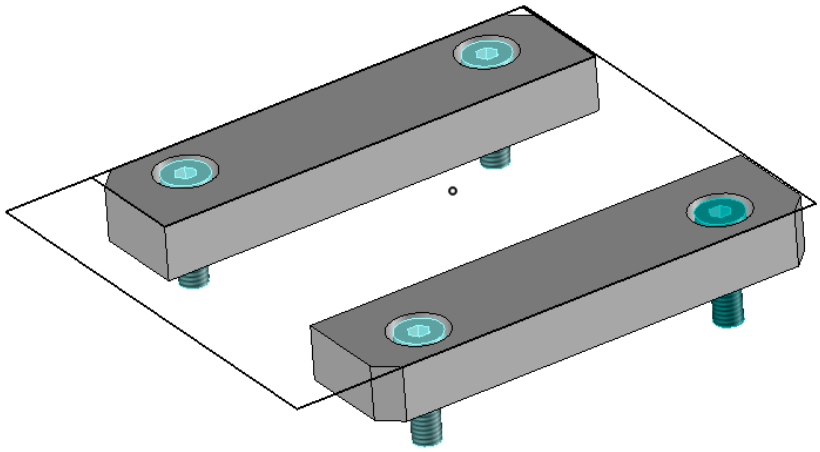


- **Repeat** the screw using the constrained linear pattern.
- **Repeat** the rail and the two screws by **symmetry** in relation to the **absolute YZ plane**.



- From the **Tools** tab's drop-down menu, select the **Processing Parts** command and select the created screws.

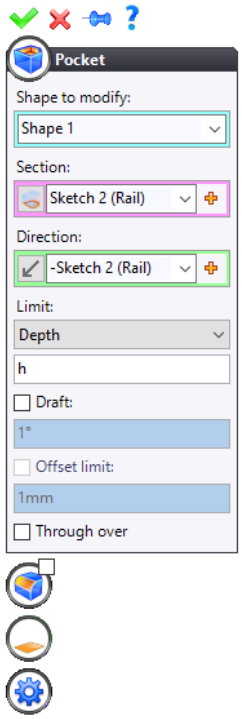


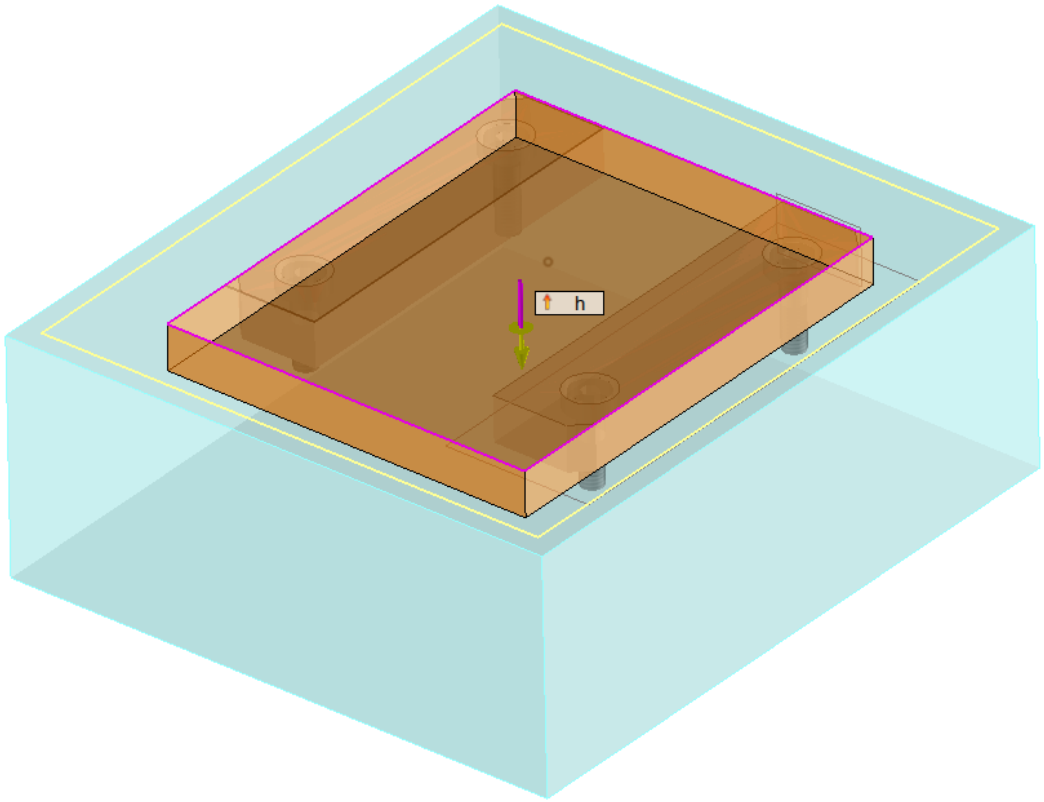


- Click on to **confirm**.
- Create a **middle plane** between the two rails, then **publish** it. Rename this publishing *Slide Middle Plane*.
- Select the bottom plane of one of the rails and **publish** it. Rename this publishing *Heel Plane*.
- Select the end plane of one of the rails and **publish** it. Rename this publishing *Slide Plane*.
- **Save** the *Rail* assembly document.

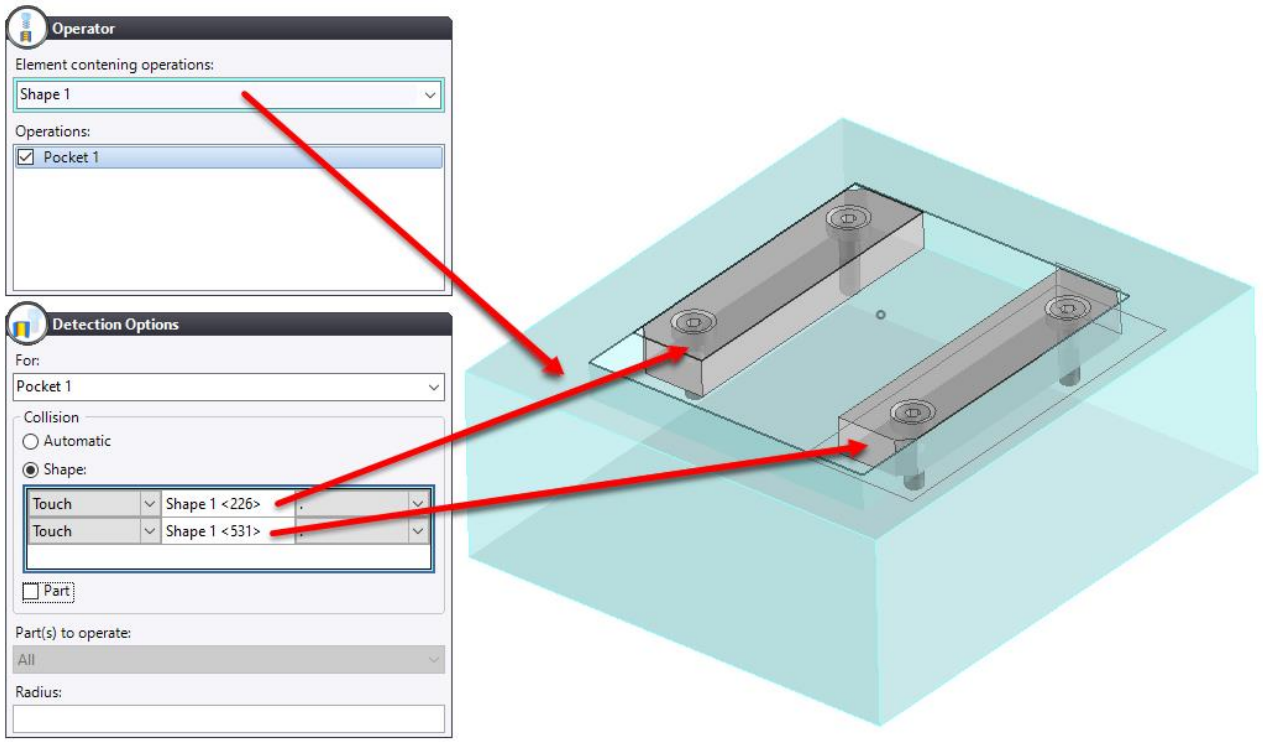
Creating the process

- From the **Tools** tab, select the **Create Process** command.
- Create a **real relay parameter** equal to the **h** parameter.
- Create a **sketch** on the same plane as the housing sketch.
- Create an **offset** of *25mm*.
- From the sketch, create an **extruded** shape with a length equal to *h+50mm*. This shape symbolizes the receiving plate of the rails.
- Adjust the **transparency** to *75%*.
- Create a **pocket** with a depth equal to **h** as shown below.





- From the **Tools** tab, select the **Operator** command and adjust the parameters as shown below.

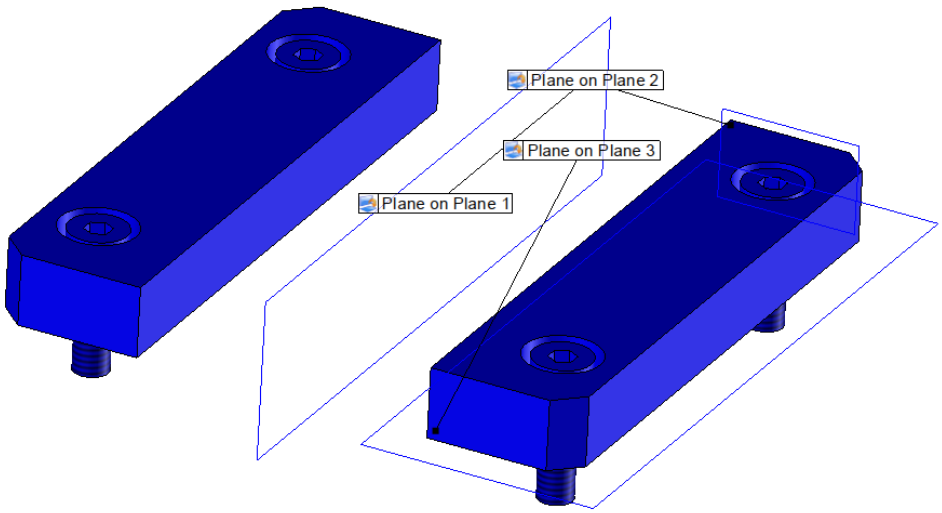


- Click on to **confirm**.
- **Save** the *Rail* process document.

Creating the wizard

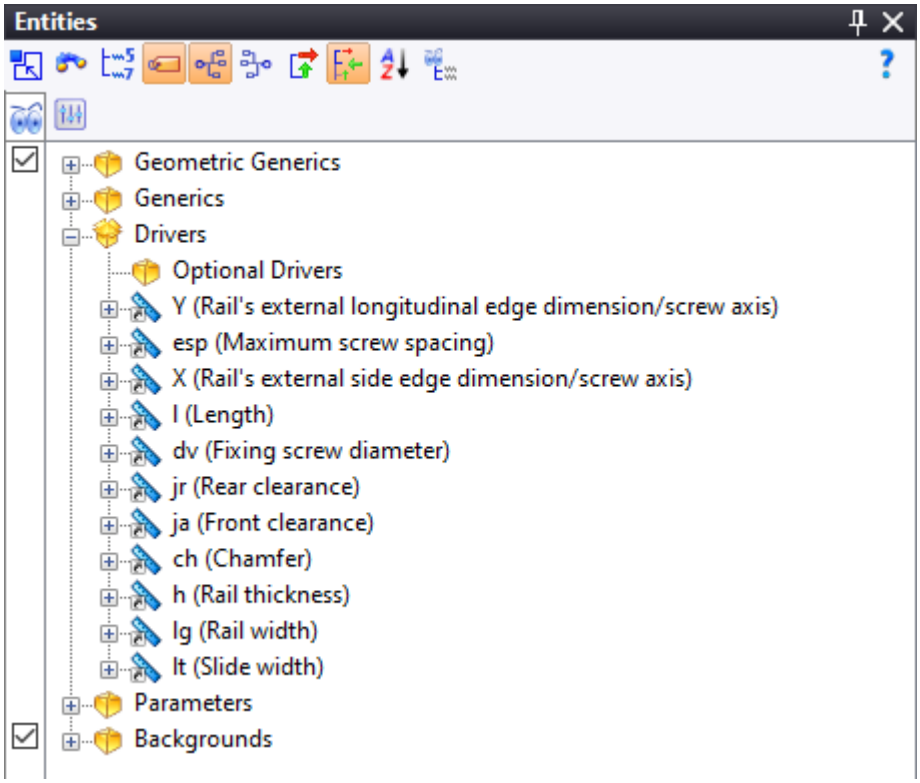
- If you closed the *Rail* assembly document, open it again.
- From the **Tools** tab, select the **Create Wizard** command.
- Create a **plane on plane** constraint between the *Slide Middle Plane* published plane and the **YZ plane**.
- Create a second **plane on plane** constraint between the *Heel Plane* published plane and the **XY plane**.
- Create a third **plane on plane** constraint between the *Slide Plane* published plane and the **XZ plane**.

You should end up with the following result.



- **Save** and **close** the *Rail* wizard document.

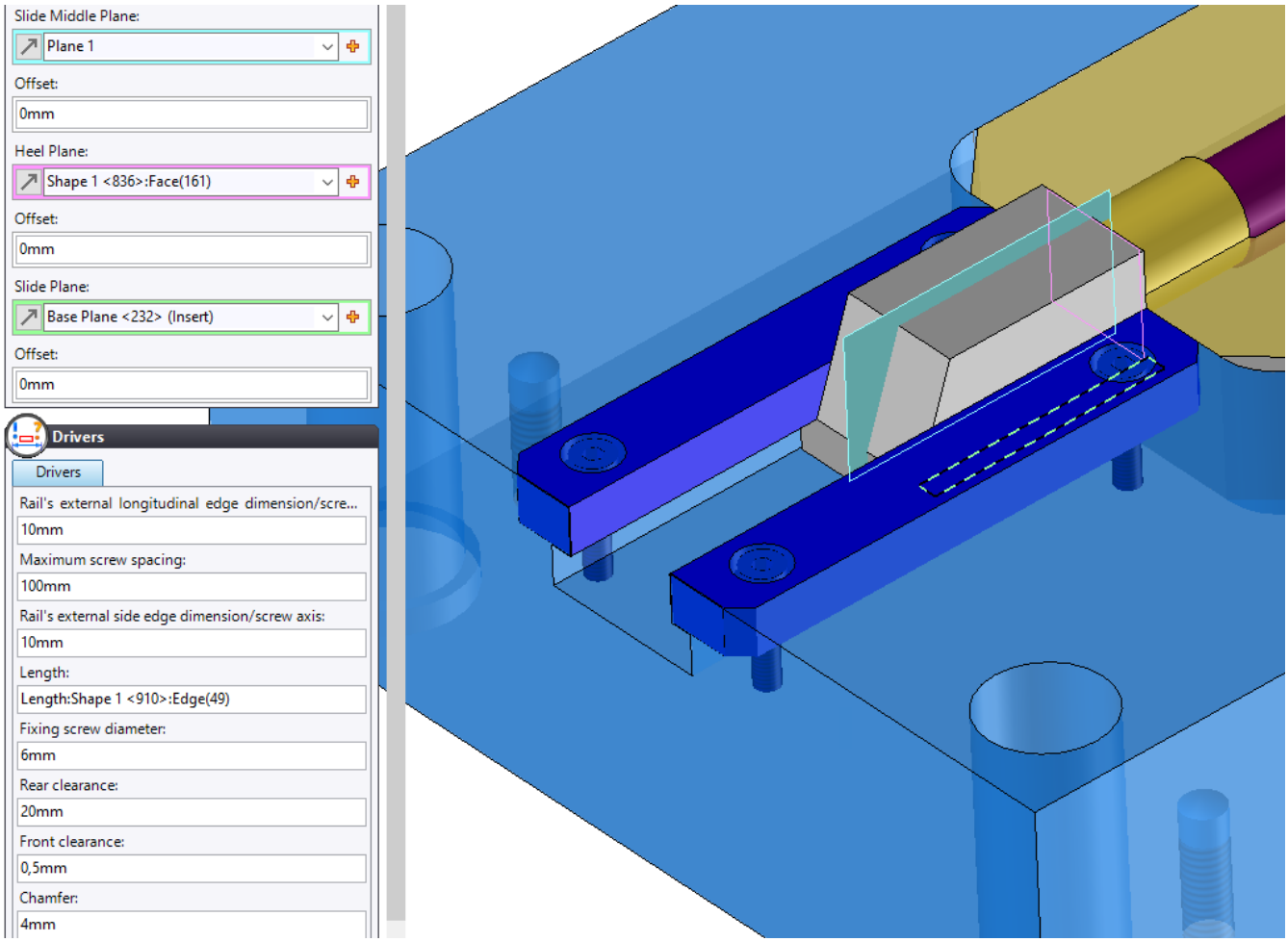
- Create a new **Family** document from the *Rail* assembly document.
- From the Entities tree's **Generics** folder, drag and drop the parameters into the **Drivers** folder.
- Order the parameters as you wish.
- Add a description for each of the parameters if necessary.



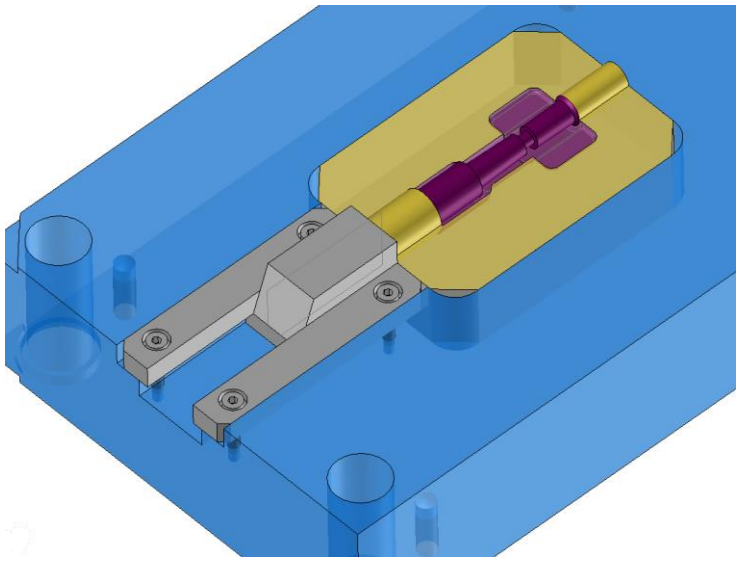
- Create a new **Drafting** document from the generic document, in the same way as for the *My slide* component.

Testing the component

- If you closed the *Test* mold document, open it again, then position the rails.

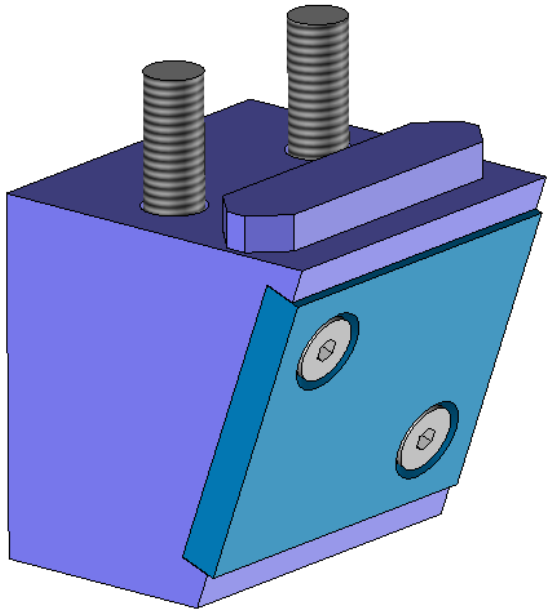


- **Confirm** the process.



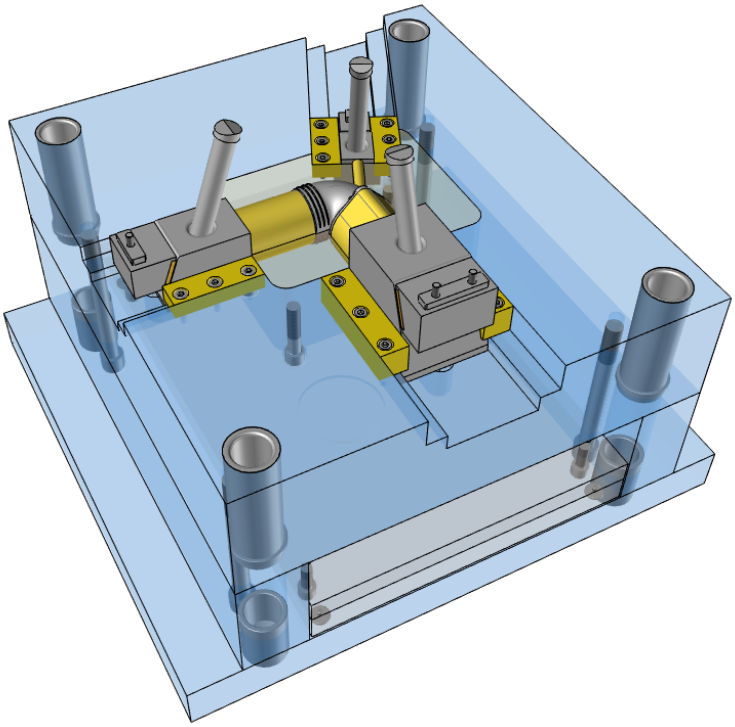
- Check the values.
- Adjust the properties for the bill of materials.
- Once all the tests are done, move the documents related to the rail (generic, wizard, process, family and drafting) to your library, **check in**, then **validate** the life cycle of the different documents.

Additional exercise: Cotter

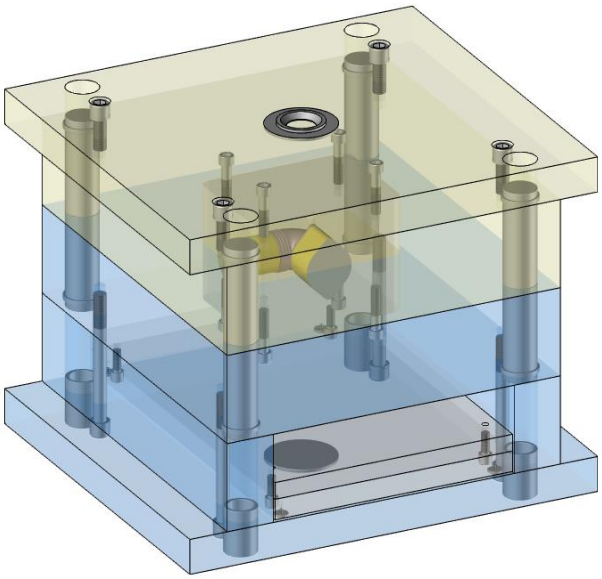


Movements

The purpose of this exercise is to determine the best strategy for creating movements in a mold.



- From the *TopSolid'Mold Training D2* project, open the *Ex03 - Movements* folder.
- From the *3- Mold* folder, open the *Elbow mold* document.



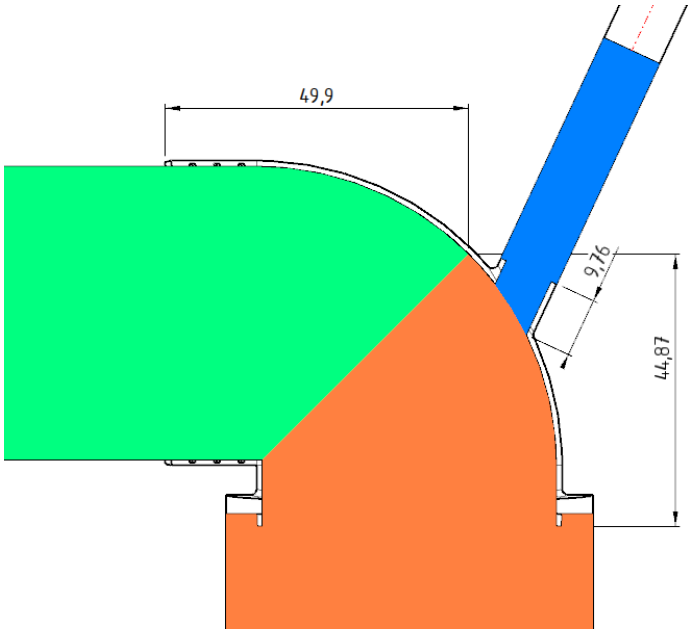
- Hide the **A side** by clicking on the icon in the graphics area.

Positioning the components

The movements consist of the following elements:

- Slide
- Cotter
- Rails
- Pin

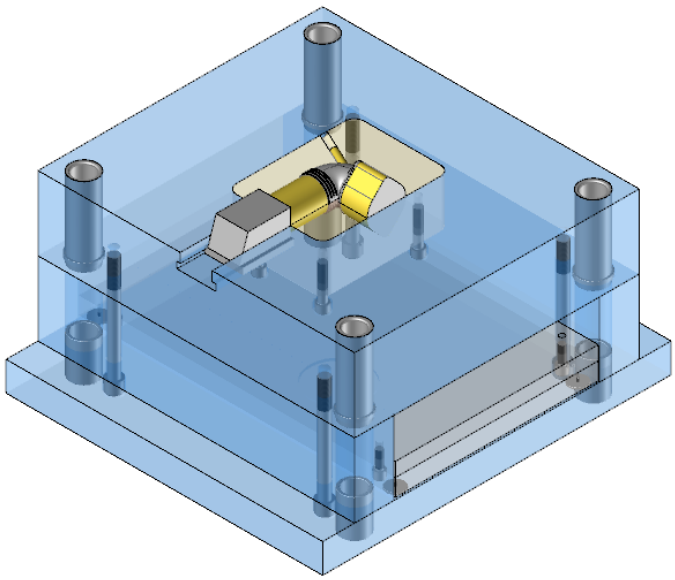
The strokes are as follows:



Movement 1: Insert on the elbow's exit side

Assembling the slide

- Select the **Quick Search** command and enter *Slide*.
- Drag and drop the *My slide* family document into the mold document.
- Adjust the values for the slide.

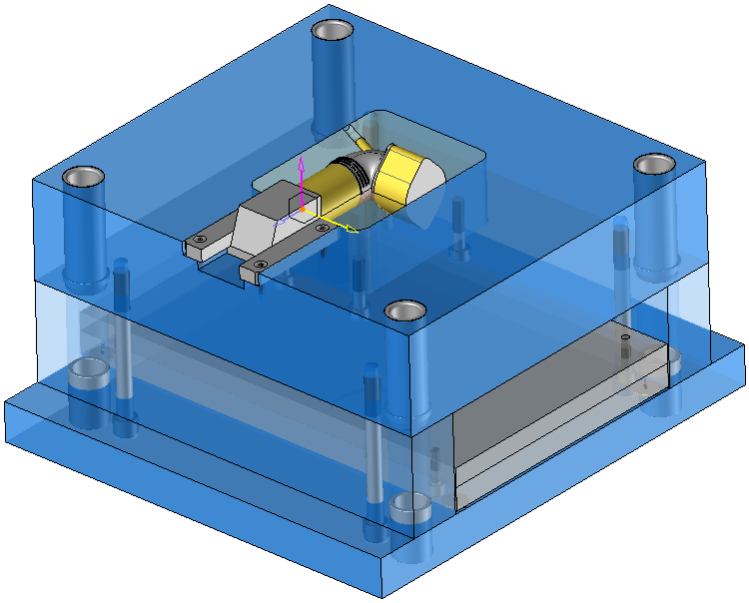


Note: If necessary, you can modify the slide process by editing it from the Operations tree. You can modify the process length for large molds (by default 500mm).

Assembling the rails

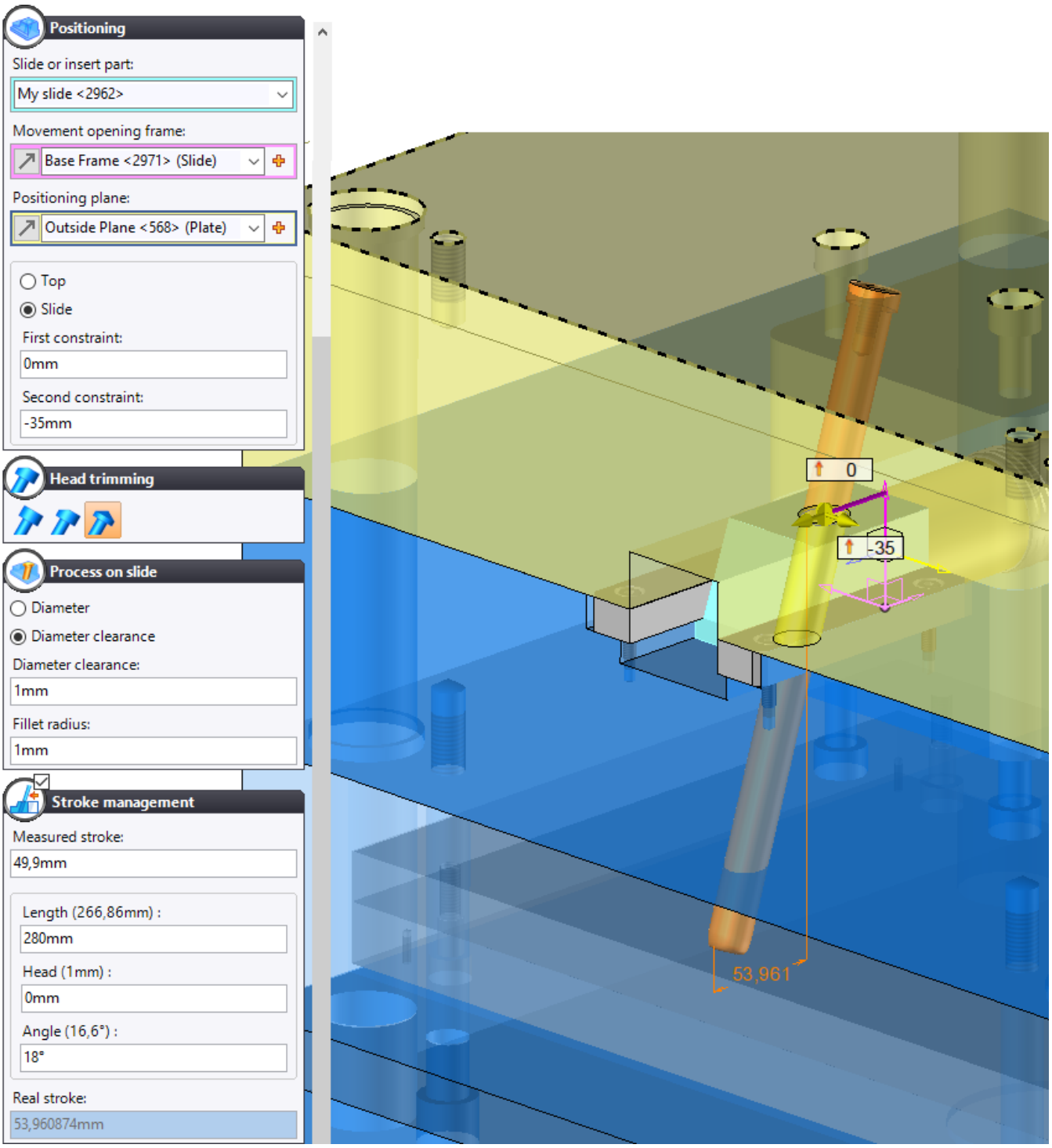
- In the quick search dialog box, enter *Rails*.
- Drag and drop the *Rails* family document into the mold document.
- Adjust the dimensions according to the slide.

The slide width can be driven associatively in relation to an edge of the slide width. The same is true for the rail height.

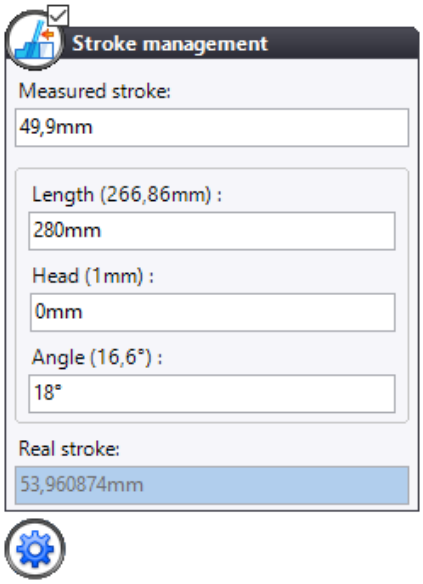


Assembling the pin

- Show the A plate.
- From the **Mold** tab, select the **Angle Pin** command.
- Select **Hasco** as the manufacturer, then select **Z 01** as the angle pin type with a diameter of *18mm*.
- Select the slide using rotary picking, then select the positioning plane of the angle pin.
- Adjust the position of the angle pin along the slide.



- Adjust the **measured stroke** to *49.9mm*.

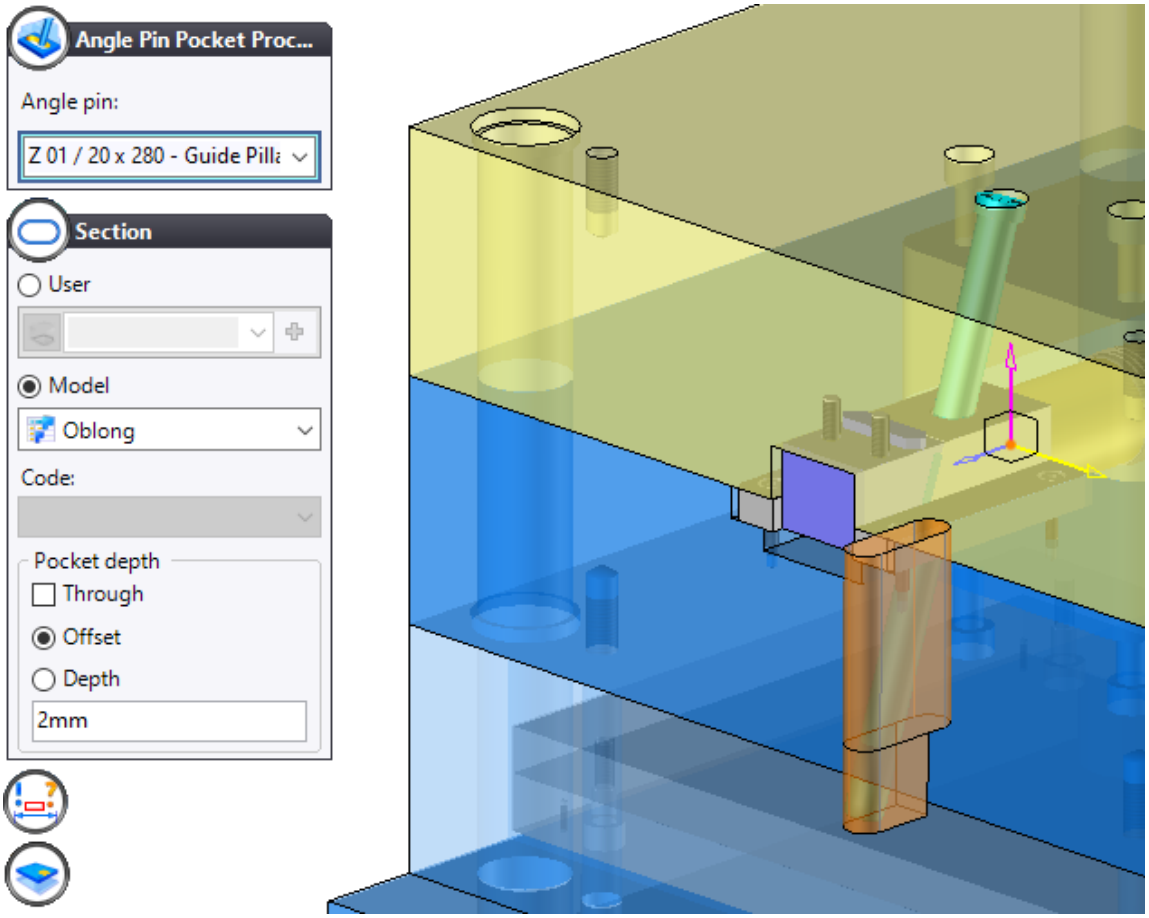


The standard length proposed by **TopSolid** is 280mm, which gives an actual stroke of 53.9mm.

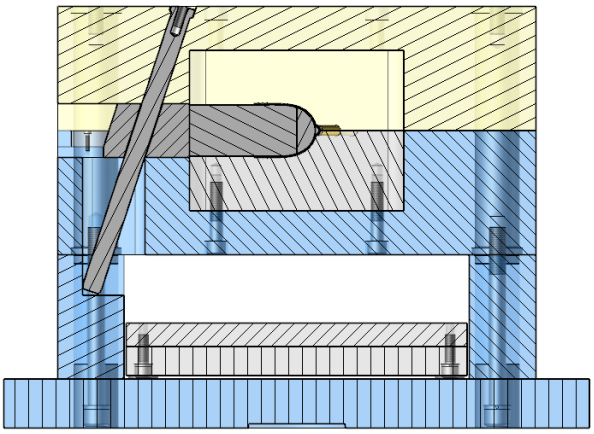
- **Confirm** the creation of the slide derivation and the process.

Angle pin pocket

- From the **Mold** tab, select the **Angle Pin Pocket** command.
- Select the **Oblong** template, adjust the **length clearance** and the **width clearance** to *2mm*, then check the **Automatic detection** box in the **Parts to process** section of the targets.

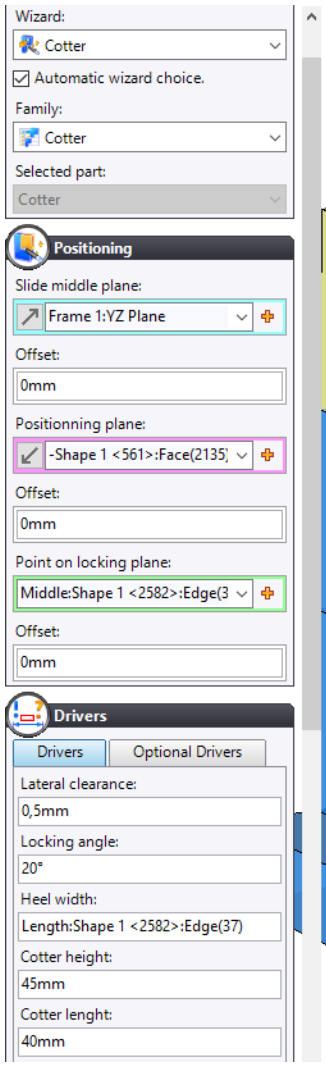


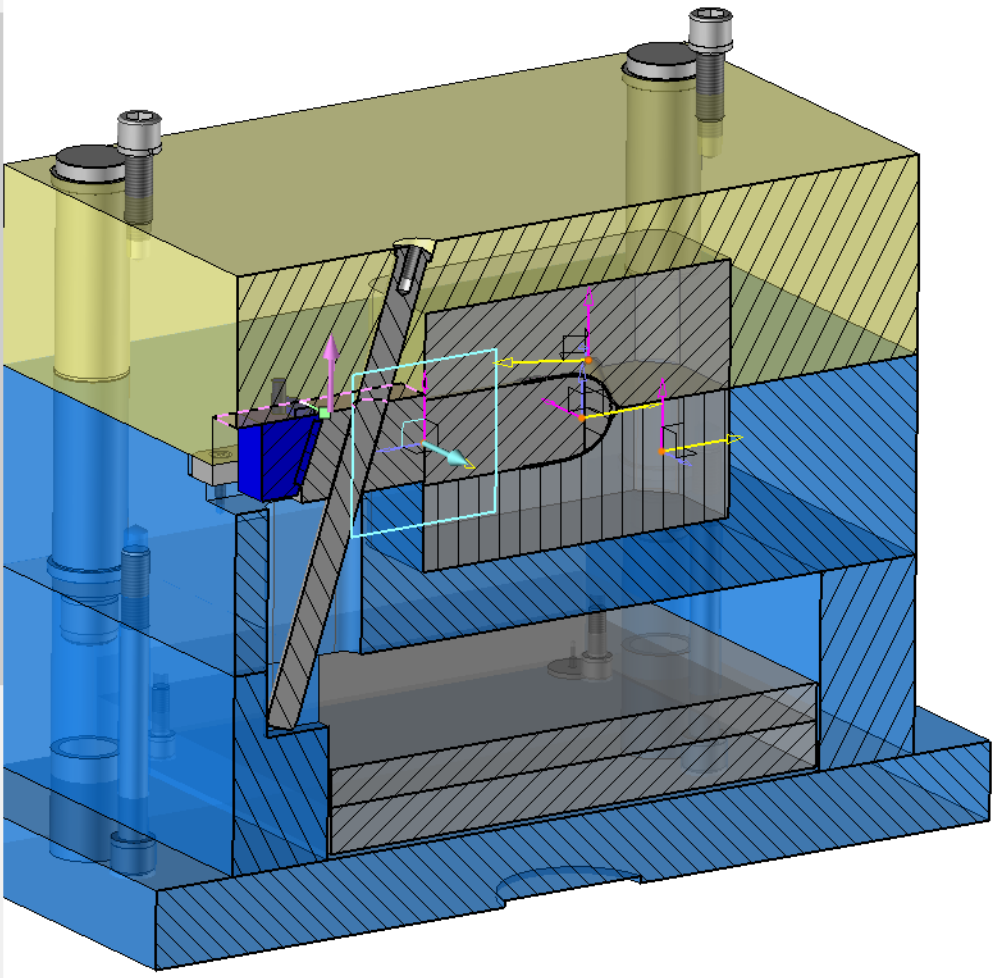
You should end up with the following cut result.



Assembling the cotter

- In the quick search dialog box, enter *Cotter*.
- Drag and drop the *Cotter* family document into the mold document.
- Position the cotter as shown below and adjust the values.





- **Confirm** the process.
- From the Operations tree, select all the operations of this movement and create a folder named *Movement 1*.
- **Save** the *Elbow mold* document.

Movement 2: Insert on the diamond-shaped carriage nose side

- Repeat the same steps as for movement 1.

Adjust the values as you wish.

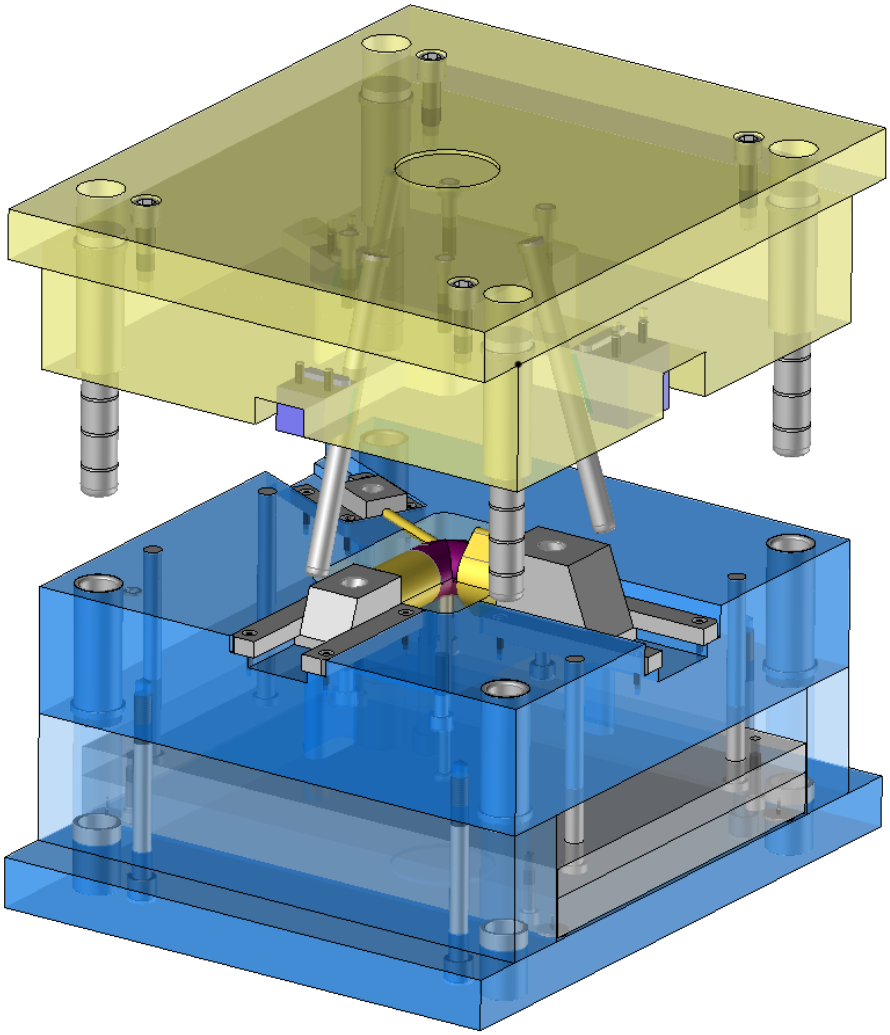
Movement 3: Insert on the inclined pin side

- Repeat the same steps as for movement 1.

Adjust the values as you wish.

- Finish the core and cavity blocks by performing **Move face** and **Fillet** operations.
- Add the cotters to the A side set.

You should end up with the following result.

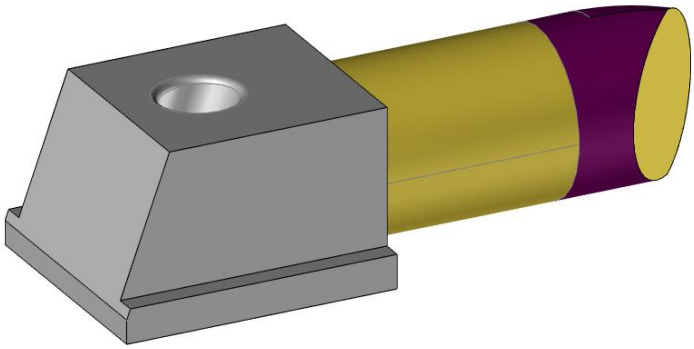


Creating the final slides

The final slide is the union of the slide with the insert.

- Double-click on the slide to edit it in place.
- Unite the slide and the insert.
- Confirm the in-place editing.

Result of the exit slide:

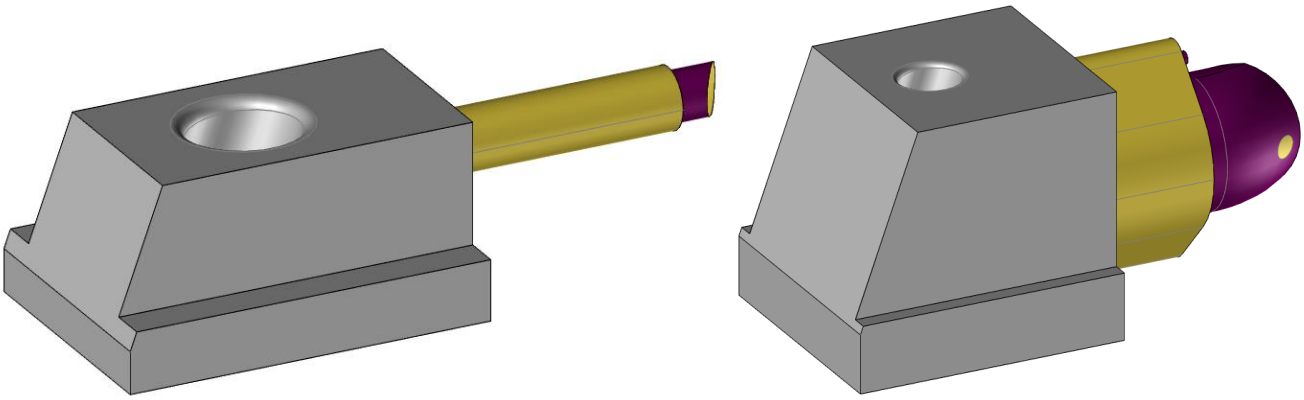


Because of the union between the insert and the slide, there is an extra part in the assembly. To avoid this, you can delete the insert as an operation.

- Hide slide 1.
- From the **Assembly** tab, select the **Removing** command and select the insert 1.

A removing operation is then added in the Operations tree.

- Repeat the procedure for the other two slides.



- Show all parts.
- **Save** the *Elbow mold* document.

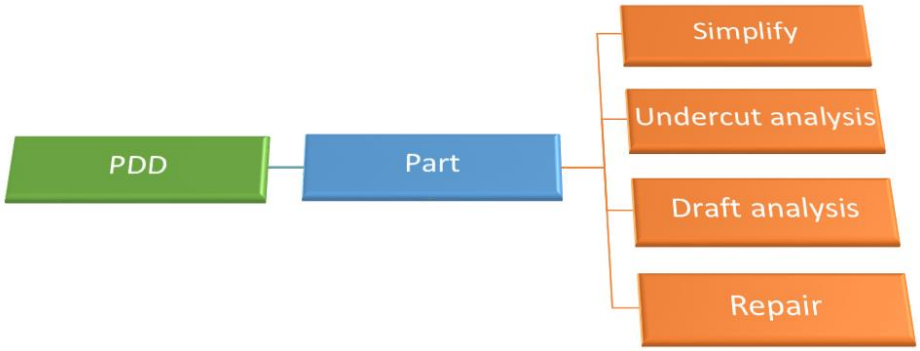
Part Replacement

The purpose of this exercise is to determine the best strategy for modifying your parts.

Overview of a mold's documents

In order to study the impact of the new part's modifications, it is recommended that you import the new PDD by using the following rules.

Functional overview for any import



Operating process

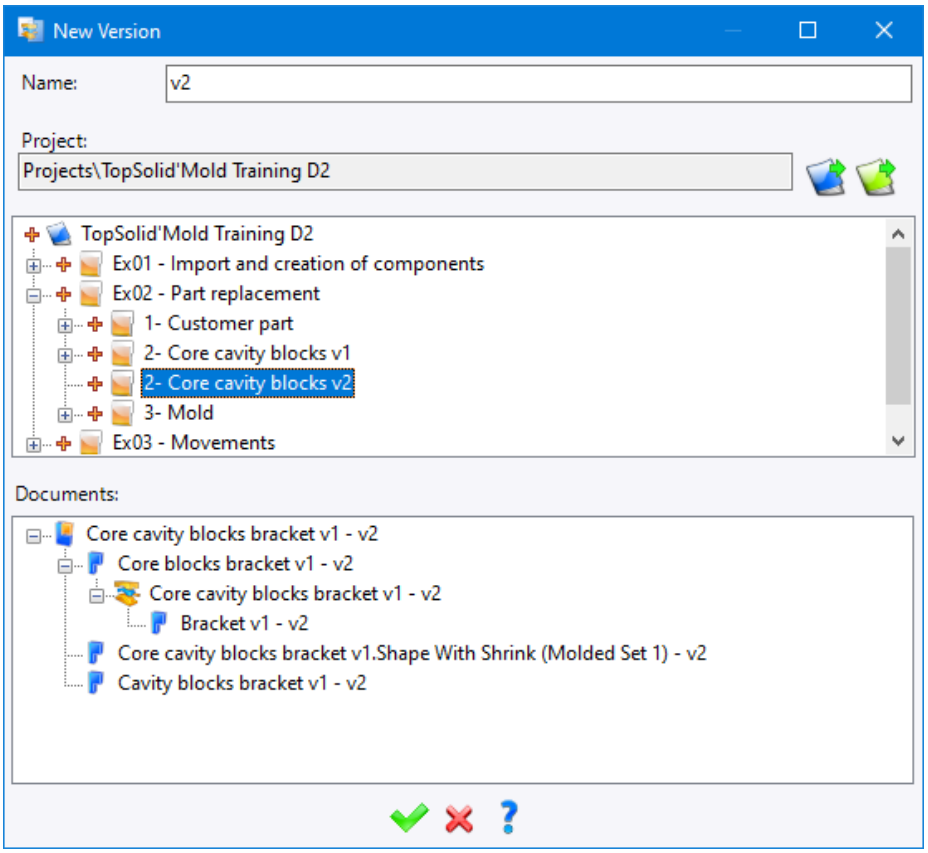
- From the *TopSolid'Mold Training D2* project, open the *Ex02 - Part replacement* folder.

Operating overview

We want to replace the *Bracket v1* part with its new version *Bracket v2*, and we want to work on a copy of the core and cavity block's assembly in order not to modify the mold immediately.
We will create a new version of the core and cavity blocks.

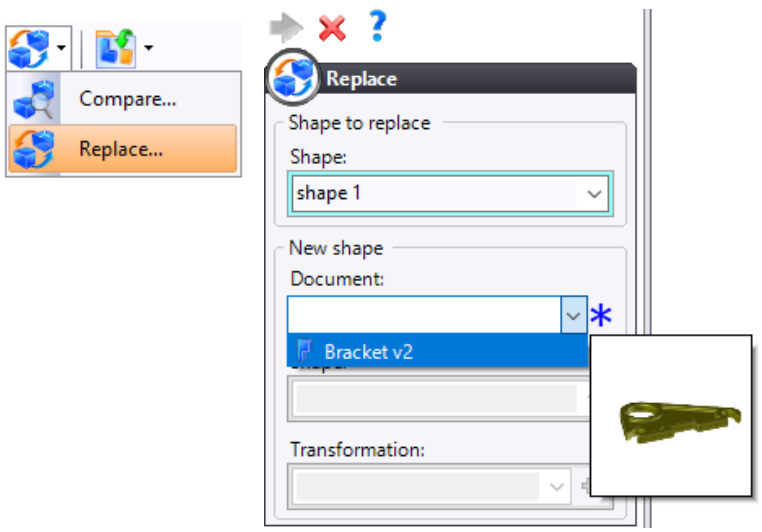
In the image below, the version 1 of the part is shown in blue.

- From the *2- Core cavity blocks v1* folder, open the *Core cavity blocks bracket v1* Split document.
- From the **Split** tab, select the **New version** command.
- Enter *v2* in the **Name** field.
- Create a subfolder named *2- Core cavity blocks v2* in the *Ex02 - Part replacement* folder.

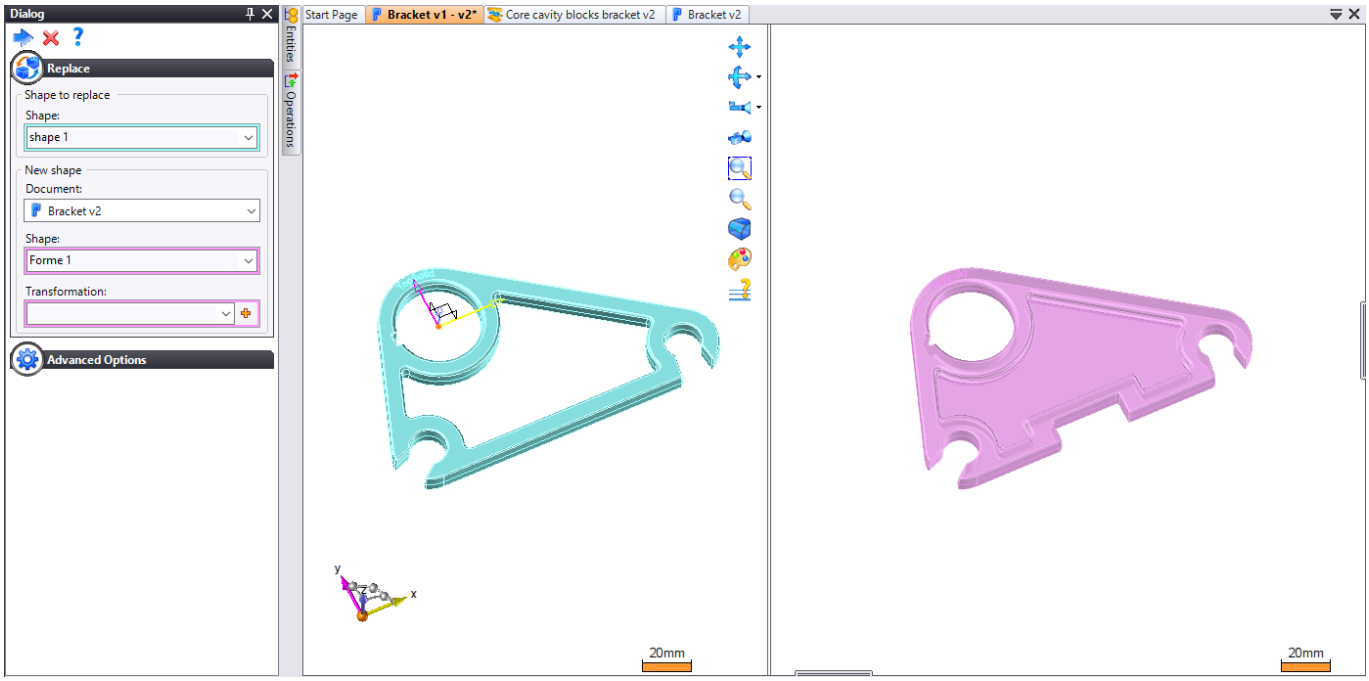


- **Confirm**.

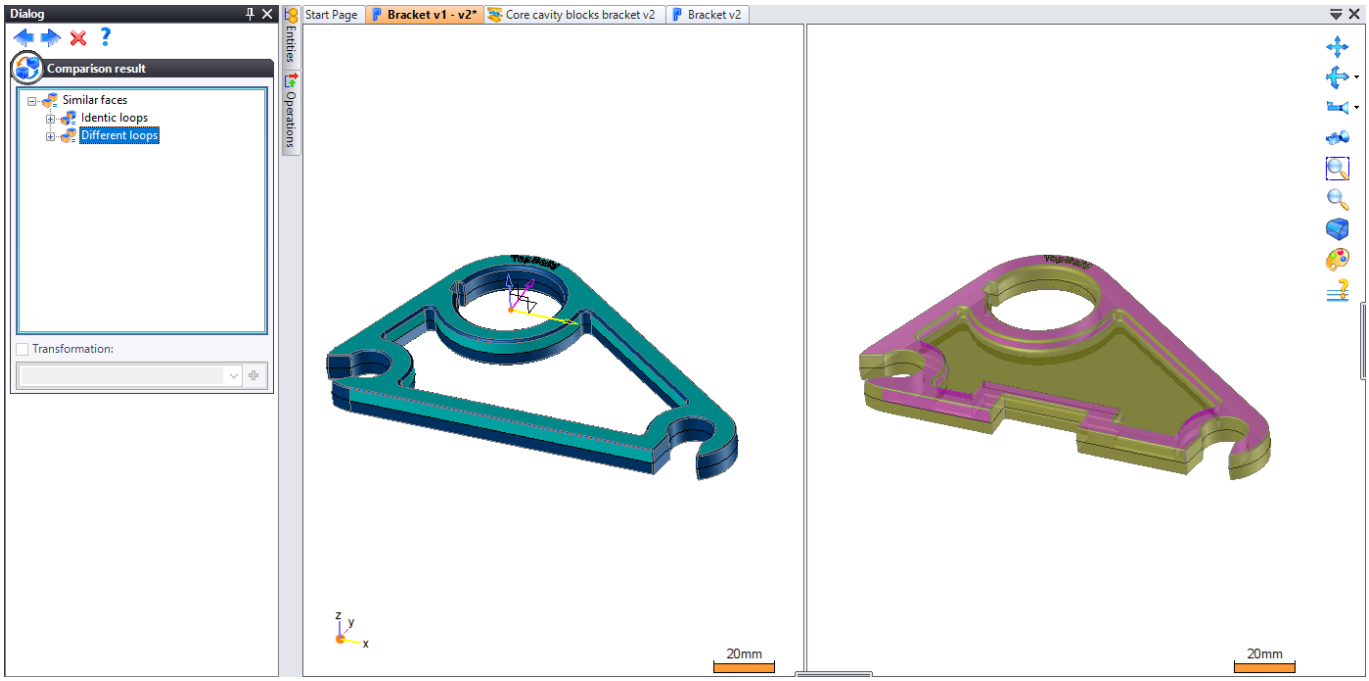
- Close the *Core cavity blocks bracket v1* Split document.
- Rename the Split and Assembly documents *Core cavity blocks bracket v2.*
- Open the *Core cavity blocks bracket v2* Split document.
- Open the *Bracket v2* part document from the *1- Customer part* folder.
- Return to the *Bracket v1 - v2* part document.
- From the **Shape** tab, select the **Replace** command.
- In the **New shape** field, select the **Bracket v2** document from the drop-down list.



The screen is then divided into two panes with a graphical synchronization of the views in both documents. Any zoom or rotation in a document will be applied in the other document.

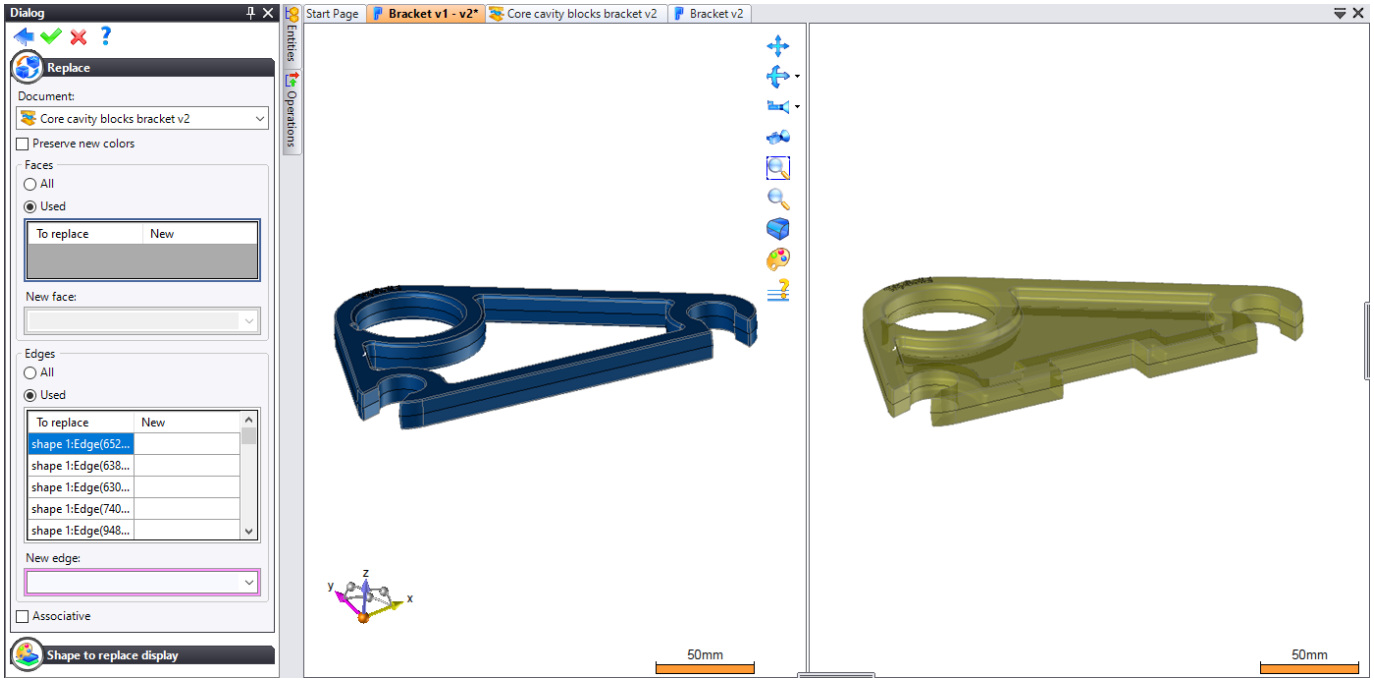


- Move to the next step.
- Click on **Different loops** to quickly identify the changes between both parts.



- Move to the next step.

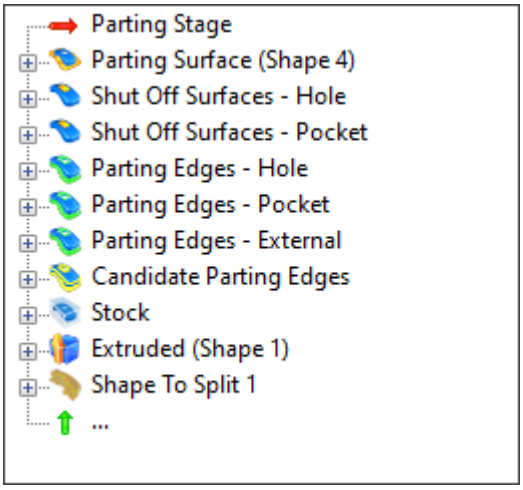
- In the **Document** field, select **Core cavity blocks bracket v2**.
- Select the **Used** option for the faces and edges.



- Click on ✔ to **confirm**, then close the *Bracket v2* document to avoid confusion.

The shape v1 is then replaced with the shape v2 in the *core cavity blocks bracket v2* Split document.

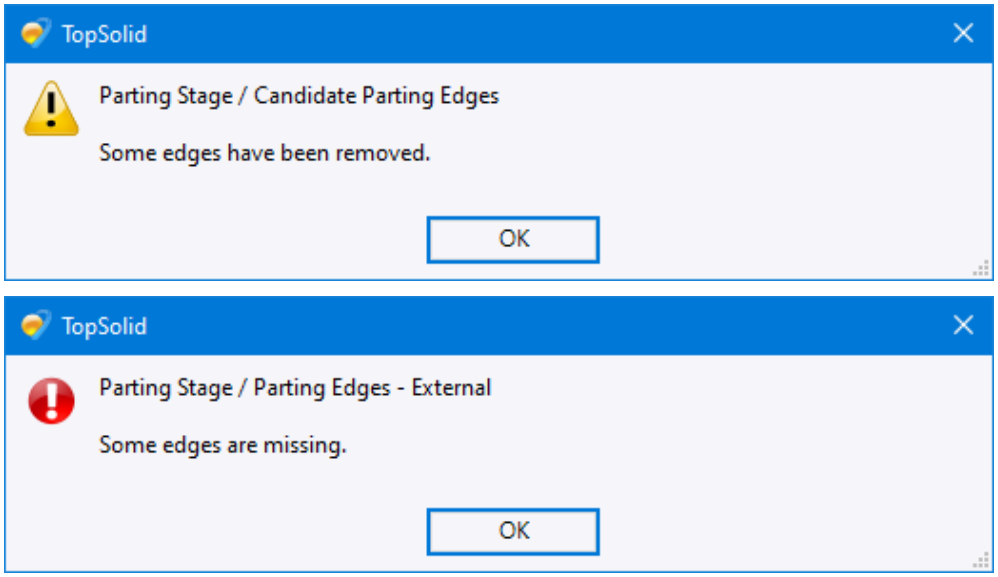
Operations performed in the first version of the core and cavity blocks:



- Open the *Core cavity blocks bracket v2* Split document.

When the document is open, the following error messages appear.

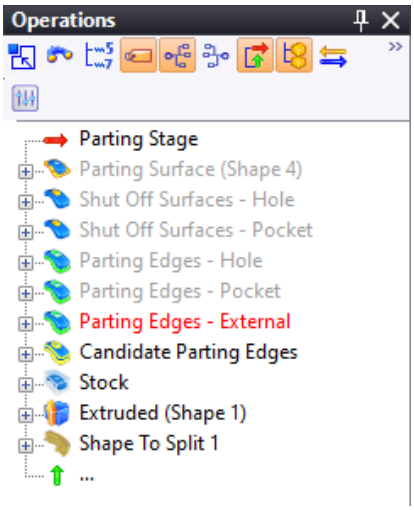
- Click on **OK** to confirm and move to the next error.



The document is then invalid.

Here is the state of the Operations tree in the parting stage:

- The candidate parting edges are no longer up to date.
- The external parting edge has been modified.
- The pocket has disappeared; the **Parting Edges-Pocket** and **Shut Off Surfaces-Pocket** operations are no longer necessary.



Repairs

- Switch to the **parting stage**.
- Move down the **Parting Stage** cursor over each operation to be modified in order to avoid the series of error messages, and move it up manually after each repair operation.
- Edit the **Candidate Parting Edges** operation to repair it.
- Edit the **Parting Edges-External** operation and select the new edges.
- Delete the **Parting Edges-Pocket** operation.
- Edit the **Parting Edges-Hole** operation and select the new edges.
- Delete the **Shut Off Surfaces-Pocket** operation.

At this stage, the parting shapes need to be recalculated.

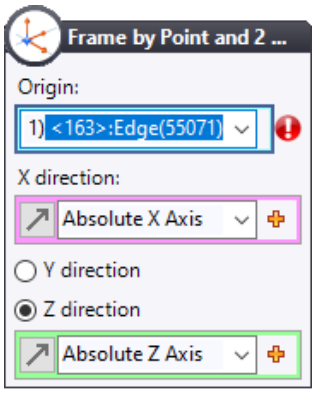
- Click on the icon in the document's tab.
- Update the parting shapes and the core and cavity blocks.
- From the core and cavity block creation dialog box, delete *v1* in the part names.
- **Save** and **close** the *Core cavity blocks bracket v2* Split document.

- Open the *Core cavity blocks bracket v2* assembly document.

The document is invalid.

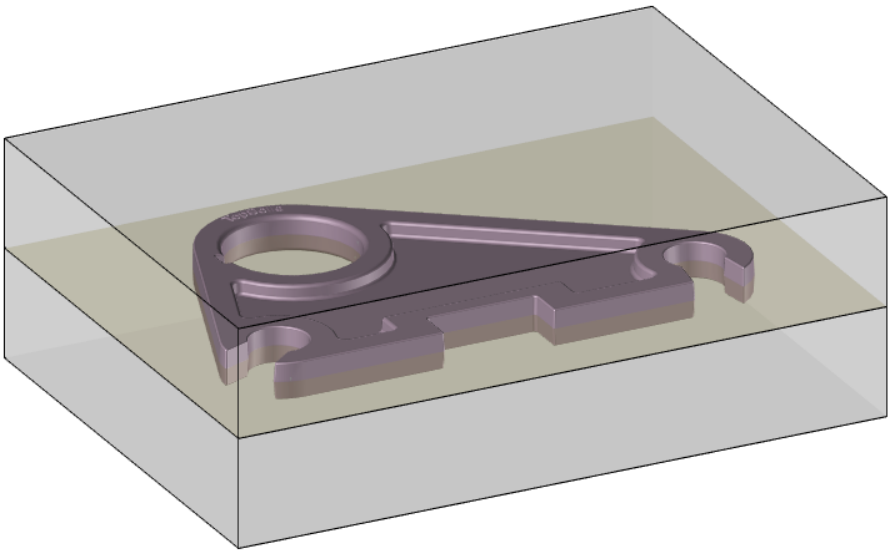
- Click on the icon in the document's tab.

The frame's origin point is invalid because of the changes.



- Click on the middle of the edge to repair.
- **Save** the *Core cavity blocks bracket v2* assembly document.

At this stage, you have defined the new core and cavity blocks version 2.



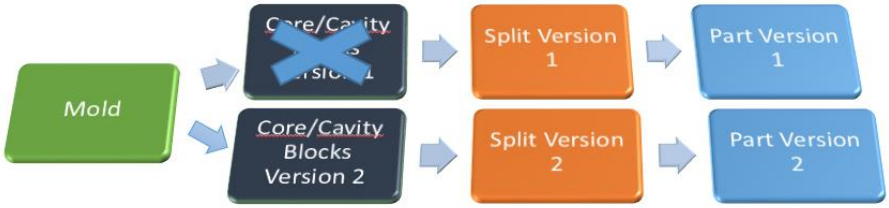
Replacing the new assembly in the mold

The mold document must be checked into the vault before any operation in order to return to a stable stage if problems arise.

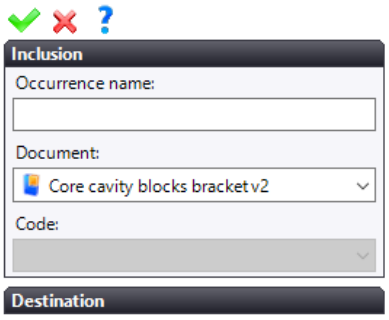
- From the *3- Mold* folder, open the *Bracket mold* document.

Note: At this stage, only the *Bracket mold* and *Core cavity blocks bracket v2* documents must be open.

We will now replace "Core cavity blocks version 1" by "Core cavity blocks version 2".



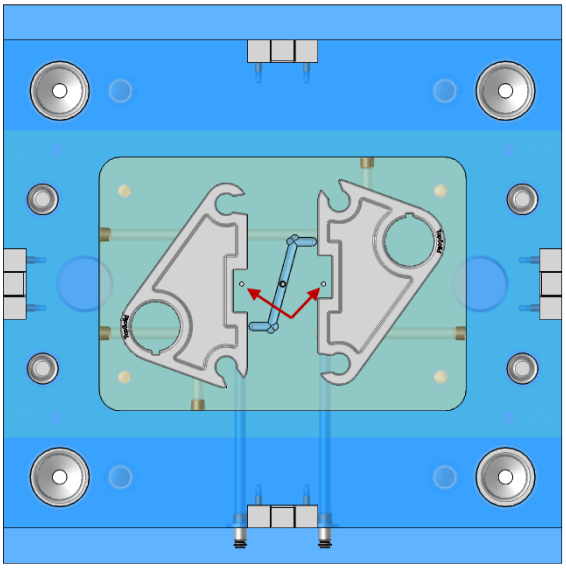
- From the Operations tree, open the node of the **Core Cavity Blocks Inclusions** and **Positioning 1** operations.
- Edit the inclusion, select the **Core cavity blocks bracket v2** document, then click on to **confirm**.



TopSolid then updates the mold.

- Hide the **A side** by clicking on the icon in the graphics area.

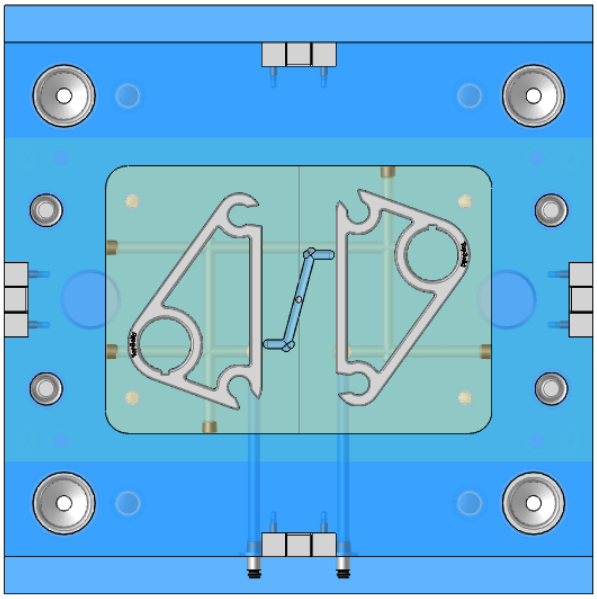
You will notice that two pins fall off after the part modification.

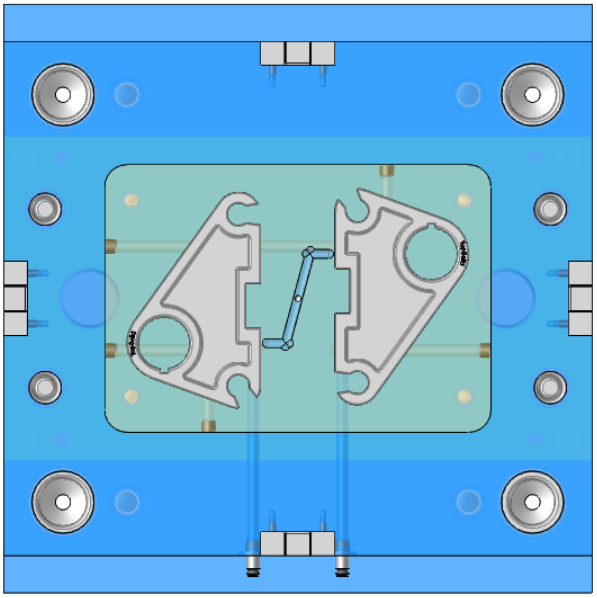


- From the Operations tree, open the **Pins** folder and delete pins 4 and 9.
- **Check** the *Bracket mold* document into the vault.

At this point, you have:

- replaced your part by its new version;
- kept track of your file before this modification.

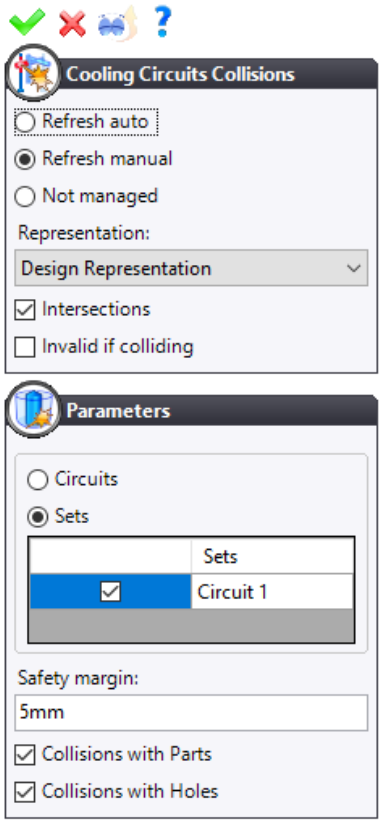


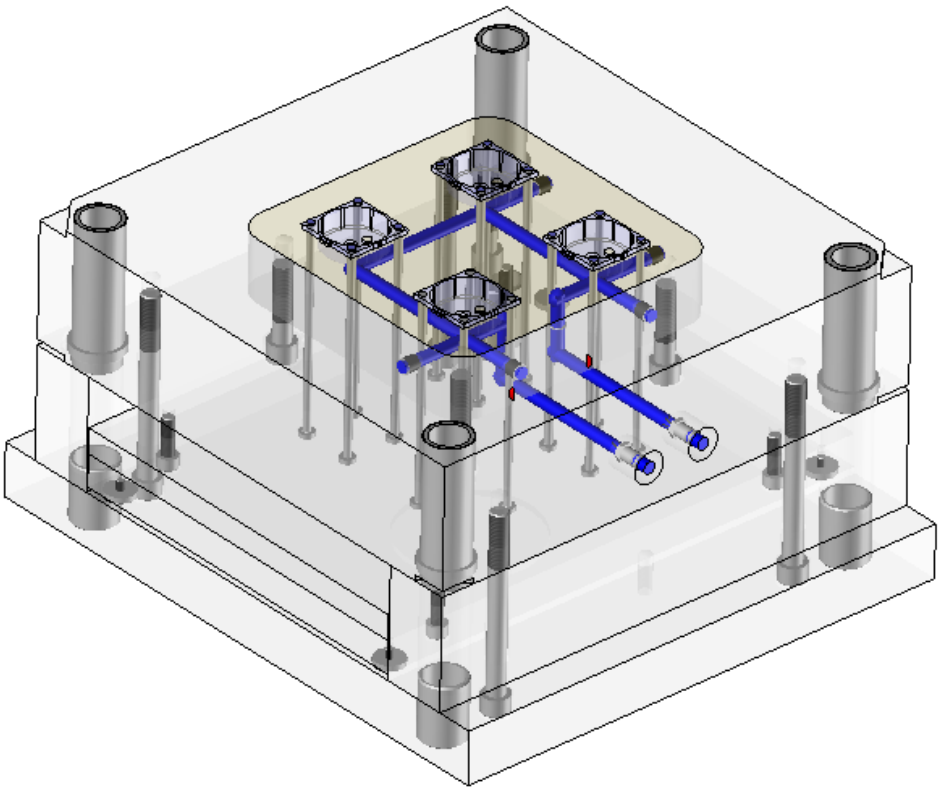


Cooling Circuit Control

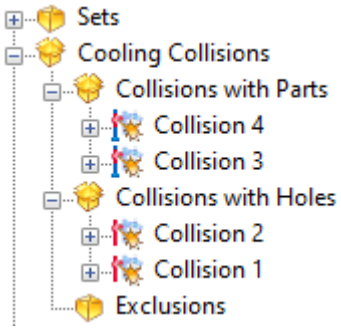
In this exercise, we will see how to control collisions between the cooling circuit and its surrounding elements: holes, pins, etc.

- Import the *TopSolid'Mold Training D3.TopPkg* project.
- From the *Ex01 - Cooling circuit control* folder, open the *Electrical outlet plate* mold document.
- Hide the **A side** by clicking on the icon in the graphics area.
- From the **Mold** tab, select the **Cooling Circuit Collisions** command.
- Adjust the **safety margin** to *5mm* and check the **Collisions with Parts** and **Collisions with Holes** boxes.

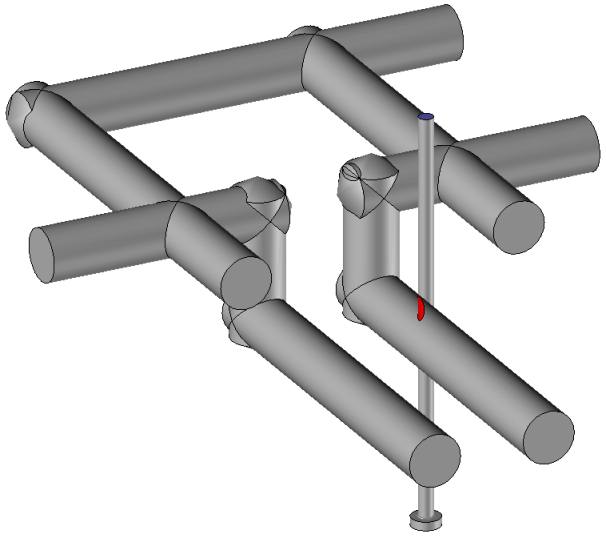




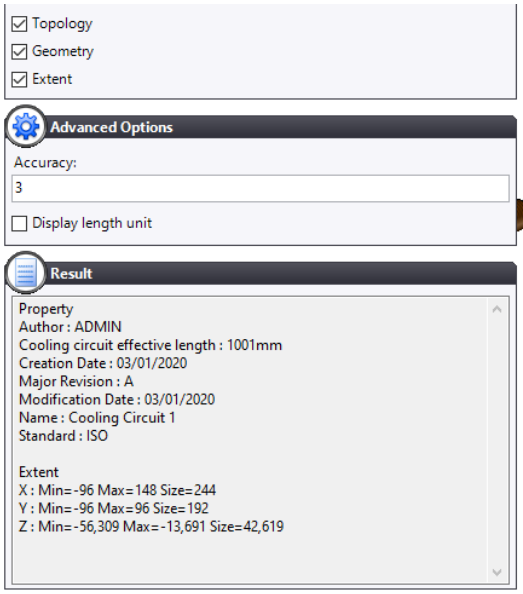
TopSolid then creates two groups of collisions in the Entities tree's **Cooling Collisions** folder.

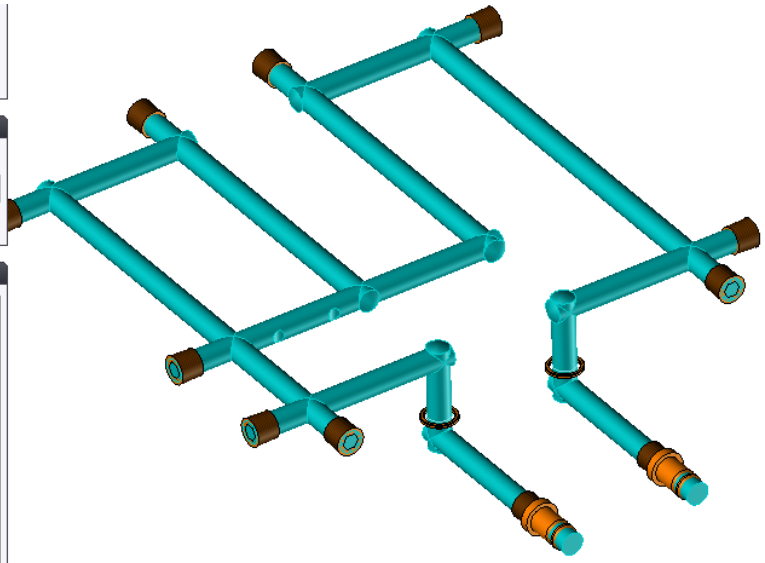


- Open the **Collisions with Parts** folder, then right-click on the first collision and select the **Show Alone** command.



- You can exclude the collisions that you deem acceptable. To do this, right-click on the collision and select the **Exclude** command. The excluded collision then moves to the **Exclusions** folder.
- To find out the effective length of the circuit, press the **F8** key, position the cursor on the circuit and press the **Shift** key.

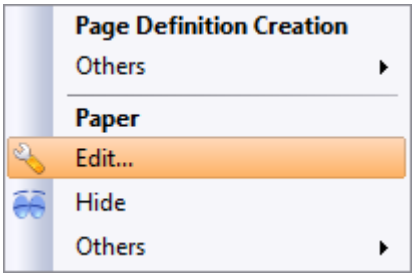




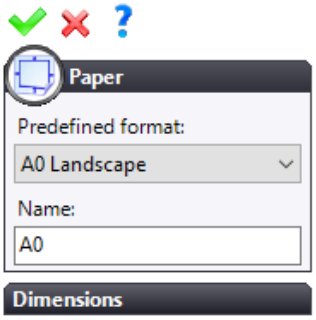
Manual Drafting Bundle

In this exercise, we will see how to create a single document that groups together all the drawings of your study.

- Open the *Ex02 - Drafting bundle* folder, and then the *Drawings* folder.
- Create a new **Drafting Bundle** document (**Advanced** tab) using a **blank template**.
- Exit the **Inclusion** command since no drafting document is open.
- Right-click on the frame shown on the screen and select the **Edit** command.

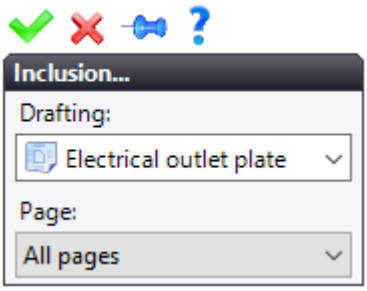


- Select the **A0 Landscape** format.



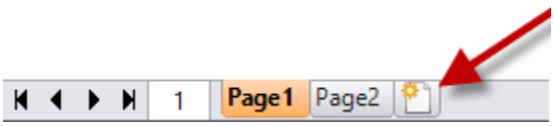
Creating pages

- From the *Drawings* folder, drag and drop the *Electrical outlet plate* drafting document into the drafting bundle document.
- For the inclusion, select **All pages**.

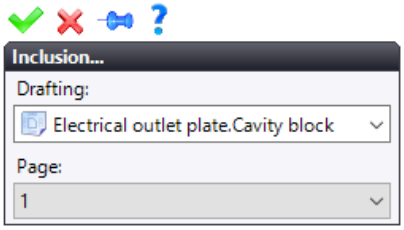


TopSolid automatically creates two pages.

- Click on the **Add page** icon at the bottom left of the graphics area.

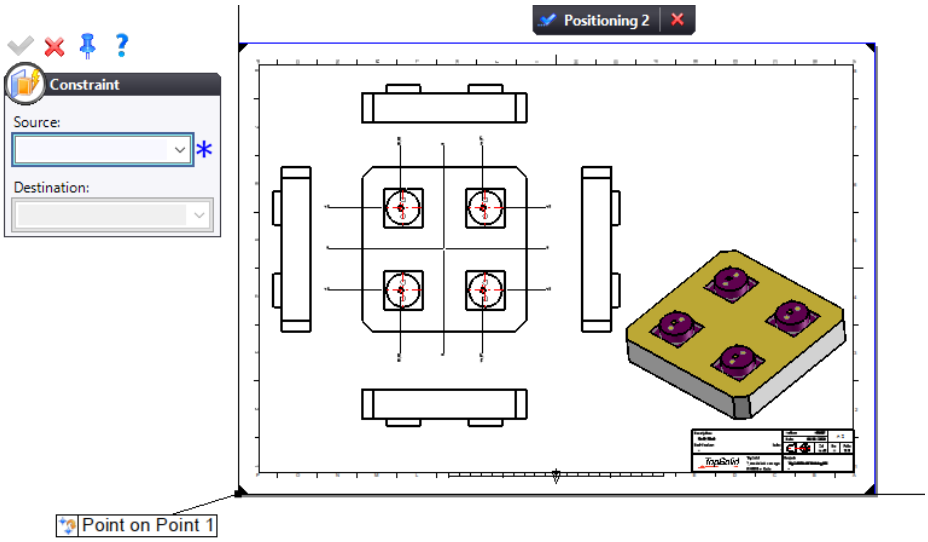


- From the *Drawings* folder, drag and drop the *Electrical outlet plate.Cavity block* drafting document into the drafting bundle document.
- Leave the **1 page** mode.

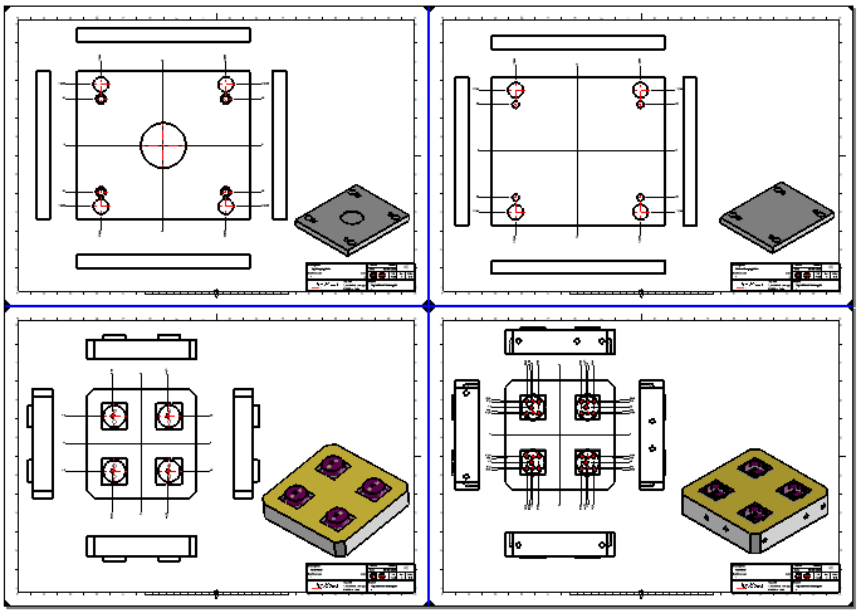


The drafting bundle behaves like an assembly document with a reduced set of commands: **Point on Point**, **Axis on Axis**, **Axis on Point**, **Point on Axis** and **Orientation**.

- Click on the bottom left point of the *Electrical outlet plate.Cavity block* document, then click on the bottom left point of the drafting bundle document.



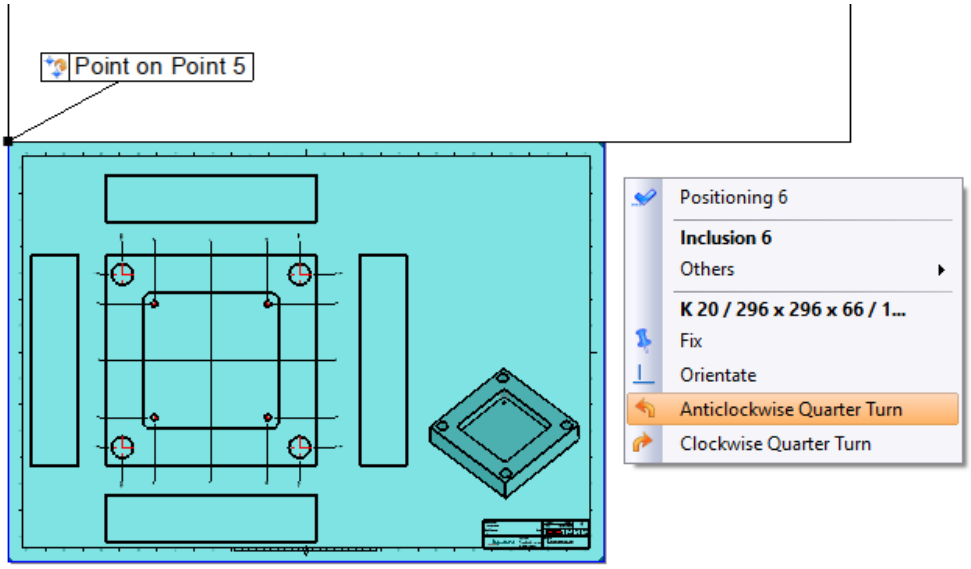
- Repeat the procedure with the other drawings, making sure to exit the constraint command each time you include a drawing.
- Drag and drop the drawings until the frame is filled as shown below.



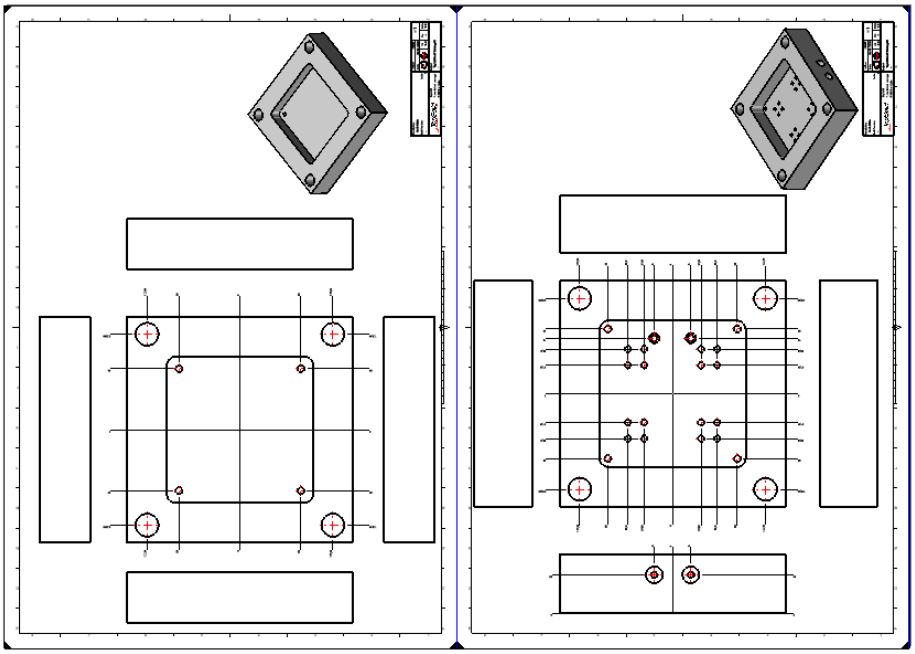
- Add a new page.

At this stage we will see how to rotate a drawing.

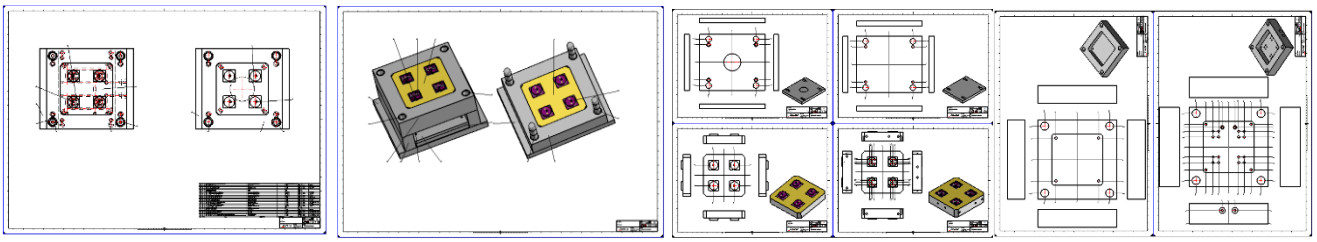
- Drag and drop the following drawing.
- Click on the top left point of the drawing and move it to the bottom left point of the drafting bundle document.
- Exit the constraint command.
- Right-click on the drawing and select the **Anticlockwise Quarter Turn** command.



- Repeat the procedure for the following drawing.

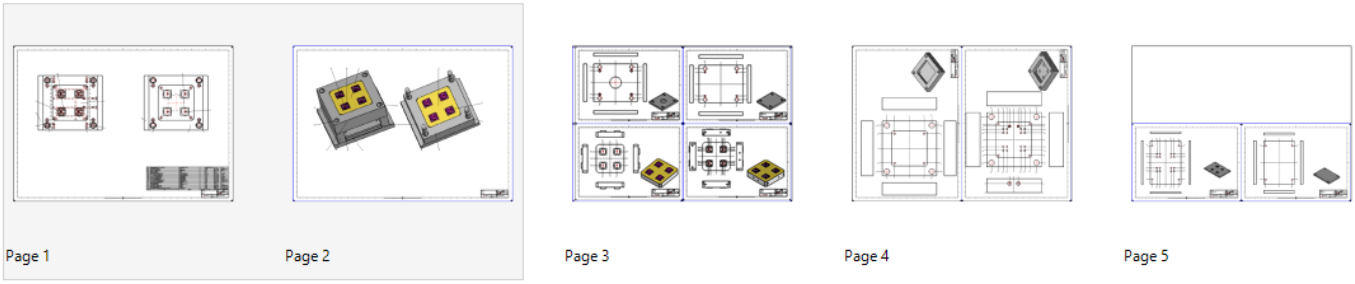


You should end up with the following result.



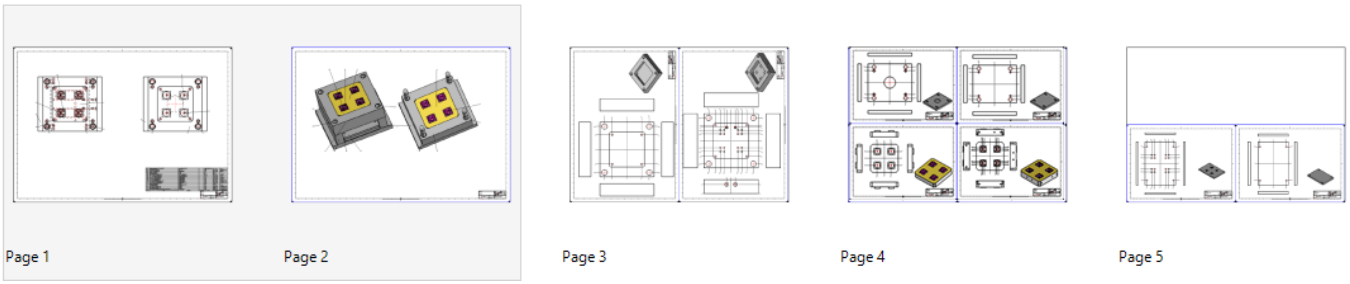
Modifying the layout

- Click on the **Page Sorter** icon at the bottom right of the drafting bundle document to access the document's layout.

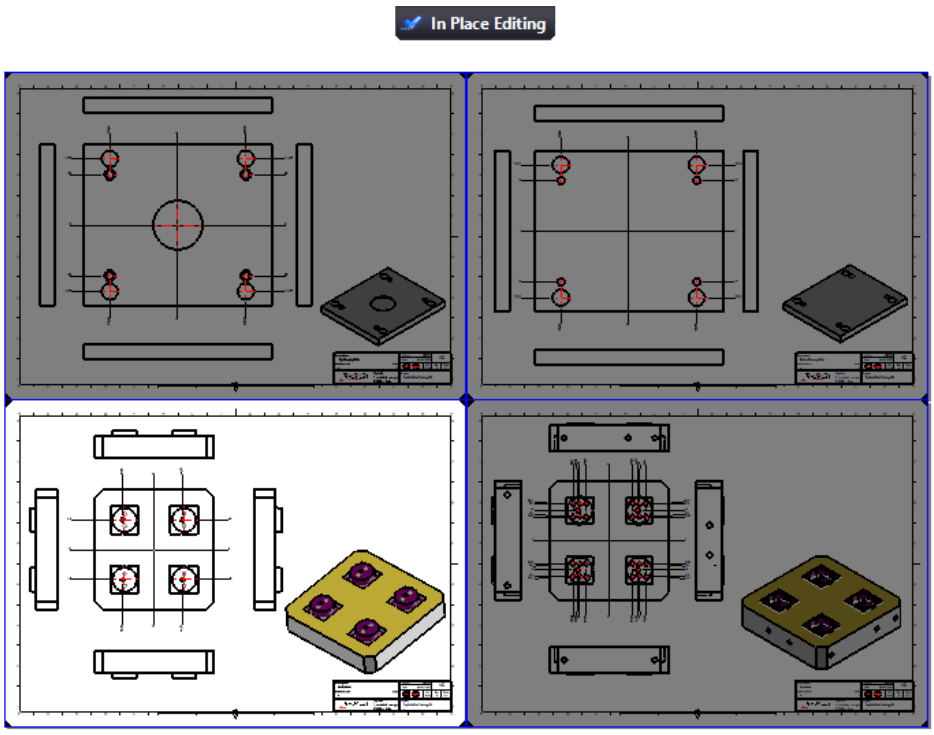


The frame around page 1 and page 2 means that these pages are linked.

- If you want to switch the page 4 to the page 3, simply drag and drop the page 4 onto the page 3.



- Click on the **Page Sorter** icon again to return to page mode.
- If you want to edit a drawing, you only have to double-click on it.
  The **In Place Editing** context of your drawing is then enabled and all the drafting commands are available.



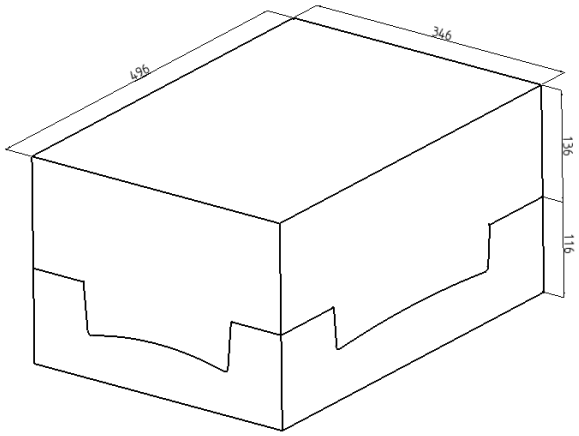
- Once changes have been made, **confirm** the in-place editing to return to the drafting bundle document.
- **Save** the drafting bundle document.

Core and Cavity Blocks from a Block

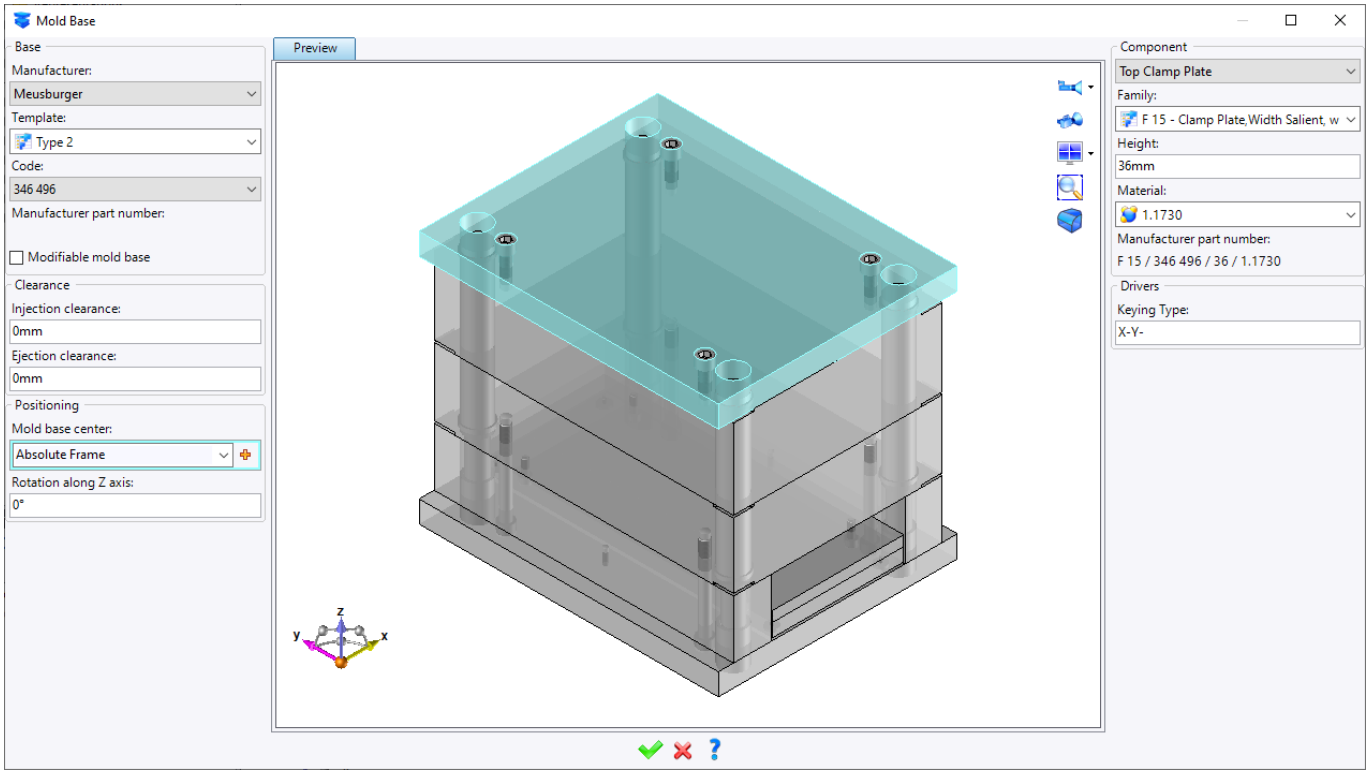
In this exercise, we will see how to design a mold with core and cavity blocks made from a block.

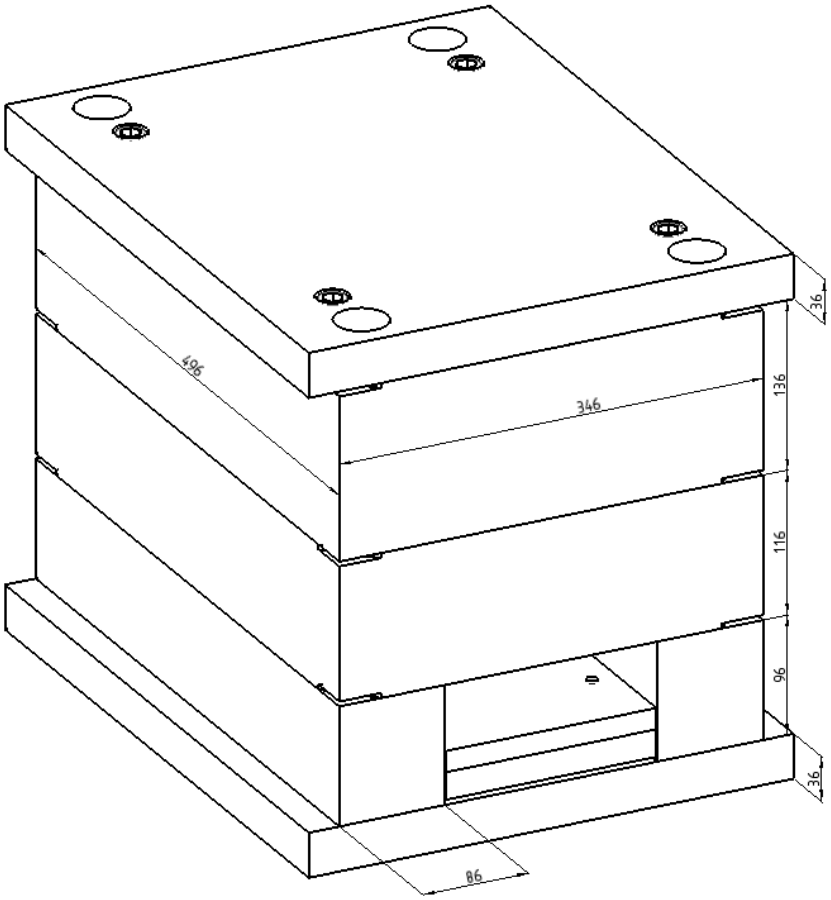
- Open the *Ex03 - Core cavity blocks* folder.

The stock of your Split document must be sized according to the dimensions you want to apply to your mold's standard plates (here, 346 x 496).

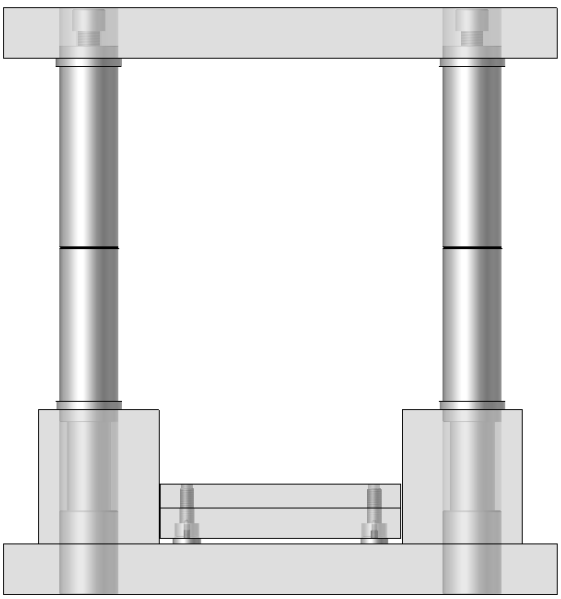


- Create a new **Mold** document using a **blank template**.
- Include a 346 x 496 **Meusburger** mold base, select **Type 2** as the template and adjust the following dimensions for the plates.



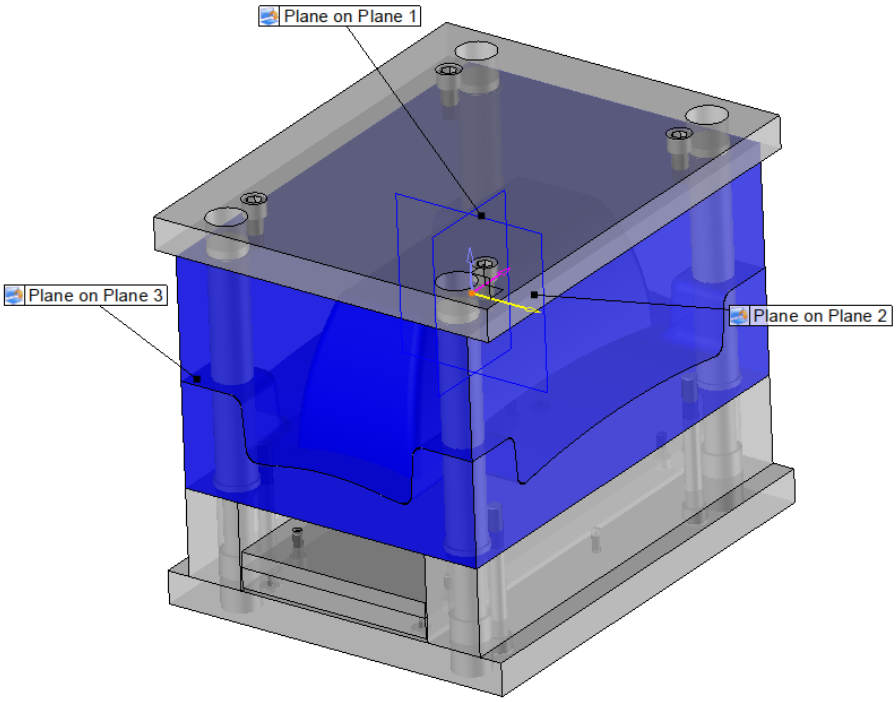


- From the **Assembly** tab, select the **Removing** command and select the two core and cavity blocks.



- From the *Split* folder, drag and drop the *Flower pot* assembly document into the mold document.

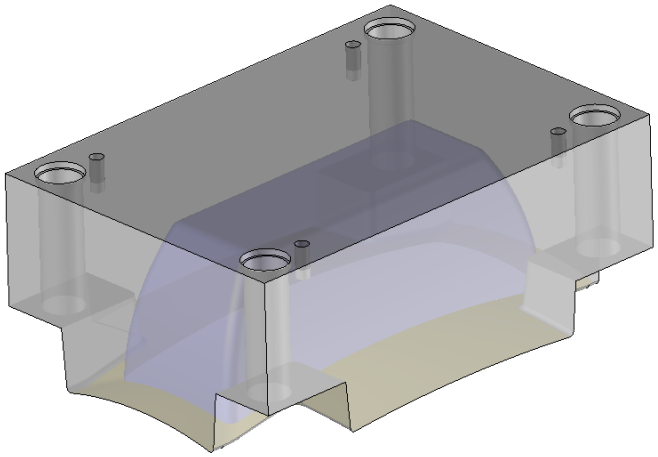
- Assemble the core and cavity blocks using the appropriate positioning constraints.



Executing the processes

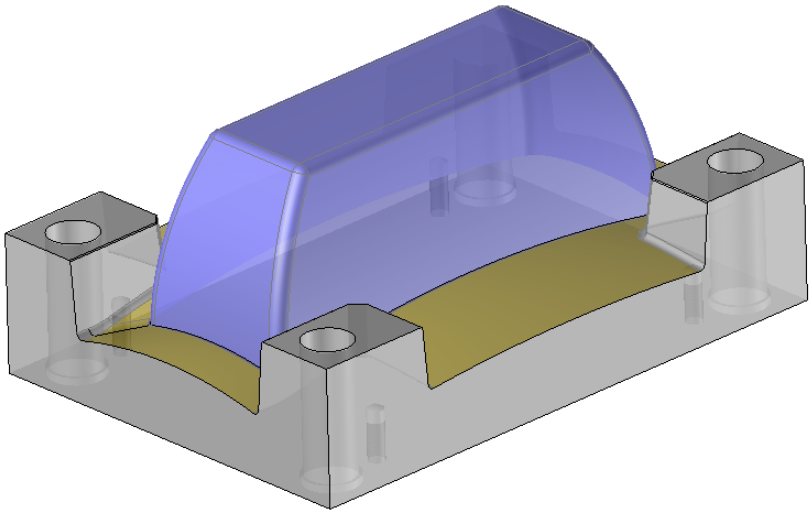
- Hide the **B side**.
- Right-click on the first fixing screw of the clamp plate and select the **Use Process** command.
- Repeat the operation for the four screws, and then for the four guide pins.

You should end up with the following result for the cavity block (A side).



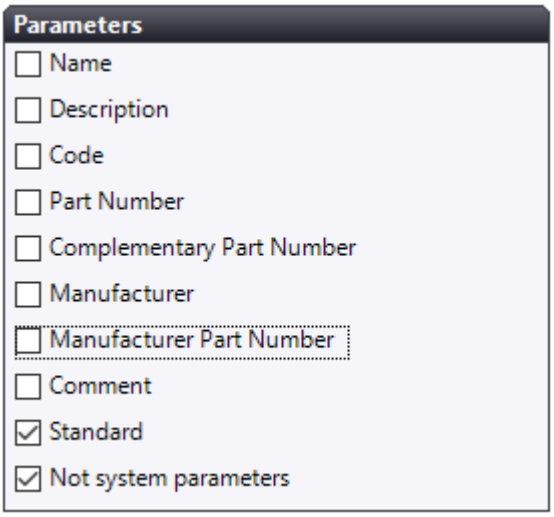
- Hide the **A side**.
- Show the **B side**.
- Right-click on the first fixing screw of the clamp plate and select the **Use Process** command.
- Repeat the operation for the four screws, and then for the four guide bushes.

You should end up with the following result for the core block (B side):

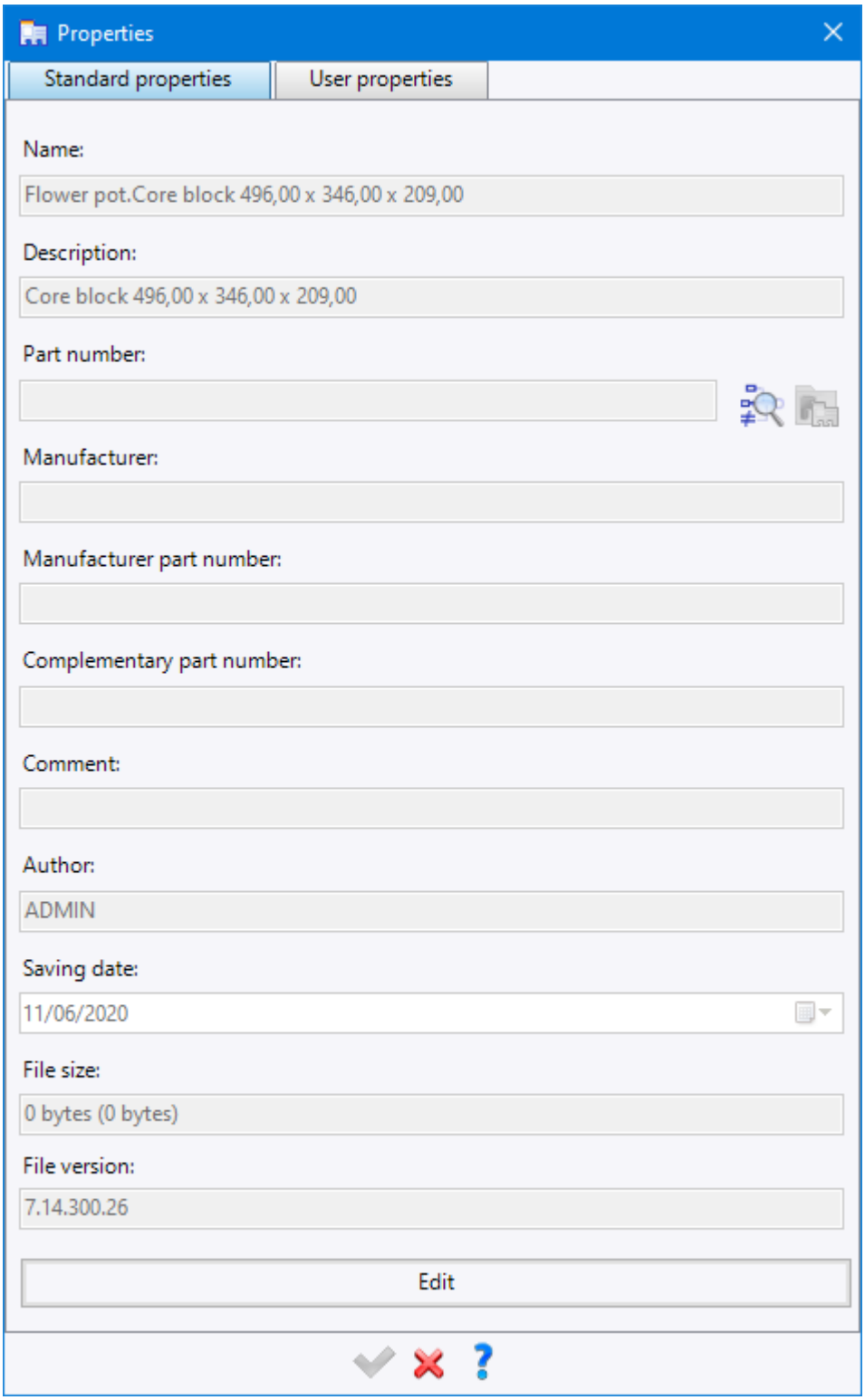


Modifying the properties

- Open the core block (B side).
- Right-click on the shape and **edit** the derivation.
- Uncheck the **Name**, **Description**, **Code**, **Manufacturer** and **Manufacturer Part Number** parameters.



- Enter the following properties.



- Repeat these steps for the cavity block (A side).
- **Save** the document.

Notes