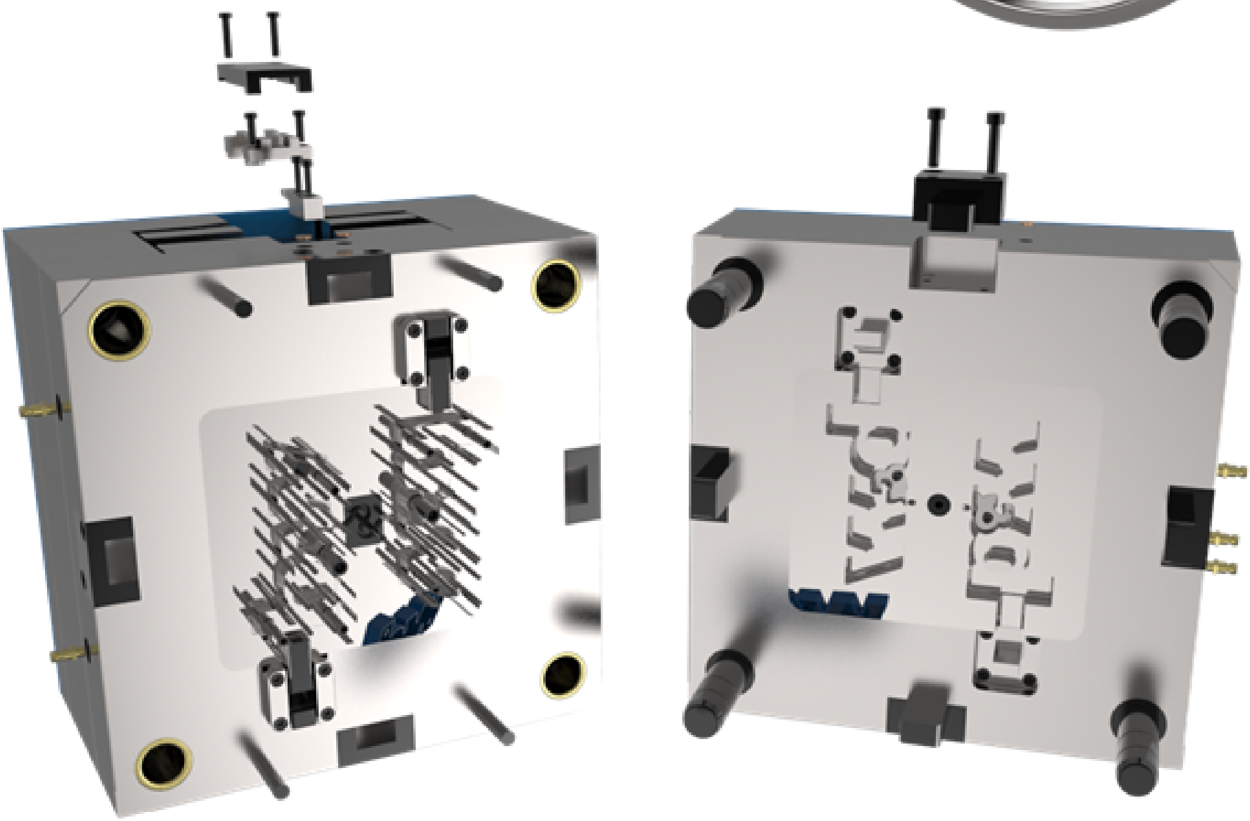
Expert Moldbase Extension 12.0

Tutorial

01.09.2019



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About Expert Moldbase Extension

1.1 About Creo Expert Moldbase Extension

Use Creo Expert Moldbase Extension (EMX) to create a mold base as a regular assembly.

1.2 About the EMX Ribbon

1.2.1 Commands

The Creo Expert Moldbase Extension commands are accessible through a the **EMX** ribbon. The content of the Creo Expert Moldbase Extension ribbon is different depending on the type of the current model and it's related modes in Creo Parametric.

1.2.2 Assembly Mode

An Creo Expert Moldbase Extension mold base project always starts with an assembly. Therefore, the Assembly Mode is the most present mode.

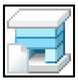
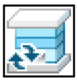
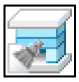
The **EMX Assembly** tab contains the following groups.

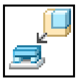
- **Preparation**:

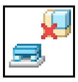
  The  **Classify** dialog box for setting up the different model types for assembly components.

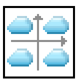
  The  **Identify trim faces** dialog box for selecting and designating quilts as trim surfaces for ejector pins and lifters.

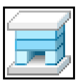
  The  **Duplicate Cavity** dialog box to create duplicated cavity insert assemblies
- **Project**:

  The user can  **Create New**,  **Modify** and  **finish** projects.
- **Structure**:

  Subassemblies can be added with  **Add Subassemblies** to the project.

  Existing subassemblies can be deleted with  **Delete Subassemblies**.
- **Mold Base**:

  The  **Cavity Layout** can be managed.

  In the  **Assembly Definition** dialog box the general design of the mold base can be defined.

In the **Cavity Cutouts** dialog box pockets can be created within the cavity paltes.

The **Component Status** allows the user to decide which type of components models should be are assembled or just predefined.

In the Group overflow you will the **Define Main Axis Offset** feature for off-centered mold bases.

- **View**:

  The predefined simplified representations **Injection Side (IS)** or **Ejector Side (ES)** can be toggled in this section.

  Components can be added to one of these predefined views via **Add Components To**.

- **Documentation**:

  The **Bill of Materials** can be managed in this group.

  You can **Edit BOM Parameters** of selected parts.

  You can **Open drawing of current model** or **Open drawing of selected model**

- **EMX Tools**:

  This is the group for a variety of handy tools.

  - **Calculating Model Size**
  - **Set Technology Colors**
  - **Set Accuracy**
  - **Export Csys Position**
  - **Calculate Clamping Force**

- **Administrator Tools**:

  Administrators and users can configure Creo Expert Moldbase Extension with the **EMX Options** dialog box.

  The **Component Editor** and the **Mold Base Editor** can be found in this group.

Add Drawing Formats, **Verify unused Models** and **Replace Parameters** in existing Creo Expert Moldbase Extension assemblies can be started from here.

- **Help**:

  Start the Creo Expert Moldbase Extension Help Center with **Help** and check your current release version with the **Info About** dialog.

The **EMX Components** tab contains:

- **Mold Base**:

  Additional access to the **Assembly Definition** and the **Component Status** .

- **Cooling Components**:

  Create **Cooling Components**.

  Start the **Waterline Designers**.

  An extracted model of all cooling circuits can be created with **Show Waterline**.

  Cooling diameters can be changed either by **Change diameter of all cooling holes** or **Change diameter of selected cooling holes**.

  With **Delete entire waterline** all cooling components will be deleted from the project assembly.

- **Ejector Pin**:

  Create **Ejector Pins**.

  Start the **Ejector Pin Designer**.

- **Components**

  Create Creo Expert Moldbase Extension Components like:

  - **Screw**
  - **Guide Component**
  - **Equipment**
  - **Stop System**
  - **Dowel Pins**
  - **Support Pillar**
  - **Slider**
  - **Latch Lock**
  - **Lifter**

- **Library**:

  Create **Library Components**.

- **Component Handling**:

  Modify, **Remove**, **Assemble Again** and **Assemble as Copy** of all kinds of Creo Expert Moldbase Extension components.

  Additionally **Assemble predefined components** from this group

1.2.3 Manufacturing Mode

The cavity insert assembly is commonly a Manufacturing model. Therefore some Creo Expert Moldbase Extension commands are available in the ribbon for the Manufacturing Mode. During the preparation of the cavity insert assembly the **Classify Dialog** and the **Duplicate Cavity Dialog** will be needed. It is recommended to select trim surfaces with **Identify trim faces** in manufacturing model.

The **EMX Manufacturing Mode** contains:

- **Preparation**:

  The **Classify** dialog box.

  The **Identify trim faces** dialog box.

  The **Duplicate Cavity** dialog box.

- **Project**:

  Create New projects.

1.2.4 Drawing Mode

The Drawing Mode in Creo Expert Moldbase Extension contains a couple of handy features for editing of the mold base drawing.

The **EMX Drawing Mode** contains:

- **Documentation**:

  The **Bill of Materials** can be used to place balloons to the drawing.

- **Ejector Pin**:

  Add Drawing Symbols for ejector pins to the drawing.

1.2.5 Part Mode

In Part Mode it is possible to predefine all different types of Creo Expert Moldbase Extensioncomponents.

The **EMX Part Mode** tab contains:

- **Preparation**:

  The **Identify trim faces** dialog box.
- **Components**:

  All kinds of components are available.
- **Library**:

  Library Components are available.
- **Component Handling**:

  Modify, **Remove**, **Assemble Again** and **Assemble as Copy** of all kinds of Creo Expert Moldbase Extension components.
- **EMX Tools**:

  Function like **Calculating Model Size** can be found.
- **Administrator Tools**:

  EMX Options and the **Component Editor** are accessible in part mode.
- **Help**:

  Start the Creo Expert Moldbase Extension Help Center with **Help** and check your current release version with the **Info About** dialog.

1.3 About the Principal Workflow

Starting from an existing cavity insert assembly the principal Creo Expert Moldbase Extension workflow contains the following steps:

1. Prepare the cavity insert assembly in Creo Parametric manufacturing:
   a. To define the functions of the components for the cavity insert assembly, click **EMX Manufacturing Mode ▸ Classify**. The **Classify** dialog box opens.
   b. If multiple cavity copies are required for a mold base project click **EMX Manufacturing Mode ▸ Duplicate Cavity**. The **Create Insert Copies** dialog box opens from which you can create dependent copies of the assembly.

c. If ejector pins and lifters need to be trimmed against quilt surfaces, identify trim faces within the reference model by clicking **EMX Manufacturing Mode** ► **Identify trim faces**.

NOTE:
If no cut quilt is selected as a trim surface, ejector pins are trimmed against the solid of the reference model. Lifters cannot be trimmed.

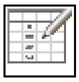
2. Create the new project:

   a. Click **EMX Manufacturing Mode** ► **Create New**. The **Project** dialog box opens.

   b. Add project information as required and click **OK** to close the dialog box and create the new project.

3. Set up the assembly structure: Click **EMX Assembly** ► **Add Subassemblies**. The **Subassembly** dialog box opens.
4. Define the layout of the cavity insert assembly pattern and assemble the cavity insert assemblies: Click **EMX Assembly** ► **Cavity Layout**. The **Cavity Layout** dialog box opens.
5. Create plates, guide components, and major equipment: Click **EMX Assembly** ► **Assembly Definition** to open the **Mold Base Definition** dialog box. Set various options in this dialog box and then click **OK** after creating the required components.
6. Load an existing mold base assembly, add, modify or delete plates, locate rings, sprue bushings, guide components, and standard screws: In the **Mold Base Definition** dialog box, click . You can also define machine data. Click **OK** after defining the required components.
7. Define the layout of the cavity insert assembly pattern and to assemble the cavity insert assemblies: Click **EMX Assembly** ► **Cavity Cutouts**. The **Cavity Cutouts** dialog box opens.
8. See the status of the component: Click **EMX** ► **Component Status** to open the **Component Status** dialog box.
9. Add screws, ejector pins, cooling system, sliders, and so on: Click **EMX Components** and then in the **Components** group, select a component. Click **Preview** to check component properties before assembly.
10. Use an editable library component: Click **EMX Components** ► **Library** ► **Assemble**. The **Library Component** dialog box opens.

11. Analyze the model and finish the design: Click **EMX Assembly** and then in the **EMX Tools** group, select a tool to use for the analysis.

12. Generate a Bill of Materials: Click **EMX Assembly** ►  **Bill of Materials**. The **Bill of Materials** dialog box opens.

2

Tutorial Introduction

Introduction
Conventions

2.1 Introduction

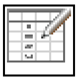
This Expert Moldbase Extension 12 [EMX] tutorial contains step by step explanations for all major features. It is recommended to go through this tutorial chapter by chapter. Many EMX dialog boxes and functions are very similar for different mold base components. So they are explained only once (the first time this dialog box is used during the mold base design). In later chapters the author assumes you have this background information already. If you "jump" within the chapters you might not understand everything.

2.1.1 The releases used in this tutorial are:

- PTC Creo Paramteric 6.0.1.0
- Expert Moldbase Extension 12.0.0.2

2.2 Conventions

2.2.1 Conventions

`MEASURE_DISTANCE`	The courier font indicates a configuration option, parameter or a path-/filename
Bold letters	Indicate elements of the EMX GUI and Ribbon
	An icon image represents a button in the Creo Parametric Ribbon or in an EMX dialog
LMB	Press the left mouse button
MMB	Press the middle mouse button
RMB	Press the right mouse button

2.2.2 Before you start

Required settings in the config.pro

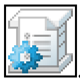
Please check if your config.pro file contains the following entries that are necessary for the correct use of EMX.

default_ext_ref_scope	ALL (to allow external references)
set_trail_single_step	NO (to make sure macros/mapkeys used within EMX can be executed without interaction of the user)
native_kbd_macros	NO (EMX macros run independent from any LANG setting)

open_simplified_rep_by_default	NO (To avoid the SimpRep-dialog box opening when retrieving a drawing)
show_dim_sign	NO (EMX will handle positive and negative dimensions like offsets internally)

2.2.3 EMX Options

To make sure that your EMX 12.0 installation will work according to this tutorial you have to set EMX configuration options.

Open the EMX Configuration options with **EMX Assembly ► EMX Options** .

Set the following options while working through the examples of this tutorial

NR_DIGITS_COMMON	5 (sets the digits for all dimensions within EMX models and components) When working with inch catalogs, value must be 5 minimum. Otherwise the inch-dimensions cannot be set properly.
NR_DIGITS_BOM	5 (Used for digits in parameter values (i.e. SIZE) in the EMX BOM only)
SAVE_DRAWINGS	YES (Creates drawings during component definition process automatically)
COMPANY_ID	- (Changing this setting will make EMX behave customer specific)

Notes:

3

The Preparation of the Cavity Insert Assembly

3.1 About the Cavity Insert Assembly as requirement for Moldbase Projects

To start working with Creo Expert Moldbase Extension an finished cavity insert assembly is required.

To create this cavity assembly a Creo Mold Design and Cast option is required.

3.2 Before Starting Your Design

Download 00_start to start from with this chapter.

3.2.1 Initial work on the cavity insert assembly

Before starting a new mold base project, some initial work on the needs to be done as described below:

- Understand the received data

  If the cavity insert assembly is resulting from Creo Mold Design Extension, make sure all required files are in the working directory. If it is an IGES or STEP file, import it into Creo and convert it into a solid model(s).

- Cavity layout planning

  You should have an idea on how many cavities are going to be used? How will the cavity layout look like? What is the overall size of the cavity?

- Type of mold base

  You should have an idea on what type of mold base is going to be used, two-plate mold or three-plate mold? Is it a hot runner mold? What is mold opening distance is required?

- Runner planning

  Start planning your runner system in advance.

- Ejector Pin planning

  Plan the ejector pin location, use Datum Points to indicate the ejector pin position in the mold reference part. You should also take down the ejection distance required for this design part.

- Slider planning

  Is one or multiple slider(s) required for this project? In what locations? What is the slider opening distance? Place a Datum Coordinate System on to the slider in Creo Mold Design Extension.

- Lifter planning

  Is a lifter required for this project? In what location? Place a Datum Coordinate System on to the reference part where the lifter must be placed.

- Waterline/Cooling system planning

  What kind of cooling system is required for this article?

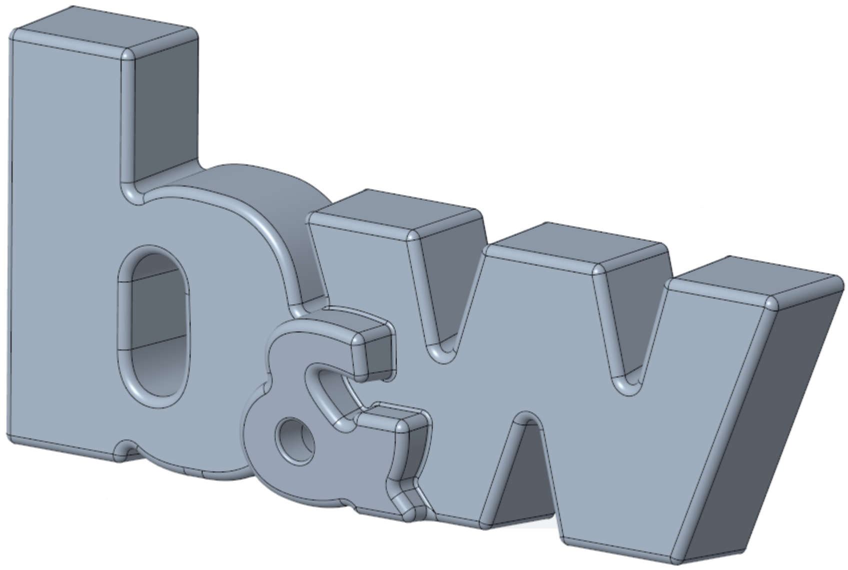
3.2.2 Set Working Directory

The usage of Creo Expert Moldbase Extensionbegins once you have finished the design of the cavity insert assembly. For this tutorial the assembly `MFG.ASM` is used. This can be found in the folder `models/tutorial_start` of the tutorial directory.

1. Copy all models from the directory `%TUTORIAL_PATH%/models/tutorial_start` to your working directory or Workspace.
2. Open `MFG.ASM`.

3.2.3 Prepare datum points for ejector pins

1. Retrieve the master design model `ARTICLE_REF.PRT`



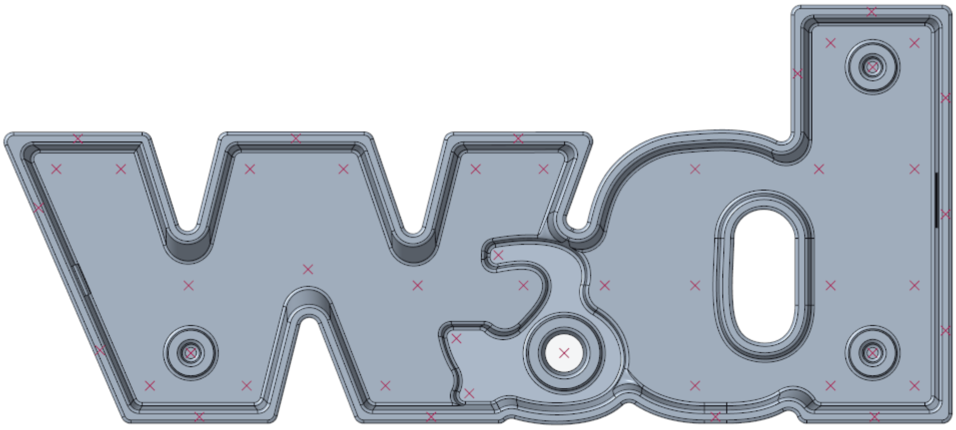
2. Make sure that Datum Points and Coordinate Systems are generally visible.

   The required Datum Points for ejector pin design have been prepared for this tutorial.

   a. PNTS_COREPIN_1: Used for trimmed core-pins at the screw bosses
   b. PNTS_SLEEVE_1: Used for according ejector sleeves at the screw bosses
   c. PNT_COREPIN_2: Used for a trimmed core-pins at the screw boss with a through hole. Additionally used for a trimmed core pin inside the cavity insert on the fixed half
   d. PNT_SLEEVE_2: Used for according ejector sleeves at the screw boss with a through hole
   e. PNTS_EPIN_ROUND_4MM: Pattern points, used to create round ejector pins with a diameter of 4 mm.
   f. PNTS_FLAT_EPIN_5-5X1MM_0-DEG: Used for flat blade ejector pins 5.5 x 1 mm and not rotated
   g. PNTS_FLAT_EPIN_5-5X1MM_90-DEG: Used for flat blade ejector pins 5.5 x 1 mm and 90° rotated
   h. PNTS_FLAT_EPIN_5-5X1MM_113-DEG: Used for flat blade ejector pins 5.5 x 1 mm and about 113° rotated

i. PNTS_FLAT_EPIN_5-5X1MM_0-DEG: Used for flat blade ejector pins 3,2 x 1 mm and not rotated

j. PNTS_SHOULDERED_EPINS_2MM: Used for shouldered trimmed ejector pins with a diameter of 2 mm and rotation fixture.



NOTE:
Usually the designer must add these datum points and consider where and how ejector pins need to be placed.

3.2.4 Prepare trim surfaces for ejector pins in case it is required

By default ejector are trimmed against the reference part itself. This can not be achieved in case is defined on a hole itself or in case the ejector cross section is not inside the reference part. In this tutorial two ejector trim surfaces are defined.

- TRIM_SURFACE_COREPIN_1: Used to trim the core-pin at the screw bosses with hole
- TRIM_SURFACE_COREPIN_2: Used to trim the core-pin at the screw bosses with hole on the fixed half side

3.2.5 Prepare coordinate systems for lifters and sliders

It is required to think about the position of lifters and sliders.

- CSYS_LIFTER: Used for lifter definition.

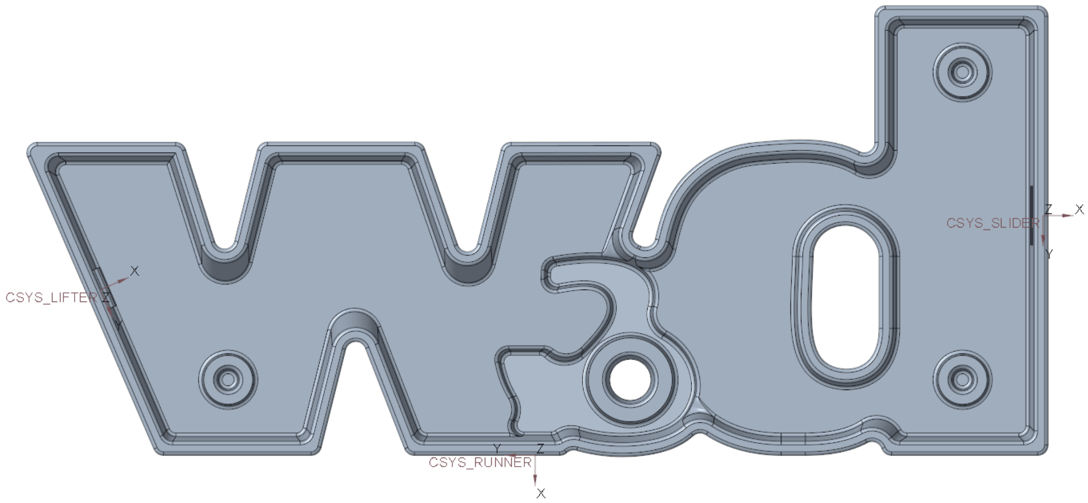
  NOTE:
  The x-axis of this coordinate system must point towards the planned opening movement of the lifter, the z-axis is in mold opening-direction

- CSYS_SLIDER: Used for slider definition.

  NOTE:
  The x-axis of this coordinate system must point towards the planned opening movement of the slider, the z-axis is in mold opening-direction

- CSYS_RUNNER: Used for runner definition.

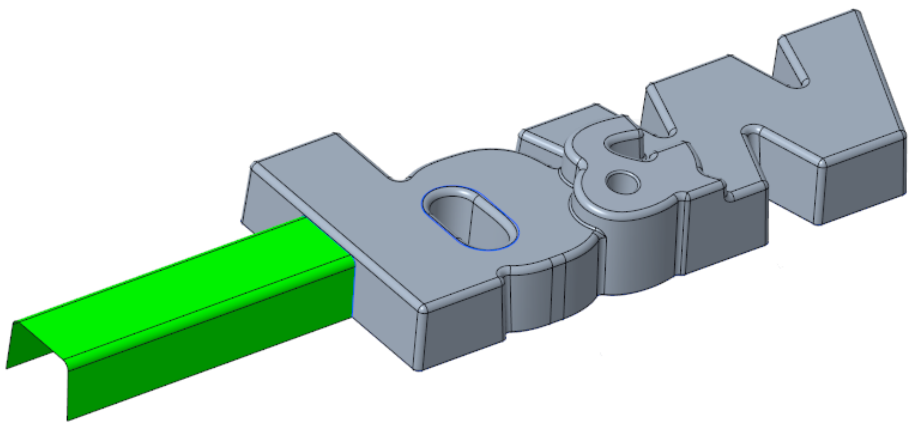
  NOTE:
  The x-axis of this coordinate system must point towards the planned demolding direction of the runner, the z-axis is in mold opening-direction



3.2.6 Prepare additional trim surfaces for sliders

1. Open the base model `ARTICLE.PRT.`

   The base model contains a quilt surface Trim 5. This surface will be used to trim the cam of the slider.



3.3 About Classifying Cavity Insert Assembly Components

The **Classify** dialog box can be open with **Classify** command.
Creo Expert Moldbase Extension must know the meaning of certain components in the cavity insert assembly. This is necessary for hand full of reasons:

- The outline of the cavity insert assembly can be calculated. These dimensions can later be user in the mold base definition to select the required size of the moldbase, to create a cavity layout or to create the cutouts in the cavity plates. Therefore it is required classify the cavity insert assembly.
- The software should avoid any geometric modification on the workpiece or on the reference part of the cavity insert assembly as this will cause regeneration failures.
- The software needs to know the reference part as this part islater used for several operations, e.g. trimming ejector pins.

1. Component can be classified by selecting a model type in the **Model** list in the **Model Type** column. The following types of models are available:
    - **Model**—General model.
    - **Workpiece**—Creo Parametric workpiece.
    - **Insert MH**—Extracts a moving half. When you select this type of model, the part ID and the part name are automatically added.
    - **Insert FH**—Extracts a fixed half. When you select this type of model, the part ID and the part name are automatically added.

- **Reference Model**—Used when performing interference checks. Important for the ejector pin designer.
- **Insert Assembly**—Used when measuring the outline of the included insert MH and insert FH models.

2. The outline calculation of the cavity insert assembly can be forced by the check box **Use the selected insert assembly size**. This is available only if an insert assembly is selected.
3. Select the **Rename Extracts** check box to rename all insert MH and insert FH components with the specific part name, and to save part ID parameters.
4. Click **OK** to recalculate the insert assembly outline, rename the insert models, and to save component settings, or click **Cancel** to close the dialog box without saving changes.

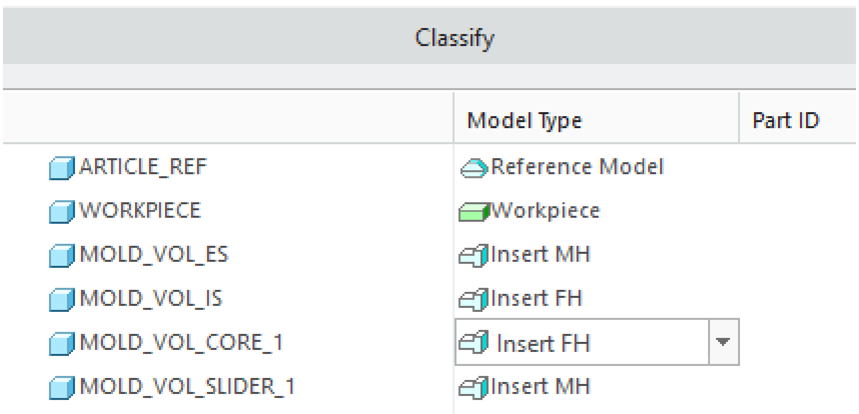
NOTE:
If **EMX Option** `ADD_TRIM_FACE_IN_CORE` is set, EMX will create a 'Solid surface Copy' to all models classified as **Reference Model** automatically.

3.4 To Classify the Cavity Insert Assembly

Download 01_To Classify the Cavity Insert Assembly to start from with this chapter

1. **Open** the Manufacturing model `MFG.ASM`
2. Select **EMX Manufacturing Mode ▸ Classify**

   The **Classify** dialog box opens showing a list of all models from the assembly including the assigned model type.
3. In the **Classify** dialog box Double-Click the **Model Type** of a component to change its type.



For this tutorial the types as shown in the image above:

- ARTICLE_REF: Reference Model
- WORKPIECE: Workpiece
- MOLD_VOL_ES: Insert MH
- MOLD_VOL_IS: Insert FH

- MOLD_VOL_CORE_1: Insert FH
- MOLD_VOL_SLIDER_1: Insert MH

4. Make sure the **Use the selected insert assembly size** check box is enabled.

   The Outline of the components classified as **Insert MH** and **Insert FH** will be determined and the result to saved to parameters in the assembly.

5. Close the dialog box with **OK**.

3.5 About Creating Duplicated Cavity Insert Assemblies

Most mold bases are used to manufacture more than one copy of the article. For multiple cavity mold bases it can be necessary to have independent model geometry for the individual inserts, i.e. in case different cooling channels should be applied to the cavity inserts.

Within the **Duplicate Cavity** feature of Creo Expert Moldbase Extension external merge features are used to create the copies. Even if you want to create a single cavity mold we recommend to create a copy of the insert assembly as i.e. the work piece can be removed so it will not appear in the design at all.

NOTE:
Cavity insert assemblies based on family tables should not be used with **Duplicate Cavity** .
In case you prefer creating the insert assembly copies manually this step can be skipped.

1. With the original manufacturing model active the **Create Insert Copies** dialog box can be opened by clicking **EMX Manufacturing Mode ▸** **Duplicate Cavity** .
2. Double-click the **Type** column to set copy options:
   - **Copy**—Creates a new copy of this component using a external merge feature
   - **Ignore**—Does not include the component in the copy.
   - **Keep**—Retains the original models in the copied assemblies.
3. In the **Quantity** box, enter a value for the number of copies to generate.
4. The **Wildcard** entry defines how the renaming of the copied assemblies and parts will take place. When you make copies of an assembly or insert, the wildcard character is added as a suffix along with a numerical increment. The placeholer `%i` represents a counter starting with 1. For instance: `modelname_copy_1.abc`, `modelname_copy_2.abc`, etc.
5. Click **OK** to create the copies of the active assembly and then close the dialog box or click **Cancel** to close the dialog box.

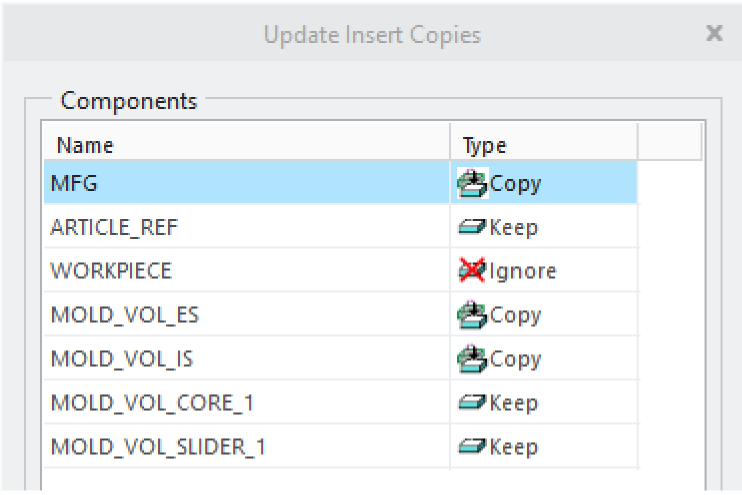
3.6 To Create Duplicated Cavity Insert Assemblies

Download 02_To Create Duplicated Cavity Insert Assemblies to start from with this chapter.

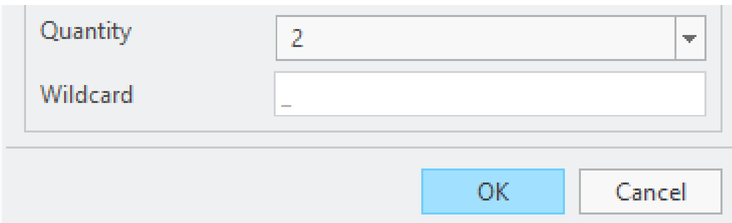
1. Click **EMX Manufacturing ▸ Duplicate Cavity** .

The **Multi Cavity** dialog box opens.

2. Double-click the **Type** operation within the table to set the type for each component in the cavity assembly.
3. Set the **Type** to Copy for:
   - `MFG.ASM` (The assembly always needs to be copied)
   - `MOLD_VOL_ES.PRT`
   - `MOLD_VOL_IS.PRT`
4. Set the **Type** to Keep for:
   - `ARTICLE_REF.PRT`
   - `MOLD_VOL_CORE_1.PRT`
   - `MOLD_VOL_SLIDER_1.PRT`
5. Set the **Type** to Ignore for:
   - `WORKPIECE.PRT`



6. Select `2` from the **Quantity** pull down to determine how many copies will be created.



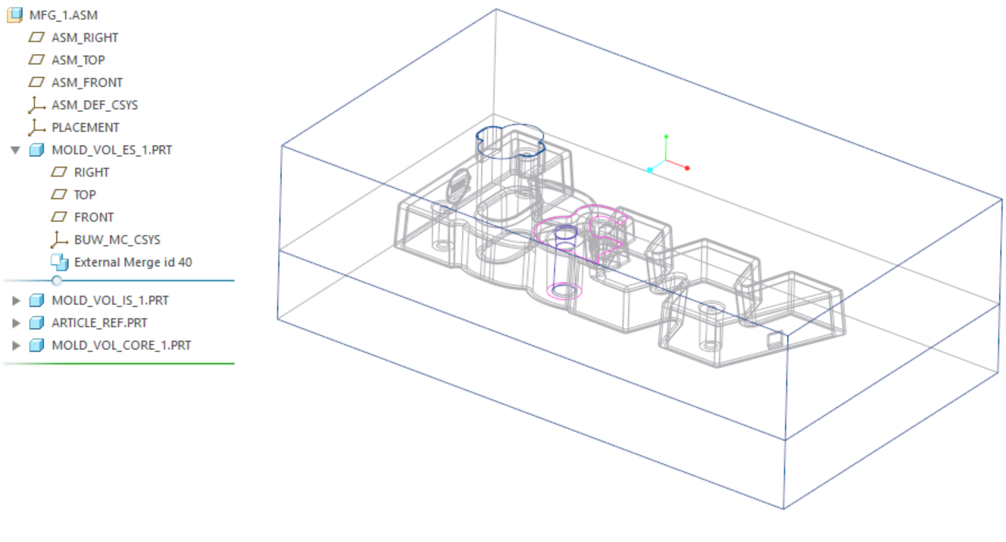
7. Close the dialog box with **OK** .

   With the current settings two new assemblies are created in session.

   Parts defined with **keep** will be assembled directly (`ARTICLE_REF`, `MOLD_VOL_CORE_1` and `MOLD_VOL_SLIDER_1`).

   Ignored parts like the `WORKPIECE` are not assembled at all.

Copied parts have an external merge feature:



8. Check the two copied cavity assemblies `MFG_1.ASM` and `MFG_2.ASM`.

3.7 About Identifying Trim Quilts in Cavity Insert Assembly

In Creo Expert Moldbase Extension Ejector Pins can be trimmed. To create a trim feature two different ways exists:

- Trim against the reference model: In case the ejector pin or lifter intersects with a part which was classified as **Reference model** in **Classify,** a **Trim to geometry** feature will be created during the trim operation.

  This default behavior is easy to use and does not require any additonal feature creation inside the reference model. Still, in certain design situation this feature is limited.

  - The feature can fail for certain free form shapes.
  - The feature does not completely trim the ejector pin in case the ejector pin cross-section is not fully covered by the reference model
  - In case the ejector pin is defined on a hole, the reference model can not be found and no feature can be created.

  To overcome these problems in can be necessary to use the alternative trim solution

- Trim against a trim quilt surface: In case the ejector pin intersects with a quilts surface which is identified in **Identify trim faces** a **Solidify** feature will be created during the trim operation.

  The disadvantage of this solution is the required additional step of creating these quilt surfaces within the reference model. On the other side the solidify feature covers all mentioned problems described above.

  - It is stable for free form shapes

- ○ Quilt surfaces can go beyond the reference model geometry and therefore easily cover the full ejector pin cross section.
- ○ Quilt surfaces can be used to fill holes in the reference model geometry.

To identify trim surfaces enter the **Trim Faces** dialog box. Trim surfaces can be identified in part, assembly and manufacturing mode

NOTE:
When there are no quilts in the list, you cannot trim lifters.

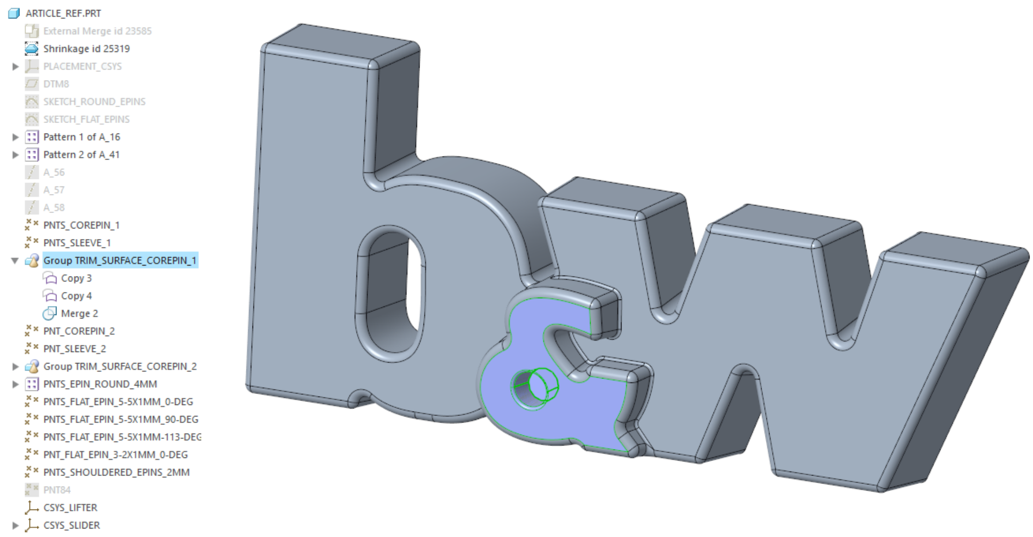
1. To open the **Trim Faces** dialog box opens click **Preparation ▸** **Identify trim faces**.
2. To remove a quilt, select it from the **Existing trim faces** list, and then click to remove the quilt from the list.
3. To add a quilt, click , select a quilt in the graphics window, and then click **Close** to close the dialog box.

3.8 To Identify the Trim Quilts in the Cavity Assembly

Download 03_To Identify the Trim Quilts in the Cavity Assembly to start from with this chapter.

3.8.1 Create the Trim Surface

1. Open the model `ARTICLE_REF.PRT`.
2. For this tutorial two trim quilt surfaces are already created: `TRIM_SURFACE_COREPIN_1` and `TRIM_SURFACE_COREPIN2`. In additional mold base design these surfaces need the be created manually.

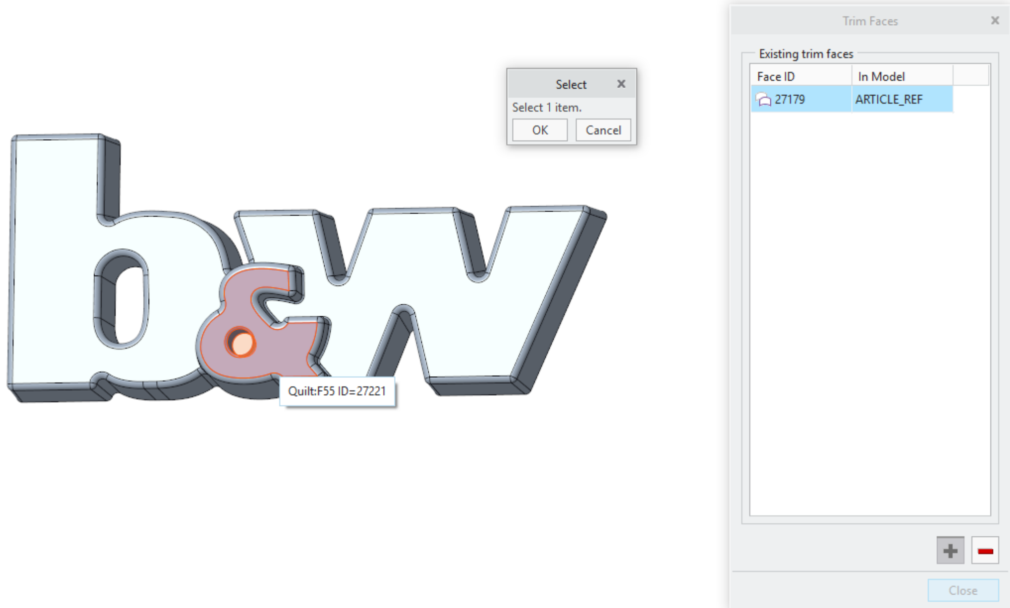


3.8.2 Select the trim surface

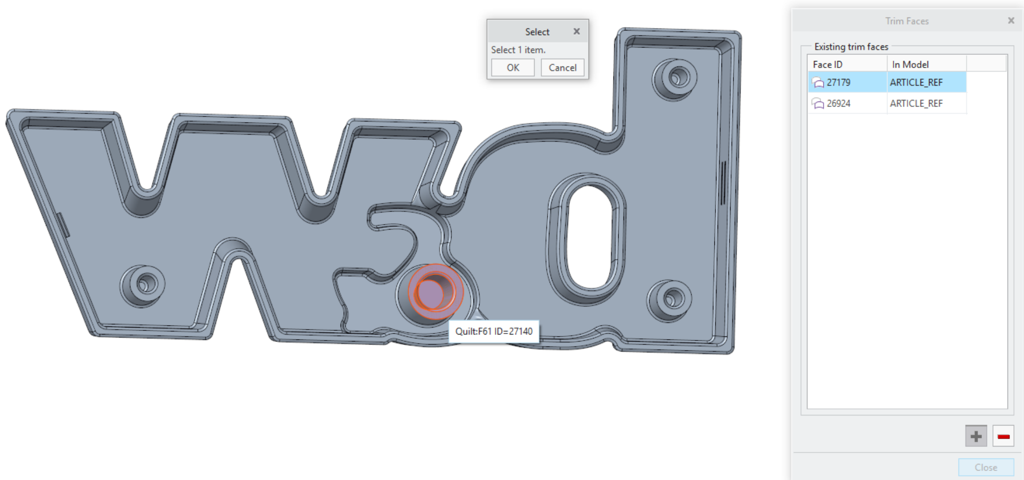
1. Stay in the part mode of the `ASM_PALM_REF.PRT` and open **EMX Part Mode ▸ Preparation ▸ Identify trim surfaces** 

   The **Ejector Trim Faces** dialog box opens.

2. Add a new surface to the list of trim surfaces with .
3. Select the quilt surface `TRIM_SURFACE_COREPIN_1` in the graphics window.



4. Select the quilt surface `TRIM_SURFACE_COREPIN_2` in the graphics window.



5. Close the **Select** dialog box with **Cancel**.
6. Close the **Ejector Trim Faces** dialog box with **Close**.

4

The Mold Base Project

4.1 About Projects

The top mold base assembly is referred to as a project. When you create a mold base design you must define general parameters and data for the mold base components.

- Click **EMX** ► **Create New** to create a new Creo Expert Moldbase Extension assembly, drawing, and report.
- After creating a project, you can perform the following actions:
  - Click **EMX Assembly** ► **Project** ► **Finish** to delete all suppressed plate features. Use only when the mold base is ready for production.
  - Click **EMX Assembly** ► **Project** ► **Archive** to save bitmap images of the current assembly and all reference models in an archive folder.

4.2 About Creating a Mold Base Project

1. The **Project** dialog box can be opened by clicking **EMX** ► **Create New** or in manufacturing mode click **EMX Manufacturing Mode** ► **Create New**.
2. Under **Data** project specific information can be provided:
   - **Project Name** — Name of the project assembly, associated drawings, and reports.
   - **Prefix** — A wildcard character, used as prefix for all parts assembled to this project
   - **Postfix** — A wildcard character, used as postfix for all parts assembled to this project
   - **User Name** — Name of the user that created this project.
   - **Date** — Date when project is created.
   - **Note** — Add notes about the project

   The default settings for almost all values and check buttons can be defined with this **EMX Options** .

Project GUI Entry	Corresponding EMX Option
Project Name	`DEFAULT_EMXNAME`
Prefix	`DEFAULT_PREFIX`
Postfix	`DEFAULT_POSTFIX`
Unit	`ASSEMBLY_UNIT`
Copy Drawings	`CREATE_ASM_DRAWING`

Copy Reports	CREATE_ASM_REPORT
Date	DATE_FORMAT

3. Under **Options** set the unit of measurement to **mm** or **inch**.
4. Under **Templates**, click to browse to the template directory. Select a previously designed EMX assembly or accept the EMX default template. If you do not use the EMX default template, then the files for the previously designed EMX assembly, are copied to the current working directory and are then renamed with the new project name, prefix, and suffix. If you use an EMX default template, select the **Copy Drawings** check box to create a drawing and select the **Copy Reports** check box to create a report.
5. In the **Project Parameter** area, select **Add local project parameters** to create a set of project-specific parameters for the new project. These parameters and their values will be inherited to all components of the mold base assembly. To enter or change a parameter value, double-click in the **Default Value** column for the parameter and type the value.

   NOTE:
   When you set the Creo Expert Moldbase Extension configuration option SAVE_LOCAL_PARAM_IN_ASM to YES, the project parameters are stored with the assembly. If you do not set the option to YES, they are written to an ASCII file in the current working directory.

   Project Parameters are defined in the **EMX Options ► Project Parameters** tab.
6. Click **OK** to create the new assembly and report or drawing in the working directory or click **Cancel**.

NOTE:
If the configuration option NAME_SUBASM_COMP is enabled during the creation of a new project and its value is different from "–", a set of subassemblies for all different types of the components will be created.

As an example the name of the automatic subassembly for cooling components is defined with the configuration option NAME_SUBASM_COOL.

All cooling components are always assembled to this subassembly automatically

4.3 To Create the Mold Base Project

Download 04_To Create the Mold Base Project to start from with this chapter.

1. Erase all objects from session.
2. Select **EMX** ► **Project** ► **Create New** .
3. Enter **Project Name** TUTORIAL
4. Enter **Prefix** TUTOR
5. Enter an empty value as **Postfix**
6. Enter the **User name** and **Date**

As a default the current date and current user is set.

7. Choose `mm` as the **Unit** system
8. The **Template Directory** is already predefined. Use the default `empty_template` assembly.
9. Enable the **Copy Drawings**check button .In the background a drawing of the current assembly is created. This drawing needs to be opened and initially saved.
10. Enter `ABC` as **Default Value** for **Project Parameter** `CUSTOMER`.
11. Enter `ARTICLE123` as **Default Value** for **Project Parameter** `ARTICLE`.
12. Start the creation process with **OK**.

    EMX copies the chosen assembly template from the **Template Directory** and renames both the assembly and the according drawing to the new name `TUTORIAL` . At this point there are two models in the assembly:

    a. TUTOR_MACHINE represents the machine plates and holmets.
    b. TUTOR_SKELETON provides all Datum Elements (Coordinate systems, Datum planes, Axes and Datum Points) that will be used for placing the Moldbase Components.

5

The Mold Base Assembly Structure

5.1 About Subassemblies

You can use Creo Expert Moldbase Extension to create and delete subassemblies. Each subassembly consists of a skeleton model with all the major datum features of the main assembly. To modify the skeleton, the EMX main assembly must be the active assembly.

Separating the entire mold base assembly into smaller subassemblies has two advantages :

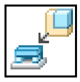
1. Two or more designers can work parallel on one design job.
2. Performance of Creo is much better when working with smaller subassemblies.

Good practice is saving a start assembly which already contains the desired set of subassemblies.

When you place a new component in an existing subassembly, make sure to set up the subassembly structure first. You cannot move a component to another subassembly when you modify it. First delete the component, and then recreate it.

5.2 About Creating a New Subassembly

You can create a new subassembly in any existing assembly. A subassembly uses the same unit of measure as the top assembly.

1. The **Subassembly** dialog box can be opened by clicking **EMX Assembly** ▸  **Add Subassemblies**.
2. Select a subassembly from the list of EMX subassemblies, as the target assembly for the new subassembly.
3. In the **Name** box, enter a name for the new subassembly, drawing, and report.
4. In the **Project Prefix** box, enter the project prefix.

   NOTE:
   If the EMX Option ENABLE_SUBASM_PREFIX is set to NO, the **Project Prefix** box is unavailable.
5. In the **Project Postfix** box, enter the suffix for the project.
6. Assign any of the following functions to the subassembly:
   - **Assembly**—Default, standard assembly
   - **Moving Half**—Defines the subassembly as moving half.
   - **Fixed Half**—Defines the subassembly as fixed half.

     NOTE:
     The correct subassembly is preselected for all components assembled from the **Mold Base Definition** dialog box, depending on the selected side. Both subassemblies, moving half and fixed half, must exist for the correct subassembly to be preselected.
   - **Ejector Pins**—Defines the subassembly as the target assembly for all ejector pins.
   - **Split FH MH**—Splits cavity assemblies which are assembled to this kind of subassembly into moving half and fixed half.

NOTE:
You can only split cavity assemblies with cavity assemblies created using the **Duplicate Cavity** command. All three types of subassemblies **Split FH MH**, **Cavity Moving Half**, and **Cavity Fixed Half** are required for the availability of the **Split the cavity assemblies into FH and MH** check box, in the **Cavity Layout** dialog box.

- **Cavity Moving Half**—Target assembly for moving half parts from the split cavity assembly. All parts classified as **Insert MH** in the **Classify Dialog** are added to this subassembly.
- **Cavity Fixed Half**—Target assembly for fixed half parts from the split cavity assembly. All parts classified as **Insert FH** in the **Classify** dialog box, are added to this subassembly.

7. Select the **Copy Drawings** check box to create a drawing from the selected template.
8. Select the **Copy Reports** check box to create a report from the selected template.
9. Under **Parameter**, in the **Parameter Name** column notice the and symbols:
   - —Indicates that the current parameter value is linked to the default value.
   - —Indicates that the current parameter value is not linked to the default value.

   To toggle the lock status, click or . You can change or enter a value in the **Value** column only if **Parameter Name** is set to .
10. In the **Value** column, double click a value to make changes.
11. Click **OK** to create the subassembly and close the dialog box or click **Cancel** to exit without saving changes.

5.3 About Deleting a Subassembly

Before you can delete a subassembly, you must delete its assembled components.

1. The **Delete Subassembly** dialog box can be opened by clicking **EMX Assembly ▶ Structure ▶ Remove Subassemblies**.
2. Select the subassembly to delete from the list on the left. The subassembly components appear in the list on the right.
3. Click to delete the subassembly.

5.4 To Set up the Assembly Structure

Download 05_To Set up the Assembly Structure to start from with this chapter.

1. Click **EMX Assembly ▶ Structure ▶ Add Subassemblies** .

The **Subassembly** dialog box appears.

2. In the **Subassembly** dialog box enter **Name** `TUTOR_MH` to create a subassembly that represents the Moving Half.
3. In the **Function** pull-down select **Moving Half** to indicate that this subassembly represents the moving half.
4. Click **OK** to close the dialog.

   EMX creates a new subassembly that has the same skeleton model included as the main assembly.
5. Repeat the steps to create a subassembly `TUTOR_FH` (for fix half). Make sure to select **Fixed Half** in **Function** pull-down
6. Repeat the steps to create a subassembly `TUTOR_SHARED` for the shared subassembly. Make sure to select **Assembly** in **Function** pull-down. This indicates that this is a regular EMX subassembly without specific function.

6

The Cavity Insert Assembly Layout

6.1 About the Cavity Insert Assembly Layout

Use the **Cavity Layout** dialog box to load cavity insert assemblies into memory and to define cavity insert assembly pattern.

1. The **Cavity Layout** dialog box can be opened by clicking **EMX Assembly ▸ Cavity Layout**
2. Select the **Subassembly** from the list of EMX subassemblies as the target assembly for the cavity insert assemblies.

   NOTE:
   You can assemble a different cavity insert assembly to two or more different subassemblies.
3. Click to load the cavity insert assembly. The loaded insert assembly is listed under **Insert Assemblies**. The current name and the placement coordinate system appear.

   NOTE:
   The first insert assembly that is loaded into memory will be automatically assigned to the first cavity pattern member.
4. By default the `PLACEMENT` coordinate system is listed as the assembly reference of the loaded cavity insert assembly..

   NOTE:
   In case no `PLACEMENT` coordinate system is found, the first coordinate system in the model tree is used as placement reference.
   Double click on the **CSYS** entry of an insert assembly opens a new graphic window and allows the manual selection of any other coordinate system in the model tree.
5. Select a **Cavity Pattern** type:
   - Single Cavity Pattern
   - Rectangle Cavity Pattern
   - Round Cavity Pattern
   - Star Cavity Pattern

   Specify the pattern data as follows:
   - Z angle and offset along x-axis and y-axis from the mold base origin
   - Quantity along the X axis and Y axis from the mold base origin.
   - Pattern size along the X axis and Y axis from the mold base origin.

     NOTE:
     The **Pattern Size** does not appear for **Single** type of cavity pattern.

- Pattern diameter for circular pattern shapes.

6. Select the **Automatically Recalculate** check box to automatically recalculate the pattern after changing parameters, or click to manually recalculate the pattern.
7. Double-click a cell to change a value in the pattern table. or indicates visibility status.
8. To assemble the previously loaded cavity insert assembly models to the cavity pattern coordinate system, double-click the cell in column **Assembly** from the table of all cavity pattern members.

   Select the cavity insert assembly from the option-menu by name.
9. Check the currently defined pattern type, size, and the assembled cavity Insert assembly using **Preview**.

   The Cavity Pattern and the assembled cavity insert assemblies appear in a 3D preview.
10. If the cavity insert assembly is assembled, you can click **Pattern first Insert Assembly** to create a pattern of the first insert assembly on all pattern members.
11. Click **Split the cavity assemblies into FH and MH** to move parts classified as **Insert MH** and **Insert FH** into the target assemblies. The target assembly must have **Cavity Fixed Half** and **Cavity Moving Half** assigned.
12. Click **OK** to close the dialog box. All changes to the cavity pattern layout are applied
13. Click **Cancel** to discard the previewed changes. The previous cavity layout is restored.

6.2 To Set up the Cavity Insert Assembly Layout

Download 06_To Set up the Cavity Insert Assembly Layout to start from with this chapter.

1. Open the `TUTORIAL.ASM` assembly.
2. Click **EMX Assembly ▶ Mold Base ▶ Cavity Layout** . The **Cavity Layout** dialog box opens.
3. Select `TUTOR_SHARED` from the **Subassembly** pull-down list.
4. Select the push button **Load Insert Assemblies** .

   The **Open File** dialog box opens.

   Select mfg_`1.asm` from the current working directory and leave dialog with **Open**.

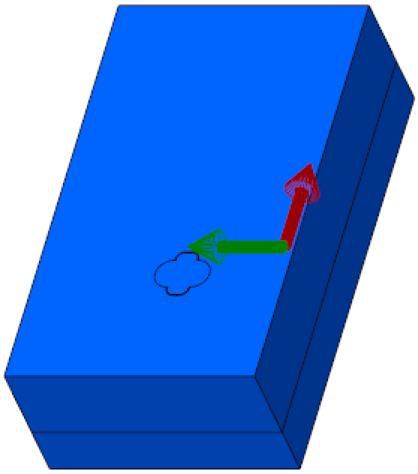
   The assembly is loaded to memory and listed in the table of insert assemblies.

Name	CSYS
MFG_1	PLACEMENT

   The assembly is automatically assigned to the first pattern member

ID	X	Y	Z	A-Z	A-X	Assemb
1	0	0	0.000	0.000	0.000	MFG_1

5. Click **Preview** to update the insert assemblies in the graphic window.



6. Repeat **Load Insert Assemblies** for the second insert assembly `mfg_2.asm` and load it to memory. `mfg_2.asm` will be listed in the table of insert assemblies as well.
7. Select the Rectangular **Cavity Pattern** Type .
8. Enter `2` for the **Quantity** in Y-direction.
9. Enter `50` for the **Pattern Size** in Y-direction.
10. Click to recalculate the pattern.

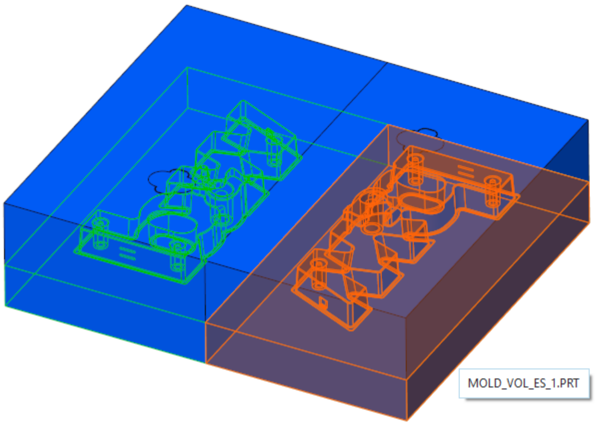
    The cavity pattern table now shows both cavity instances.
11. Double-click the **Assembly** column in the second row of the pattern table. An Option menu appears which lists all insert assemblies in memory.

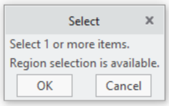
    Assign `MFG_2` to the second pattern member.
12. Change the value of the **A_Z** -column to `180` in the first row.

ID	X	Y	Z	A-Z	A-X	Assembly
1	0	-25.000	0.000	180	0.000	MFG_1
2	0	25.000	0.000	0	0.000	MFG_2

-
MFG_1
MFG_2

13. Click **Preview** again to update the insert assemblies in the graphic window.





14. Leave the dialog box with **OK**.
15. The Cavity pattern is now adjusted in the `TUTOR_SKELETON.PRT` model and contains two coordinate systems `CAVITY_1` and `CAVITY_2`.

Notes:

7

The Moving and Fixed Half Views

7.1 About the Moving and Fixed Half Views

By default two simplified representations for the fixed and the moving half of the mold base are created. The names of these simplified representation can be defined with the **EMX Options** `NAME_MOVING_HALF` and `NAME_FIX_HALF`.

Set the `CREATE_LAYER_PER_PART` and `CREATE_GROUP_LAYER` configuration options to `YES`, to automatically create a layer for each model and a group layer for component groups.

Most EMX components are automatically added to the correct simplified representation. Still it can be necessaray to add components to a certain side manually.

1. To add a component to the moving half click **EMX Assembly ► Views ► Add component to ► Moving Half** and select the desired component in the graphic window.
2. To add a component to the fixed half click **EMX Assembly ► Views ► Add component to ► Fixed Half** and select the desired component in the graphic window.
3. The simplified representation of the moving half can be activated with **EMX Assembly ► Views ► Show ► Moving Half View**
4. The simplified representation of the fixed half can be activated with **EMX Assembly ► Views ► Show ► Fixed Half View**
5. To switch back to the master view click**EMX Assembly ► Views ► Show ► Master View**
6. Click **EMX Assembly ► Views ► Toggle Wireframe Style** to toggle componenty like plates and insert parts to wire frame style.

In case it is prefered to work with layers instead of simplified reprenetations the **EMX Options** `USE_LAYER_FOR_SIDE_VIEWS` can be activated.

NOTE:
Not do not change this option in the middle of on going projects There can either be simplified representations or layers, but a mixture is not recommended.

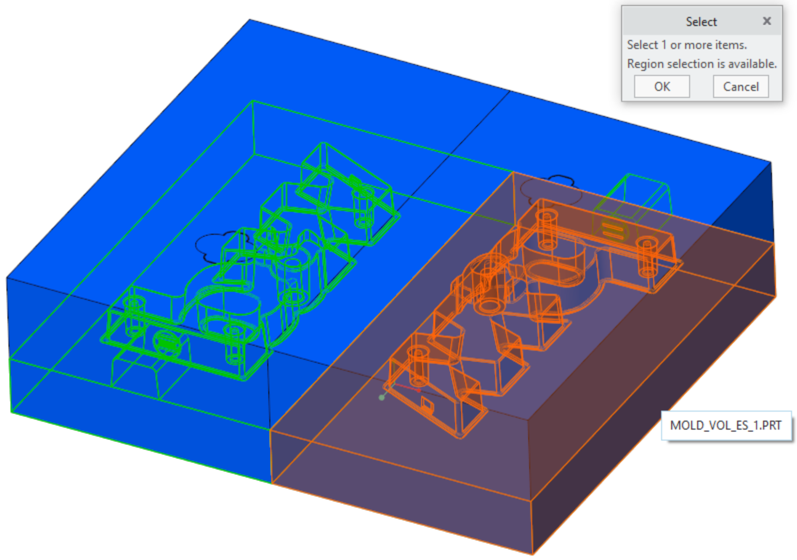
7.2 To Assign Components to Moving and Fixed Half Views

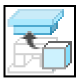
Download 07_To Assign Components to Moving and Fixed Half Views to start from with this chapter.

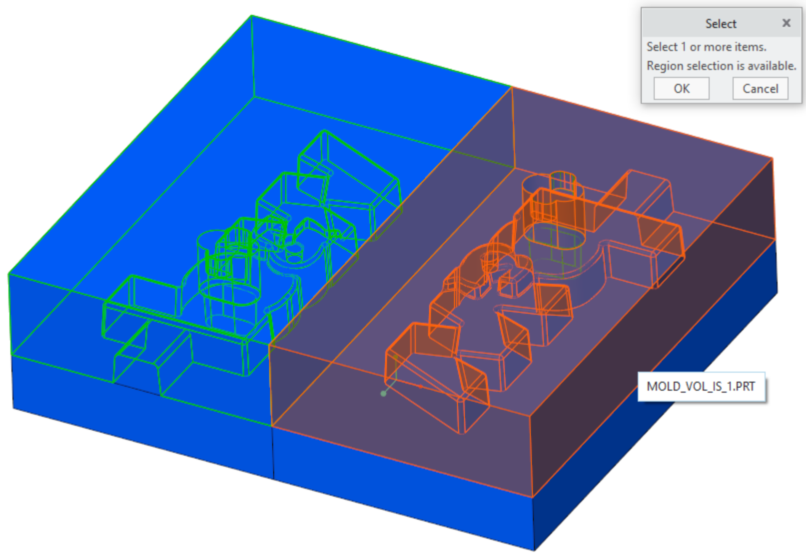
1. Open the `TUTORIAL.ASM` assembly.
2. Click **EMX Assembly ► View ► Show ► Moving Half View**

Note that still all components are displayed. Components must be assigned to moving and fixed half assembly manually.

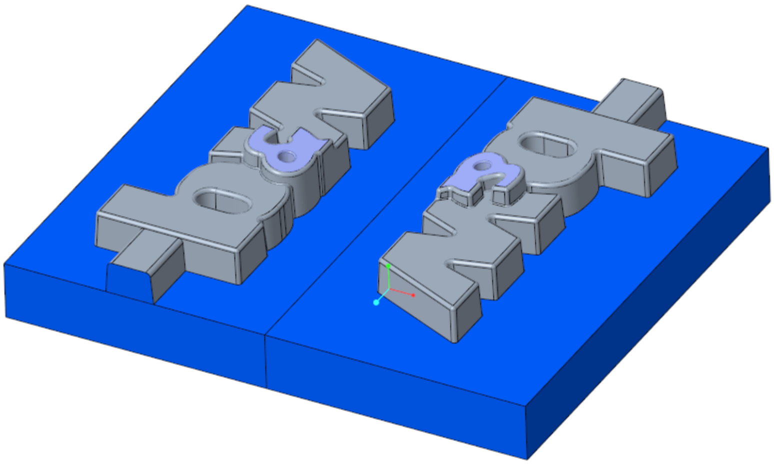
3. Click **EMX Assembly ► Views ► Add component to ►**  **Moving Half** and select the components `MOLD_VOL_ES_1`, `MOLD_VOL_ES_2` and two times `MOLD_VOL_SLIDER_1`.



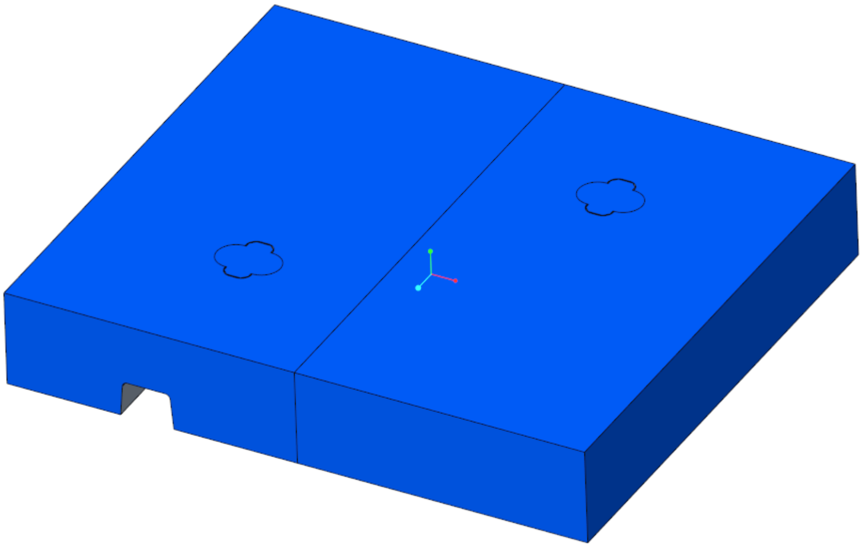
4. Click **EMX Assembly ► Views ► Add component to ►**  **Fixed Half** and select the components `MOLD_VOL_IS_1` , `MOLD_VOL_ES_2`and `MOLD_VOL_CORE_1` in both cavity insert assemblies.

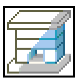


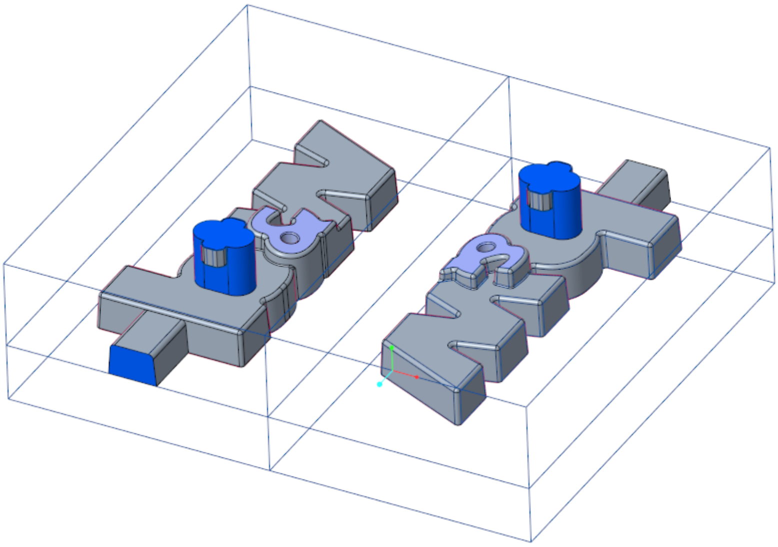
5. Click **EMX Assembly ► Views ► Show ►**  **Moving Half View**



6. Click **EMX Assembly ► Views ► Show ►**  **Fixed Half View**



7. Click **EMX Assembly ► Views ► Show ►**  **Master View** and select **EMX Assembly ► Views ►**  **Toggle Wireframe Style**



Note that components like plates and insert parts are set to wire frame style.

Notes:

8

The Mold Base Defintion

To Assemble Side Interlocks

8.1 About Defining a Main Axis Offset

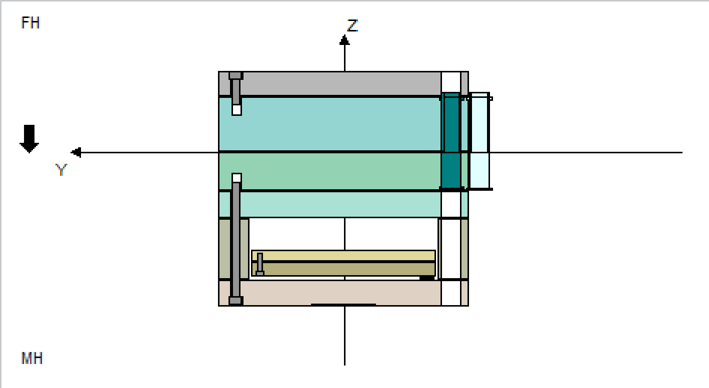
1. Click **EMX Assembly ► Mold Base ► Define Main Axis Offset** to open the **MAIN_ AXIS Position** dialog box.
2. Enter values for **Offset X** and **Offset Y**.
3. Click **OK** to close the dialog box. The main axis in the skeleton and existing cutouts in plates are be moved to the new position.

8.2 About Defining the Mold Base Assembly

A mold base is the set of plates and major components that hold the core/cavity insert.

1. The **Mold Base Definition** dialog box can be opened by clicking **EMX Assembly ► Assembly Definition.**
2. Define the following variables in the **Mold Base Definition** dialog box:
   - The **Unit** of measurement is the same as the project assembly.
   - Select a **Supplier** from the list of available mold base suppliers.
   - Select the **Size** from the list of mold base sizes. Existing size-dependent component dimensions are automatically updated when the size is changed.
3. Select or clear the **Show Cavity** check box to show or hide the cavity insert assembly outline in the top and side view.

   NOTE:
   The outline of the cavity insert assembly, represented by a blue box, was calculated while **Classifying** the Insert assembly and is displayed in the **Side View**. The representation enables the user to set up the cavity layout and select the best fitting mold base size.
4. Click an icon at the top of the dialog box to access other dialog boxes in which you can define the component:
   - —Opens the **Load EMX Assembly** dialog box.
   - —Opens the **Save EMX Assembly** dialog box.
   - —Clears all the components in the mold base
5. A dynamic view of the current mold base definition appears in the preview window.



6. Use the mouse wheel for zooming and middle-mouse button for panning in the view.
7. From the preview window you can view or position new components, and identify components to modify or delete.
8. Right-click a component in the preview window to add guides and plates or to modify and delete the component.

   Double-click a component to open the **Component** dialog box.
9. A summary tree appears under the dynamic view. This summary tree contains all defined components inside of the mold base definition.

Plates	Type				
Clamping Plate	F20	446	346	36	1.1730
Support Plate	F60	346	346	36	1.1730
Rail	F70	346	346	86	1.1730
Clamping Plate	F20	446	346	36	1.1730
Guides	Type				
Guide Bush (1x)	E1110/22-36				
Screws and Pins	Type				
Clamp Screw MH (4x)	E1200/12x160				
Clamp Screw FH (4x)	E1200/12x40				
Ejector Screw MH (4x)	E1200/8x25				
Stop Disc MH (4x)	E1500/20				
Equipment	Type				

   The tree is divided into four nodes:

   - **Plates**—All plates, including the type, the major dimensions and the material value
   - **Guides**—All guides, including the ordering number and the number of assembled instances. Additionally the color of the selected guide pattern appears
   - **Screws and Pins**—All screws and pins, including the ordering number and the number of assembled instances
   - **Equipment**—All other equipment components, including the ordering number and the number of assembled instances
10. Right-click a node to add guides or plates or to modify and delete the component. Double-click a component to open the **Component** dialog box.

11. Click to reset the zoom for side and top views.
12. Click and select a component to delete. The component is highlighted in red. Click **Yes** at the prompt to delete it.
13. Select **MH** (moving half) or **FH** (fixed half) to toggle the view for the bottom preview window.
14. To add a new plate, right-click an existing plate and select either **Add Plate Above** or **Add Plate Below**. The new plate is placed either above or underneath between the selected plate. You can also select the new plate and right-click outside the stack to place the new plate on top of the **Moving Half** or **Fixed Half** stack. The **Plate** dialog box opens.
15. Alternatively, add a new plate, by selecting a plate type from the list under the summary tree, and then select an existing plate. The new plate is placed between the selected plate and the splitting plane. You can also select the new plate and click outside the stack to place the new plate on top of the **Moving Half** or **Fixed Half** stack. The **Plate** dialog box opens.
16. To add guide components right-click a plate and click **Add Guide**.
17. Add any of the following types of components to the assembly:
    - **Locating Ring MH**
    - **Locating Ring FH**
    - **Thermal Insulation Plate MH**
    - **Thermal Insulation Plate FH**
    - **Sprue Bushing**
    - **Support Bushing**
    - **Knockout**
    - **Side Interlock**
    - **Top Interlock**

    a. When you select a component, a dialog box opens for the selected component in which you can define the component. Equipment components are automatically placed in the correct location. Before you add a guide component, select the active main guide pattern on the left side of the dialog box.

    NOTE:
    You can define up to three different patterns for the main guides.

    b. Choose a guide component:
    - **Centering Sleeve**
    - **Centered Guide Bush**
    - **Guide Bush**

- **Centered Leader Pin**
- **Leader Pin**
- **Ejector Guide Bush**
- **Ejector Leader Pin**
- **Puller Pin**
- **Puller Bolt**

c. Choose additional components such as screws, stop system components, and so on to add from the default component list under the top preview window:

- **Clamp Screw MH**
- **Clamp Screw FH**
- **Ejector Screw MH**
- **Ejector Screw FH**
- **Rail Screw**
- **Rail Dowel Pin**
- **Return Pin/Back Pin**
- **Stop Disc MH**
- **Stop Disc FH**
- **Stop Pin MH**
- **Stop Pin MH**
- **Tubular Dowel**
- **Compression Spring**

d. Define the new components in the dialog box. The components are automatically placed.

18. Select **Front** or **Top** to toggle view type. The view changes from side to top view. In the top view all major patterns are displayed as circles and rectangles.
19. The summary tree is updated and the list of patterns inside the mold base appears. The major nodes are listed below:
    - **Guide Patterns**—A list of all guides related patterns, including the number of active pattern members, the visibility status, the width and length, and the number in the width and length direction.
    - **Other Patterns**—A list of all other patterns, including the number of active pattern members, the visibility status, the width and length, and the number in the width and length direction.

20. To change pattern settings, either double-click a pattern inside the view or the summary tree or right-click a pattern. The **Pattern** dialog box opens.
21. Select a guide pattern from the summary tree under the bottom preview window. There are four main guide patterns and two ejector guide patterns (or) in the list. A dialog box for the selected guide in which you can set pattern data, opens.
22. Select a screw or stop system pattern from the list under the bottom preview window. A dialog box for the selected item opens. Define the pattern before placing a new component:
    - **Clamp Screw MH**
    - **Clamp Screw FH**
    - **Ejector Screw MH**
    - **Ejector Screw FH**
    - **Rail Dowel Pin**
    - **Rail Screw**
    - **Return Pin/Back Pin**
    - **Stop System MH**
    - **Stop System FH**
    - **Side Interlock**
    - **Top Interlock**
23. Click the list box next to the zoom tool to select the active main guide pattern. A new guide component is added to the active pattern that appears in the bottom preview window. Set the correct pattern before adding a new guide component.
24. Click to access the **Machine** dialog box.
25. Click **Close** to close the dialog box.

8.3 To Set up the Mold Base Size and Assemble Plates

Download 08_To Set up the Mold Base Size and Assemble Plates to start from with this chapter.

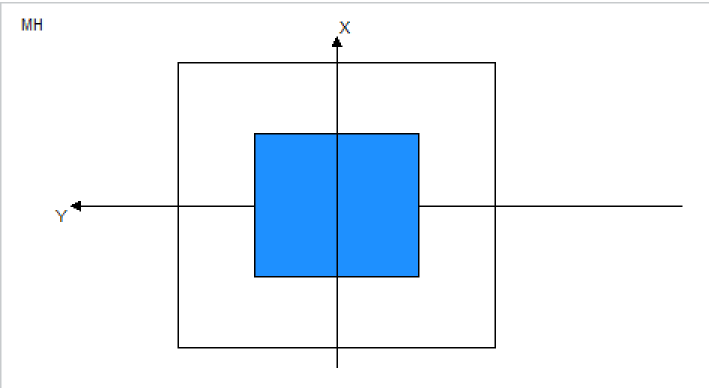
8.3.1 Recalculate Outline of Cavity Insert Assemblies considering new Cavity Layout

In case the cavity layout was modified. The Outline of the cavity insert assemblies need to be recalculated in the mold base assembly context.

1. Click **EMX Assembly ► Preparation ►** **Classifying** to open the **Classify** dialog box.
2. Make sure the check button **Use the selected insert assembly size** is enabled.
3. Click **OK** to leave the dialog box.

8.3.2 Select Supplier and Set Size

1. Click **EMX Assembly ► Mold Base ►** **Assembly Definition** .The **Mold Base Definition** dialog box opens.
2. Choose `meusburger` as mold base **Supplier**.
3. Select **Top** to switch to the top view. The current mold base size is displayed as rectangle to ensure that the selected mold base fits the cavity insert assemblies.
4. From the **Size** pull down select `446x396`.



A message dialog box appears. As you have not defined any plates at this point you can close the dialog wit `YES`. Generally, changing the mold base size is possible, but be aware that this will influence your complete design.

8.4 About Defining a Plate

To add a new plate, right-click a plate and select **Add Plate Above** or **Add Plate Below**. Alternatively, in the **Mold Base Definition** dialog box, add a new plate by selecting a plate type from the plate list and then clicking a plate. The **Plate** dialog box opens.

1. Set the unit of measurement as **mm** or **inch**. The list of available suppliers is updated.
2. Select a supplier from the list. All available plates appear in the plate list. The selected plate also appears in the preview window.
3. Select the assembly to add the plate from the list on the top right of the dialog box.

4. Plate dimensions appear in the table next to the preview. Double-click the **Value** column to perform the actions listed below:
   - Edit the dimension value.
   - Select a valid value from a list.
   - Select a valid value from a list or enter a value.
   - You cannot change disabled values.

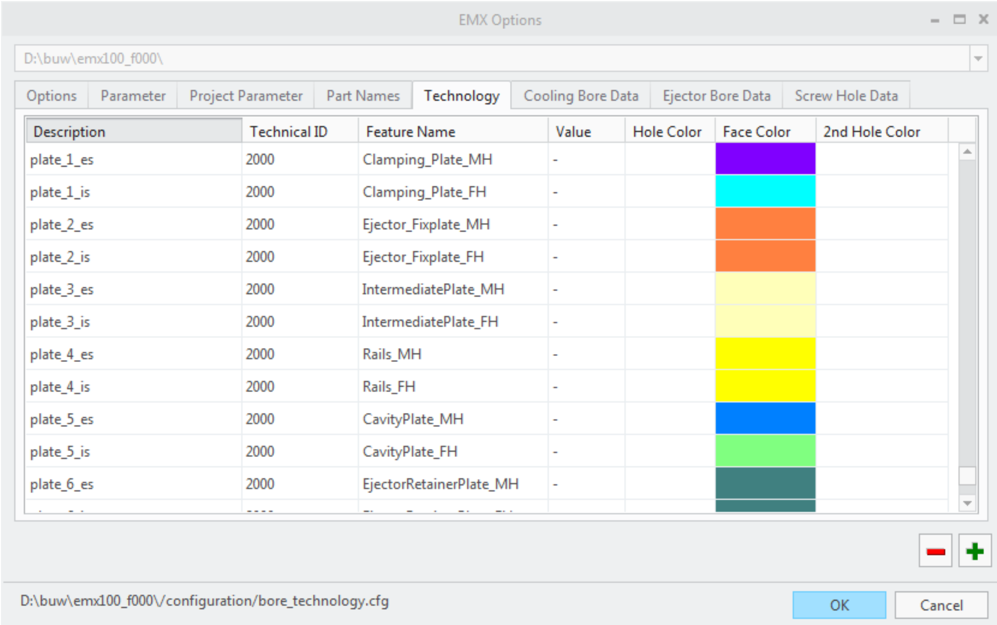
   NOTE:
   If a **Length**, **Width** or **Thickness** value is entered that is not listed in the catalog, be aware that the ordering number might change. Manual changes are indicated in the summary tree with a warning icon .

   Use the **Stock Allowance** dimension to reduce the thickness of a standard plate. This results in the correct **Order Number** but allows machining of the ordered plate.
5. Enter a plate name in the **Part Name** box. To enter an alternate name, select the check box. Both default names are defined in **Part Names** on the **EMX Options** dialog box.
6. Click a tab to preview all available features of the chosen plate type.
7. To set plate property options, select the **BOM Data**, **Parameter**, and **Relation** tabs.
8. Select the **Copy Drawings** check box on the **Options** tab to create a drawing from a template.

   NOTE:
   To save the plate drawings automatically when the plate is assembled to the mold base, set the Configuration Option `SAVE_DRAWINGS` to `YES`.
9. The dimensions of the active feature are displayed in the table. Double-click the **Value** or **Tolerances** column to change the dimension value. You cannot change the dimension name.
10. Select the **Add to BOM** check box to include the component in the Bill of Materials.
11. Click **OK** to close the dialog box and update the assembly with the new plate, or click **Cancel** to close the dialog box, and return to the **Mold Base Definition** dialog box.

8.4.1 Background Information

- The plate colors can be defined in **EMX Options ▸ Technology**.



Description	Technical ID	Feature Name	Value	Hole Color	Face Color	2nd Hole Color
plate_1_es	2000	Clamping_Plate_MH	-			
plate_1_is	2000	Clamping_Plate_FH	-			
plate_2_es	2000	Ejector_Fixplate_MH	-			
plate_2_is	2000	Ejector_Fixplate_FH	-			
plate_3_es	2000	IntermediatePlate_MH	-			
plate_3_is	2000	IntermediatePlate_FH	-			
plate_4_es	2000	Rails_MH	-			
plate_4_is	2000	Rails_FH	-			
plate_5_es	2000	CavityPlate_MH	-			
plate_5_is	2000	CavityPlate_FH	-			
plate_6_es	2000	EjectorRetainerPlate_MH	-			

- The plate-templates are located in `components/%UNIT%/plates`. The length, width and thickness of plates can be added as parameter to the models when setting the according configuration options:
  - `PARAM_PLATE_LENGTH`
  - `PARAM_PLATE_WIDTH`
  - `PARAM_PLATE_THICKNESS`
- If the configuration option `RECALC_MASS_PROPS` is set to `YES`, the center of gravity will be calculated for the entire mold base assembly when the **Mold Base Definition** dialog box is closed.

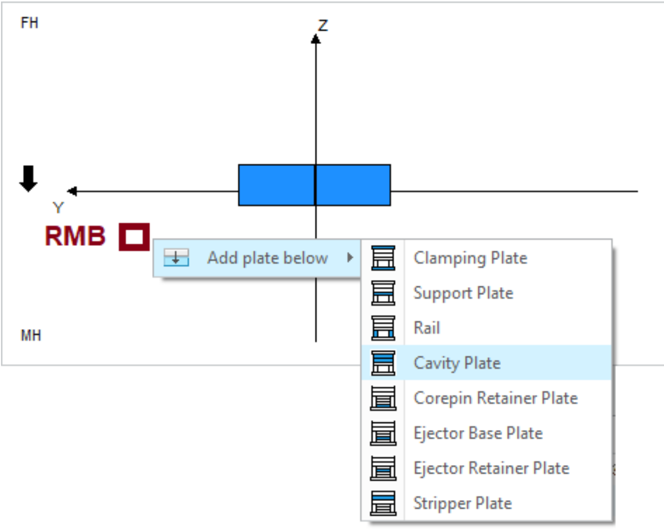
  The position of the point `CENTER_OF_GRAVITY` in the skeleton model is then moved to the correct position, so it can be used to drive the position for eyebolts. The default for this option is set to `NO` as the calculation might take some time for large mold base assemblies.
- For each component type extra layer names can be defined in **EMX Options ▸ Part Names**. In case a component is assembled, layers or summarize layers will are created and the new component will be put on the layer. Activate the configuration option `CREATE_GROUP_LAYER` and `CREATE_LAYERS_PER_PART` if extended layer functionality is required.
- Usually the tolerance mode for all dimensions of the template models is not modified when assembling such a model. Most customers want to see all dimensions being NOMINAL. In this case set the configuration option `EMX_TOL_MODE`.

8.5 To Assemble Plates

Download 09_To Assemble Plates to start from with this chapter.

8.5.1 Add Plates

1. Select **Front** to switch to the side view.
2. Click below the splitting plane with right-mouse-button (RMB) and selected **Add plate below ▸ Cavity Plate**.



   The **Plate** dialog box opens.

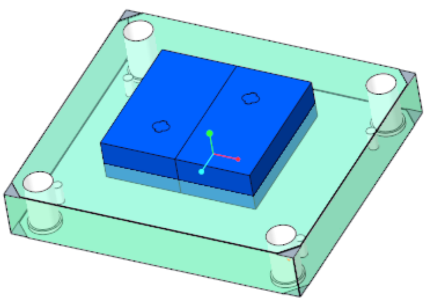
3. Select `TUTOR_MH` from the **Subassembly** pull down in the top right corner to define the target subassembly.
4. Set the **Thickness (T)** to `76`.

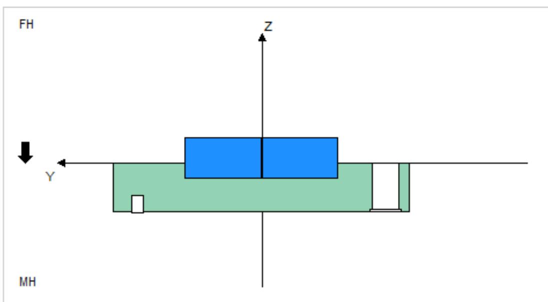
   While defining plate properties, this plate is displayed in the **Mold Base Definition** side view in preview (using white color). This indicates that the model has not yet been assembled in the Creo assembly.

5. Click **OK** in the **Plate** dialog box to assemble the plate.

   EMX copies the template of the chosen plate, renames it to the given **Part Name** and assembles it to the Creo assembly `TUTOR_MH`.

   The plate color in the **Mold Base Definition** is set to the model color. This indicates the plate exist as a real Creo model now.



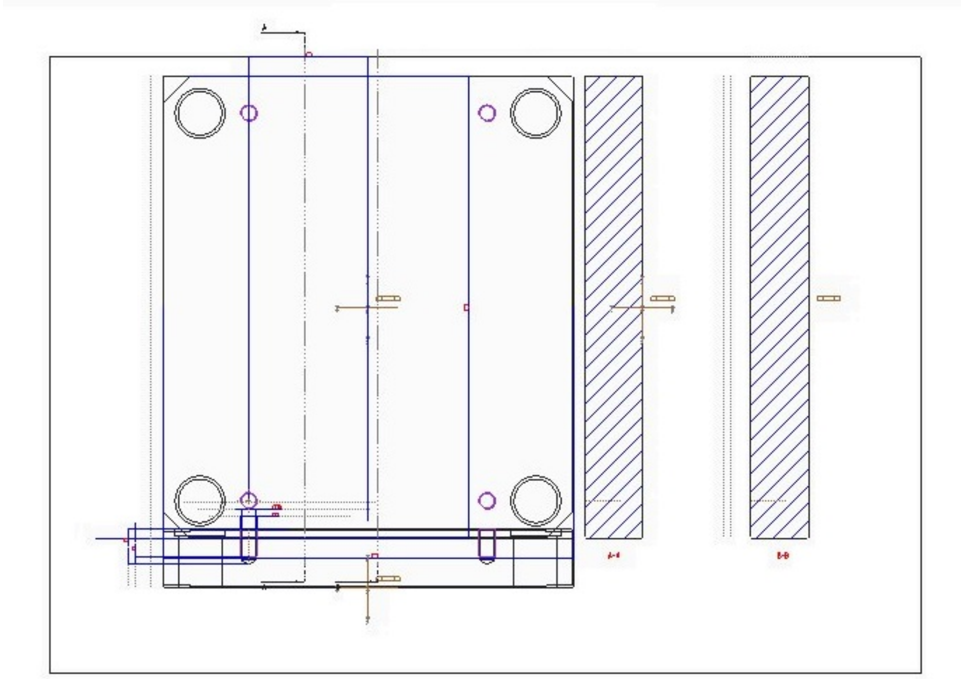


6. Close the **Mold Base Definition** dialog box with **Close**.

8.5.2 Save Plate Drawing

1. As the **Copy Drawings** check button in the **Plate** dialog box was activated, EMX has not only created the plate part model, but also a drawing with the same name.

2. Open this drawing with **EMX Assembly ► Documentation ►**  **Open drawing of selected model** and click the plate `TUTOR_CAV_PLATE_MH001.PRT` in the graphics window. The drawing opens.



3. Save the drawing and close this window

8.6 To Edit Plate Properties

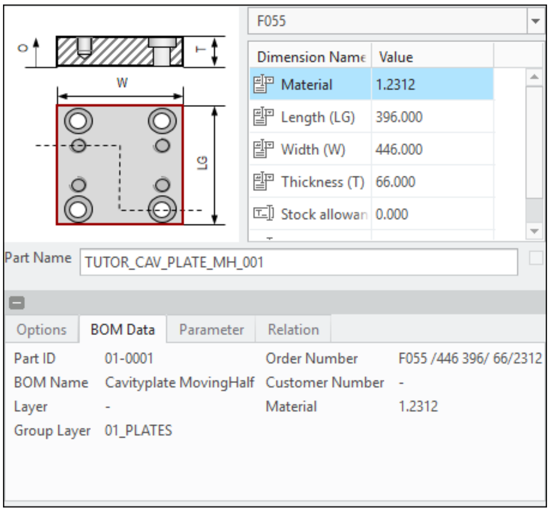
Download 10_To Edit Plate Properties to start from with this chapter.

1. To modify the properties of a plate, double-click on the plate in the side view of **Mold Base Definition**. The current targeted plate is highlighted in the side view and the graphics windows

   The **Plate** dialog box opens.

2. Switch the **Plate Type** to `F055`.
3. Change the **Thickness** to `66`.
4. Click the **BOM Data** sheet to check the BOM Information for this plate.

The ordering number is updated immediately when you modify plate information



5. Apply the modifications to the plate with **OK**.
6. Repeat the above steps and reset the plate properties back to `F050` with a thickness of `76` mm.

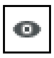
NOTE:
Alternatively, you can click on the plate with the right-mouse button and select **Modify** to edit the plate properties. Additionally, all plates can be selected in the summary tree.

8.7 About Plate Features

In the Creo Expert Moldbase Extension plate templates a set of suppressed features is predefined. By suppressing and resuming these features the software can build most of the basic plate shapes. Relevant features are listed in the bottom area of the **Plate** dialog box.

The features and the according visibility status is defined for all plate-types in `components/%UNIT%/plates/%SUPPLIER%/udf/feature_data.txt`. Additionally, all dimension rules to drive these feature are defined in the feature data file based on the supplier and the unit.

The visibility status can be:

-  **Visible**: The feature is visible. In case the mold base size is changed and the default value is defined as suppressed in the feature data, the feature will be suppressed
-  **Visible Locked**: The feature is always visible. In case the mold base size is changed the feature will still be visible even if the default value is defined as suppressed.
-  **Suppressed**: The feature is suppressed. In case the mold base size is changed and the default value is defined as visible in the feature data, the feature will be visible.

NOTE:
The visibility status can be modified manually. This is advanced functionality and not recommended for standard plates from suppliers. Manual changes can result in models that are different from the shipped supplier plates.

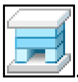
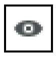
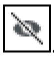
NOTE:
Manual changes on the visibility status and the feature dimension are set back to default values from the `feature data.txt` when changing the plate type or the mold base size.

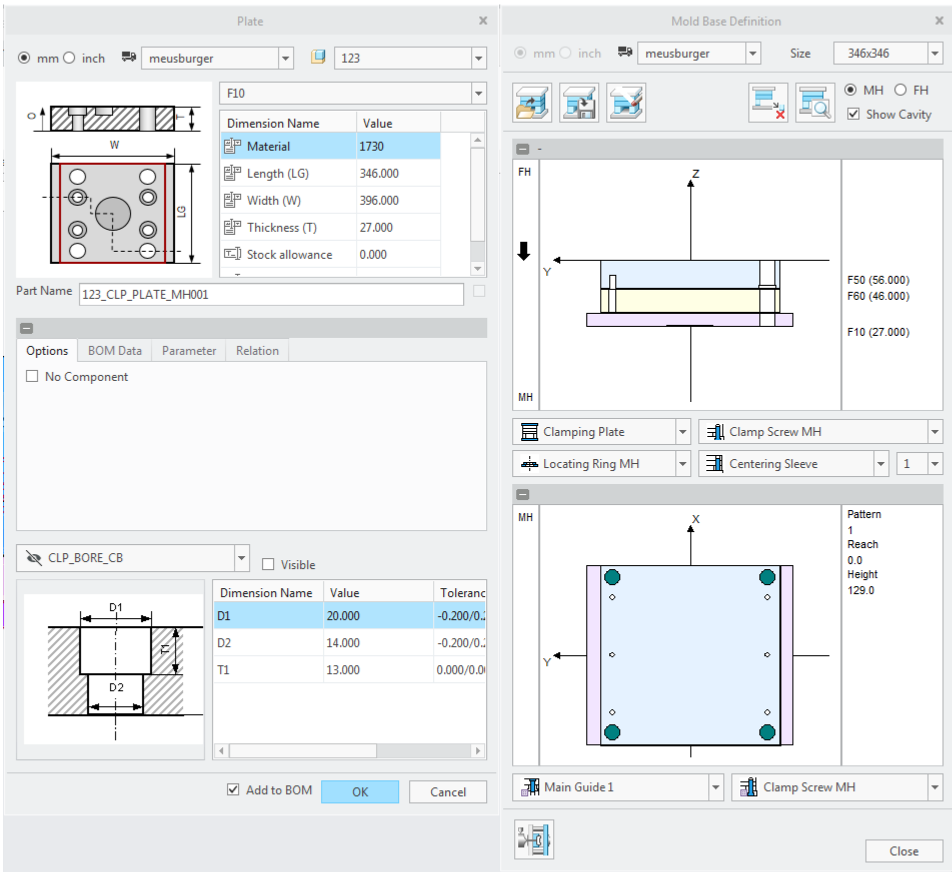
For the pryslot feature Creo Expert Moldbase Extension offers a special configuration option to suppress or resume them by default with `SHOW_PRYSLOT` set to `NO` or `YES`.

8.8 To Understand whether Cutouts in Plates are driven by Component or Plate

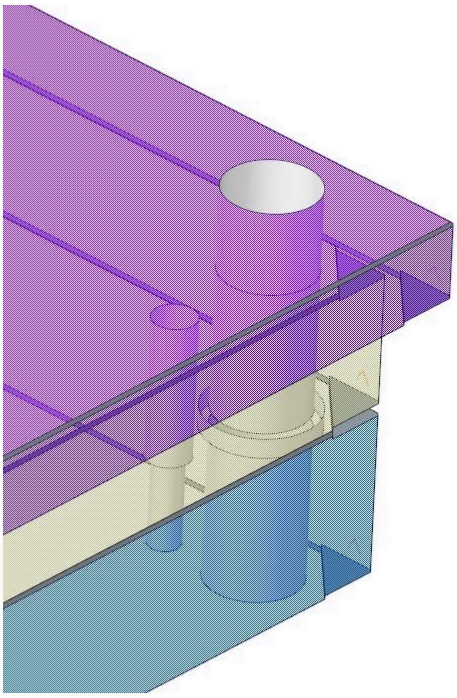
Download 00_to_cut_or_not_to_cut to start from with this chapter.

8.8.1 Hide Clamping Screws Plate Feature in Plate

1. Load the assembly `to_cut_or_not_to_cut.asm`.
2. Enter the **Mold Base Definition** dialog box with **EMX Assembly ► Assembly Definition** .
3. RMB on the **F10** plate in the side view.
4. Select the **CLP_BORE_CB** feature in the **Plate** dialog box to toggle its visibility from  to .



5. Close the **Plate** dialog box with **OK**.Now the default cutout for the clamping screw is gone from the clamping plate.

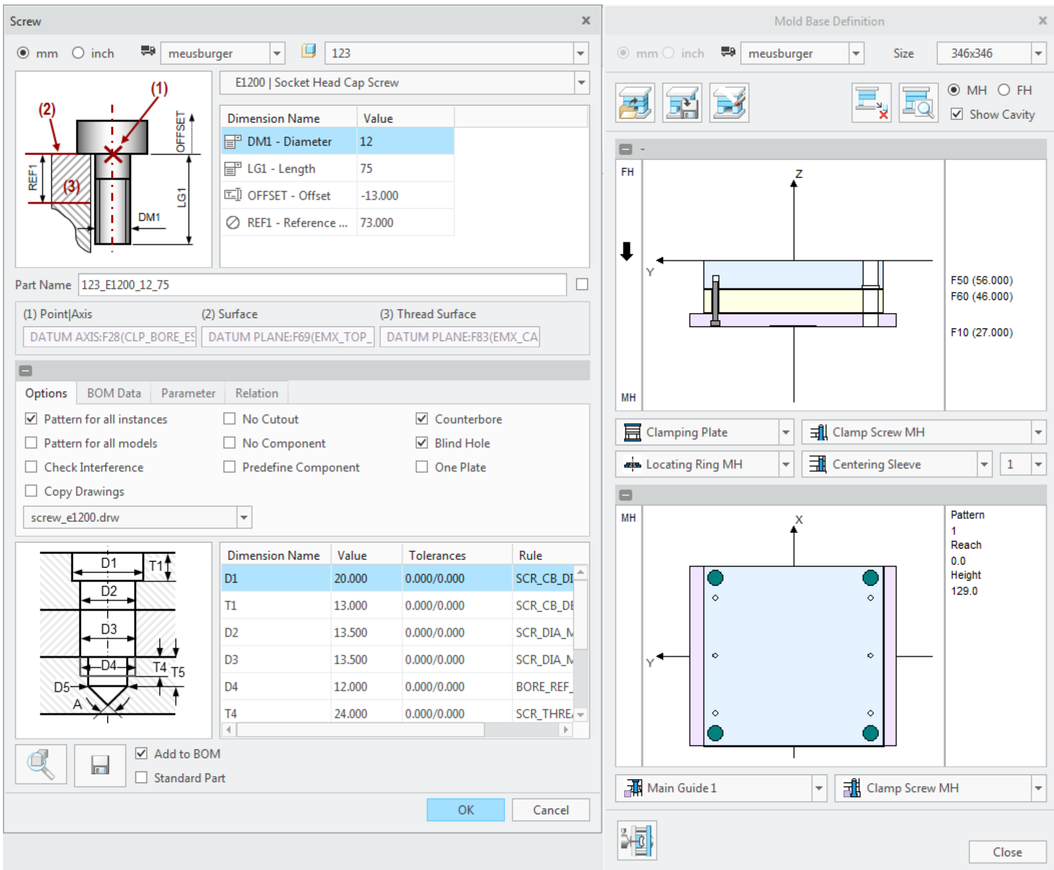


8.8.2 Add Clamping Screws

1. Select **Clamp Screw MH** from the screw list.

   The **Diameter** and **Length** in the **Screw** dialog box are set by the mold base data from the `size_346x346` plate file.

   NOTE:
   The **Reference Distance** is not set properly when adding default components in **Mold Base Definition**.

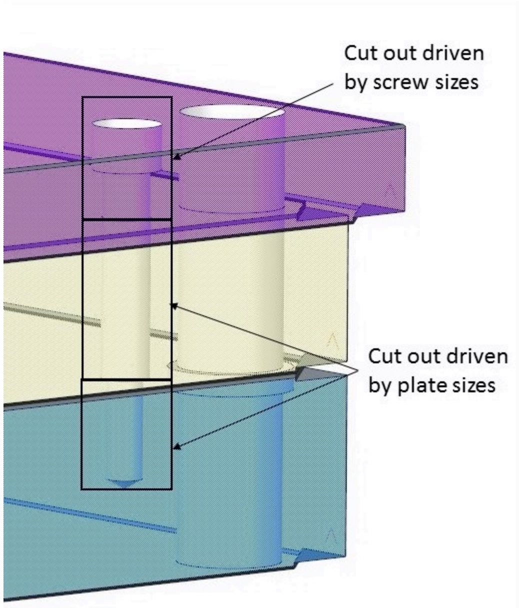


2. Leave the **Screw** dialog box with **OK**As NO counter bore in the F10 plate exist, an additional bore is added to the clamping plate CUT_CLP_PLATE_MH000.PRT. This bore is now driven by the screw cut out UDF, NOT by the plate size file.

   On the other side the counter bores in the plate F50 and F60 already existed. This features are driven by the plate size file.

   Therefore NO additional cut outs are added to the F50 and F60 plates.

NOTE:
Be aware that this can cause different cut out results.



The additional UDFs counter bore in the F10 plate is driven by the UDF's dimensions from the **Screw** dialog box.

The bores in F50 and F60 are driven by the feature dimensions in the **Plate** dialog box.

If you use P-Plates without any predefined cutouts and bores you can realized this by setting the **EMX Option** `USE_P_PLATE_STACK` to `YES`. Then all cutouts are generated by screws, leader pins individually.

8.8.3 Reset the Normal State

1. RMB on the F10 plate in the side view.
2. Select the **CLP_BORE** feature in the **Plate** dialog box to toggle its visibility from to .
3. Close the **Plate** dialog box with **OK**.
4. RMB on the clamping screw MH in the **Side View**.
5. Assemble the screw again WITHOUT ...**Counter Bore**.
6. ... and the assemble the screw again WITH **Counter Bore**.Now all cutouts in all plates are driven by the **Plate** size data only.

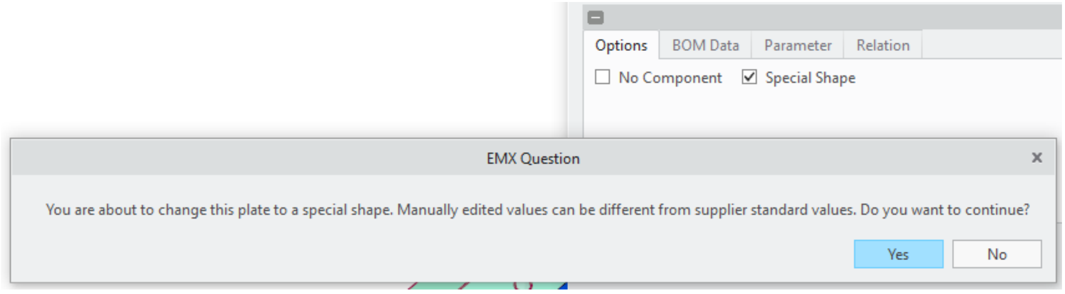
   NOTE:
   When adding components that create their cutout with cut quilts, you can avoid this cuts to be added into standard parts like screws, leader pins with **EMX Option** `CUT_PLATES_ONLY` set to `YES`.

8.9 Interlude: To Edit Plate Features

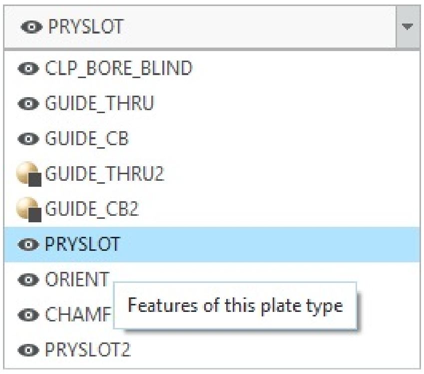
Download 11_Interlude: To Edit Plate Features to start from with this chapter.

NOTE:
The visibility status can be modified manually. This is advanced functionality and not recommended for standard plates from suppliers. Manual changes can result in models that are different from the shipped supplier plates.

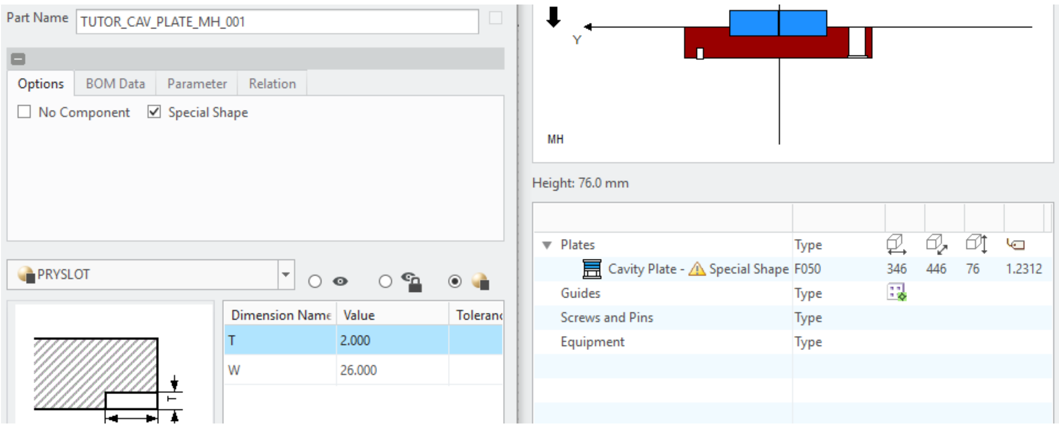
1. Click on the cavity plate in **Summary Tree** of the **Mold Base Definition** with right-mouse button and click **Modify** to open the **Plate** dialog box.
2. Click the **Options** and select **Speacial Shape** and confirm the EMX Question dialog box with **Yes**.



3. Select **PRYSLOT** from the feature list.

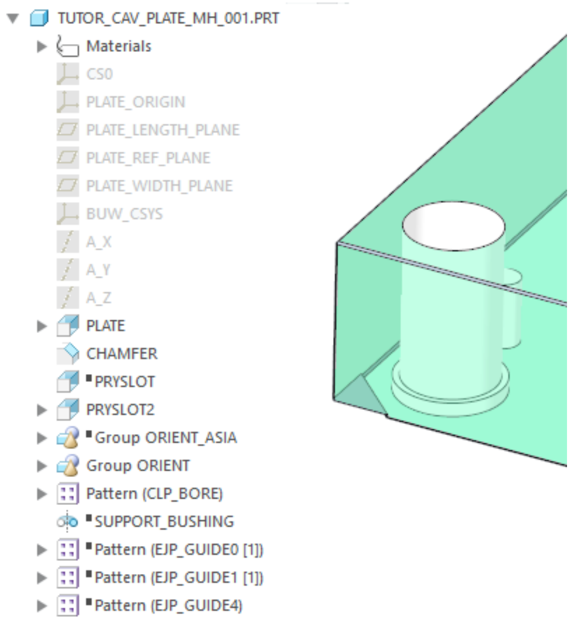


4. Select  **Suppress** from the radio group. **Special Shape** is displayed in the summary tree. This means that the plate does not conform to the standard.



5. Click **OK**.

The feature will be suppressed in the plate model.



6. Double-click the cavity plate in **Side View** to open the **Plate** dialog box again.
7. Select **PRYSLOT2** from the feature list.
8. Select **Visible** from the radio group and modify the **W** value to `20`.
9. Close the dialog with **OK.**

   The width is set to `20` mm.

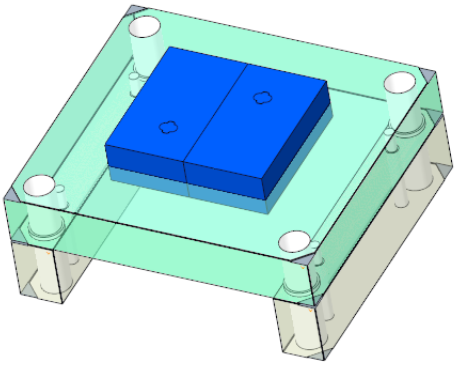
8.10 To Assemble Rail Plates

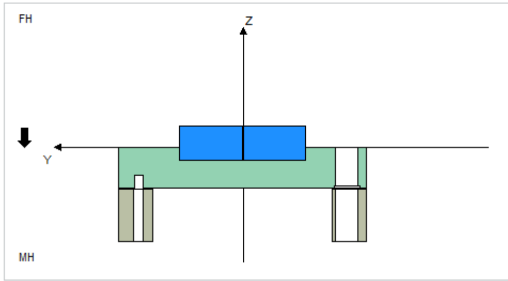
Download 12_To Assemble Rail Plates to start from with this chapter.

1. Select **Cavity Plate** with right-mouse button in the list of plates in the **Summary Tree**.
2. Click **Add plate below** ▸ **Rail** from the right-mouse menu.

   The **Plate** dialog box opens. The subassembly `TUTOR_MH` is already preselected as it is designated as the moving half subassembly.
3. Select **Thickness** to `116`.
4. Close the **Plate** dialog box with **OK**.

   The model will be created.





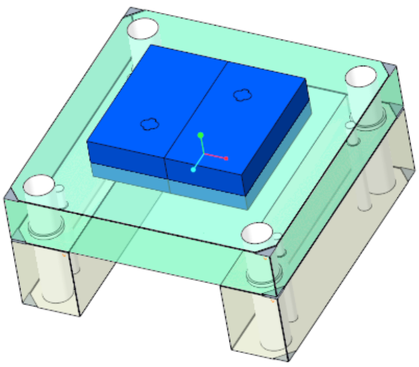
NOTE:
The rail width is set in the template, but some suppliers offer rails in two different sizes. You will get the wider size, if you have selected the `"s"` type during mold base size definition.

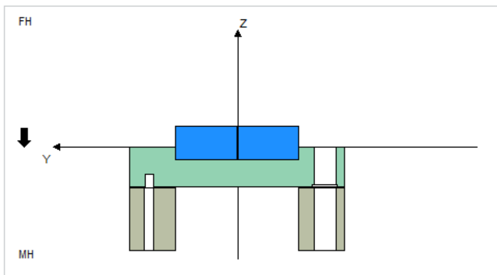
5. Select `396x396s` from the **Size** pull down menu.

   The mold base assembly will be regenerated with the new size and wide rails.

6. Answer the question **Use this size and update all existing components"** with **YES**.

   The mold base assembly will be regenerated with the new size and wide rails.





8.11 Interlude: To Assemble Non-Standard Rails

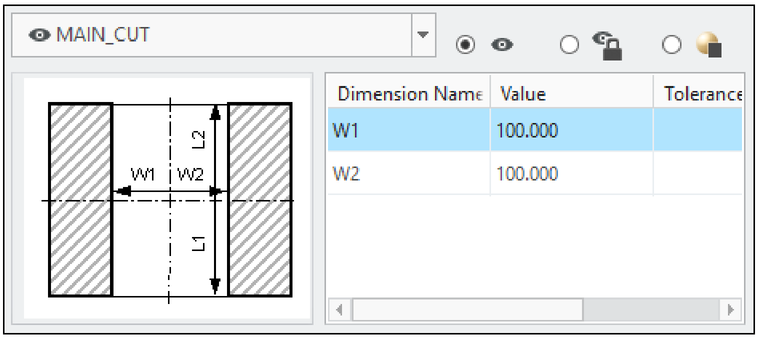
Download 13_Interlude To Assemble Non-Standard Rails to start from with this chapter.

1. Select **Cavity Plate** with right-mouse button in the list of plates in the **Summary Tree**.
2. Click **Add plate below ▸** **Rail** from the right-mouse menu.
3. Select `P` plate from the **Type** pull-down list.

   NOTE:
   Non-Standard plates should always be a `P` plate.

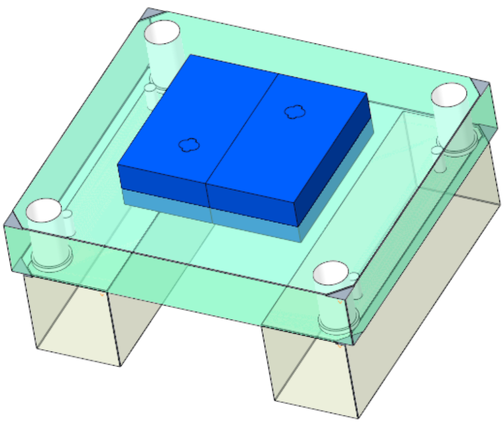
4. Activate the **Special Shape** check button in case you are working with non-standard plates

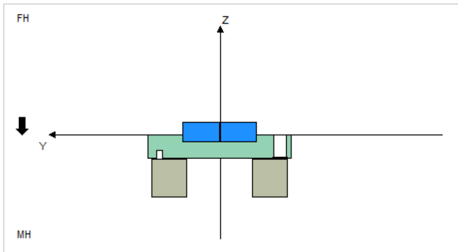
5. Set the **Length** to `390`.
6. Set the **Width** to `420`.
7. Set the **Thickness** to `120`.
8. Select the `MAIN_CUT` from the plate features pull-down list an set **W1** and **W2** to `100`.



9. Close the **Plate** dialog box with **OK**.

   The models will be regenerated.





The **Special Shape** is displayed in the summary tree.

Plates	Type				
Cavity Plate	F050	396	446	76	1.2312
Rail - ⚠ Special Shape	P	390	420	120	1.1730

8.12 Interlude: To Assemble Rail in One Model

Download 14_Interlude To Assemble Rail in One Models to start from with this chapter.

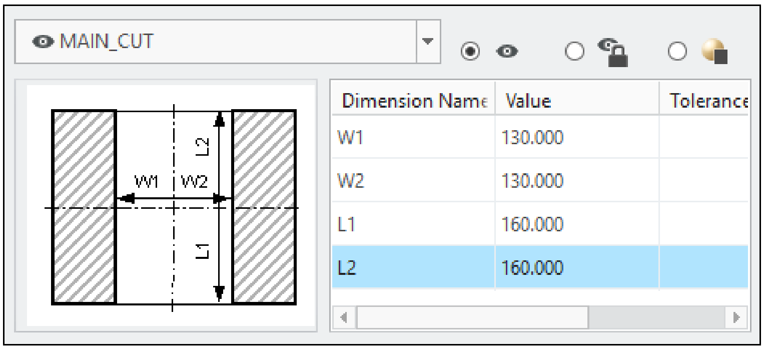
1. Select **Cavity Plate** with right-mouse button in the list of plates in the **Summary Tree**.
2. Click **Add plate below ▶** **Rail** from the right-mouse menu.
3. Select `P` plate from the **Type** pull-down list.
4. Select a **Thickness** of `116`.

5. Activate the **Special Shape** check button in case you are working with non-standard plates
6. Activate the **Both rails in one model** check button.

   NOTE:
   Both rails in one model is only enabled for `P`—Plates to avoid invalid order numbers
7. Select the `MAIN_CUT` from the plate features pull-down list.

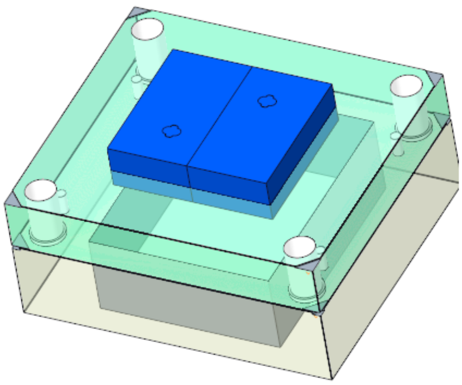
   Set **W1** and **W2** to `130`.

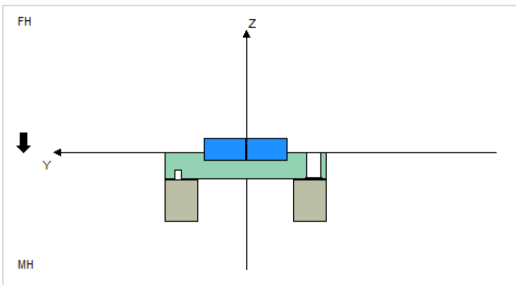
   Set **L1** and **L2** to 160.



8. Close the **Plate** dialog box with **OK**.

   The model will be regenerated.





8.13 About Mold Base Patterns

The Creo Expert Moldbase Extension mold base assembly is controlled by patterns which are defined inside the skeleton model.

These patterns are manipulated by the **Mold Base Definition** dialog box. Usually the number and dimensions of the patterns are defined based on the supplier mold base data. Still it can be useful to manipulate these values, e.g. in case additional guide or screw instances are necessary.

NOTE:
Be aware that modifying the standard patterns value can lead to geometry which does not fit the supplier standard.

The skeleton patterns define the position of the cut outs, but also the position of components assembled by the **Mold Base Definition** dialog box.

NOTE:
Even if components are NOT created from the **Mold Base Definition** dialog box it makes sense to use the datums of these patterns as placement reference.

In the skeleton model the following guide patterns exists:

- `MAIN_GUIDE1` : Position of all main guide components and related cut outs on the first pattern scheme. Displayed in the top view and summary tree with this icon or color .
- `MAIN_GUIDE2` : Position of all main guide components and related cut outs on the second pattern scheme. Displayed in the top view and summary tree with this icon or color .
- `MAIN_GUIDE3` : Position of all main guide components and related cut outs on the third pattern scheme. Displayed in the top view and summary tree with this icon or color .
- `MAIN_GUIDE4` : Position of all main guide components and related cut outs on the 4th pattern scheme. Displayed in the top view and summary tree with this icon or color .
- `EJP_GUIDE` : Position of guide components on the ejector package and related cut outs on the moving half.
- `EJP_GUIDE_IS`: Position of guide components on the ejector package and related cut outs on the fixed half.

Additionally, these other patterns exists:

- `CLP_BORE_ES` : Position of main clamping screws and related cut outs on the moving half.
- `CLP_BORE_IS` : Position of main clamping screws and related cut outs on the fixed half.
- `EJP_BORE` : Position of ejector package clamping screws and related cut outs on the moving half.
- `EJP_BORE_IS` : Position of ejector package clamping screws and related cut outs on the fixed half.
- `STOP_BORE_IS` : Position of stop system and the related cut outs on the fixed half.
- `RAIL_SCREW` : Position of rail screw and the related cut outs.
- `RAIL_PIN` : Position of rail dowel pin and the related cut outs.
- `RETURN_PIN` : Position of return pins and the related cut outs on the moving half.
- `STOP_BORE` : Position of stop system and the related cut outs on the moving half
- `STOP_BORE_IS` : Position of stop system and the related cut outs on the fixed half.

- `INTERLOCK` : Coordinate system pattern for the position of the side interlock assemblies.
- `TOPLOCK` : Coordinate system pattern for the position of the top interlock assemblies.

Patterns are displayed and modified using the top view of the assembly in the **Mold Base Definition**dialog box

1. Click **Top** to switch to the assembly top view.

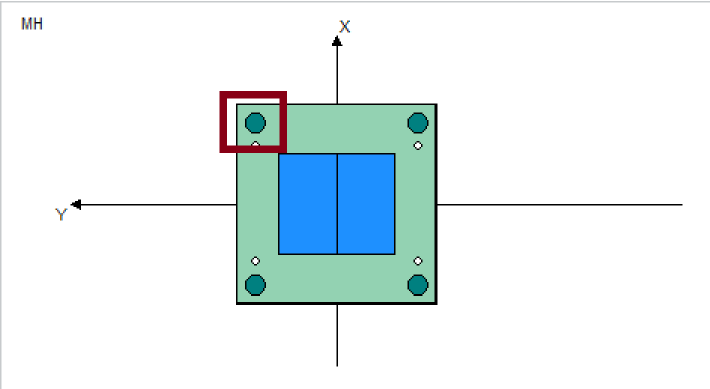
   The **summary tree** changes its content. All patterns and their numbers and major dimension are listed within the tree
2. Use **MH** and **FH** to view the mold base assembly patterns either from the bottom or from top.
3. Double-click the pattern either in the top view or from the pattern list. The relevant pattern dialog box opens.
4. Enter new values for **Quantity** and **Pattern Size** along the `X`-axis and `Y`-axis from the mold base origin.
5. To manually recalculate the pattern, click

   NOTE:
   Select the **Automatically Recalculate** check box to automatically recalculate the pattern after the parameters are edited. .
6. Double-click a cell to change a value in the pattern table. or indicates visibility status.
7. For **Main Guide 1** pattern only: Use the 1+3 Option to automatically create a guide component setup which has `1` instance on the first main guide component pattern and `3` instance on the second main guide component. This setup is used to assemble guide components with different inner diameters to avoid incorrectly rotated assembly of the mold base.
8. Click **OK** to close the dialog box, to update the assembly and to return to the **Mold Base Definition** dialog box, or click **Cancel** to close the dialog box, and to return to the **Mold Base Definition** dialog box.

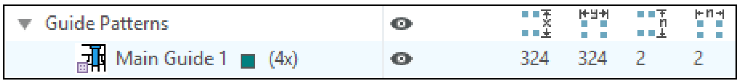
8.14 To Edit Mold Base Patterns

Download 15_To Edit Mold Base Patterns to start from with this chapter.

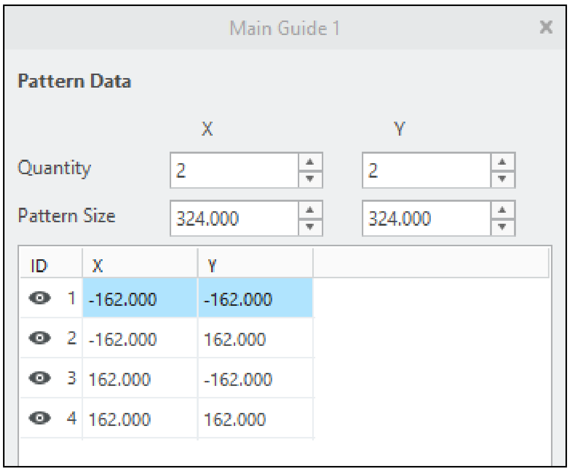
1. Click **Top** to switch to the top view of the mold base.
2. Select the **Main Guide 1** pattern.
   - Either by double-click inside the top view



   - Or by double-click on the listed pattern inside the summary tree.

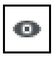


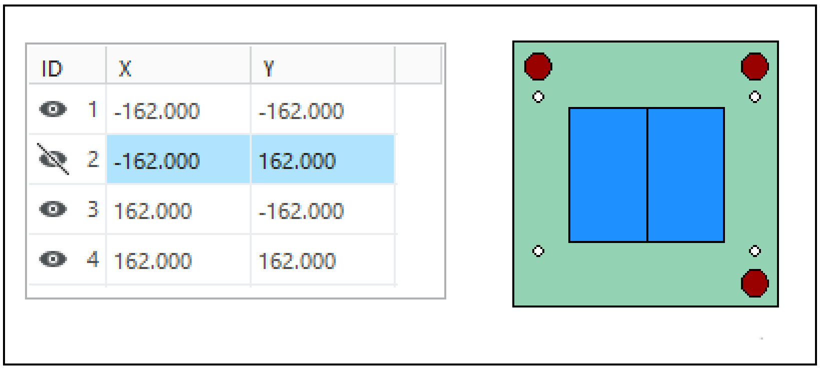
The **Pattern** dialog box appears.



If a plate supplier provides default dimensions for a certain pattern in its catalog, this values will be used as default in the entries **Quantity** and **Pattern Size**.

NOTE:
Be careful with changing pattern dimensions provided by suppliers. For guide patterns its very unusual to change. The following steps are just described as an example.

3. Click the visibility icon symbol  in the first column. The icon switches to  which indicates that this pattern instance should be removed from the axis-pattern.

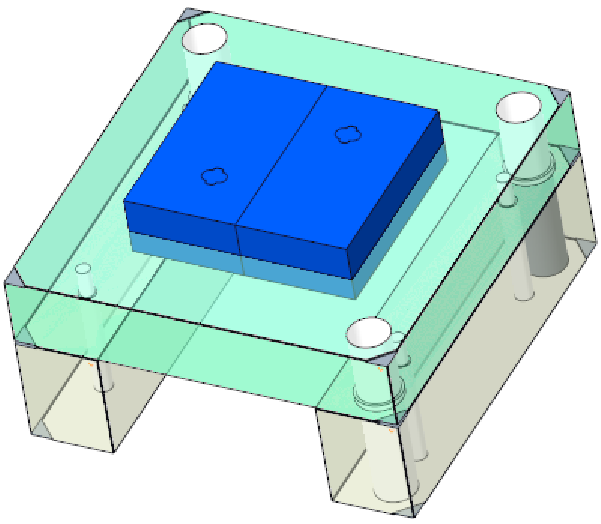


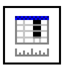
ID	X	Y
1	-162.000	-162.000
2	-162.000	162.000
3	162.000	-162.000
4	162.000	162.000

NOTE:
Leader pins, bushings etc. assembled on this axis pattern will also appear only on the visible members of this pattern.

4. Close the dialog with **OK**.

   The modification will be applied to the mold base assembly model.



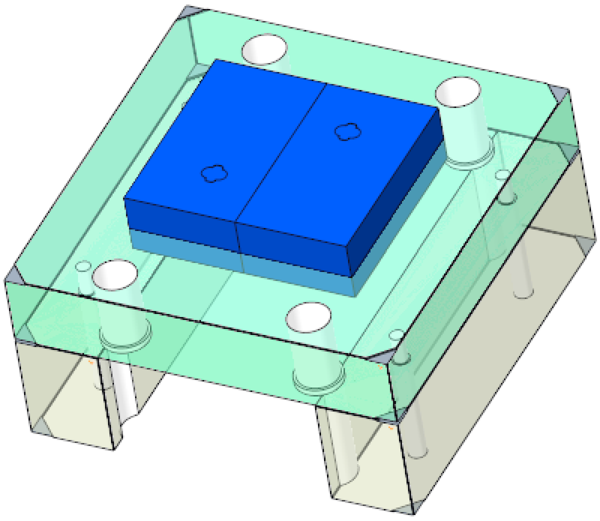
5. Reopen the **Main Guide 1** pattern dialog box
6. Modify the **Y** entry for **Pattern Size** to `200`.
7. Recalculate the pattern members with the  button.

   The preview in **Mold Base Definition** shows the new position automatically without regeneration of the 3D model.

   NOTE:
   Hidden members will become visible again when recalculate the pattern!

8. Close the dialog with **OK**.

   The modification will be applied to the mold base assembly model.



9. Reopen the **Main Guide 1** pattern dialog box
10. Reset the **Y** entry for **Pattern Size** to `324`.
11. Click again.
12. Close the dialog with **OK**.

    The pattern is set back to the default values.

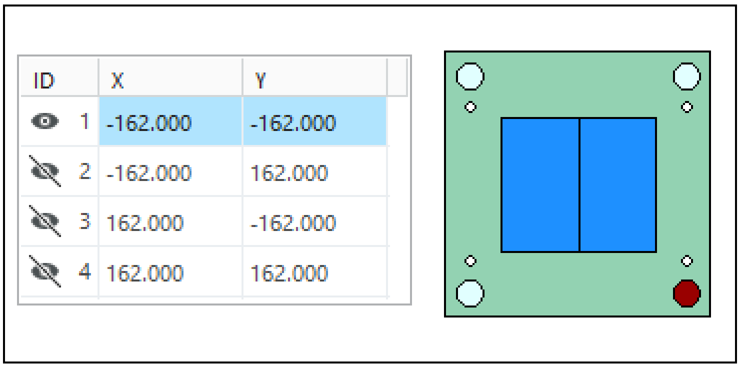
    NOTE:
    When changing the mold base **Size**, all default patterns are reset to the values from catalog. Only patterns that are not explicitly defined by the plate supplier (i.e. ejector guide patterns for Meusburger) will remain to the values you have specified.

8.14.1 Using 1+3 patterns

A common technique to avoid assembling mistakes of the mold base stack is to assemble leader pins and bushings with different inner diameter at one pattern instance.

1. Reopen the **Main Guide 1** pattern dialog box
2. Click 1+3.

   Only the pattern leader of **Main Guide 1** remains visible. In addition the pattern **Main Guide 2** is added with same Quantity and Pattern Size. In Main Guide 2 only the first position is suppressed. The different patterns are represented in **Mold Base Definition** by different colors.



3. Close the **Main Guide 1** dialog box with **OK**.

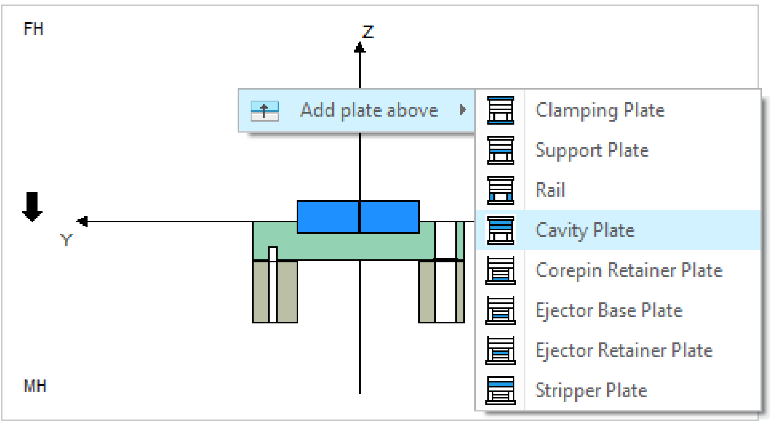
   The suppressed pattern `MAIN_GUIDE2` is resumed in the skeleton model, position and amount of axis match to the definition you have made.

8.15 To Finish the Plate Stack

Download 16_To Finish the Plate Stack to start from with this chapter.

Add a cavity plate to the fixed half.

1. Open the **Mold Base Definition** dialog box.
2. Click with right-mouse button within the fixed half of the side view and click **Add plate above ▸ Cavity Plate**



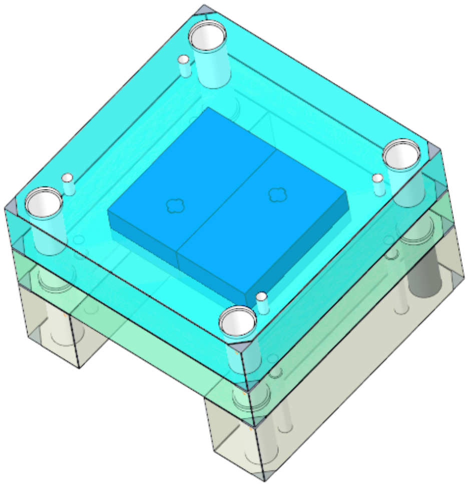
   The **Plate** dialog box opens.

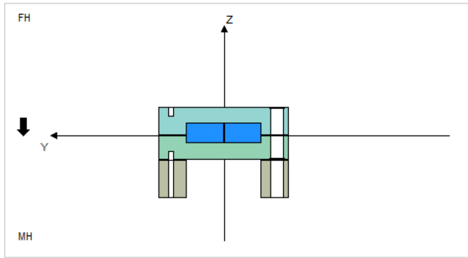
3. Select subassembly `TUTOR_FH`.

   NOTE:
   The subassembly designated with the function **Fixed Half** is automatically preselected.

4. Select type `F50`.
5. Select **Thickness** `86`.
6. Close the **Plate** dialog box with **OK**.

The cavity plate is assembled.

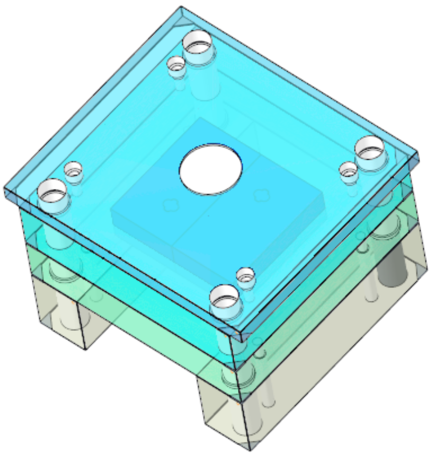


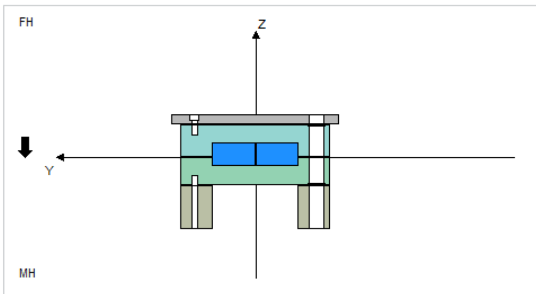


NOTE:
The cavity insert interferes with the cavity plates as the required cutout is still missing.

Add a clamping plate to the fixed half.

1. Click with right-mouse button on the cavity plate within the fixed half of the side view and click **Add plate above** ► **Clamping Plate**
2. Select `TUTOR_FH` as **Subassembly**.
3. Click **OK** to leave the **Plate** dialog box.

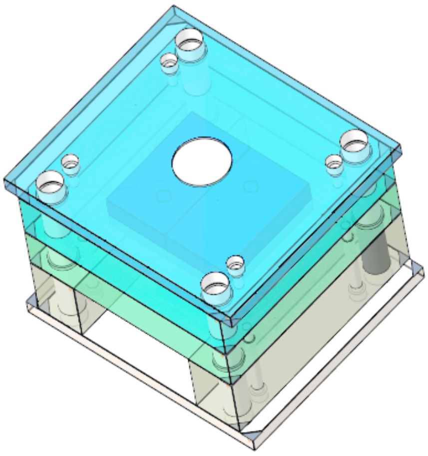


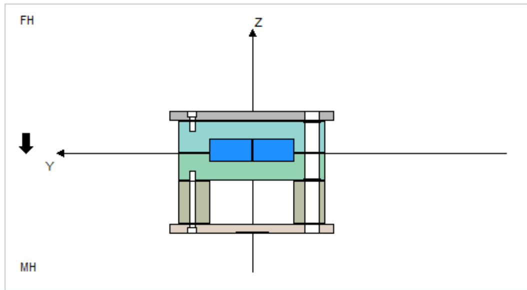


Add a clamping plate to the moving half.

1. Click with right-mouse button on the rail within the moving half of the side view and click **Add plate below ► Clamping Plate**
2. Select `TUTOR_MH` as **Subassembly**.

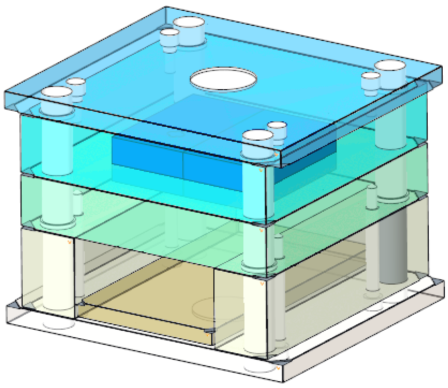
   NOTE:
   The subassembly designated with the function **Moving Half** is automatically preselected.
3. Click **OK** to leave the **Plate** dialog box.

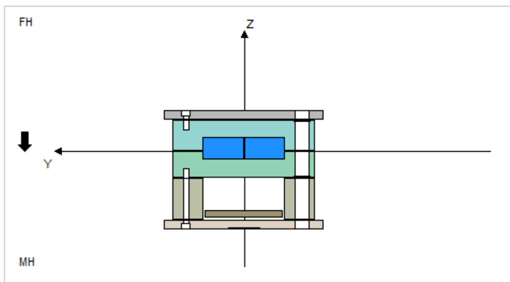




Add a core pin retainer plate.

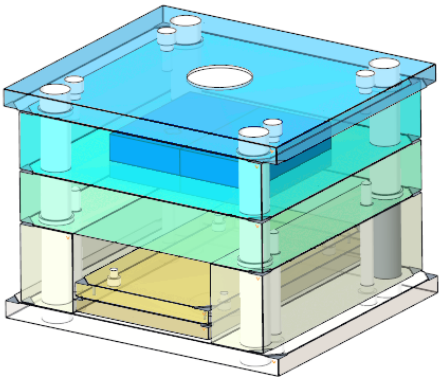
1. Click with right-mouse button on the clamping plate within the moving half of the side view and click **Add plate above ► Corepin Retainer Plate**
2. Select `TUTOR_MH` as **Subassembly**.
3. Set **Reference Distance** to `4` mm to leave some space for the stop system.
4. Close the dialog with **OK**.

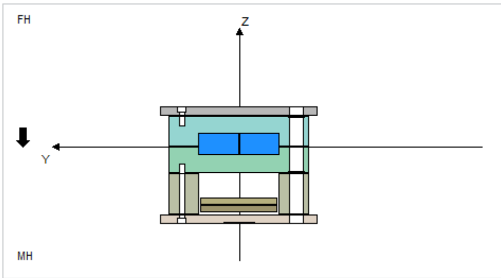




Add an ejector base plate.

1. Click with right-mouse button on the core pin retainer plate within the moving half of the side view and click **Add plate above ► Ejector Base Plate**
2. Select `TUTOR_MH` as **Subassembly**.
3. Set **Reference Distance** to `0` mm.
4. Close the dialog with **OK**.

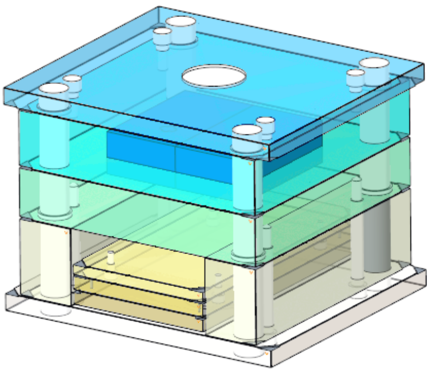


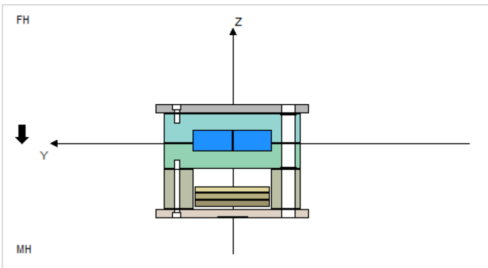


Add an ejector retainer plate.

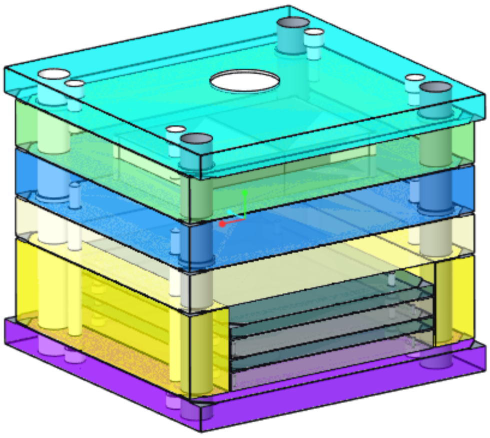
1. Click with right-mouse button on the ejector base plate within the moving half of the side view and click **Add plate above ► Ejector Retainer Plate**
2. Select the **Ejector Base Plate** as placement reference in the **Side View**.
3. Select `TUTOR_MH` as **Subassembly**

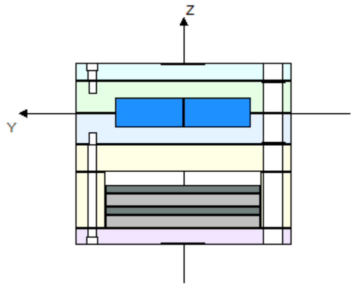
   Close the dialog with **OK**.





NOTE:
In case a second ejector package is needed repeat the assembly process for another ejector base plate and another ejector retainer plate. For the second ejector base plate selected the first ejector retainer plate as a reference in the side view of the **Mold Base Definition** dialog box.

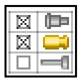
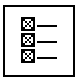
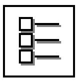




In some design cases it might be necessary to add ejector packages to the fix half. Additional two ejector packages are supported in the **Mold Base Definition** dialog box.

8.16 About Assembling and Disassembling Components

You can assemble and disassemble all components except those that create cuts in a Cut by quilt feature.

1. To assemble or disassemble a component, click **EMX** ▶  **Component Status**. The **Component Status** dialog box opens.
2. Select or clear the part check boxes. This feature is not available for function units (slide or lift assemblies) or for library components.
3. Click  or  to select or clear all available check boxes.
4. Click **OK** to update the assembly and close the dialog box or click **Cancel** to exit the operation.

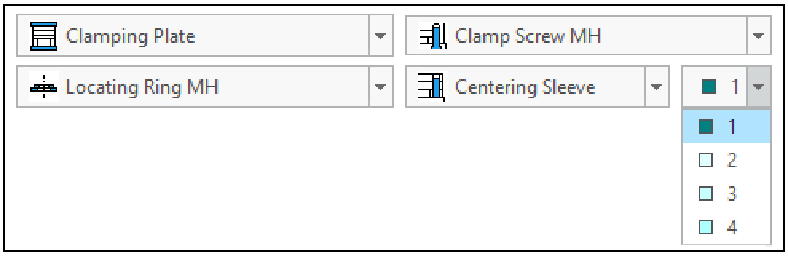
8.17 To Assemble Main Guide Components

Download 17_To Assemble Main Guide Components to start from with this chapter.

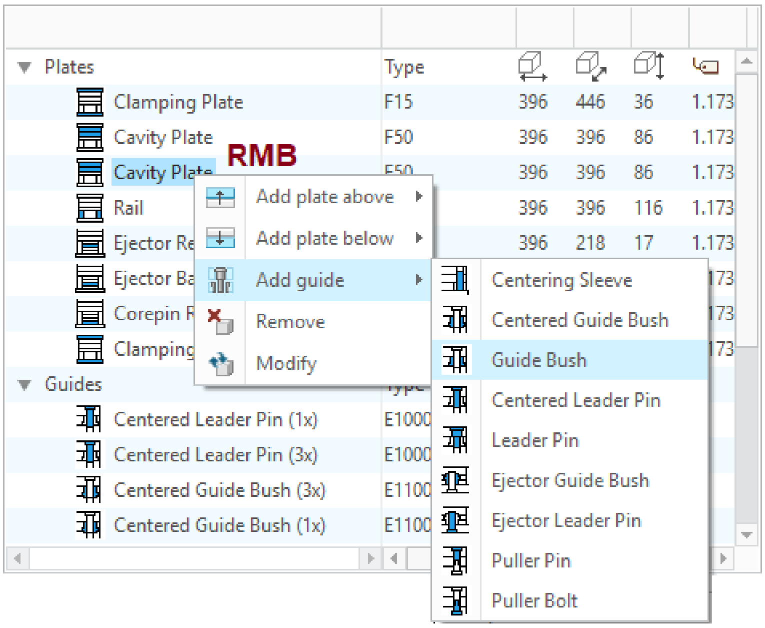
8.17.1 Add Centered Guide Bush to first Guide Pattern Scheme

Before adding a guide component, specify the guide-pattern scheme to be used as assembly reference.

1. Open the **Mold Base Definition** dialog box.
2. Select **1** from the **Pattern Scheme** pull down in the right bottom corner of the dialog.



3. Click with the right-mouse button on the moving half cavity plate in the summary tree and click **Add guide ▸** **Centered Guide Bush** .



The **Guides** dialog box appears.

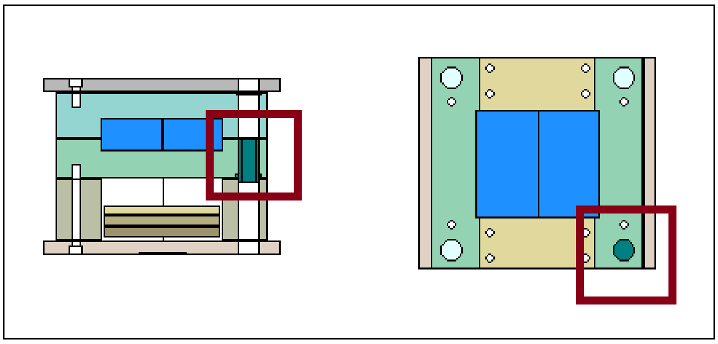
NOTE:
The nominal dimensions **DM1**, **DM2**, **LG** and **OFFSET** are set automatically using the Guide information of the plate cutouts and the thickness of the chosen plate.

4. Make sure the `TUTOR_MH` subassembly is selected before you continue.

NOTE:
As the counter bore for this centered guide bush already exist in the cavity plate, the cutouts are not created and the dimensions in cutout section of the **Component** dialog box in the bottom part of the dialog box are not applied to the cutouts. The dimensions are only used, in case a guide bush is placed on in a P-plate without predefined cuts. Then the counter bore displayed with its dimensions **D2**, **T2** and **D3** would be added. See chapter "To Understand Whether Cutouts in Plates are Driven by Component or Plate on page 68" for details.

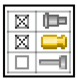
5. Close the dialog box with **OK**.

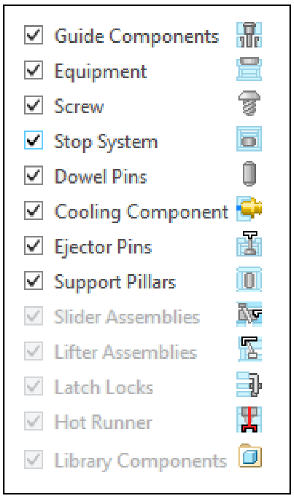
   The guide bush is displayed in the same color like the pattern in the top view. So you can see which components are related to which pattern position.



8.17.2 Set the Assembly Status of Components

There is no solid guide component to the assembly `TUTOR_FH`assembled yet.

1. Close the **Mold Base Definition** dialog box
2. Click **EMX Assembly ► Mold Base ►**  **Component Status**
3. Check all component types with .

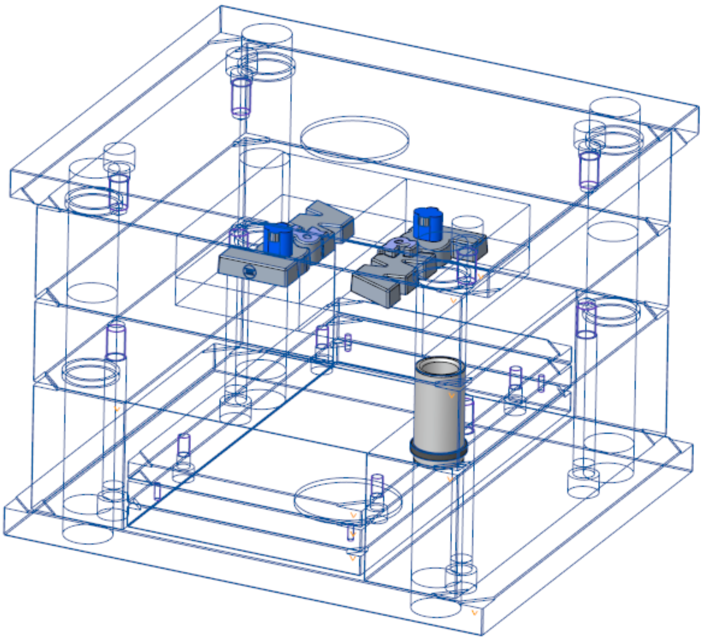


4. Close the dialog box with **OK**.

From now on all solid component model will be assembled directly.

5. Click **EMX Assembly ► View ►** **Wireframe Style**

   The centered guide bush component is now assembled.



8.17.3 Add Additional Centered Guide Bushes to second Guide Pattern Scheme

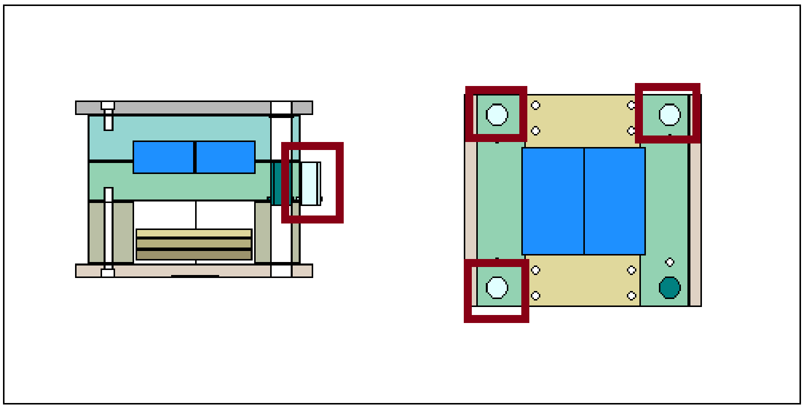
Add additional bushes with a different inner diameter on the second axis pattern.

1. Open the **Mold Base Definition** dialog box.
2. Select **2** from the **Pattern Scheme** pull down in the right bottom corner of the dialog.
3. Click again with the right-mouse button on the moving half cavity plate in the summary tree and click **Add guide ►** **Centered Guide Bush**.

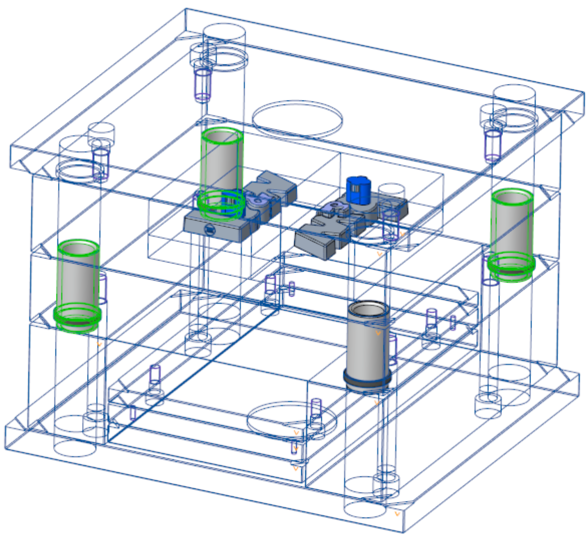
   The **Guide** dialog box opens.

   The second inner diameter is set to 32 mm for **DM2** automatically.
4. Click **OK** to assemble the guide components.

The bush on the second pattern is displayed beside the bush on first pattern in the side view. The color is equals to the color of the second pattern.



Three additional guide bushes are assembled to in the mold.



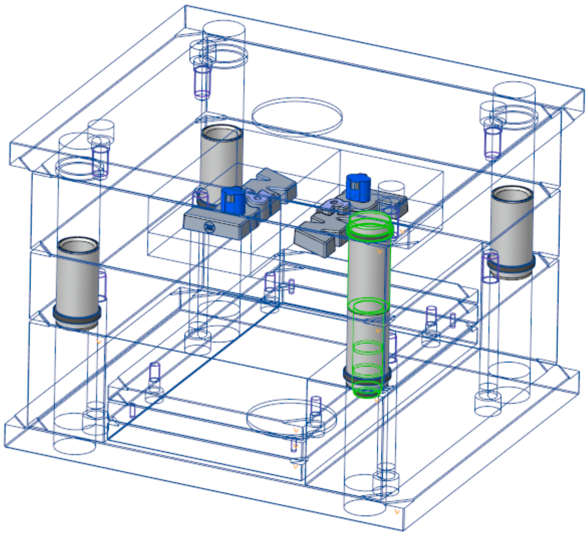
8.17.4 Add Leader Pins

1. Select **1** from the **Pattern Scheme** pull down in the right bottom corner of the dialog.
2. Click with the right-mouse button on the fixed half cavity plate in the summary tree and click **Add guide** ► **Centered Leader Pin**.

   The **Guide** dialog box opens.

   `TUTOR_FH` is already preselected as the target subassembly.

   The inner diameter **DM2** must be set to `30` mm to fit to the according bush. this is already preselected.
3. Select `95` mm for **LG2**
4. Click **OK** to leave the **Guides** dialog box.



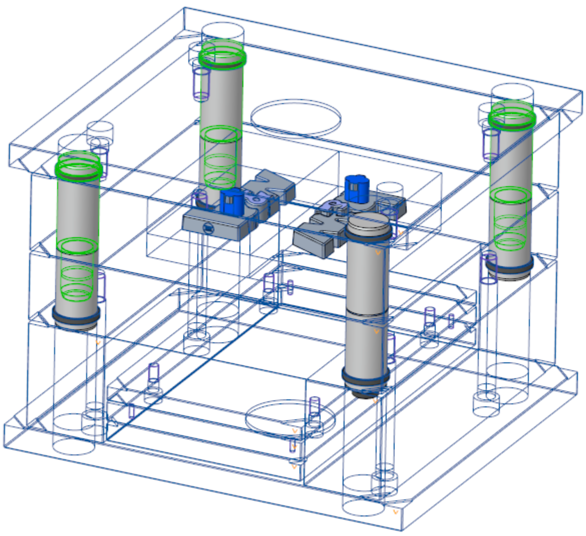
5. Select ▢ **2** from the **Pattern Scheme** pull down in the right bottom corner of the dialog.
6. Select the **Cavity Plate MH** with LMB in the **Side View** as placement reference.

   Make sure `TUTOR_MH` is the target **subassembly**. **DM2** must be `32` mm now.
7. Click with the right-mouse button on the fixed half cavity plate in the summary tree and click **Add guide** ▸ **Centered Leader Pin**.

   The **Guide** dialog box opens.

   `TUTOR_FH` is already preselected as the target subassembly.

   The inner diameter **DM2** must be set to `32` mm to fit to the according bush. This is already preselected.
8. Select `95` mm for **LG2**
9. Click **OK** to leave the **Guide** dialog box.

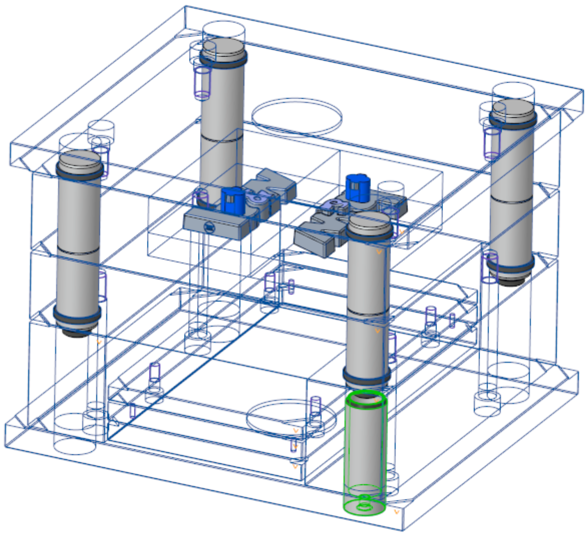


8.17.5 Add Centering Sleeves

1. Select **1** from the **Pattern Scheme** pull down in the right bottom corner of the dialog.
2. Click with the right-mouse button on the moving half clamping plate in the summary tree and click **Add guide ▸** **Centering sleeve**.

   The **Guide** dialog box opens.

   `TUTOR_MH` is already preselected as the target subassembly.
3. Set the **LG** value to `120`.
4. Click **OK** to leave the **Guide** dialog box.

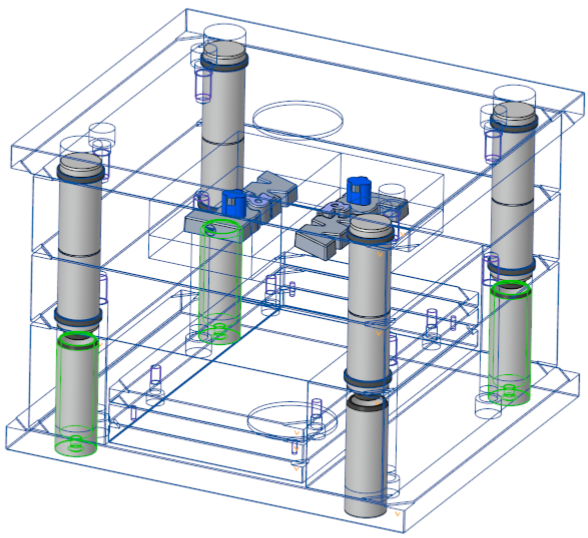


5. Select **2** from the **Pattern Scheme** pull down in the right bottom corner of the dialog.
6. Click with the right-mouse button on the moving half clamping plate in the summary tree and click **Add guide ▸** **Centering sleeve**.

   The **Guide** dialog box opens.

   `TUTOR_MH` is already preselected as the target subassembly.

7. Set the **LG** value to `120`.
8. Click **OK** to leave the **Guide** dialog box.



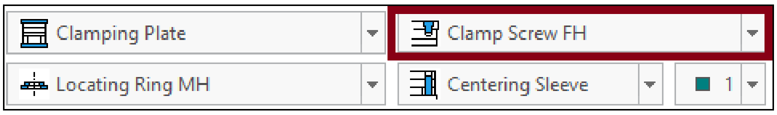
8.18 To Assemble Clamping Screws

Download 18_To Assemble Clamping Screws to start from with this chapter.

8.18.1 Add Clamping Screws FH

To fixture all plate clamping have to be added to the mold.

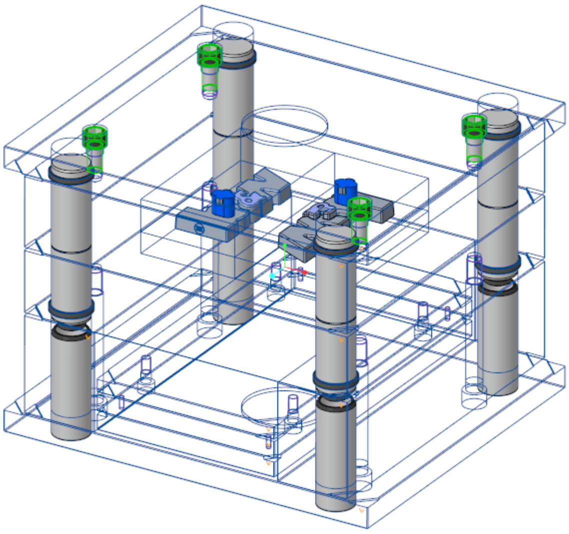
1. Open the **Mold Base Definition** dialog box.
2. Select **Clamp Screw FH** from the **Screw** pull-down to add the clamping screws to the fixed half.



The **Screw** dialog box appears.

Screw dimensions like **DM1** and **LG1** are automatically predefined based on the catalog defaults from the supplier catalog. The **OFFSET** displayed is the default counter bore depth in the clamping plate.

3. Click **OK** to leave the **Screw** dialog box.

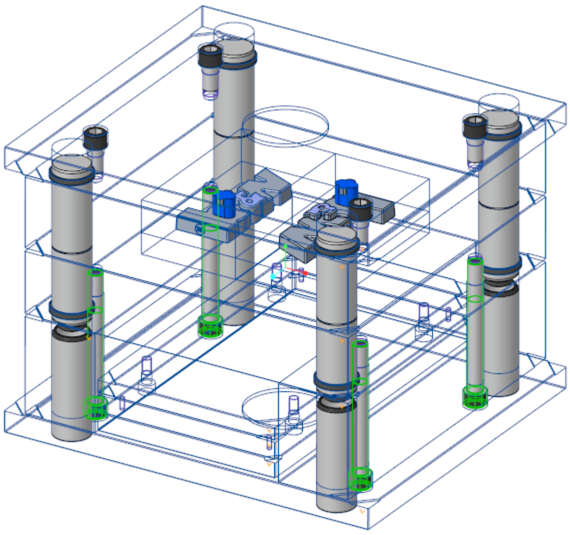


8.18.2 Add Clamping Screws MH

1. Select **Clamp Screw MH** from the **Screw** pull-down to repeat the same steps for the moving half.

   The **Screw** dialog box appears with predefined settings for the screw type, diameter and length.

2. Click **OK** to leave the **Screw** dialog box.



8.18.3 Add Ejector Clamping Screws

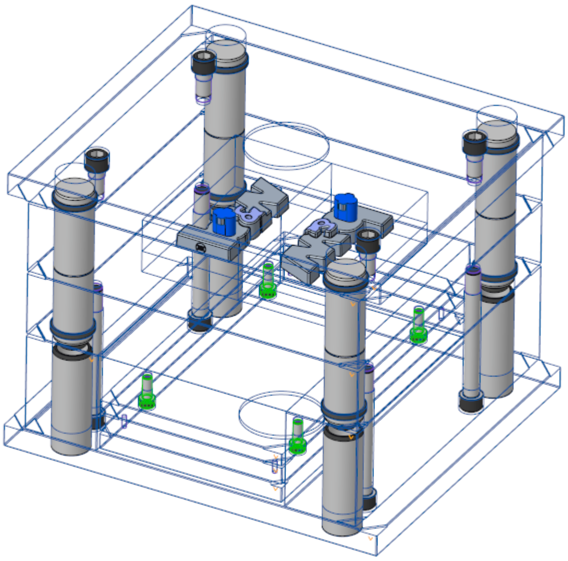
The ejector plate package needs to be fixed as well

1. Select **Ejector Screw MH** from the **Screw** pull-down to the ejector plates.

   The **Screw** dialog box appears.

   `TUTOR_MH` is preselected as the target subassembly.

2. Click **OK** to leave the **Screw** dialog box.



All Clamping Screws are now assembled.

8.19 To Assemble Insulation Plates

Download 19_To Assemble Insulation Plates to start from with this chapter.

8.19.1 Add Insulation Plates to the Moving Half

In some case it is necessary to add thermal insulation plates to the mold base. This avoids thermal interaction between the molding machine and the mold base assembly.

1. Select **Thermal Insulation Plate MH** in the **Equipment** pull-down.

   The **Thermal Insulation Plate** dialog box appears.

   The size of the according clamping plate was measured and the proper insulation plate size is preselected.

   `TUTOR_MH` is preselected as the target **subassembly**.

2. Enable the option **Include leader pin cuts**.
3. Click **OK** to leave the **Thermal Insulation Plate** dialog box.

NOTE:
The option **Include leader pin cuts** handles the visibility of additional bores following the dims of the clamping plate cutouts.

8.19.2 Add Insulation Plates to the Fix Half

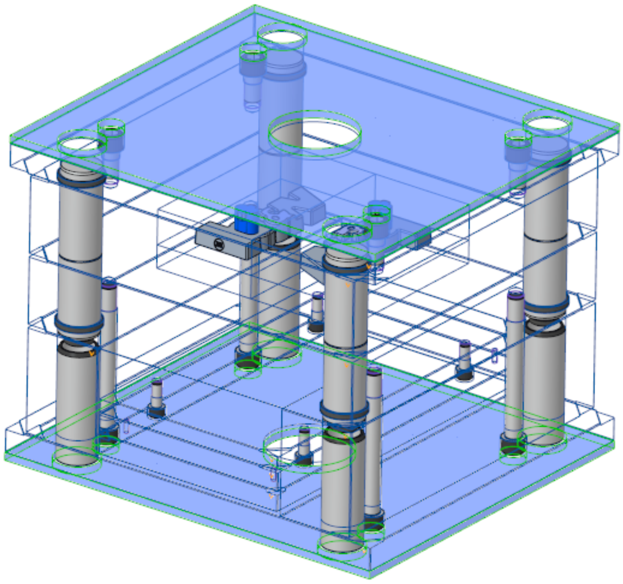
Repeat the same steps for the fixed half.

1. Click **Thermal Insulation Plate FH** from the **Equipment** pull-down list.

The **Thermal Insulation Plate** dialog box appears.

`TUTOR_FH` is preselected as the target **subassembly**.
Make sure `TUTOR_FH` is the target **Subassembly**.

2. Enable the option **Include leader pin cuts**.
3. Click **OK** to leave the **Thermal Insulation Plate** dialog box.



8.20 To Modify Clamping Plates and Adjust Components

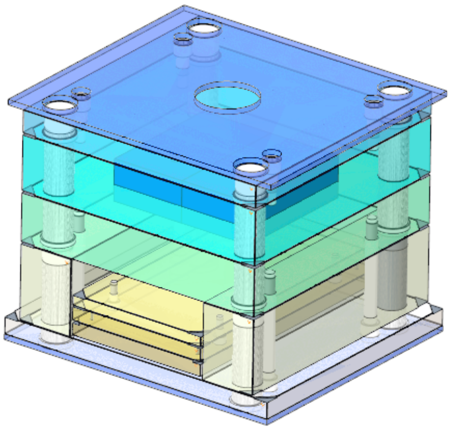
Download 20_To Modify Clamping Plates and Adjust Component to start from with this chapter.

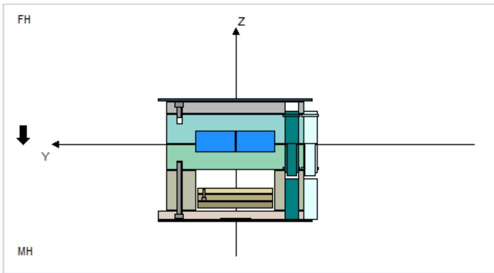
8.20.1 Modify Clamping Plates

If a invalid plate was assembled to the mold base it's type can easily be changed.

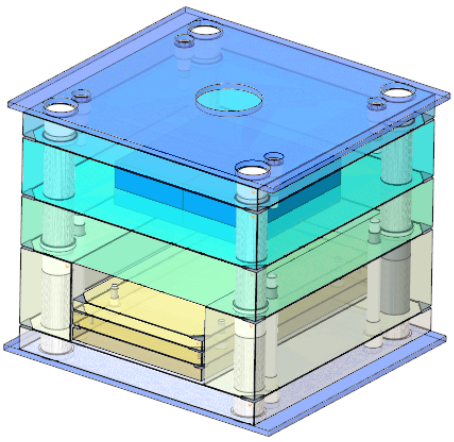
1. Open the **Mold Base Definition** dialog box.
2. Click with right-mouse button on the fixed half clamping plate in the side view and click **Modify**.

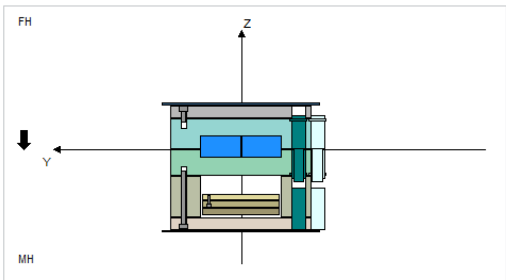
   The **Plate** dialog box appears.
3. Switch the **Type** to `F45`.
4. Select a **Thickness** of `36` mm.
5. Click **OK** to leave the dialog box.





6. Repeat the same procedure for the **Clamping Plate MH**





8.20.2 Adjust Clamping Screws due to new Plate Thickness

Due to new the new thickness of the clamping plates the screw need to be adapted manually.

1. Double-click on the **Clamping Screw FH** either in the side view or in the summary tree.

The **Screw** dialog box opens.

2. Set the length **LG1** to `40` mm.
3. Close the dialog box with **OK**.

Repeat the same steps for the moving half clamping screw.

1. Double-click on the **Clamping Screw MH** either in the side view or in the summary tree.

   The **Screw** dialog box opens.

2. Set the length **LG1** to `160` mm.
3. Close the dialog box with **OK**.

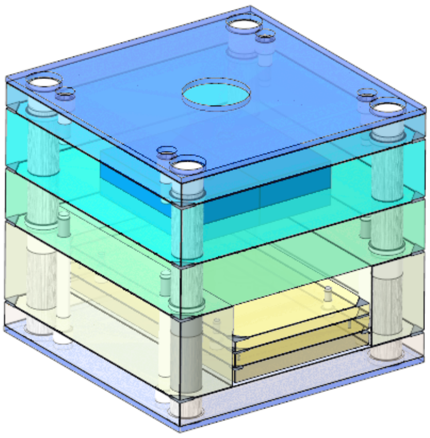
8.20.3 Adjust the Insulation Plate Size

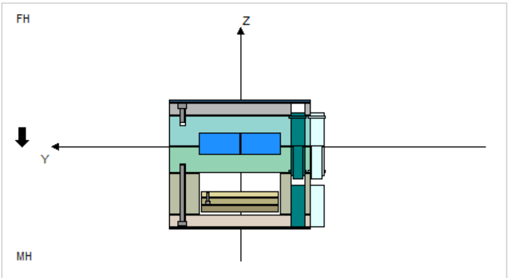
The insulation plates need to be changed as well.

1. Double-click on the **Thermal Insulation Plate FH** either in the side view or the summary tree

   The **Thermal Insulation Plate** dialog box appears.

2. Change the **INSULATION_ES_WIDTH** value to `396` that it fits to the clamping plate.
3. Close the dialog box with **OK**.
4. Repeat the same procedure for the **Thermal Insulation Plate MH**.





NOTE:
By default changes to the related components need to be done manually. Still the are configuration options available which can automate the update procedure. See chapter About Modifying the Mold Base Size on page 334 as a reference.

8.21 To Assemble Ejector Guide Bushings and Pins

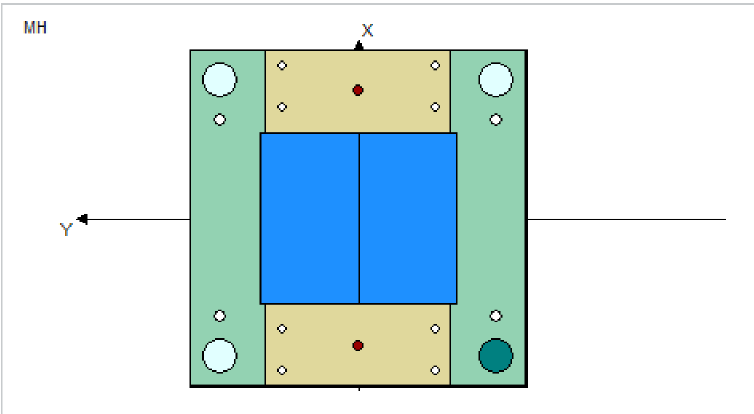
Download 21_To Assemble Ejector Guide Bushings and Pins to start from with this chapter.

8.21.1 Set up the Ejector Guide Pattern

1. Open the **Mold Base Definition** dialog box.
2. Click **Top** to switch to the top view.
3. Double–click the **Ejector Guide MH** either in the top view or in the summary tree.

   The **Ejector Guide MH** dialog box opens.
4. Enter **Quantity** `2` in both **X** direction.
5. Enter **Quantity** `1` in both **Y** direction.
6. Enter a value of `280` in **X** direction.
7. Click **Recalculate** to update the instances.

ID	X	Y
1	-150.000	-0.000
2	150.000	-0.000



8. Close the **Ejector Guide MH** dialog box with **OK**.

8.21.2 Add Ejector Guide Bushings

1. Click **Front** to switch to the side view.
2. Click with right-mouse button on the **Ejector Base Plate** either in the side view or in the summary tree. Click **Add guide ▸** **Ejector Guide Bush**.

   The **Guides** dialog box opens.

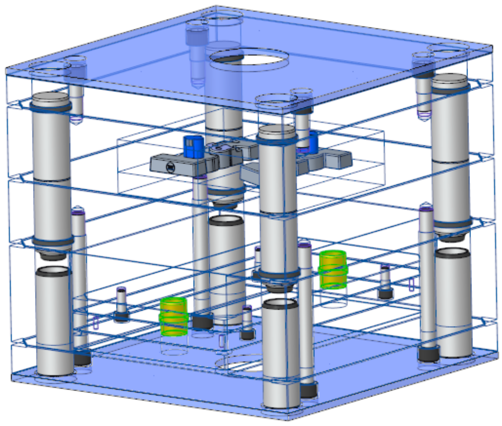
   The `TUTOR_MH` subassembly is already preselected .
3. Set the **DM2** value to `24`.
4. Set the **LG1** value to `22`.
5. Set the **LG2** value to `17`.

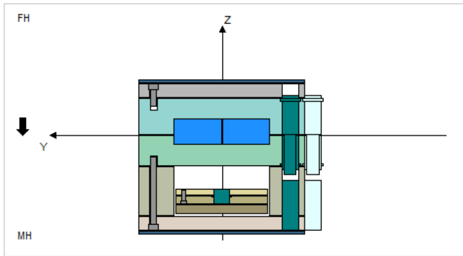
   NOTE:
   In the summary tree all plate thicknesses are shown. This helps you to choose the right sizes for your components.
6. Make sure that the Option **Toggle Direction** is enabled.

This option specifies wether the bush shoulder is located in the ejector base plate or the ejector retainer plate.

7. Close the **Guides** dialog box with **OK** and answer the message box with **OK**.

   The ejector guide bushes are assembled to the mold base assembly.

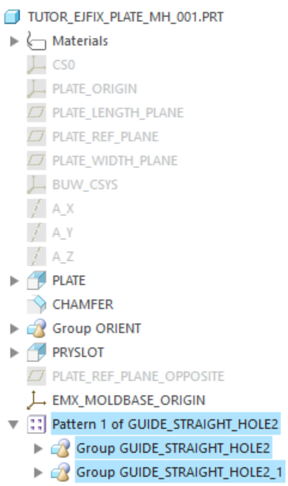


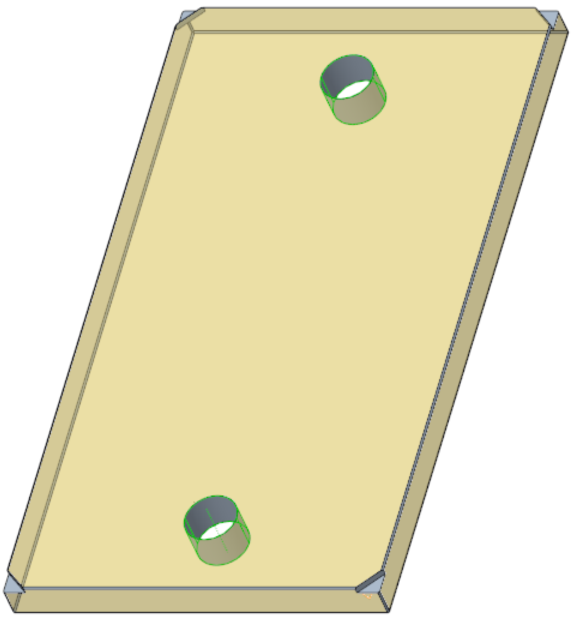


8. Close the **Mold Base Definition** with **Close**.

8.21.3 Remove Cutouts from Ejector Core Pin

1. Open the part `TUTOR_EJFIX_PLATE_MH_001.PRT`
2. Delete the group `GUIDE_STRAIGHT_HOLE`



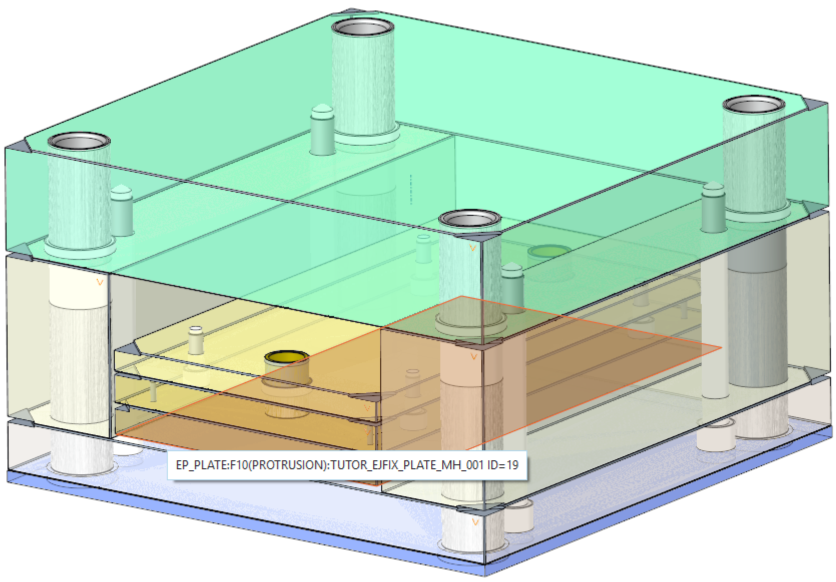


NOTE:
All cutout UDFs created with Creo Expert Moldbase Extension can be deleted with default Creo Parametric functionality if required.

8.21.4 Add Ejector Guide Bush with Circlip to the Core pin Plate

1. Open the `TUTOR_MH` subassembly.
2. Select **EMX Components ► Components ► Guide Component**.

   The **Guides** dialog box opens.
3. Select `TUTOR_MH` as target subassembly from the **subassembly** pull down list.
4. Select `E1110_` from the **types** pull down list.
5. Set the **DM1** value to `30`.
6. Set the **DM2** value to `24`.
7. Set the **LG** value to `22`.
8. As the first assembly reference **(1) Axis|Point** select the datum axis `EJP_GUIDE` from the pattern inside the `TUTOR_SKELETON.PRT` Model.
9. As the second assembly reference **(2) Surface** select the bottom side of the core pin plate.

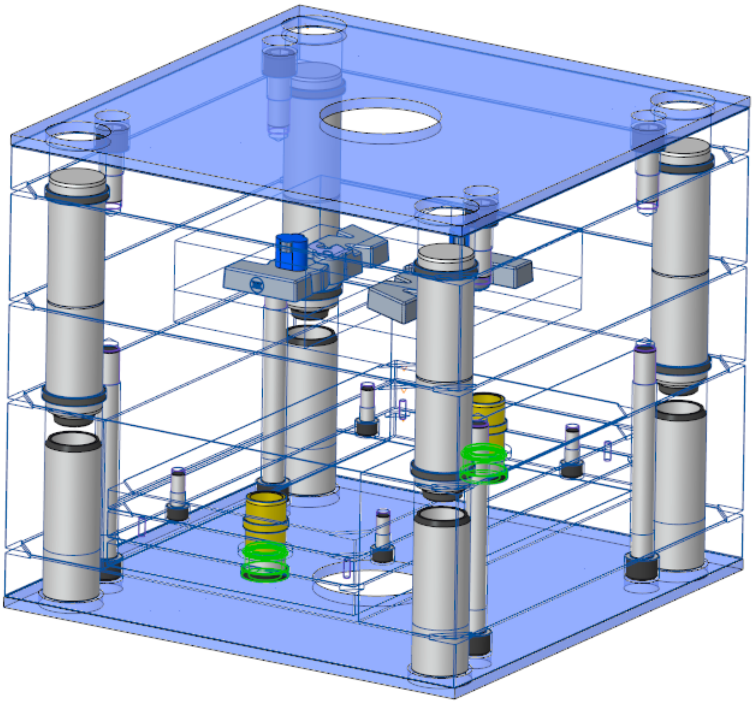


10. Enable the option **One Plate**.

    A different UDF Images appears in the cut out section. Circlip and shoulder cutouts are added to the same plate.
11. Switch to the **Options** tab and check the option**Pattern for all instances** to create a guide component for each axis inside the pattern.
12. Close the **Guide** dialog box with **OK**.

The guide bushed and the according cutouts for the core pin plate are assembled to the moving half assembly.

13. Close the `TUTOR_MH.ASM` and switch back to the main assembly.



NOTE:
When placing the combination of leader pins and guide bushes, you must always place the bush first. If you start with the leader pins, this would create the smaller cutouts. Then the required bush cutouts would miss as the plates to be cut can not be identified.

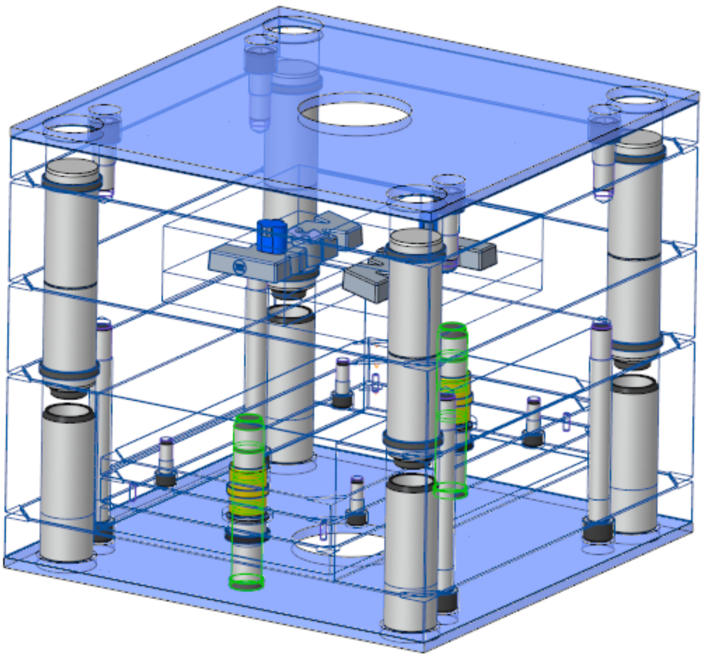
8.21.5 Add Ejector guide Pins

1. Open the **Mold Base Definition** dialog box.
2. Click with right-mouse button on the **Clamping Plate MH** either in the side view or in the summary tree. Click **Add guide ▸** **Ejector Leader Pin**.

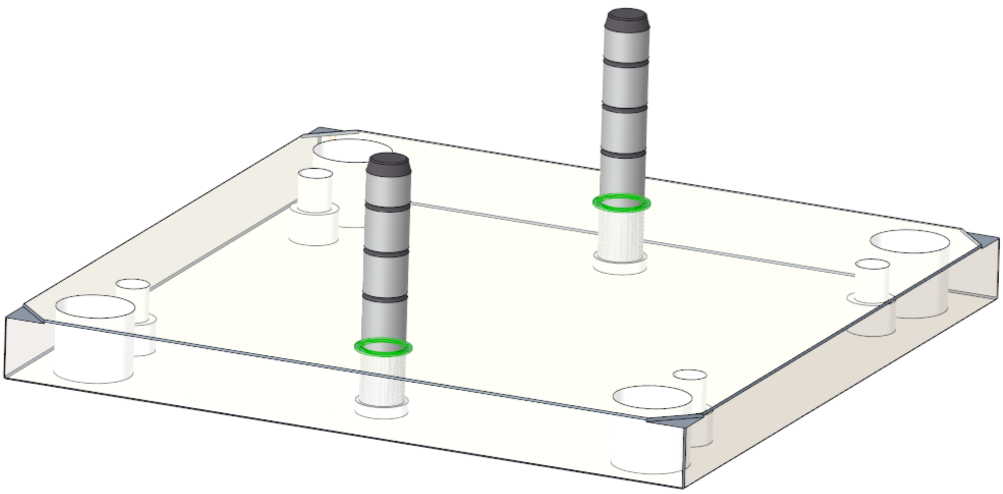
   The **Guides** dialog box shows up.

   The `TUTOR_MH` subassembly is already preselected .
3. Set the **DM1** value to `24`.
4. Set the **LG** value to `160`.
5. Close the **Guide** dialog box with **OK** and answer message box with **OK**.
6. Close the **Mold Base Definition** with **Close**.

The leader pins are assembly to the mold base.



To fix the leader pins, circlips can be used. The image bellow shows a design example.



8.22 To Assemble the Return Pin / Back Pin

Download 22_To Assemble the Return Pin Back Pin to start from with this chapter.

8.22.1 Define the Return Pin / Back Pin Pattern

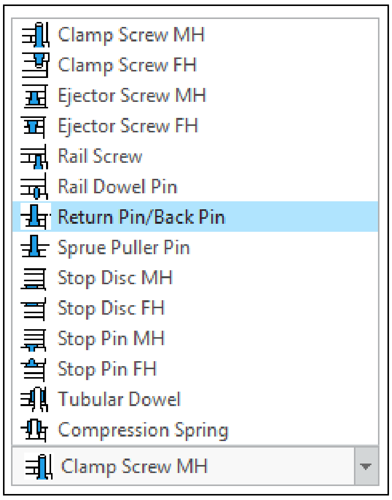
1. Open **Mold Base Definition** dialog box.
2. Click **Top** to switch to the top view.
3. Double-click on **Return Pin/Back Pin** in the summary tree.

   The **Return Pin/Back Pin** pattern dialog box opens.

   For supplier `meusburger` no default pattern information is defined in the catalog.
4. Set **X** and **Y Quantity** to `2`.
5. Enter **Pattern Size** `358` and `180`.
6. Click to update the number of instances.
7. Click **OK** to close the pattern dialog.

8.22.2 Assemble the Return Pin / Back Pin

1. Click **Front** to switch to the side view.
2. Select **Return Pin / Back Pin** from the **Screw** pull-down.

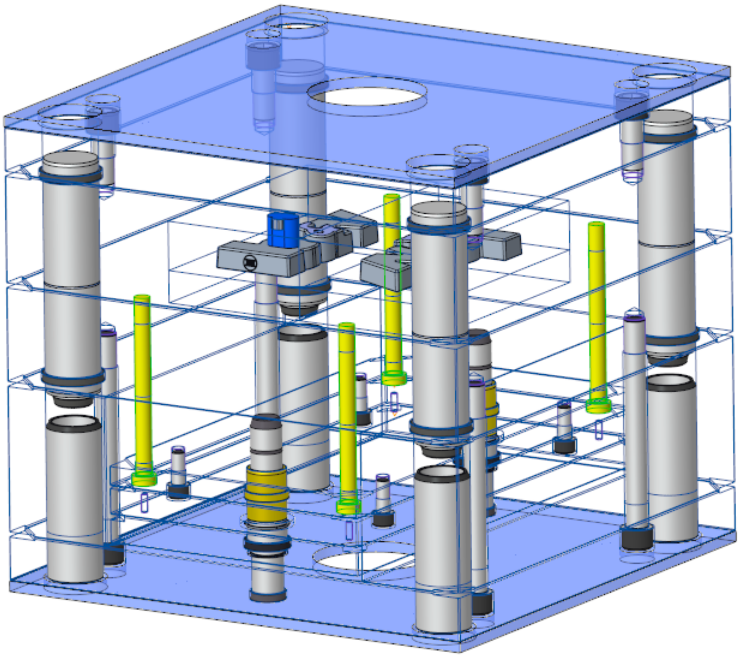


   The **Ejector Pin** dialog box appears.

   `TUTOR_MH` is preselected as target **subassembly**
3. Select a cylindrical ejector pin of the **Type** `E1710`
4. Set the **DM1** value to `12`

   References and the length are preselected automatically. The measured offset **REF1** is the distance between the splitting plane and the ejector retainer plate.
5. Leave the dialog box with **OK**.

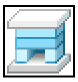
The return pins / back pins are assembled to the mold.



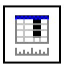
8.23 To Assemble Side Interlocks

Download 23_To Assemble Side Interlocks to start from with this chapter.

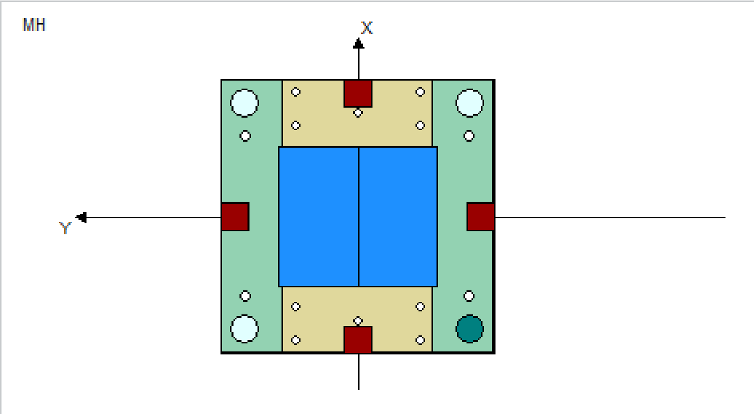
8.23.1 Modify the Side Interlock Pattern

1. Open **Mold Base Definition** dialog 
2. Click  **Top** to switch to the top view.
3. Double-click on **Side Interlock** in the summary tree.

   The **Side Interlock** pattern dialog box opens.

   The **Pattern size** values in the **Side interlock** dialog box are set to the current mold base size by default.
4. Enter **Quantity** to 2 in both X- and Y-direction.
5. Select  to update the number of instances.

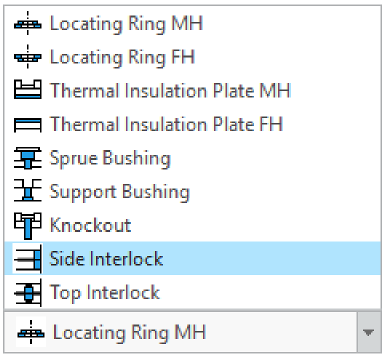
6. Check the placement of the Side Interlocks in the **Top View** of the **Mold Base Definition** dialog box.



7. Click **OK** to close the dialog and regenerate the pattern in the skeleton model.

8.23.2 Define the Side Interlock Properties

1. Select **Side Interlock** from the **Equipment** pull-down.

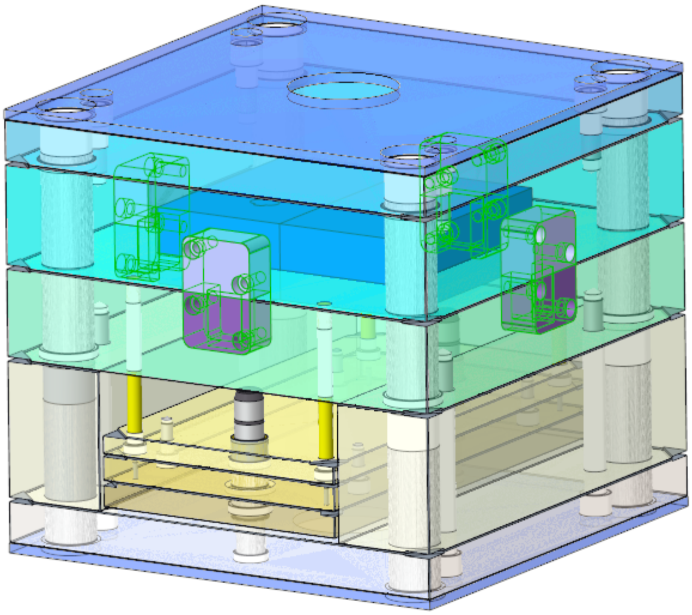


The **Side Interlock** dialog box opens.

2. Select the main assembly `TUTOR_SHARED` as target **subassembly**
3. Select `hasco` from the **supplier** pull down list.
4. Select `Z07_17` from the **types** pull down list.
5. Select a value of `46` mm for **B_**.
6. Do not change any other default settings in the **Side Interlock** dialog box.

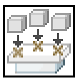
NOTE:
Side interlock components create their cutouts using a ‘Cut by Surface’ feature. In this case just one Overview image is displayed in the UDF area, the quilt cut has no further dimensions.

7. Click **OK** to close the **Side Interlock** dialog box.
8. Close the **Mold Base Definition** with **Close**.



8.23.3 Assemble Predefined Screws in Side Interlock Assembly

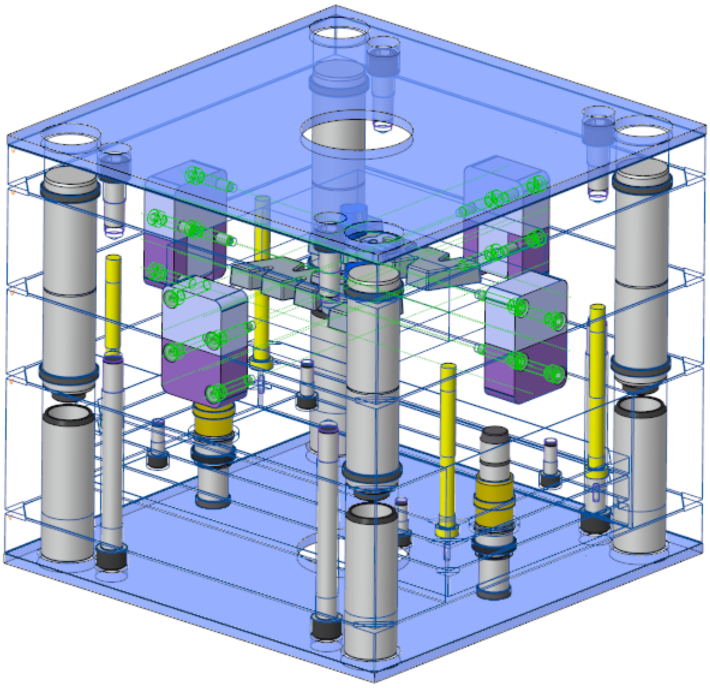
For the Hasco Z07/17 side interlock assembly screws are already predefined.

1. Click **EMX Components** ► **Component Handling** ►  **Assemble predefined components**.

   The Selection dialog appears.
2. Select the subassembly `TUTOR_SHARED.ASM`.

   All predefined components within this assembly are assembled.

   NOTE:
   The predefined components are search recursively. The screws are assembled to the next top subassembly which is found in the model tree.



3. Leave the selection dialog with middle-mouse button.

9

The Cavity Insert Assembly Cutouts

9.1 About Creating the Cavity Insert Assembly Cut-Outs

Use the **Cavity Cutout** dialog box to define cavity plate cutouts.

1. The **Cavity Cutouts** dialog box can be opened opened by clicking **EMX Assembly ▸** **Cavity Cutouts**.

   The insert assembly outline dimensions automatically appear in the **Cavity Size** boxes and preview window.

   The preview orientation is the same as that in the **Mold Base Definition** dialog box preview. Correct initial data appears only if the cavity insert assembly was assembled to the cavity layout pattern. When the values are changed, the old values continue to appear until the insert assembly outline is remeasured in the **Classify** dialog box.

   In the top area the **Cavity Size** (outline of the insert assembly) is shown. The size (**+,-X; +,-Y;+,-Z**) can be edited here if the outline does not fit your requirements.

2. Select cutout definitions:
   - No cutout — Removes existing cavity insert cuts.
   - One rectangular cavity cutout — Enter values for the x- or y- offset, radii or corner mouse ears, or both.
   - Counterbore — One cutout is created for each cavity insert assembly individuall
   - Rectangular cutout pattern — One cutout is created for each cavity insert assembly individually.

     NOTE:
     A rectangular cutout can have radii or corner mouse ears.
   - Workpiece outline cutout — Creates a solid surface copy in all workpiece components. The quilts are used to create Cut by Quilt features in models that interfere with the quilts.

3. Click **Preview** to see the model in 3D so you can decide whether or not to keep the changes you made.
4. Click **OK** to apply changes to the model or **Cancel** to restore the previous cutout shape. The dialog box closes.

9.2 To Create the Cavity Insert Assembly Cut-Out

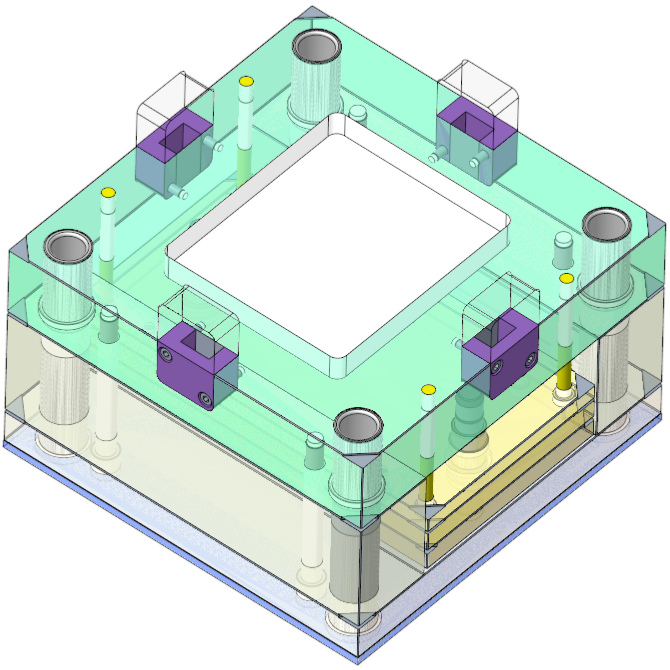
Download 24_To Create the Cavity Insert Assembly Cut-Out to start from with this chapter.

1. Click **EMX Assembly ► Mold Base ►** **Cavity Cutouts**.

   The **Cavity Cutout** dialog box opens.

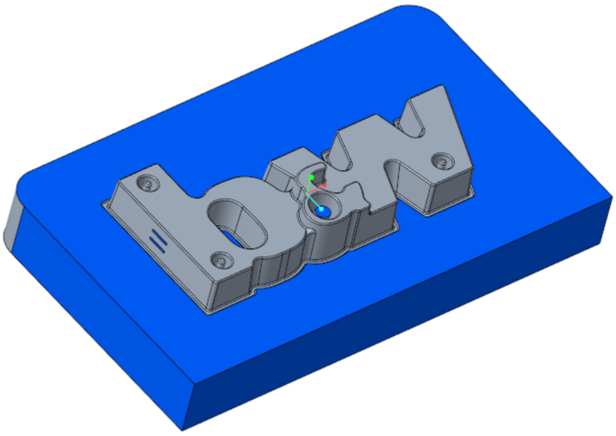
2. Select the cut-out type **Single rectangular** .
3. Set **Cut Radius** to `13` mm.
4. Click **Preview** to update the cutouts in the graphics window.
5. Click **OK**.

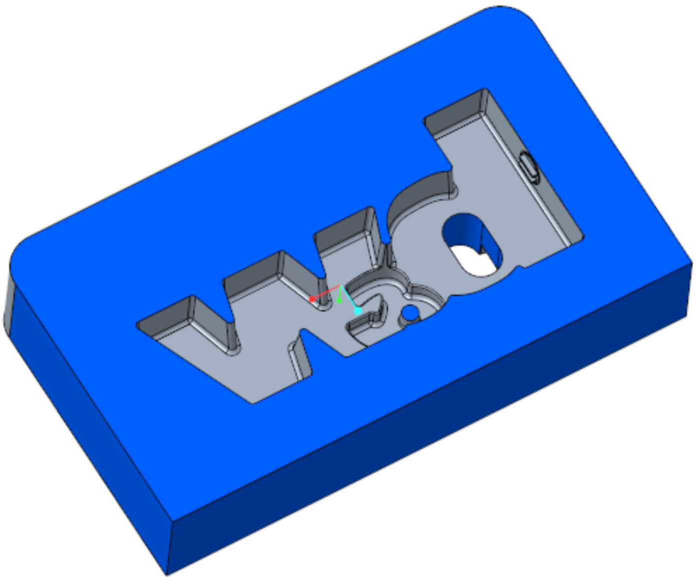
   The cutouts are added to both cavity plates.



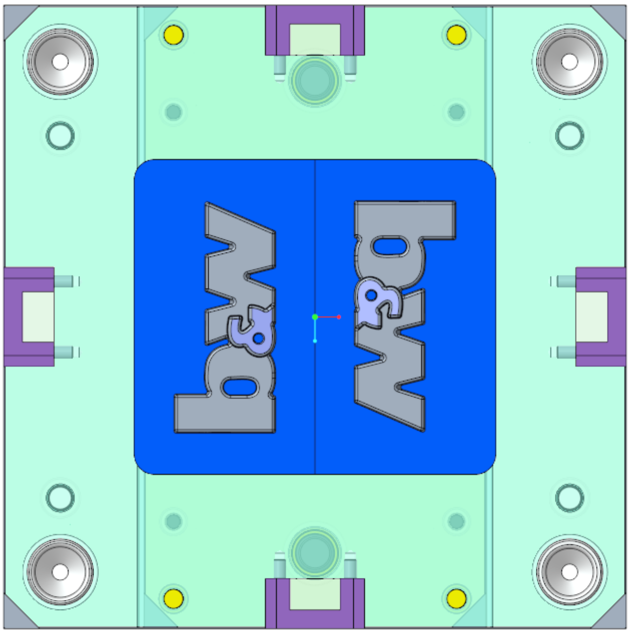
6. The Cavity plates and the insert parts still interfere:

   Open `MOLD_VOL_ES.PRT` and `MOLD_VOL_IS.PRT`.
7. Add round features of `13` mm to all the corners of the insert parts.





The inserts assembly do no interfere with the cavity plates



Notes:

10

About Component Handling

10.1 About Component Operations

There are five default operations for all component types:

- Define a component—From the **EMX Components** tab in the **Components** group, select one of the components as shown in the list below. A dialog box for the selected component opens.
  - Screw
  - Dowel Pin
  - Guide Component
  - Support Pillar
  - Stop disc
  - Stop pin
  - Equipment
  - Slider
  - Lifter
  - Latch Lock

  The component definition dialog box opens. Define the component properties, and click **OK** to close the dialog box. A reference group is created for the new component where component information is stored as a parameter of this reference group. All cutouts and the component itself are assembled as children of this reference group.

- Modify a component —Click **EMX Components** ▶ **Modify** or right-click the component in the graphics window or Model Tree and select **Modify**. Select an existing reference group. A component definition dialog box opens. Modify dimensions in the dialog box or in the graphics window.
- Delete a component —Click **EMX Components** and then in the **Component Handling** group, click **Remove**, or right-click the component in graphics window or Model Tree and select **Remove**. Click **OK** to delete the reference group including the component dependent geometry.

  NOTE:
  All other related components are also deleted.

  NOTE:
  When deleting a component, you can force EMX also to delete the model files from the current working directory.

  Set the configuration option `DELETE_FILES_AFTER_DELETE` to `YES`.

  Its even possible to remove the files when simply disassemble the component in Component Status dialog box with `DELETE_FILES_AFTER_DISASSEMBLE`.

  Use these options with care.

NOTE:
When deleting a component, all referring cut-outs will be removed without any warning. To highlight them before deleting them, set configuration option `PRE_HIGHLIGHT_BEFORE_DELETE` to `YES`.

- Reassemble an existing component with new references—Click **EMX Components** and then in the **Component Handling** group click **Assemble again**, or right-click the component in the graphics window or Model tree and select **Assemble again**. A dialog box in which you can define the component opens. Select this component type in the **Component Status** dialog box. Select new placement references to place the component and create the cutouts.
- Assemble an existing component as a copy with new references—Click **EMX Components** and then in the **Component Handling** group click **Assemble as a copy** or right-click the component in the Model Tree and select **Assemble as a copy** to create a copy of the existing component. The component definition dialog box opens. The settings of the existing component are displayed, but the component has a new unique ID. Select new placement references. A copy of the component is placed with new references and cutouts.

NOTE:
Fast placement of additional cooling components is possible if the configuration option `AUTO_REASSEMBLE` is set to `YES`.

After assembling an initial component the reference will be prompted instantly for the second instance in the message area.

After selecting the appropriate reference the next on will be prompted.

Continue selecting references to place further instances of t a component.

Stop the loop by Clicking **Cancel** in the **Selection** dialog box or by using the middle-mouse button.

With this option multiple components of the same type can be created quickly in a row.

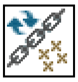
NOTE:
If this loop function should also be available for other type of components, set also the configuration option `AUTO_REASSEMBLE_ALL_TYPES`.

10.2 About Assembling Predefined Components

To assemble predefined EMX components, click **EMX Components** ▸ **Assemble predefined components**. Select an assembly or part that has reference groups with predefined components. The component information is read from the reference group and the assembly is assembled and cut.

You can predefine any component by selecting the **Predefine Component** check box during its definition.

10.3 About Redefining Placement Point References

1. Click **EMX Components ► Component Handling ►**  **Redefine point reference**.
2. Click any EMX reference point. Creo Expert Moldbase Extension opens the regular redefine dialog box for the component corresponding to the selected point.
3. Make modifications to the component and click **OK** to apply them. All components and cutouts placed on the selected point are updated.

11

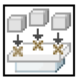
About Components

About Components

11.1 About Component Options

From the dialog box in which you confine a component, select or clear the following check boxes when you place a component:

- **Pattern for all models**—Adds the new component to each instance of the owner of the first placement reference.
- **Pattern for all instances**—Places the new component on each instance of a reference pattern of the first placement reference.
- **First Cut In Face Owner**— The cut out UDF defined on first position will be added to the owner model of the first surface selected as placement reference.
- **No Cutout**—No cuts are generated.
- **No Component**—The component is not assembled.
- **Predefine Component**—The component can be predefined in part mode, to be assembled later in the main assembly.

  NOTE:
  When the **Predefine Component** check box is selected, component data is saved with the reference group. Click **EMX Components** ▸  **Assemble predefined components** to assemble the component later.
- **Copy Drawings**: If a drawing template exist for the current component, you can let EMX copy this drawing template together with the part/assembly-file.

  NOTE:
  The button **Copy Drawings** is only checked automatically, if a drawing template exist and the component type is listed in the configuration `EMX_CHECK_DWG_TYPES`.

Background Information

With the **EMX Option** `EMX_NO_CUT_PARAM` you can specify an INTEGER-Parameter name. If this Parameter exist at a certain model, it will never get a quilt cut, even it interferes with the slider quilts.

The Option `CUT_PLATES_ONLY` also drives, which components will be cut. If the option is set to YES, only EMX plates get cuts.

If EMX assembles components with cut quilts as a real pattern, the **EMX Option** `PATTERN_CUT_QUILTS` set to YES will force EMX to create all cuts also as reference patterns. If the option is set to `NO`, the cuts are added individually for each instance of the component pattern. This is required if instances interfere with a different amount of models.

The last relevant option is `PROMPT_FOR_CUTTING_PARTS`. If this is set to YES, a dialog box is displayed with all interfering parts. So the user can drive for each part if the quilt cut is added or not.

11.2 About UDF Component Dimensions

When you double-click a UDF dimension, it is highlighted in the UDF preview window. All dimensions of all required UDFs for the current component appear in an overview list.

11.3 About the Component Detail Viewer

To activate the **Component Details** viewer, click in the definition dialog box for the component. The **Component Details** dialog box opens.

- Click to view the previous image.
- Click to view the next image.
- Click **Close** to close the dialog box.

11.4 About Editing a Subcomponent

Define assembly subcomponents or modify default Bill of Materials (BOM) values by selecting any of the subcomponents in the **Subcomponents** tab on the **Component Editor** dialog box.

1. Click **EMX Assembly ► Administrator Tools ► Component Editor**. The **Component Editor** dialog box opens.
2. Enter a component name in the **Part Name** box, or select the check box and type another name. The component appears in the preview window.
3. The **BOM Data** tab opens automatically. Enter the following information:
   - **Part ID**—Unique part ID string.
   - **BOM Name**—Full description string.
   - **Group Layer**—Name of the component layer. This layer is created in the active (sub) assembly when the configuration option `CREATE_LAYERS_PER_PART` is set to `YES`.
   - **Part ID**—Unique part ID string.
   - **BOM Name**—Full description string.
   - **Order Number**—Order number.
   - **Customer Number**—Customer specific identifier.
   - **Material**—Material.

NOTE:

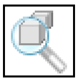
- You cannot set parameters and relations for subcomponents.
- There is no UDF information for subcomponents.

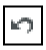
11.5 About Defining Components

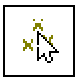
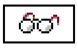
1. Click **EMX Components** and then in the **Components** group, select a component. A definition dialog box for the selected component opens.
2. Set the unit of measurement to **mm** or **inch**. By default, the unit of measurement is the same as that of the project assembly. You can, however, use multiple measurement types in an assembly such as using metric plates in an inch assembly.
3. Select a supplier from the list.
4. Select the assembly in which to assemble the equipment component. You cannot change this selection while redefining a component.
5. Select the subtype of the component from the list. A simplified image of the selected component type is displayed in the preview window. The dimensions appear in the table.
6. Right-click a row in the table to open a shortcut menu from which you can measure diameter, length, or offset.
7. Double-click the **Value** column to enter a new value  or to select a value from the list . You cannot change values marked .

   NOTE:
   You can use the configuration option `MON_DIM_DISPLAY` to reduce the visible description of the driving dimension.
8. Enter a component name in the **Part Name** box. To enter an alternate name, select the check box or type the name.

   NOTE:
   Change the default part name format or reset the part name, in the **Part Names** tab in the **Options** dialog box.
9. Select references such as **Axis|Point**, **Thread Surface**, **Surface**, and so on and select the assembly placement constraints in the graphics window. A constraint name that does not appear in boldface indicates that this constraint was set automatically and does not need further definition.

   NOTE:
   If you move your mouse above the panel the selected reference will be highlighted in the graphics window.
10. Set Component options on page 120, component BOM data, parameters and relations, and UDF dimensions on page 121.
11. Select the **Standard Part** check box to prevent editing.
12. Select the **Add to BOM** check box to include the component in the Bill of Materials.
13. Click  to open a detail image of the current component.
14. Click  to save current settings as the default for the type of component.

    NOTE:
    You can also automatically save the last component used by setting the configuration option `AUTO_SAVE_LAST_COMP` to `YES`.
15. Click  to reset save current settings.

16. Select a placement plane, and then click  to create assembly reference datum points on the fly. The model reorients normal to the placement plane. A circle appears with the main diameter of the component. Click to place the point, and then right-click to exit the tool.
17. Click  to preview the component.
    - The mold base is in wire frame mode.
    - The new component is in shaded mode.
    - Changes after editing a component dimension.

    NOTE:
    In preview the new component is always displayed in shaded mode. The configuration option `PREVIEW_VIS_MODE` drives how the other components should be displayed (wireframe, hiddenvis etc).
18. Click  or  to collapse or expand the options tab when the dialog is too big such as for screens with lower resolution.
19. Click **OK** to create the new component and cuts or click **Cancel** to close the dialog box without making changes.

11.5.1 Background Information

- When placing components through the **Mold Base Definition** dialog box it is not required to select the position as their placement is defined automatically.
- The point or coordinate system inside an reference group is renamed so it can be identified for later redefine.

  By default the name is <component_type>_<id> (i.e, GUIDE_51).
  Set the configuration option `USE_INSTANCE_IN_REFNAME` to `YES` to force a naming format like <component_type>_<id>_<instance> (i.e, GUIDE_51_E1140_42_160).

11.6 About Measuring Dimensions

1. Right–click a line in the dimension table of a definition dialog box for a component or a plate. A shortcut menu opens.
2. Select a dimension, diameter, length, or offset to measure from the shortcut menu, or choose a dimension to modify directly from the table.

Notes:

12

The Mold Base Stop System Components

12.1 About Stop System Components

To define stop components, stop discs, or stop pins, click **EMX Components** and then in the **Components** group, click **Stop disc** or the **Stop pin** to define a new stop disc or a new stop pin.

12.2 To Assemble the Stop System

Download 25 To Assemble the Stop System to start from with this chapter.

12.2.1 Define the Stop System Pattern

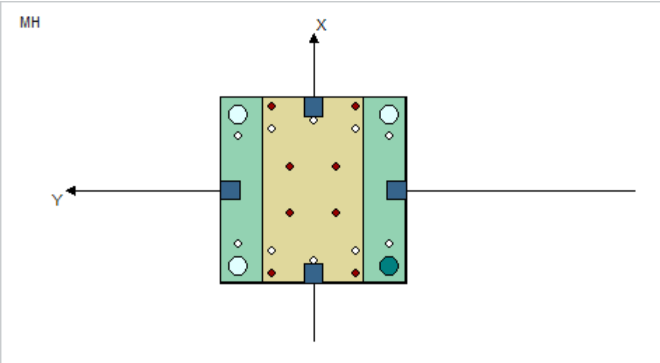
1. Open **Mold Base Definition** dialog box.
2. Click **Top** to switch to the top view.
3. Double-click on **Stop System MH** in the summary tree.

   The **Stop System MH** pattern dialog box opens.

   For supplier `meusburger` no default pattern information is defined in the catalog.
4. Set **X Quantity** to `4`.
5. Set **Y Quantity** to `2`.
6. Enter **Pattern Size** `358` and `180`.
7. Click to update the number of instances.
8. Change instance 3 to 6 as seen in the image below.

ID	X	Y
1	-179.000	-90.000
2	-179.000	90.000
3	-50.000	-50.000
4	-50.000	50.000
5	50.000	-50.000
6	50.000	50.000
7	179.000	-90.000
8	179.000	90.000

9. In the **Mold Base Definition** dialog box enable the option **Show Cavity** to display the full stop system pattern in the top view.



10. Click **OK** to close the pattern dialog.
11. Leave the **Mold Base Definition** dialog box with **Close**.

12.2.2 To Assemble the Stop Pins

1. Open the moving half assembly `TUTOR_MH.ASM`.
2. Click **EMX Components ► Components ►** **Stop pin**

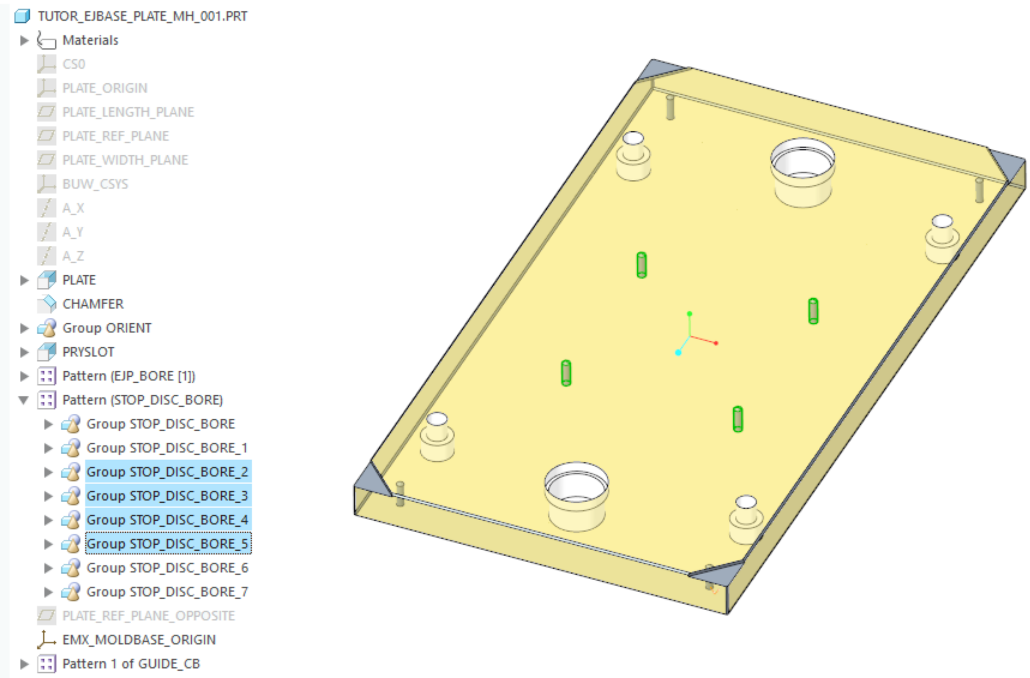
   The **Stop pin** dialog box opens.
3. Set the **DM3** value to `30` mm.
4. As the first assembly reference **(1) Point|Axis** select the first axis instance `STOP_BORE` from the stop system pattern in the `TUTOR_SKELETON` model.
5. As the second assembly reference **(2) Surface** select the top surface of the clamping plate.
6. Switch to the **Options** tab and enable the option **Pattern for all instances**.
7. Click to preview the stop pins.
8. Click **OK** to leave the **Stop Disc** dialog box.

   The Stop System will be added to the mold base.
9. Close the `TUTOR_MH.ASM` to return to the main assembly.

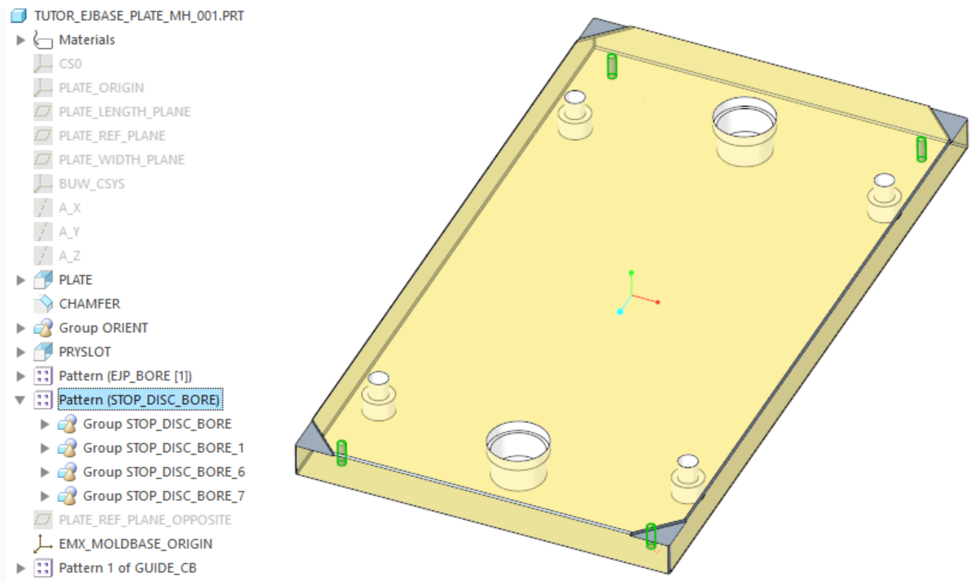
12.2.3 Remove Additional Stop Pin Bores from Ejector Base Plate

The new stop bore instance should not be added to the ejector base plate. They need to be removed from the model

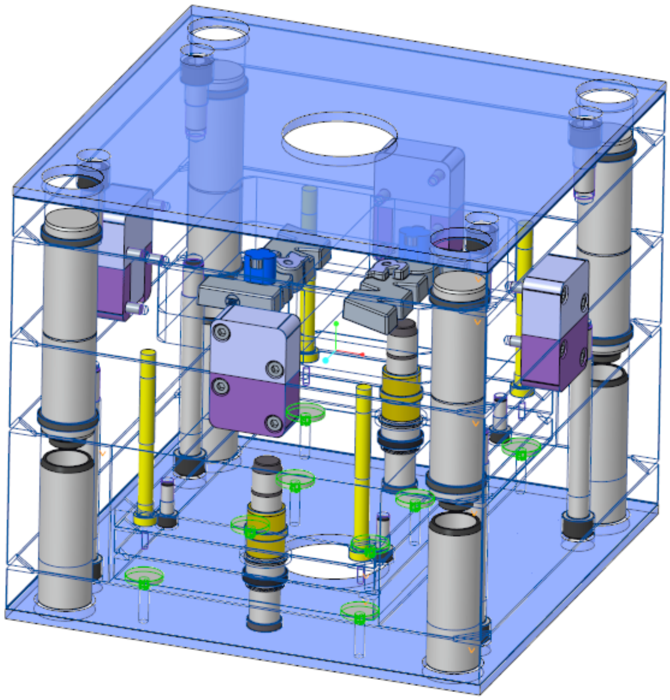
1. Open the ejector base plate `TUTOR_EJBASE_MH_001.ASM`.
2. Modify the pattern `STOP_DISC_BORE`.



3. Suppress the instances 3 to 6



4. Close the `TUTOR_MH.ASM` to return to the main assembly.



Notes:

13

The Mold Base Locating Ring

13.1 About Equipment

Some components are classified as equipment. These components can be locating rings, insulation plates, sprue bushings, or knockouts. To create, modify or define equipment features, click **EMX Components**, and then in the **Components** group, click the arrow next to **Equipment** to see the equipment list described below:

Icon	Equipment	Function
	Locating ring	Defines a new locating ring
	Thermal Insulation plate	Defines a new insulation plate
	Knockout	Defines a new knockout
	Side Interlock	Defines a new side interlock
	Top lock	Defines a new top interlock
	Compression spring	Defines a new spring
	Sprue bushing	Defines a new sprue bushing
	Support bushing	Defines a new support bushing
	Tubular Dowel	Defines a new tubular dowel

NOTE:
Some equipment, including sprue or support bushings, can have both blind or straight holes.

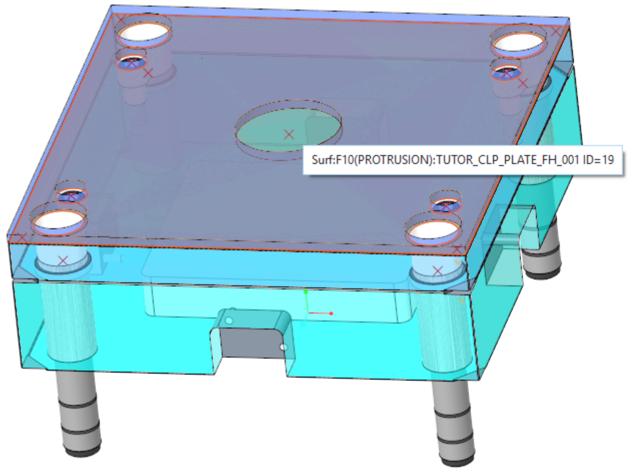
13.2 To Assemble the Locating Ring

Download 26 To Assemble the Locating Ring to start from with this chapter.

13.2.1 Assemble the Locating Ring

1. Open the fixed half assembly `TUTOR_FH.ASM`
2. Click **EMX Components** ▸ **Components** ▸ **Locating ring**

   The **Locating Ring** dialog box opens.
3. Select `E1370` from **types** pull down list.
4. Select `125` mm for **DM3**.
5. Select `20` mm for **LG**.
6. Enter a value of `-6` mm for **OFFSET**.
7. As first assembly reference **(1) Axis|Point** select `AXIS_LOCATING` from the `TUTOR_SKELETON.PRT` model.
8. As the second assembly reference **(2) Surface** select the top surface of clamping plate.



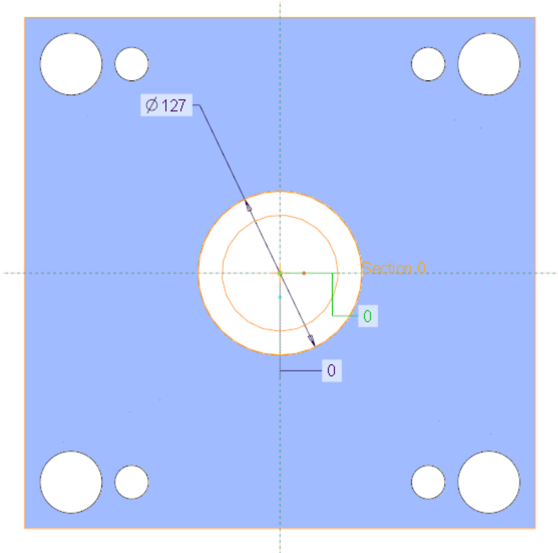
9. Close the dialog box with **OK**.

   The locating ring is assebmly to the fixed half assembly.

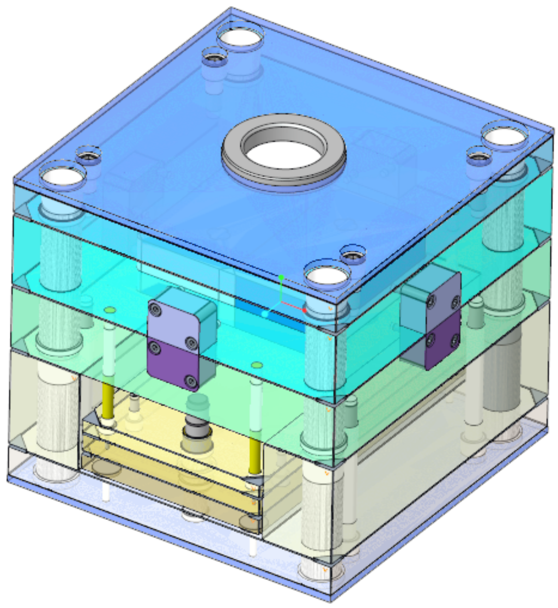
13.2.2 Adapt the locating cutout in the insulation plate

The insulation plate template already contains a locating cutout feature. This feature needs to be adapted to the assembled locating ring.

1. Open the insulation plate `TUTOR_INSULATION_4.PRT`
2. Edit the feature `LOCATING.`
3. Set the diameter to `127` mm.



4. Close the `TUTOR_INSULATION_4.PRT`
5. Close the `TUTOR_FH.ASM`



14

The Mold Base Sprue Bushing

To Assemble the Sprue Bushings

14.1 To Assemble the Sprue Bushings

Download 27 To Assemble the Sprue Bushings to start from with this chapter.

14.1.1 Assemble the Sprue Bushing

1. Open the fixed half assembly `TUTOR_FH.ASM`.
2. Click **EMX Components** ▶ **Components** ▶ **Sprue Bushing**

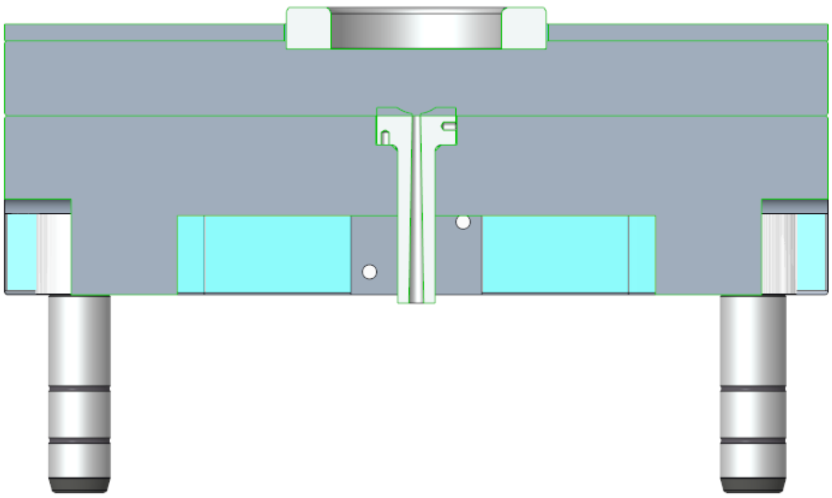
   The **Sprue Bushing** dialog box appears.
3. Select `E1605` from the **Types** pull down list.
4. Set the **D_2** value to `18`.
5. Set the **OFFSET** value to `4`.

   The sprue bush will interfere with the clamping plate.
6. Set **LG1** to `76`.

   The length needs to set to a value that the sprue bush reaches the splitting plane of the mold.
7. As the first assembly reference **(1) Axis|Point** select the `AXIS_LOCATING` from the `TUTOR_SKELETON.PRT` model.
8. As the second reference **(2) Surface** select the top surface of the cavity plate.
9. Close the **Sprue Bushing** dialog with **OK**.

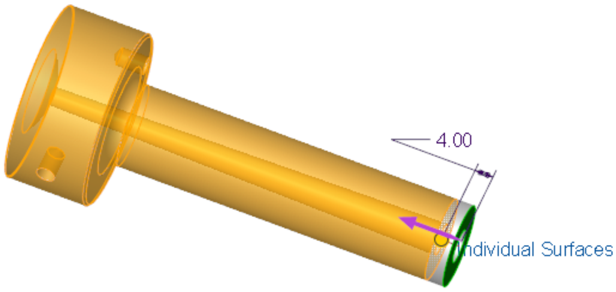
   The sprue bush is assembled to the fixed half and cut-outs are added to the cavity plate.

   However, additional cutouts are required in the clamping plate and the cavity insert parts, which will be designed in the following steps.



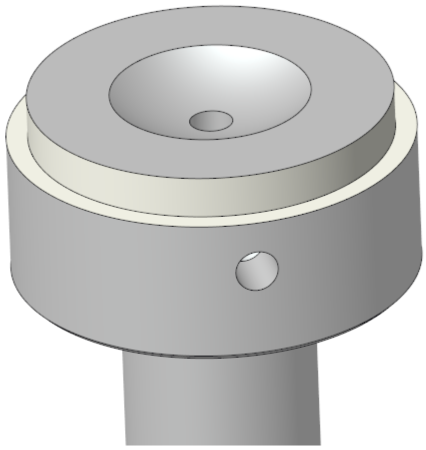
14.1.2 Shorten the Sprue Bush

1. Open the `TUTOR_E1605_18_76_4.PRT`.
2. Create an **Offset** feature to shorten the sprue bushing about 4 mm.



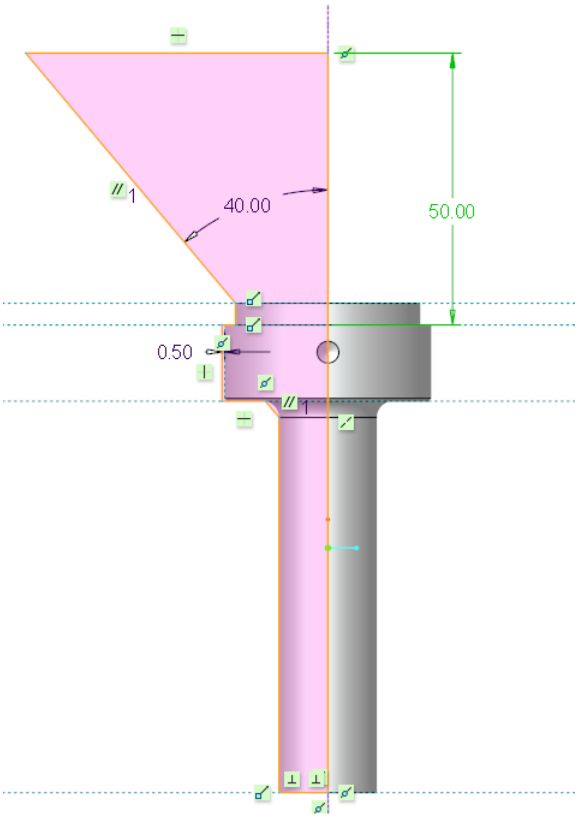
14.1.3 Create a Groove for fixing the Sprue Bush in Clamping Plate

1. Create a groove of 2x4 mm by using a **Revolve** feature.



14.1.4 Create a Quilt Surface Additional Cut-Outs

1. Create a quilt surface with a **Revolve** feature.

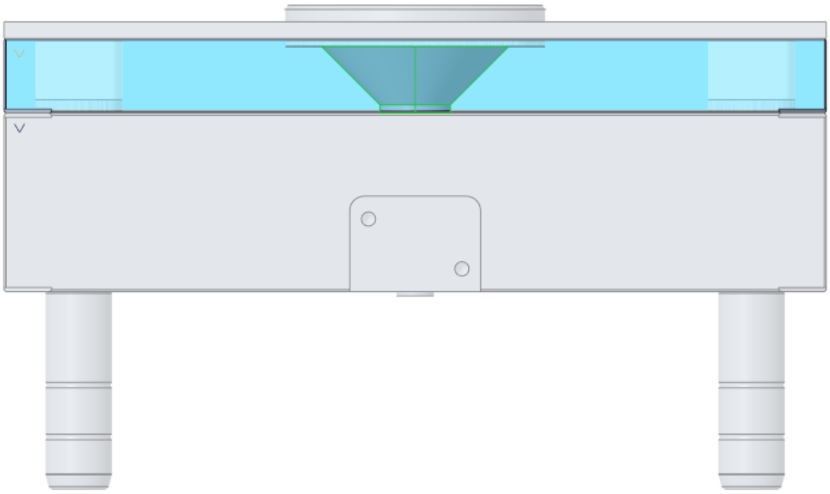


2. Set the quilt surface color to transparent.

3. Close the sprue bushing `TUTOR_E1605_18_76.PRT` model.

14.1.5 Create Cut-Out in Clamping Plate

1. Activate the clamping plate `TUTOR_CLP_FH_001.PRT`
2. Select the previously create quilt surface and create a **Solidify** feature with **Remove material** option enable

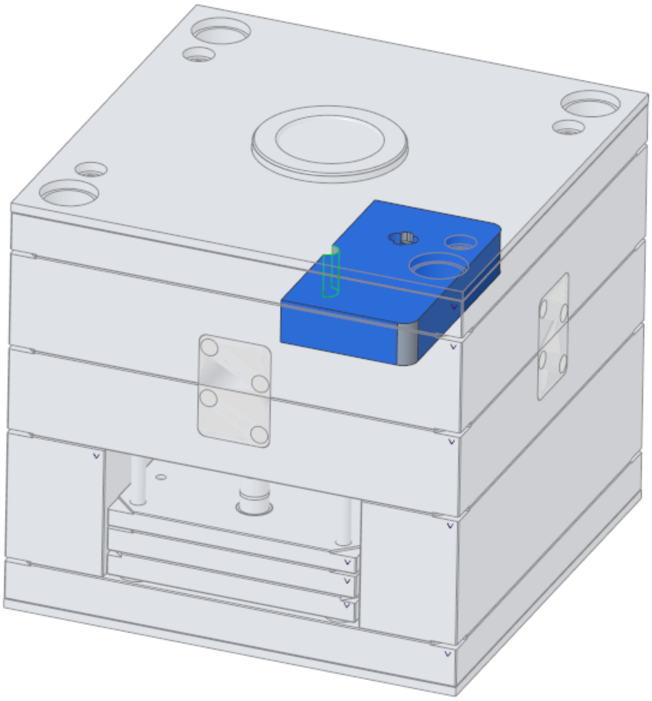


3. Activate `TUTOR_FH.ASM`
4. Close `TUTOR_FH.ASM` and return to the main assembly.

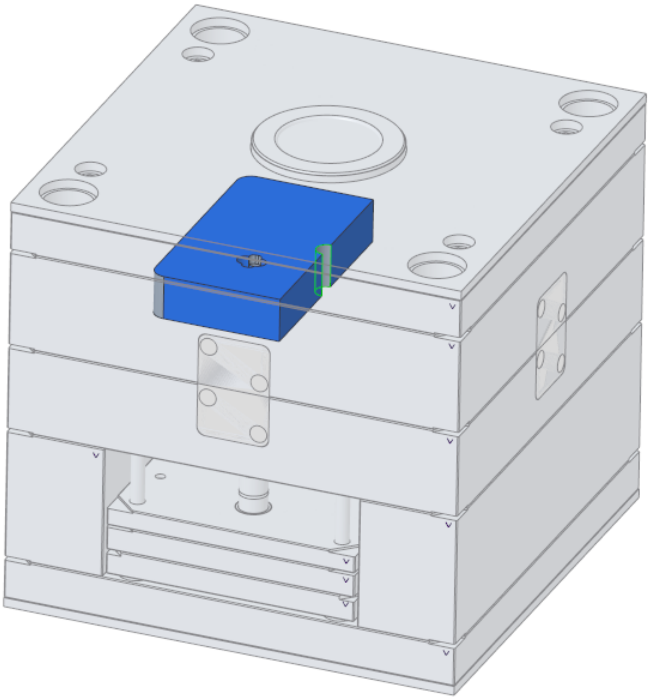
14.1.6 Create Cut-Outs in Cavity Insert Partse

1. Within the main assembly activate the insert parts `MOLD_VOL_IS_1.PRT` within the first cavity insert assembly `MFG_1.ASM`.

2. Select the previously create quilt surface in the sprue bushing model and create a **Solidify** feature with **Remove material** option enable

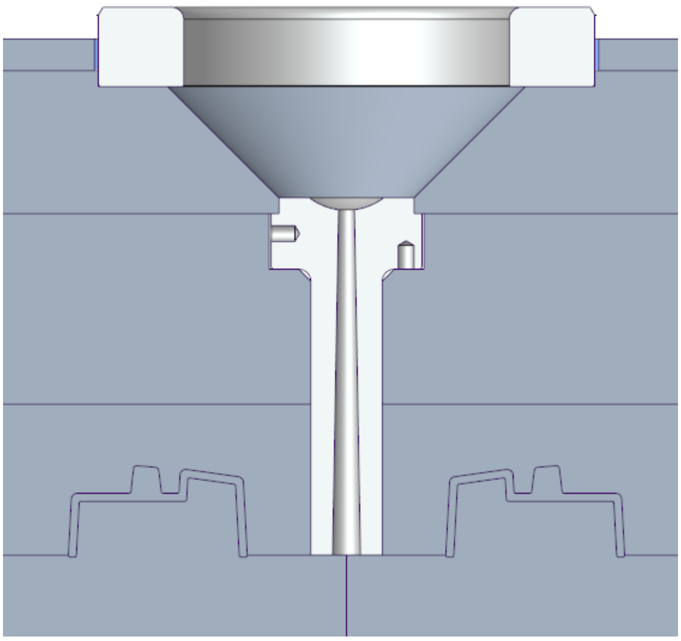


3. Repeat the steps for MOLD_VOL_IS_2.PRT.



4. Activate TUTORORIAL.ASM.

   The Sprue bushing is now full placed.



Notes:

15

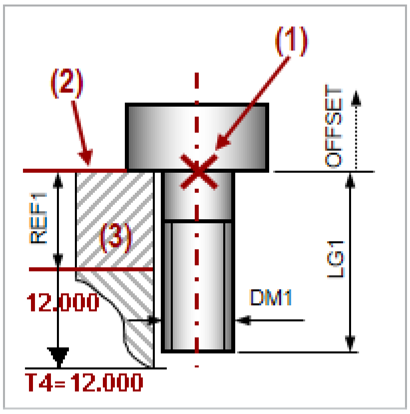
Screws

15.1 About Screws

Use screws to join components and use shoulder screws that are similar to latch locks to control plate movement. Click **EMX Components** ► **Screw** to define a new screw.

You can place screws with a counterbore in the first component, and a straight hole or blind hole in the last component. When you select **One Plate** the screw is assembled on only one plate.

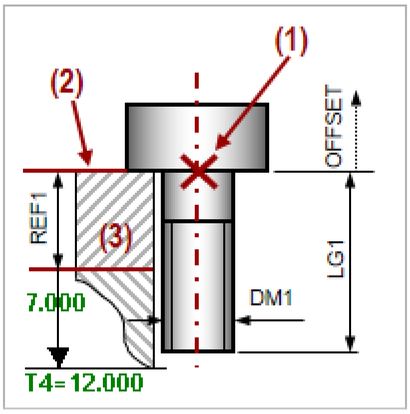
The placement references for screws always contain a **(2) Surface** and a **(3) Thread Surface**.



The offset of the two surfaces is measured and displayed in the **REF1 Reference Distance**, i. e. `20` mm.

Within the screw image the meaning of the reference distance is displayed by the dimension **REF1**.

The red dimension `12.0` indicates that the fixing length of the screw in the tapped hole is deeper than the default thread depth. Either the length `LG1` or the thread depth in the cut-out dimension `T4/T5` need to be adapted.



In case a shorter screw instance is selected the threaded depth dimension shows value `7.000` and switches to green, which indicates that the selected length is appropriate.

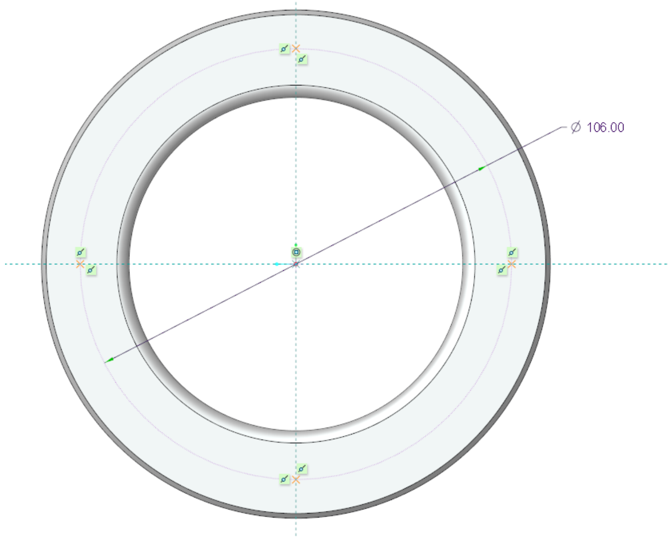
This dimension is calculated by the offset of the two surfaces and the selected screw length. If the result is larger than `0` and smaller than `T4` of the bore, it is considered to be okay.

15.2 To Assemble Screws

Download 28 To Assemble Screws to start from with this chapter.

15.2.1 Create a Datum Point

1. Open the mold base `TUTOR_FH.ASM`.
2. Open the locating ring `TUTOR_LOCATING_RING_2001.PRT`.
3. Create a sketch of datum points as seen in the picture below.



4. Close the current window `TUTOR_LOCATING_RING_2001.PRT` and return to the `TUTOR_FH.ASM`.

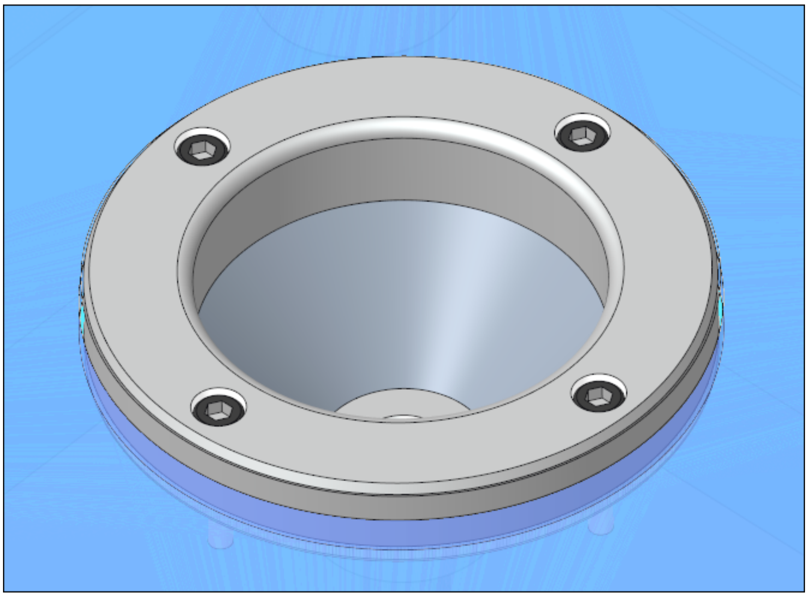
15.2.2 Assemble the Screw

1. Select **EMX Components ► Components ►** **Screw** .
2. In the **Screw** dialog box select the desired screw supplier **meusburger** and screw type **E1200 | Socket head cap screw**.
3. Define the references.
   a. Click **(1) Point|Axis** and select the previously created datum point on the locating ring `TUTOR_LOCATING_RING2002.PRT`.
   b. Click **(2) Surface** and select the top face of the model `TUTOR_LOCATING_RING_2001.PRT`.
   c. Click **(3) Thread surface** and select the locating cut face of the top clamping plate.

   The default length for the screw is calculated by the rule defined in the configuration Option `MIN_SCREW_LENGTH_RULE`.

   The default value for this rule is `1.5*diameter+REF1+OFFSET`.
4. Select **Pattern for all instances**.
5. Double click the **DM1 Diameter Value** and select `6`.
6. Double click the **LG1 Length Value** and select `20`.

7. Close the **Screw** dialog box with **OK**.



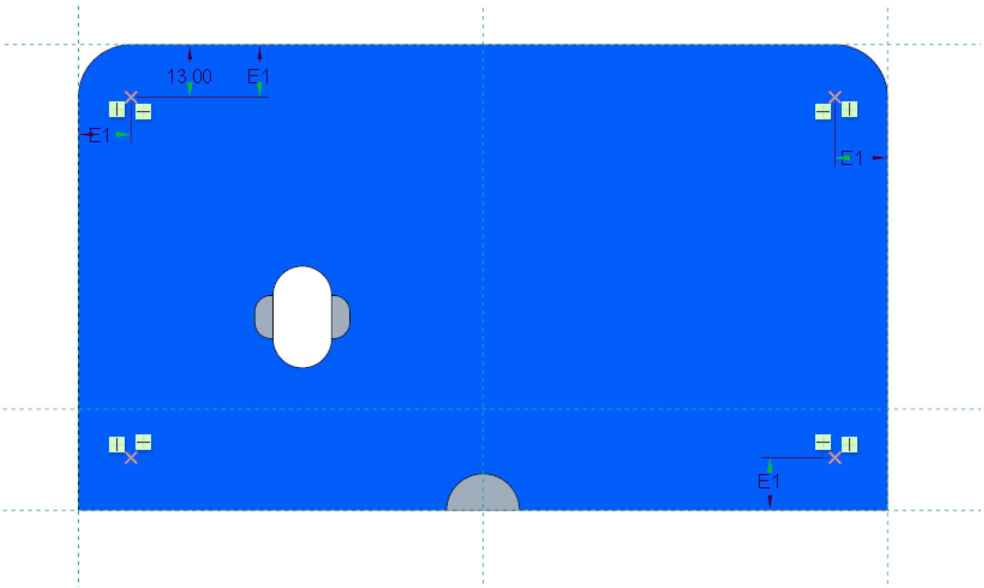
8. Close the mold base TUTOR_FH.ASM and return to main assembly.

15.3 To Assemble Screws on Insert Parts

Download 29 To Assemble Screws on Insert Parts to start from with this chapter.

15.3.1 Create Points on Insert Parts

1. Open MOLD_VOL_IS.PRT.
2. Create sketched datum points on the top surface of the model as seen in the image below.



3. Close MOLD_VOL_IS.PRT and return to TUTORIAL.ASM.

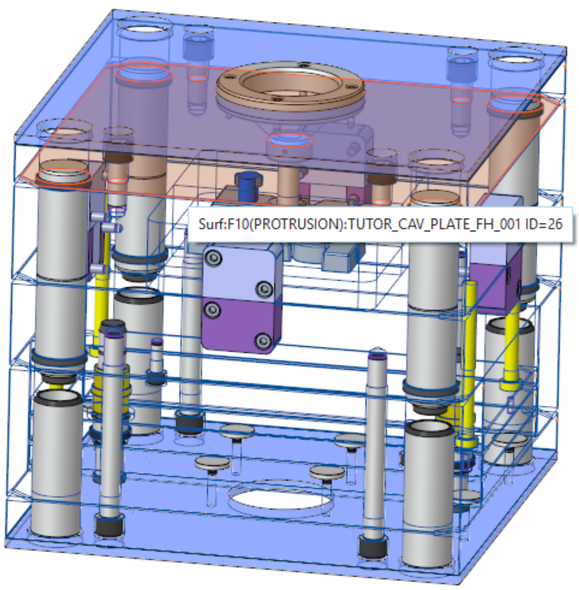
NOTE:
You can hide the Layers **00_BUW_SCREWS**, **00_BUW_EJECTORS**, **00_BUW_GUIDES** or **00_BUW_EQUIPMENT** etc. to reduce the amount of points displayed in the assembly.

4. Repeat these steps for the `MOLD_VOL_ES.PRT` model.
5. Regenerate the Assembly `TUTORIAL.ASM`.

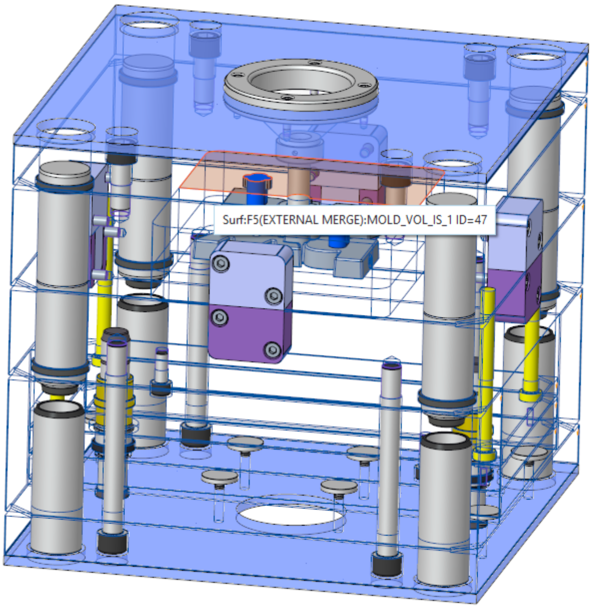
   The points will be added to the parts `MOLD_VOL_IS_1.PRT`, `MOLD_VOL_IS_2.PRT`, `MOLD_VOL_ES_1.PRT` and `MOLD_VOL_ES_2.PRT`.

15.3.2 Assemble the Screws for the Fixed Half insert Parts

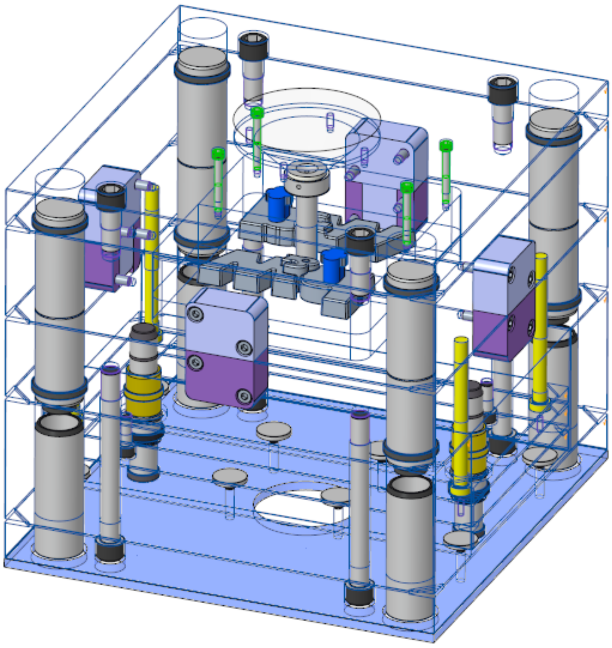
1. Click **EMX Components ► Components ► Screw** .
2. Select `TUTOR_FH` as target subassembly form the **subassembly** pull-down list.
3. Define the references.
   a. Click **(1) Point/Axis** and select the recently defined points from `MOLD_VOL_IS_1.PRT`
   b. As **(2) Surface** select the top surface of the cavity plate `TUTOR_CAV_PLATE_FH_0001.PRT`



   c. As **(3) Thread Surface** select the top surface of the `MOLD_VOL_IS_1.PRT` insert.



4. Select **DM1 — Diameter** 8 mm.
5. Select **LG1 — Length** 50 mm.
6. Close the **Screw** dialog box with **OK**.



7. Repeat these steps for the MOLD_VOL_IS_2.PRT model.

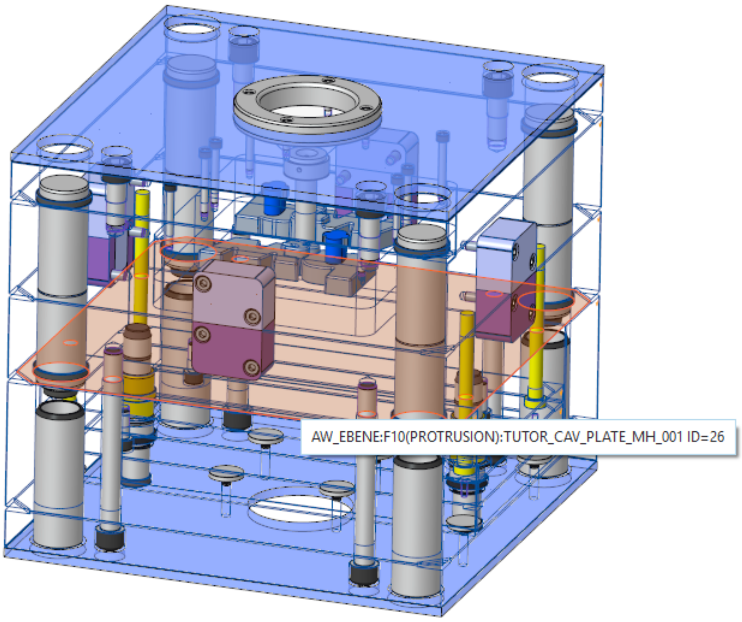
15.3.3 Assemble the Screws for the Moving Half insert Parts

1. Click **EMX Components ► Components ► Screw**.
2. Select TUTOR_MH as target subassembly form the **subassembly** pull-down list.

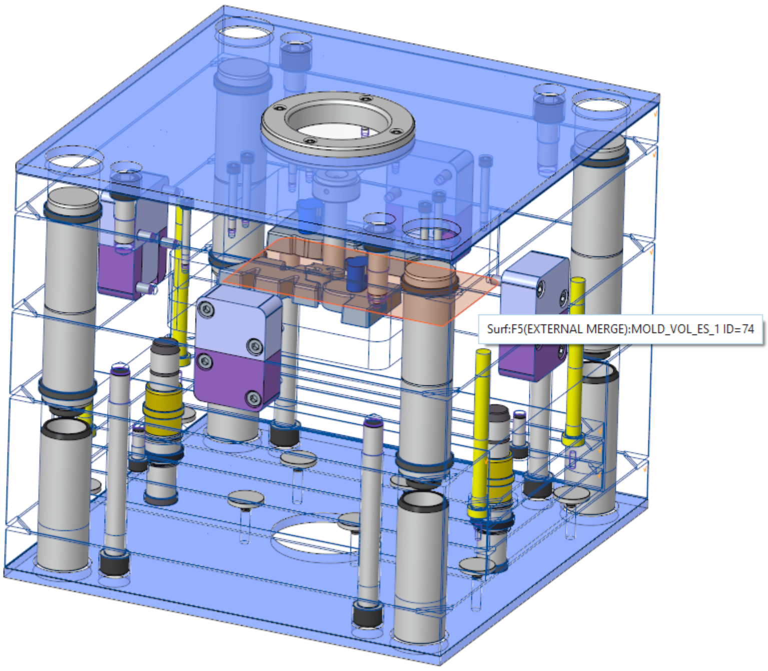
3. Define the references.

   a. Click **(1) Point/Axis** and select the recently defined points from `MOLD_VOL_ES_1.PRT`

   b. As **(2) Surface** select the bottom surface of the cavity plate `TUTOR_CAV_PLATE_MH_001.PRT`



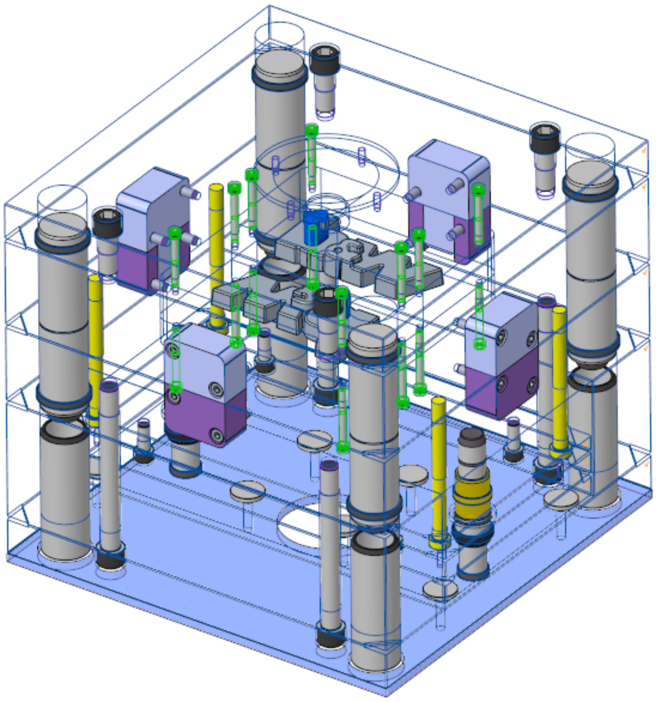
   c. As **(3) Thread Surface** select the bottom surface of the `MOLD_VOL_ES_1.PRT` insert.



4. Select **DM1 — Diameter** 8 mm.

5. Select **LG1 — Length** `45` mm.
6. Close the **Screw** dialog box with **OK**.
7. Repeat these steps for the `MOLD_VOL_ES_2.PRT` model.

   In total 16 screws should be assembled to the mold base.



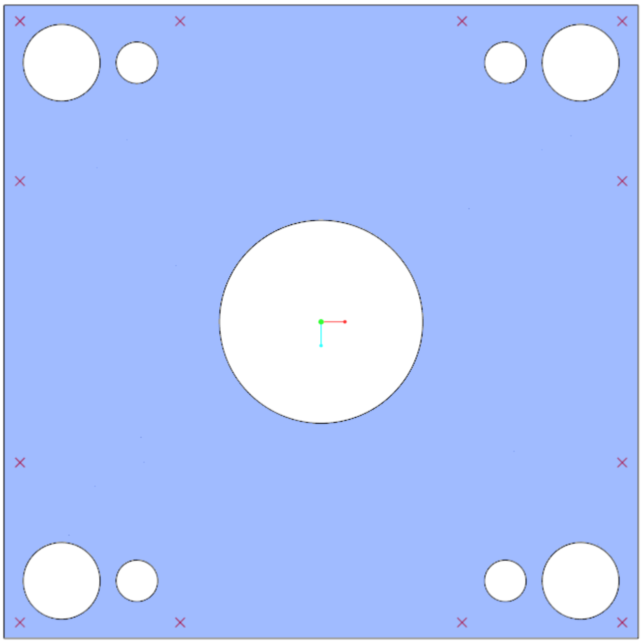
15.4 To Assemble Screws on Insulation Plates

Download 30 To Assemble Screws on Insulation Plates to start from with this chapter.

15.4.1 Create Points on Insulation Plates

1. Open `TUTOR_FH.ASM`
2. Open `TUTOR_INSULATION_4.PRT`

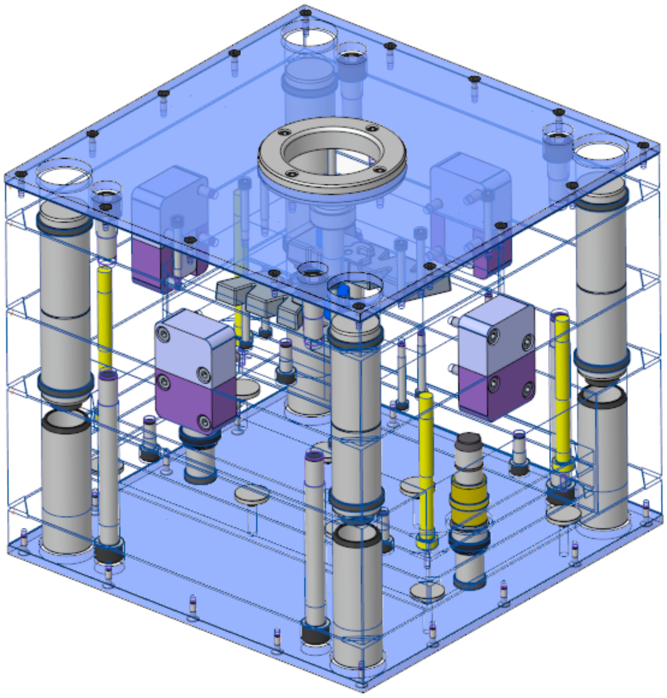
3. Create sketched datum points on the top surface of the model as seen in the image below.



4. Close TUTOR_INSULATION_4.PRT and return to TUTOR_FH.ASM.

15.4.2 Assemble the Screws to the Insulation Plates

1. Click **EMX Components ▸ Components ▸ Screw** .
2. Select the screw E1220 from **Types** pull-down list.
3. Define the references.
   a. Click **(1) Point/Axis** and select the recently defined points from TUTOR_INSULATION_4.PRT
   b. As **(2) Surface** select the top surface of the insulation plate TUTOR_INSULATION_4.PRT
   c. As **(3) Thread Surface** select the top surface of the clamping plateTUTOR_CLP_PALTE_FH_001.PRT insert.
4. Select **DM1 — Diameter** 5 mm.
5. Select **LG — Length** 16 mm.
6. Close the **Screw** dialog box with **OK**.
7. Close TUTOR_FH.ASM and return to main assembly.
8. Repeat these steps for the INSUALTION_PLATE_3.PRT in the moving half.



16

Dowel Pins

16.1 About Dowel Pins

Use dowel pins to prevent two components from moving relative to each other. Click **EMX Components** and then in the **Components** group click **Dowel Pin** to define a dowel pin.

You can place dowel pins with a straight hole, a counter bore or a blind hole in both sides.

16.2 To Assemble Dowel Pins

Download 31 To Assemble Dowel Pins to start from with this chapter.

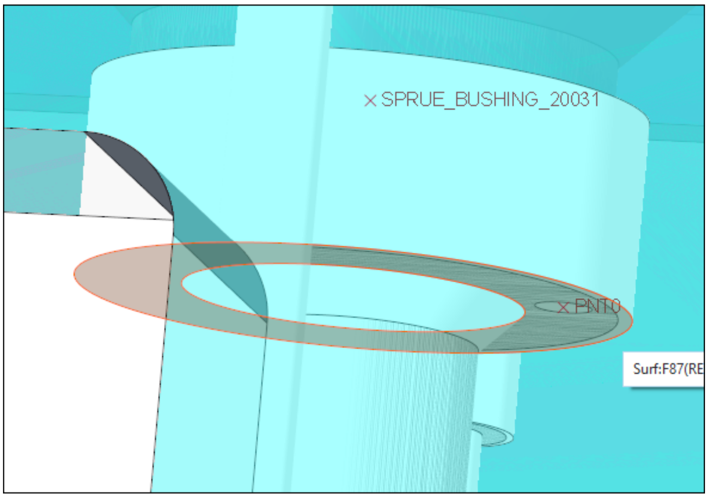
16.2.1 Create a Datum Point

1. Open the mold base `TUTOR_FH.ASM`.
2. Open the sprue bushing `TUTOR_E1605_18_76_4.PRT`.
3. Define a point on the existing dowel pin hole on the bottom side of the bush head.

4. Close the current window `TUTOR_E1605_18_76_4.PRT` and return to the `TUTOR_FH.ASM`.

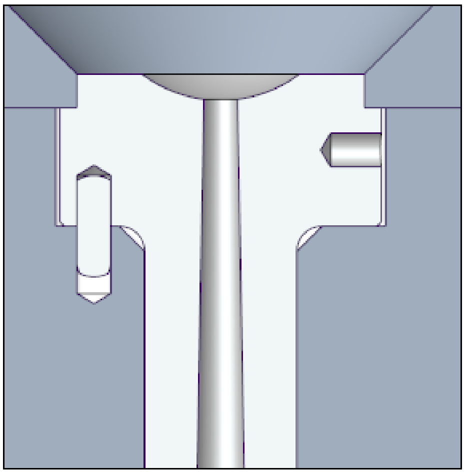
16.2.2 Assemble the Dowel Pin

1. Select **EMX Components ► Components ► Dowel Pin**.
2. In the **Dowel Pin** dialog box select the desired supplier `meusburger` and type `E1300`.
3. Define the references.
   a. Click **(1) Point|Axis** and select the previously created datum point in the sprue bushing `TUTOR_E1605_18_76_4.PRT`.
   b. Click **(2) Surface** and select the flat cut-out surface of the sprue bushing.



4. Select `4` mm for **DM1**.
5. Select `12` mm for **LG**.

   NOTE:
   For dowel pins the offset is initially always set to half of the length.
6. Activate the option **Blind Hole Bottom**.
7. Close the **Dialog** dialog box with **OK**.



8. Close the mold base `TUTOR_FH.ASM` and return to main assembly.

Notes:

17

Ejector Pins

17.1 About Ejector Pins

Use an ejector pin to push the cold plastic part out of the cavity insert when the mold base is opened.

Their placement can be specified using datum points in the reference model.

If no reference model exists the placement references for ejector pins can be defined in the core model of the Creo Expert Moldbase Extension assembly.

Click **EMX Components ► Ejector Pin** and click one of the following commands:

- **Ejector Pin Designer**—Define Ejector Pins dynamically
- **Ejector Pin**—Defines a new ejector pin.
- **Update existing Ejector Pins**—Updates existing ejector pins in case reference distance has changed..
- **Delete all Ejector Pins**—Deletes all ejector pins in the current assembly.
- **Trim Ejector Pin Heads**—Trims ejector pin heads against each other.
- **Identify ejector pin owner models** —Prepares the selected parts for predefining ejectors with layout curve on page 190
- **Add Ejector Bores**—Checks and adds the cutouts for every assembled ejector pin if these are not present.
- In Drawing Mode Only: **Add drawing symbols**—Places drawing symbols on all ejector pin cutouts in the drawing model. Add notes to the drawing to see the ejector ID.

NOTE:

- You can use **Trim to quilt/refmodel** on all ejector pin types. To define the quilts, click **EMX Part Mode ► Preparation ► Identify trim faces** or **EMX Assembly ► Preparation ► Identify trim faces**.
    - If a trim surface is defined then Creo Expert Moldbase Extension uses the **Solidify** feature to create the trimmed surface.
    - If no trim surface is defined then Creo Expert Moldbase Extension trims the ejector pin against the solid using the Mold Design **Trim to Geometry**.
- When you use an ejector pin as a core pin, you must assemble a different hole type to the extracts. Select these hole types from a list of fixed rotation hole types.

17.2 To Assemble Ejector Pins

Download 32 To Assemble Ejector Pins to start from with this chapter.

17.2.1 Assemble the Ejector Pin Target Subassembly

1. Click **EMX Assembly ▸ Structure ▸ Add Subassemblies**

   The **Subassembly** dialog box opens.
2. Select TUTOR_MH from the **Subassembly** pull-down list.
3. Enter TUTOR_MH_EPINS into the **Name** input panel.
4. Select **Ejector Pins** from the **Function** pull-down list.
5. Click **OK** to leave the **Subassembly** dialog box.

   A new EMX Subassembly will be added to the mold base assembly.

   With the assigned **Ejector Pins** function all upcoming ejector pins will by default be assembled to this subassembly.

17.2.2 Define the Ejector Pin Properties

1. Click **EMX Assembly ▸ View ▸ Show ▸ Moving Half**.
2. Select **EMX Components ▸ Ejector Pin ▸ Ejector Pin** .

   The **Ejector Pin** dialog box opens.

   TUTOR_MH_EPINS is preselected as **Subassembly**.
3. Select knarr from the **Supplier** pull-down list.
4. Select 37111 from the **Types** pull-down list.
5. Set the **DM1** value to 4 mm.
6. Define the references

   a. For **(1) Point** select a point from the multi point feature PNTS_ROUND_EPIN_4MM in the ARTICLE_REF.PRT model.

      After the Point has been selected the offset between the point and the default reference surface will measured.

      This offset (157.541 mm) is shown in the **Reference Distance** line of the Dimensions.

      The next valid instance is chosen automatically in case an instance of this ejector pin exists, where the **LG1** exceeds the measured distance.

      NOTE:
      A warning dialog appears in case the required length for the selected diameter does not exist in the catalog.

b. The **(2) Surface** reference will be selected automatically and is used to align the head of the ejector pin. By default the bottom face of the first ejector retainer plate in the mold base is used.

   NOTE:
   A different surface can be selected by clicking the **Surface** button and select the desired face in the model.

c. The **(3) Orientation surface** is preselected. The `MOLDBASE_Y_Z` of the skeleton model is used by default for all ejector pins.

7. Click **Preview** to visualize the ejector pins.

   On each point of the multiple point features an ejector pin is previewed.

   NOTE:
   It is not possible to place a single ejector component on a single datum point within a point feature containing multiple point instances. The same ejector pin model will be placed on all points.

8. Enable the option **Individual ejector models on each point**.

   This option is necessary to be able to Trim the ejector pin heads on page 194 after finishing the ejector pin placement.

   NOTE:
   With this option an individual ejector pin model will be placed on each point.

   This automatically is used for:

   - Ejector pins defined on multiple point features with different heights.
   - Trimmed ejector pins on multiple point features.

9. Make sure **Auto Length** is activated.

   With this option the length of the ejector is set to the **Reference Distance** value displayed in the dialog box.

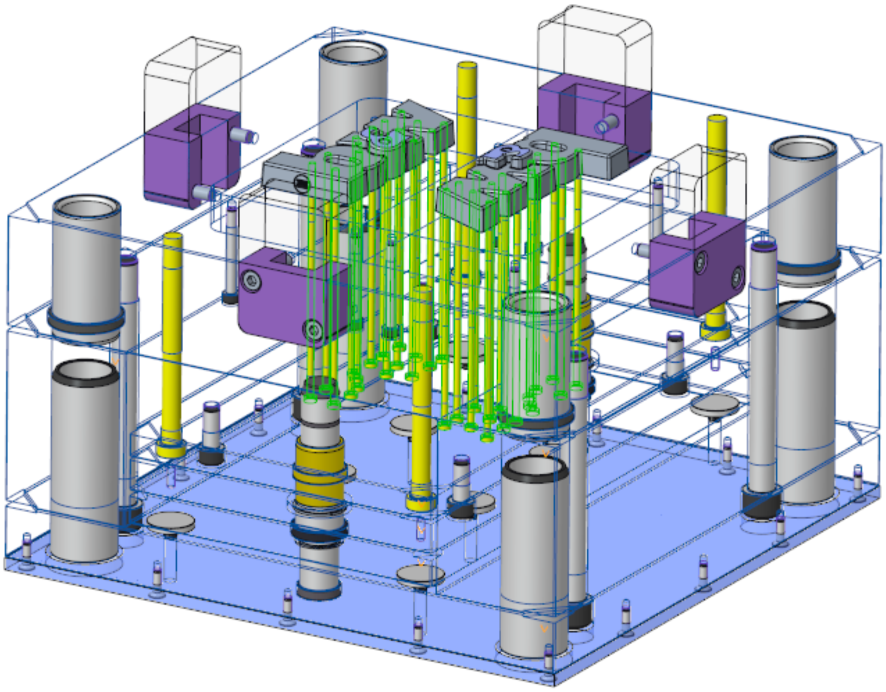
10. Switch to the **Options** tab and click **Pattern on all models** to add the ejector pins (and their cutouts) to both `ARTICLE_REF.PRT` models.

11. Click **Preview** again to visualize the ejector pins.

    On each point of the multiple point features and on both reference models an ejector pin is previewed.

12. Close the **Ejector Pin** dialog box with **OK**.

    The ejector pins and their according cutouts are assembled to the mold base.



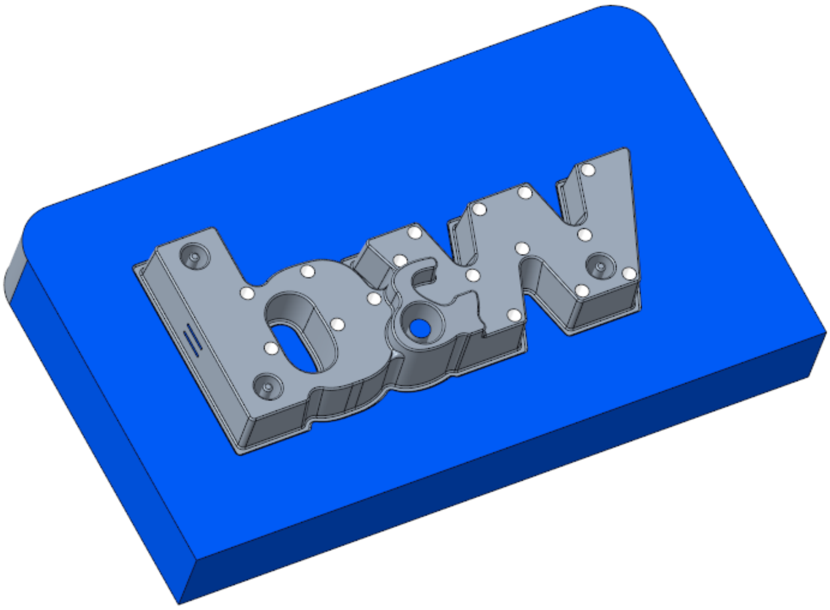
NOTE:
This ejector pin length is stored in the Parameter with the rule `&remark` to the model, by default this is the `NOTE` Parameter.

Use the two configuration option `EJP_NOTE_PRE_STRING` and `EJP_NOTE_POST_STRING` to specify additional text for this `NOTE` Parameter before and after the number.

13. Open the `MOLD_VOL_ES_1.PRT` model.

    All cut-outs are created on part level.

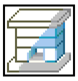
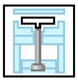
The groups `EJP_CB_ANGLE` can be treated and handled like any other feature..



14. Close `MOLD_VOL_ES_1.PRT` and return to the main assembly.

17.3 To Assemble Trimmed Ejector Pins

Download 33 To Assemble Trimmed Ejector Pins to start from with this chapter.

1. Click **EMX Assembly** ▸ **View** ▸  **Wireframe Style**.
2. Click **EMX Assembly** ▸ **View** ▸ **Show** ▸  **Moving Half**.
3. Click **EMX Components** ▸  **Ejector Pin**.

   The **Ejector Pin** dialog box opens.

   The `TUTOR_MH_EPINS` assembly is preselected.
4. Select `knarr` from the **Supplier** pull-down list.
5. Select `36111` from the **Types** pull-down list.
6. Set the **DM1** value to `2` mm.
7. Define the references

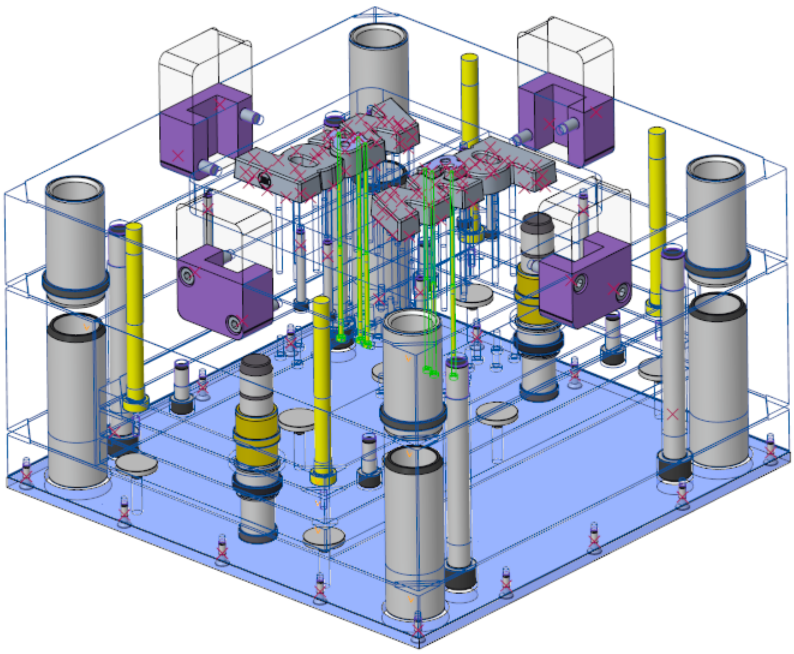
   a. For **(1) Point** select a point from the multi point feature `PNTS_SHOULDERED_EPINS_2MM` in the `ARTICLE_REF.PRT` model.

      The **LG1** value is automatically set to `200` mm.

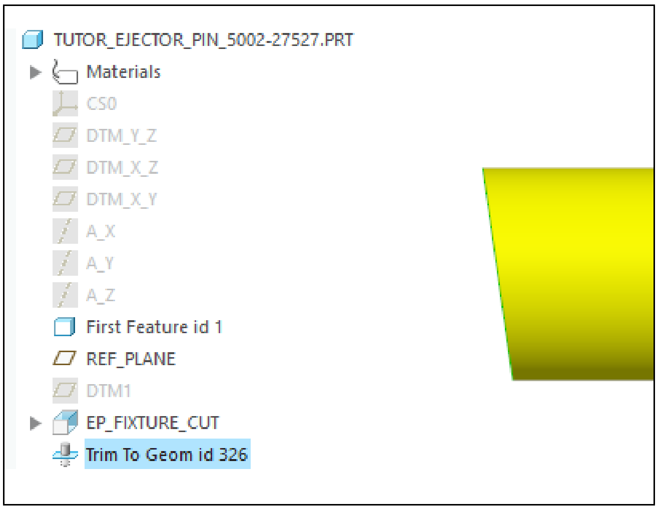
8. Activate **Trim to quilt/refmodel** option.
9. Select **Rotfix1** from the **Fix Rotation** pull-down list.

   The former counter bore in the ejector retainer plate is replaced by a new UDF with the rotation fix geometry. The ejector pin geometry is also modified to fit to the new cutout.
10. Switch to the **Options** tab and activate the **Pattern for all models** option.
11. Close the **Ejector Pin** dialog box with **OK**.

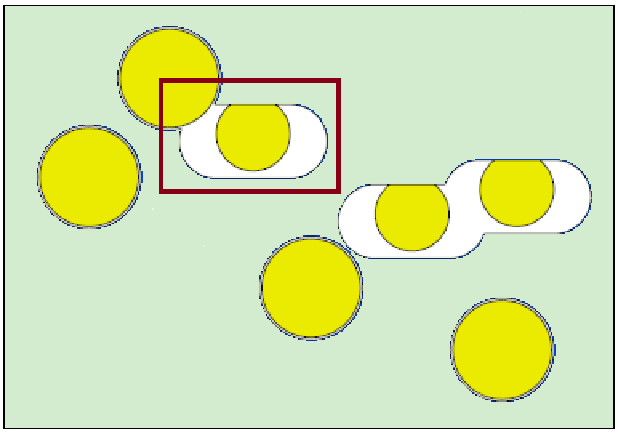
    The cut-outs are generated and the ejector pin is assembled.



A **Trim To Geom** feature is added to each ejector pin to trim it against the reference model.



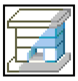
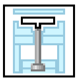
The rotation fixture UDF is added to the ejector retainer plate and the ejector models a modified.



17.4 To Assemble Ejector Sleeves

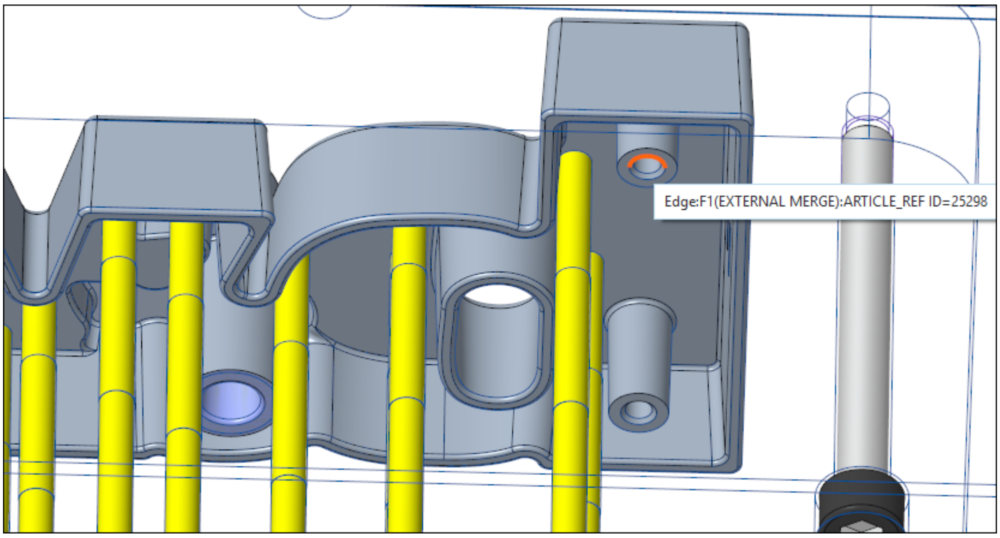
Download 34 To Assemble Ejector Sleeves to start from with this chapter.

17.4.1 Assemble the Small Ejector Sleeves

1. Click **EMX Assembly** ► **View** ►  **Wireframe Style**.
2. Click **EMX Assembly** ► **View** ► **Show** ►  **Moving Half**.
3. Click **EMX Components** ►  **Ejector Pin**.

   The **Ejector Pin** dialog box opens.

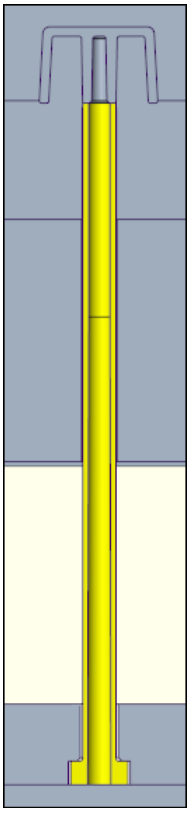
   The `TUTOR_MH_EPINS` assembly is preselected.
4. Select `knarr` from the **Supplier** pull-down list.
5. Select `38188` from the **Types** pull-down list.
6. Click RMB on the **Value** column of **Diameter**.
7. Pick **Measure Diameter** from the Measure Popup Menu.
8. Select the edge as displayed below. The measured diameter will be displayed in an EMX message box with `4.221` mm



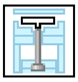
9. Set the inner diameter **DM1** value to `4.5` mm which is the next larger diameter comapre to the measured value.
10. Set the outer diameter **DM3** value to `7 mm`.
11. Define the references
    a. For **(1) Point** select a point from the multi point feature `PNTS_SLEEVE_1` in the `ARTICLE_REF.PRT` model.

       The **LG1** value is automatically set to `150` mm.
12. Make sure **Pattern for all models** is enabled.
13. Make sure **Individual ejector models on each point** is enabled.
14. Make sure that the **Auto Length** check box is enabled.
15. In the UDF Dimension set the **LL** value to `25` mm.
16. Close the dialog Box with **OK**.

The ejector sleeves are added to the moldbase

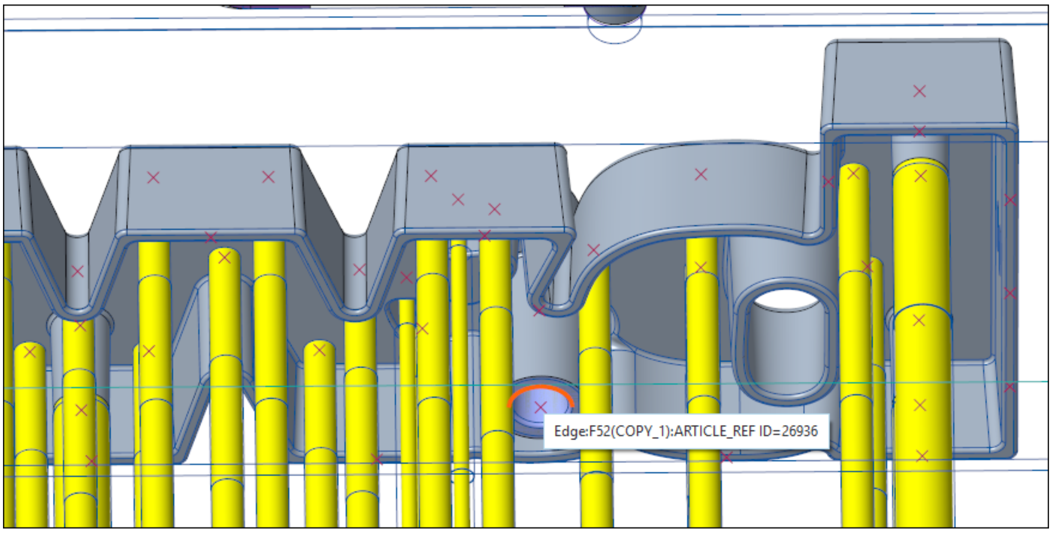


17.4.2 Assemble the Large Ejector Sleeves

1. Click **EMX Components** ►  **Ejector Pin**.

   The **Ejector Pin** dialog box opens.

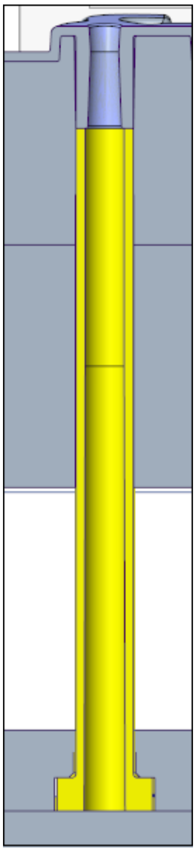
   The `TUTOR_MH_EPINS` assembly is preselected.
2. Select `knarr` from the **Supplier** pull-down list.
3. Select `38188` from the **Types** pull-down list.
4. Click RMB on the **Value** column of **Diameter**.
5. Pick **Measure Diameter** from the Measure Popup Menu.
6. Select the edge as displayed below. The measured diameter will be displayed in a message box with `8.023` mm



7. Set the inner diameter **DM1** value to `8.2` mm which is the next larger diameter compare to the measured value.
8. Define the references
   a. For **(1) Point** select a point from the multi point feature `PNTS_SLEEVE_2` in the `ARTICLE_REF.PRT` model.

      The **LG1** value is automatically set to `150` mm.
9. Make sure **Pattern for all models** is enabled.
10. Make sure that the **Auto Length** check box is enabled.
11. In the UDF Dimension set the **LL** value to `30` mm.
12. Close the dialog Box with **OK**.

The ejector sleeves are added to the mold base



17.5 To Assemble Shims for Core Pins from the Library

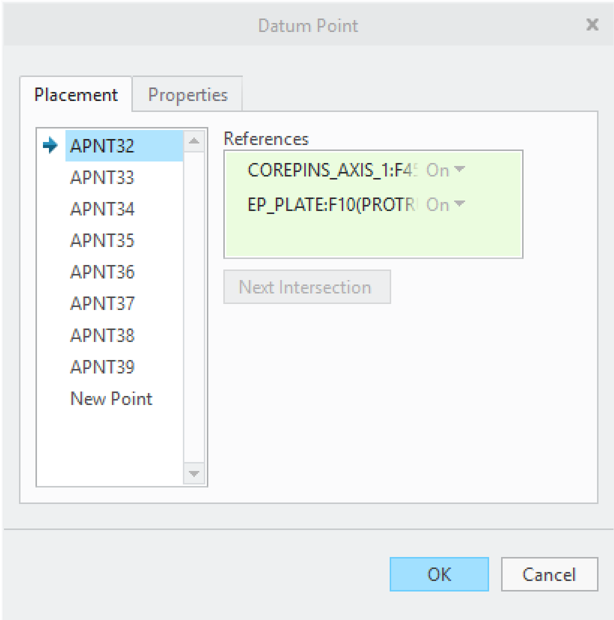
Download 35 To Assemble Shims for Core Pins from the Library to start from with this chapter.

17.5.1 Create the Placement Coordinate Systems

1. Open the `TUTORIAL.ASM`
2. Activate the `TUTOR_MH.ASM`
3. Click  **Point** to create a datum point feature.

   The **Datum Point** dialog box opens.

4. Create 8 points in total within this feature.



For each point select as the first placement references the following axes:

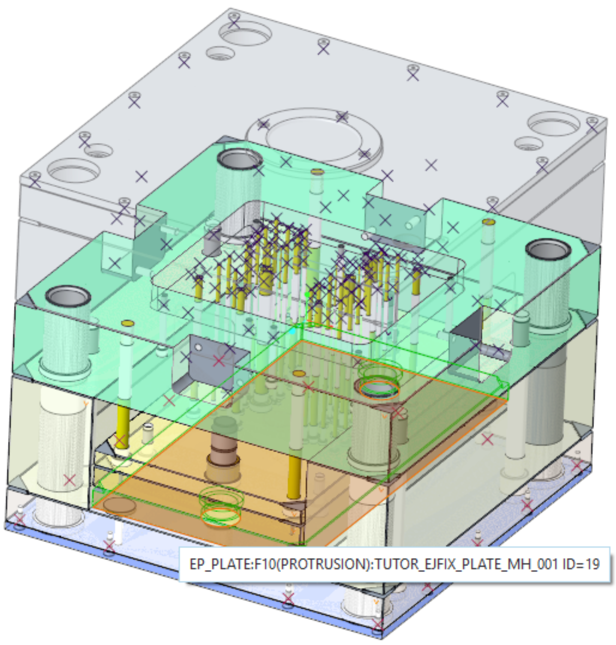
a. `MFG1.ASM` — `ARTICLE_REF.PRT` — `COREPINS_AXIS_1` (3 axis in pattern)

b. `MFG1.ASM` — `ARTICLE_REF.PRT` — `COREPINS_AXIS_2` (1 single axis)

c. `MFG2.ASM` — `ARTICLE_REF.PRT` — `COREPINS_AXIS_1` (3 axis in pattern)

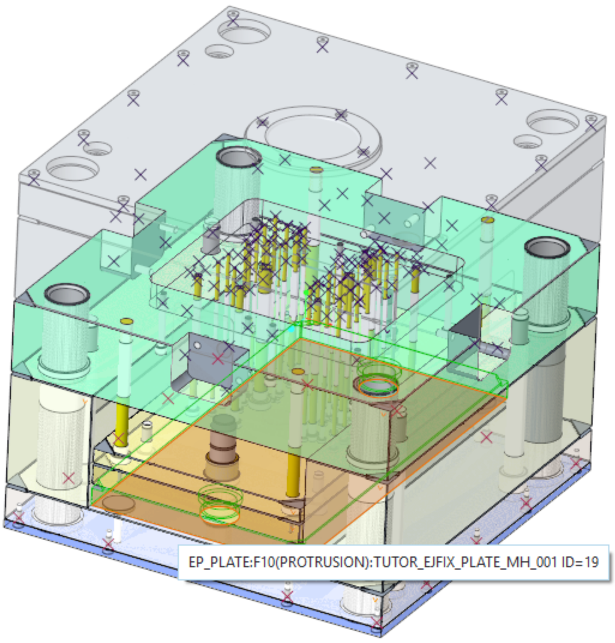
d. `MFG2.ASM` — `ARTICLE_REF.PRT` — `COREPINS_AXIS_2` (1 single axis)

As second placement reference select the bottom surface of core pin retainer plate `TUTOR_EFIX_PLATE_MH_001.PRT`.

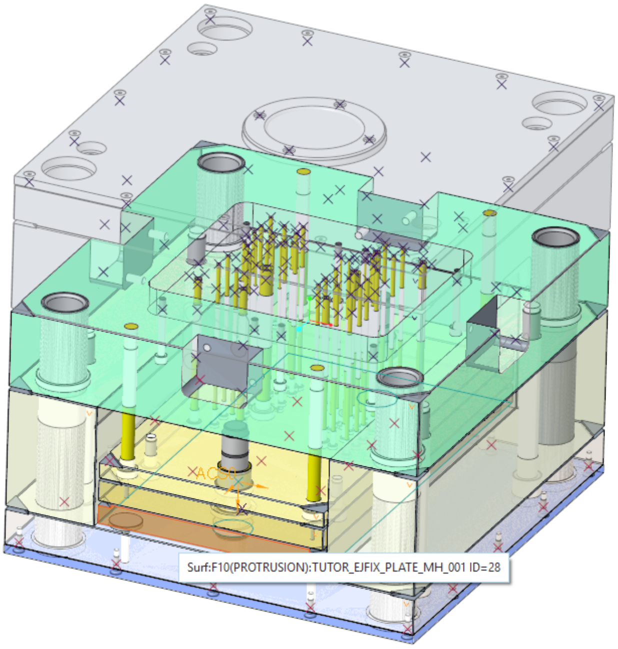


5. Click **OK** to create the datum point feature
6. Click **Coordinate System**.

   The **Coordinate System** dialog box opens.
7. Select the previously created datum points as placement references.
8. Switch to the **Orientation** tab
9. Select as first orientation plane the bottom surface of core pin retainer plate `TUTOR_EFIX_PLATE_MH_001.PRT` and set the direction to **Z**.



10. Select as second orientation plane the side surface of core pin retainer plate TUTOR_EFIX_PLATE_MH_001.PRT and set the direction to **X**.



11. Make sure the orientation is correct for the library component that will be assembled in the next section.
12. Click **OK** to create the coordinate system feature
13. Select the recently created coordinate system
14. Click on **Pattern**.
15. Set **Point** from the **Select Pattern Type** pull-down list.
16. Select **From datum point** for **Set Type Settings**.
17. Select the previously created datum point feature.
18. Click **OK** to create the pattern.
19. Activate the main assembly TUTORIAL.ASM

17.5.2 Assemble the Shims from the Library

1. Open the moving half subassembly TUTOR_MH.ASM
2. Click **EMX Components ▸ Library ▸ Assemble**

   The **Library Component** dialog box opens.
3. Select the folder meusburger. from the tree.
4. Double–click on the E2680 library component in the list of **Library Components**.

   The **Component** tab is activated.
5. Select a **Length (LG)** value of 40 mm.

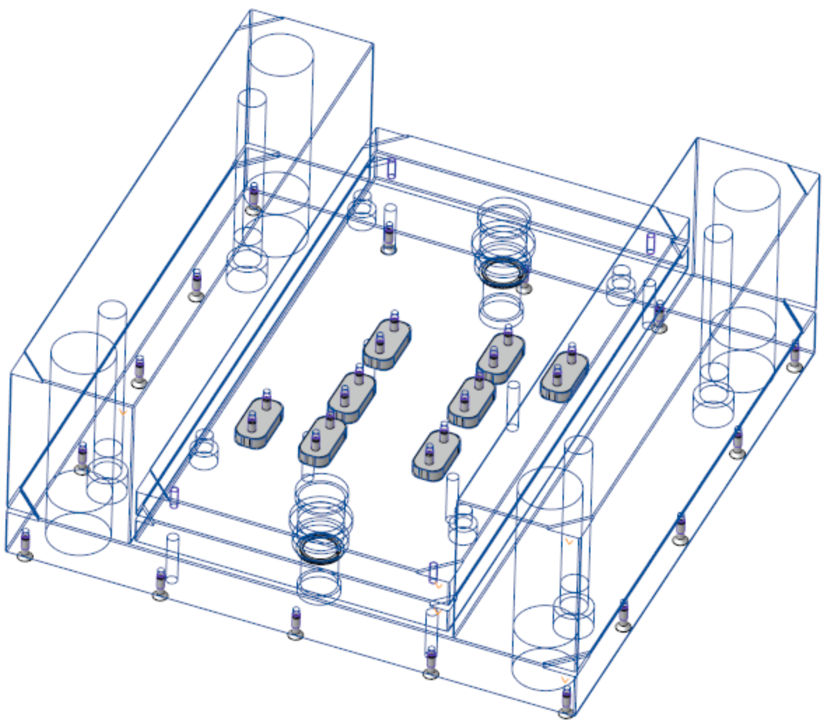
6. As the assembly reference **(1) CSYS** select the previously created patterned coordinate system.
7. Enable the option **Pattern Component**.
8. Click on **Preview** and check if component is correctly oriented.
9. Click **OK** to assemble the shims.

17.5.3 Assemble the Predefined Screws defined on the Shims

1. Open the moving half subassembly `TUTOR_MH.ASM`
2. Click **EMX Components ► Component Handling ► Assemble predefined components**

   The Selection window opens.
3. Select the first instance in the pattern of the previously create shim.

   All predefined screws and the cut-outs are assembled.
4. Leave the Selection window with **Cancel**.
5. Close the `TUTOR_MH.ASM` to return to the main assembly.



17.6 To Assemble Ejector Core Pins

Download 36 To Assemble Ejector Core Pins to start from with this chapter.

17.6.1 Add the Trimmed Core Pins to the Large Ejector Sleeve

1. Click **EMX Assembly ▸ View ▸** **Wireframe Style**.
2. Click **EMX Assembly ▸ View ▸ Show ▸** **Moving Half**.
3. Click **EMX Components ▸** **Ejector Pin**.

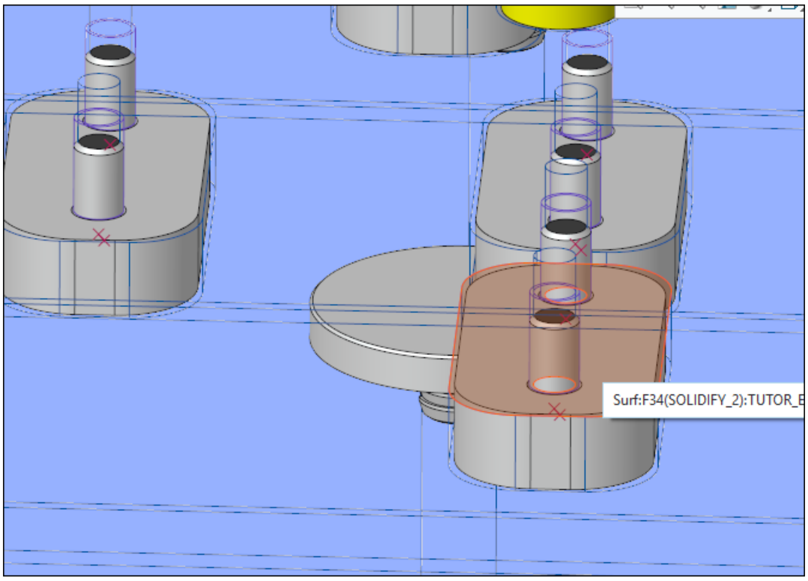
   The **Ejector Pin** dialog box opens.

   The `TUTOR_MH_EPINS` assembly is preselected.
4. Select `knarr` from the **Supplier** pull-down list.
5. Select `37111` from the **Types** pull-down list.
6. Set the **DM1** value to `4.5` mm
7. Set the **REF_ANGLE** value to `90` degree
8. Define the references
   a. Click **(1) Point** to select the point from the multi point feature `PNTS_COREPIN_1` in the `ARTICLE_REF.PRT` model.

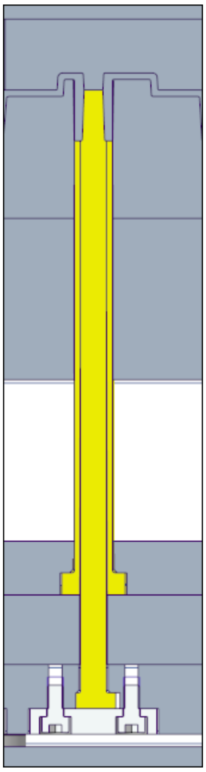
      The **LG1** value is automatically set to `160` mm.

   b. Click **(2) Surface** to select a different placement plane.

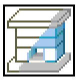
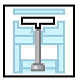
      Select the top surface of the cutout created by the shims in `TUTOR_EFIX_PLATE_MH001.PRT`.



      The **LG1** value is updated to `200` mm.
9. Make sure that the **Trim to quilt/ refmodel** check box is enabled.
10. Select **Rotfix1** from the **Fix Rotation** pull-down list.
11. Make sure **Pattern for all models** is enabled.
12. Close the dialog box with **OK** to assemble the trimmed core pins.



17.6.2 Add the Trimmed Core Pins to the Small Ejector Sleeve

1. Click **EMX Assembly** ► **View** ►  **Wireframe Style**.
2. Click **EMX Components** ►  **Ejector Pin**.

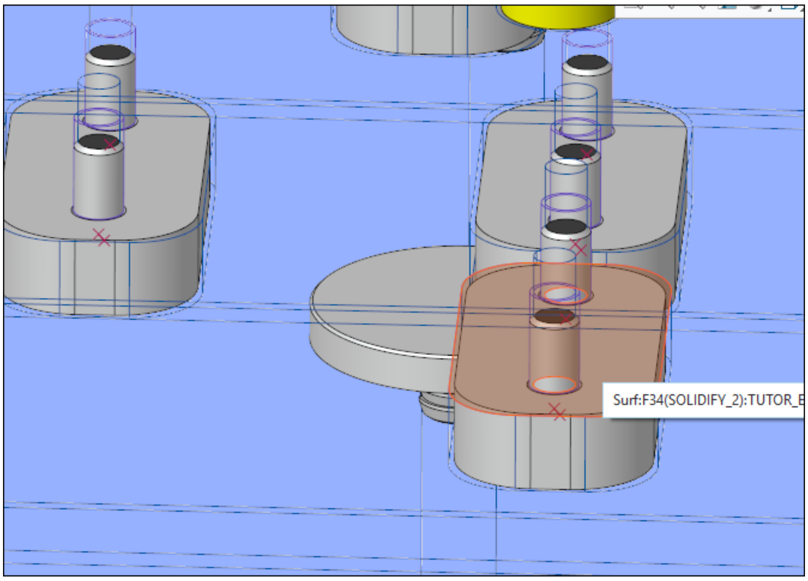
   The **Ejector Pin** dialog box opens.

   The TUTOR_MH_EPINS assembly is preselected.
3. Select knarr from the **Supplier** pull-down list.
4. Select 37111 from the **Types** pull-down list.
5. Set the **DM1** value to 8.2 mm
6. Set the **REF_ANGLE** value to 90 degree
7. Define the references
   a. Click **(1) Point** to select the point from the multi point feature PNTS_COREPIN_2 in the ARTICLE_REF.PRT model.

      The **LG1** value is automatically set to 160 mm.

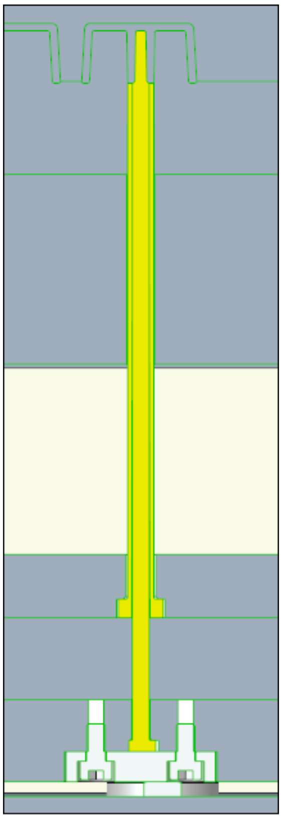
   b. Click **(2) Surface** to select a different placement plane.

      Select the top surface of the cutout created by the shims in TUTOR_EFIX_PLATE_MH001.PRT.



The **LG1** value is updated to `200` mm.

8. Make sure that the **Trim to quilt/ refmodel** check box is enabled.
9. Select  **Rotfix1** from the **Fix Rotation** pull-down list.
10. Make sure **Pattern for all models** is enabled.
11. Close the dialog box with **OK** to assemble the trimmed core pins.



17.7 To Assemble Core Pins in Insert Parts using the Option “As Core Pin in One Plate”

Download 37 To Assemble Core Pins in Insert Parts using the Option As Core Pin in One Plate to start from with this chapter.

17.7.1 Assemble Core Pin to Insert Parts On the Fixed Half

1. Click **EMX Assembly ► View ► Wireframe Style**.
2. Click **EMX Components ► Ejector Pin**.

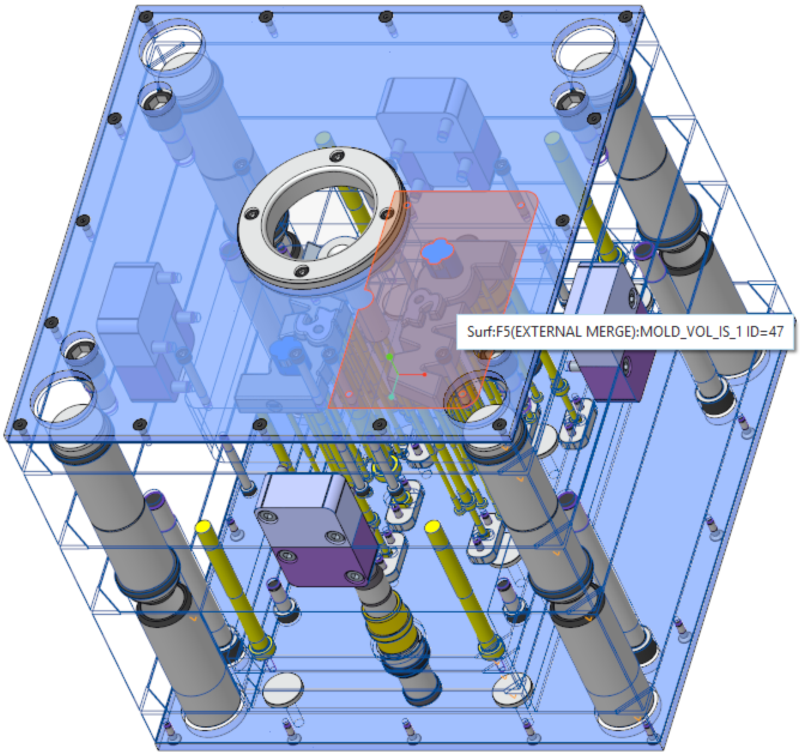
   The **Ejector Pin** dialog box opens.
3. Select `TUTOR_SHARED` as target subassembly from the **subassembly** pull-down list.
4. Select `knarr` from the **Supplier** pull-down list.
5. Select `37111` from the **Types** pull-down list.
6. Set the **DM1** value to `8.2` mm
7. Define the references

   a. Click **(1) Point** to select the point from the multi point feature `PNTS_COREPIN_2` in the `ARTICLE_REF.PRT` model.

      The **LG1** value is automatically set to `160` mm.

   b. Click **(2) Surface** to select a different placement plane.

      Select the top surface of the insert part `MOLD_VOL_IS`.

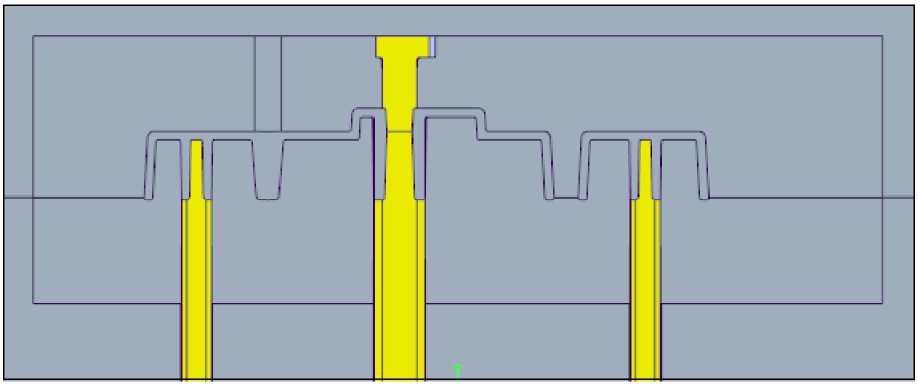


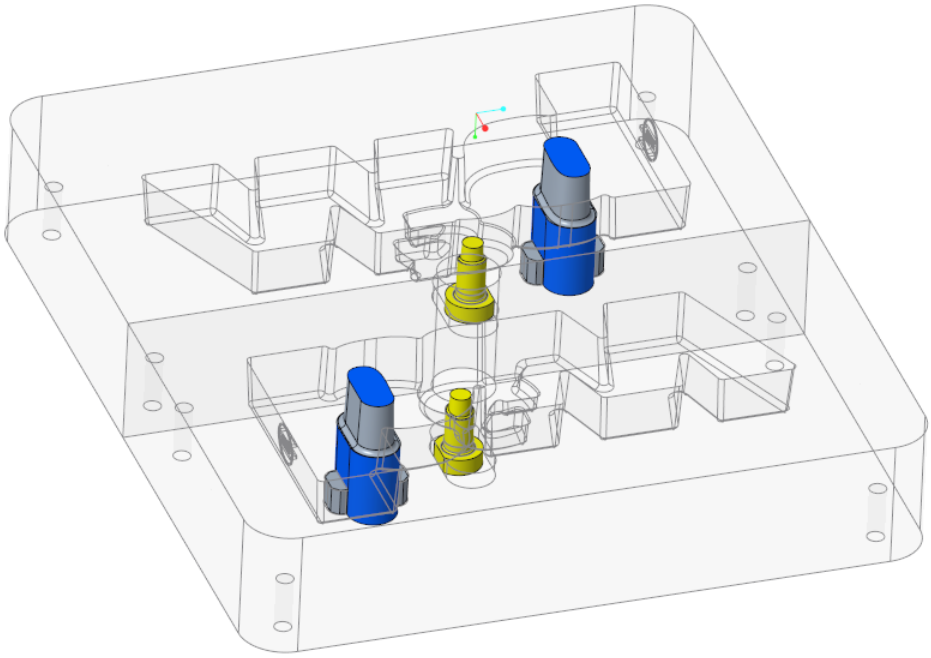
The **LG1** value is updated to `100` mm.

8. Make sure that the **Trim to quilt/ refmodel** check box is enabled.
9. Make sure that the **As core pin in one plate** check box is enabled.

   NOTE:
   As soon as this option is enabled the UDF image changes. A new UDF will be used which created the ejector guide bore and the ejector head bore in one plate.

10. Select **Rotfix1** from the **Fix Rotation** pull-down list.
11. Make sure **Pattern for all models** is enabled.
12. Close the dialog box with **OK** to assemble the trimmed core pins.





17.8 About Creating Ejector Pins with the Ejector Pin Designer

Use the **Ejector Pin Designer** to speed up the process of defining ejector pins. It takes fewer clicks to predefine ejector pins on reference models and to finally assemble ejector pin models and cuts to the mold base.

1. Make sure that all cavity insert assemblies are classified correctly.

   The **Ejector Pin Designer** only works on models that are classified as reference models.

2. The **Ejector Pin Designer** dialog box can be opened by clicking **EMX Components ▸**  **Ejector Pin Designer**.

   A simplified representation is activated that only contains the classified reference models.

   The **Ejector Pin Designer** dialog box contains the two tabs described below:

   - **Design**—Contains functions for predefining and assembling ejector pins.
   - **Options**—Contains filters and options to help you design the ejector pins.

   During the initialization of the dialog the **Reference Models** list is populated.

   The number of entries depends on how many reference models have been assembled to the mold base.

3. In the **Design** tab click the **Reference Model** box, and select your active reference model.

   The active working model is the model in which you can define, modify, or delete ejector pins.

4. In the **Placement Location** area, manage your active placement location for the ejector pin head.

   The **Ejector Pin Designer** supports up to four different placement planes on both sides of the mold base, the moving half, and the fix half.

   To setup the active placement location select **MH** or **FH**. The list of valid placement planes is populated depending on the current mold base design.

   The following plates are supported:

   - **Ejector Plate #1**: Ejector Pin Head is placed to the first ejector pin plate.
   - **Ejector Plate #2**: Ejector Pin Head is placed to the second ejector pin plate.
   - **Core Pin Plate**: Ejector Pin Head is placed to core pin plate.
   - **Clamping Plate**: Ejector Pin Head is placed to clamping plate.

5. Notice the ejector pin that appears in the **Type Preselection** area which specifies the shape of the preview that is displayed while placing a new ejector pin.

   The following information is listed for the preselected component:

   - Shape—Round or rectangular
   - Size—Diameter or length multiplied by width
   - Rotation Angle
   - Unit system
   - Supplier
   - Instance Type
   - Target Subassembly

6. Click to preselect another ejector pin type.

   The **Ejector Pin** Dialog Box opens. Select new ejector pin type and click **OK**.

   NOTE:
   Click **EMX Assembly ▸ Options** and set the configuration option `EJP_DESIGNER_DEFAULT_COMPONENT`. By default, what you set is assigned to the preselected component. A valid format for this EMX option is `unit|supplier|instance_name`, such as `mm|meusburger|E1725_2_8_0_5_80`

7. In the **Ejector Pin Designer** dialog box, click the **Options** tab to set snapping rules and collision control to support your waterline creation.

   - Select or clear the check boxes under **Snap Rules** for support during your waterline creation process.

     The **Ejector Pin Designer** recognizes all point features in the reference model and helps you to define new ones based on their positions.
   - Select the **Grid** check box and then set **Width** to a specified value.

     This sets the accuracy of the decimal digits.
   - Set the **Snap Tolerance**.

If the snap tolerance is higher than the grid width, EMX can snap to existing points.

8. From the **Ejector Pin Designer** dialog box on the **Design** tab under **Ejector Pin Definition**, you can create, modify, and delete predefinitions for ejector pins.

   As you create ejector pins, the ejector pins for the currently selected reference model appear in a list on the bottom of the dialog box.

   The steps for creating, modifying, and deleting ejector pins and for assembling predefined components of the ejector pins, are described below:

   - To create an ejector pin, follow the steps below:

     a. Click to start defining a new ejector pin.

        The coordinates of your pointer position are visible with the origin on the reference model standard coordinate system.

        In the graphics window, the ejector pin cross section is previewed at the pointer position.

     b. Click anywhere within the outline boundaries of the reference to define the intersecting point of the ejector pin with the reference model.

        The point feature and a layout curve are assembled to the plate.

        The layout curve represents the dimensions of the predefined ejector pin. The **Ejector Pin** dialog box opens.

     c. Finish your ejector pin type selection. You can select different suppliers, types, and dimensions. Click **Preview** to preview the layout curve.

     d. Click **OK** to complete component predefinition and close the dialog box for the component or click **Cancel** to stop the definition process.

   - To create an ejector pin on existing points follow the steps below:

     a. Click to add a new ejector pin on existing points.

     b. Select a sketch point or datum points inside of the reference model.

        The **Ejector Pin Designer** dialog box opens.

     c. Finish the type selection for your ejector pin.

        You can select different suppliers, types, and dimensions.

     d. Click **Preview** to preview the layout curve.

     e. Click **OK** to complete component predefinition and close the dialog box for the component or click **Cancel** to stop the definition process.

     NOTE:
     When creating ejector pins on existing points, you can define multiple ejector pins in one step, depending on the number of points defined in the sketch or the datum point feature.

     The position of the ejector pin is updated if sketched points are modified.

   - To modify an ejector pin follow the steps below:

     a. Click to activate the modify mode for existing ejector pins.

b. Place the pointer on one ejector pin until the color of the knot changes to red and then click the knot and move the pointer.

The selected ejector pin follows the pointer within the plate.

c. Click again to place the selected ejector pin at the current position of the pointer.

The ejector pin point is updated to the new position. Middle–click when you want to stop modifying.

d. Right–click one of the ejector pins that changed to red to open the **Ejector Pin** dialog box.

In the **Ejector Pin** dialog box you can redefine the ejector pin type and dimension.

Click **OK** to close the **Ejector Pin** dialog box.

- To delete a waterline, follow the steps below:

a. Click and then place the pointer on the ejector pin to be removed.

The ejector pin changes to red.

b. Click the ejector pin to remove it.

The ejector pin, the component, and the ejector pin cut out are deleted from the mold base.

9. After you finish defining the ejector pins within the reference model, you can assemble all components including cut outs for all defined ejector pins in the current reference model.

Click **Update Ejector Pins in 3D** to assemble the ejector pins and the cut outs after defining ejector pins, and when applying modifications.

If the ejector pin solid model is already in the assembly, the representing line in the table of all ejector pins is locked .

If a modification should be applied to the ejector pin definition, double-click the icon and set the line to +unlocked .

Only unlocked ejector pins can be modified and reassembled with **Update Ejector Pins in 3D**.

10. As ejector pins are created, they appear in the bottom of the **Ejector Pin Designer** dialog box.

Review the bullets listed below for information on customizing your list of ejector pins:

- In the list, an ejector pin is represented with a name in the **Instance** column.

  Double-click the instance name to open the **Ejector Pin** dialog box where you can define the ejector pin and the associated ejector pin cut out.

- Directly change the position of the ejector pin by editing the cell values in the **X** and **Y** columns.

- Directly rotate the ejector pin can by editing the cell value in the column.

- The trim behavior of one ejector pin is visualized in the column.

  The green hook represents trimming.

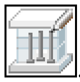
Double–click to toggle this value directly within the table.

17.9 To Assemble Ejector Pins using the Ejector Pin Designer

Download 38 To Assemble Ejector Pins using the Ejector Pin Designer to start from with this chapter.

17.9.1 Start the Ejector Pin Designer

1. Open the assembly `TUTORIAL.ASM`

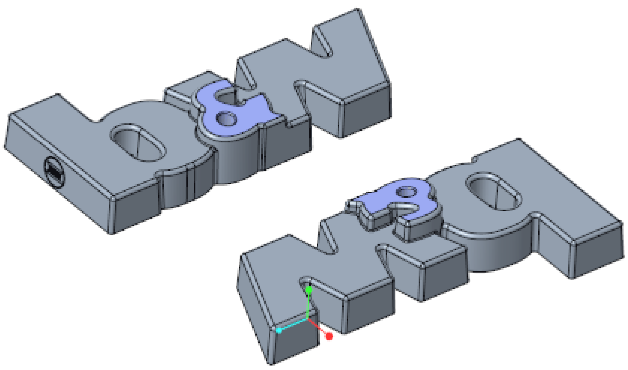
2. Click **EMX Components ► Ejector PinsEjector ► Pin Designer** .

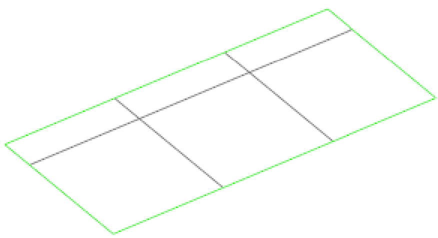
   The **Ejector Pin Designer** opens.

   A simplified representation which only contains the reference models and all ejector pins defined with the **Ejector Pin Designer** is activated.

   So far no ejector pins are defined by the Ejector Pin Designer.

   The outline of the reference model is displayed as a green rectangle.





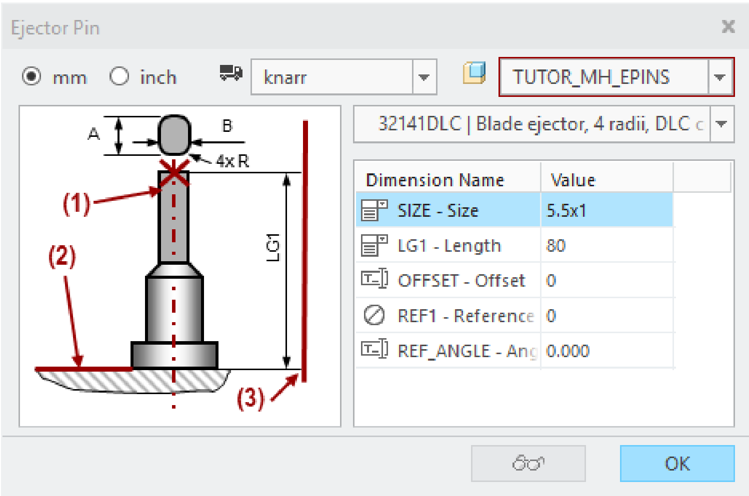
1. Use the default setting:

- Define on the reference model `ARTICLE_REF.PRT`.
- Assemble to the moving half **MH**.
- Place the ejector to the first ejector package **Ejector Plate #1**.

17.9.2 Manage Type Preselection

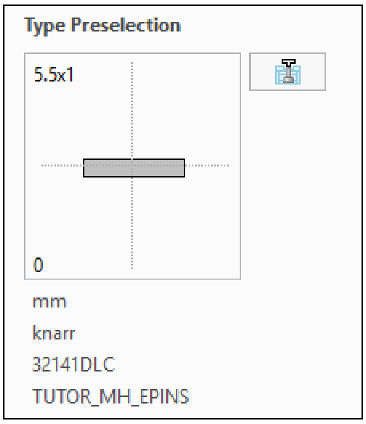
1. Click to preselect another ejector pin type.

   The **Ejector Pin** Dialog Box opens.
2. Select `knarr` from the **Supplier** pull-down list.
3. Select `32141DLC` from the **Types** pull-down list.
4. Select a **SIZE** of `5.5x1`.



5. Click **OK** to leave the dialog box.

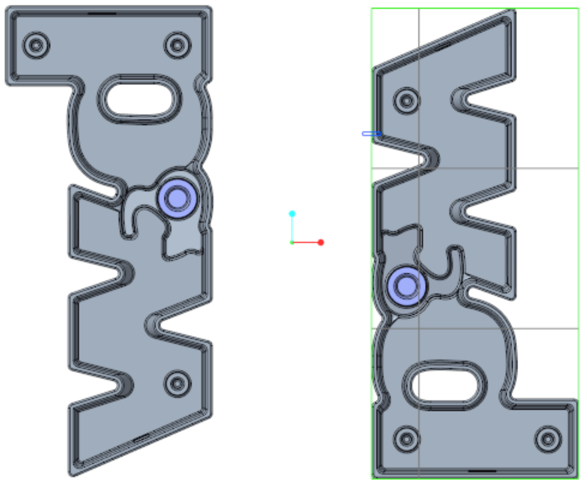
   The preselected ejector pin information will be displayed in the dialog.



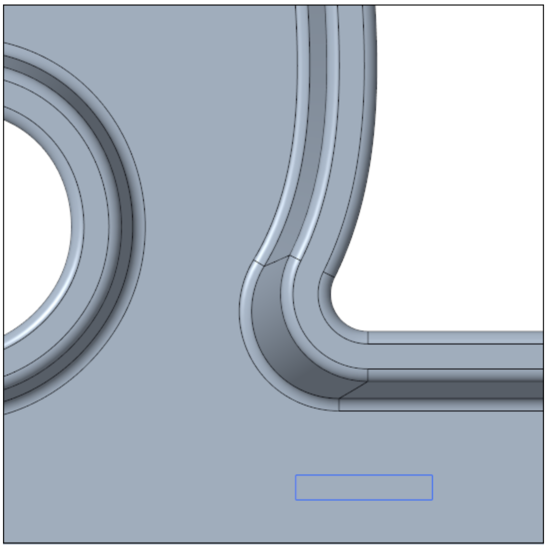
17.9.3 Predefined Ejector Pin dynamically

1. Click to start defining a new ejector pin.

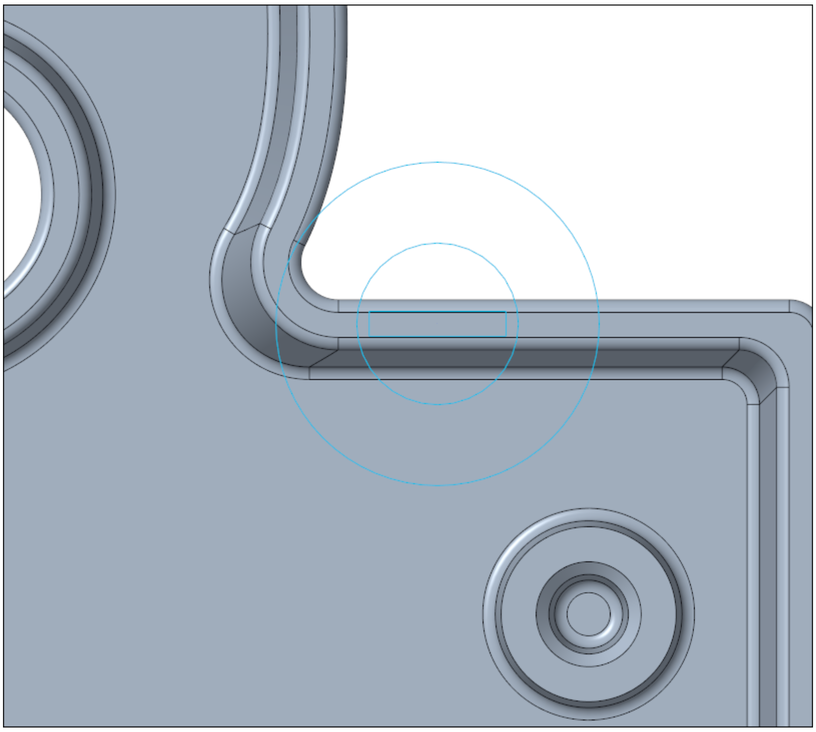
The view rotates to the bottom.



2. In the graphics window, the ejector pin cross section is previewed at the pointer position.



3. Click anywhere within the outline boundaries of the reference to define the intersecting point of the ejector pin with the reference model.



The point feature and a layout curve are assembled to the plate.

The layout curve represents the dimensions of the predefined ejector pin.

The **Ejector Pin** dialog box opens.

The previously preselected type is displayed in the **Ejector Pin** dialog box.

NOTE:
You can select different suppliers, types, and dimensions

4. Click **OK** to complete component predefinition and close the dialog box for the component.

   The ejector pin will be added to the list of ejector pins.

5. Click middle-mouse button to leave the interactive predefinition.

17.9.4 Working with list of Ejector Pins

As ejector pins are created, they appear in the bottom of the **Ejector Pin Designer** dialog box.

	Σ	Instance	X	Y				
	1x	32141DLC_5_5_1_160	23.155	48.195	0.000	MH		

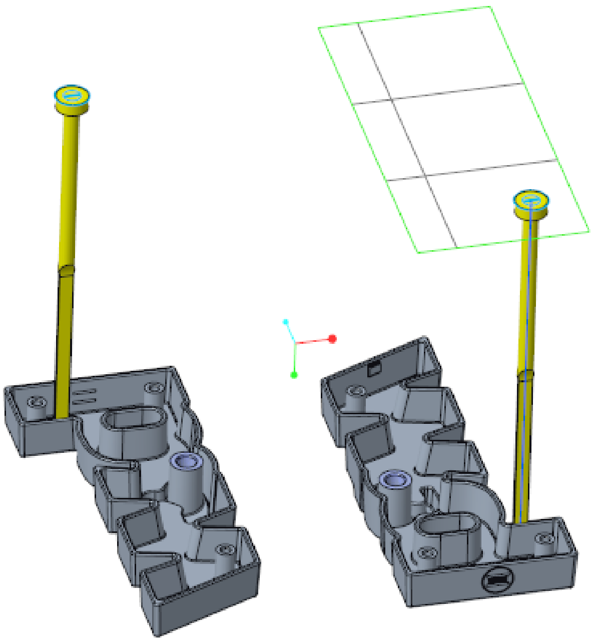
1. Double-click on **X** value and set it to `23.100`
2. Double-click one **Y** value and set it to `48.000`.

   The curve will be moved top the new position. In case the position is invalid the old value remains.

17.9.5 Assemble the predefined Ejector Pins

1. Click **Update Ejector Pins in 3D**

   The ejector pins will be assembled and create the cut-outs in all interfering plates and components.



17.9.6 Delete Ejector Pins

1. Click with right-mouse button on the recently created ejector pin in the table and click **Delete Ejector Pin**

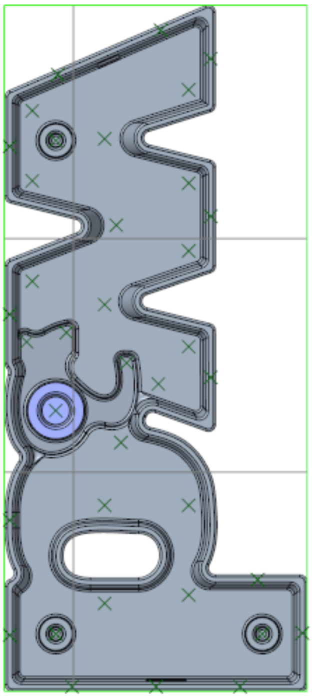
   The ejector pin, the component, and the ejector pin cut out are deleted from the mold base.

17.9.7 Predefined Flat Ejector Pins with 0° Rotation on existing points

1. Make sure all points in the reference model are visible.
2. Click to predefine a new ejector pin on existing points.

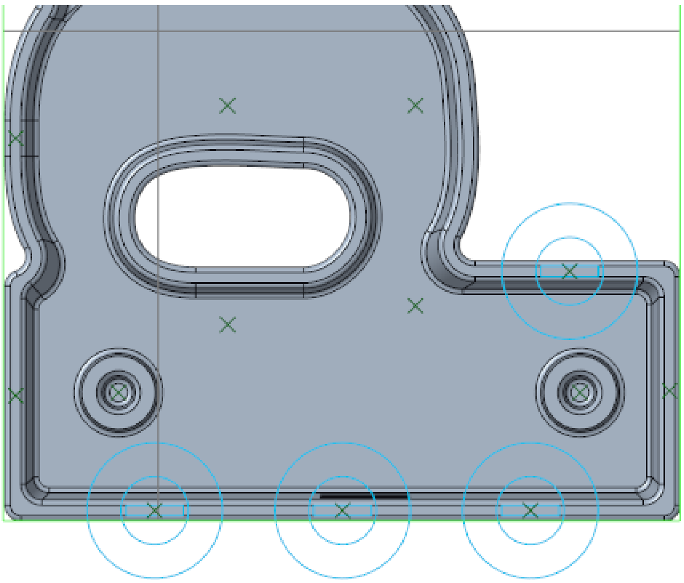
   the view is rotated to the bottom view.

   The Selection Window opens.



3. In the graphics widow select a point from the multiple point feature PNTS_FLAT_EPIN_5-5X1MM_0-DEG.

   The point feature and a layout curve are assembled to the plate.



4. Make sure **Trim to quilt/refmodel** is activated.
5. Click **OK** to finish the predefinition.

   The ejector pins are added to list.

17.9.8 Predefined Flat Ejector Pins with 90° Rotation on existing points

1. Click to predefine a new ejector pin on existing points.

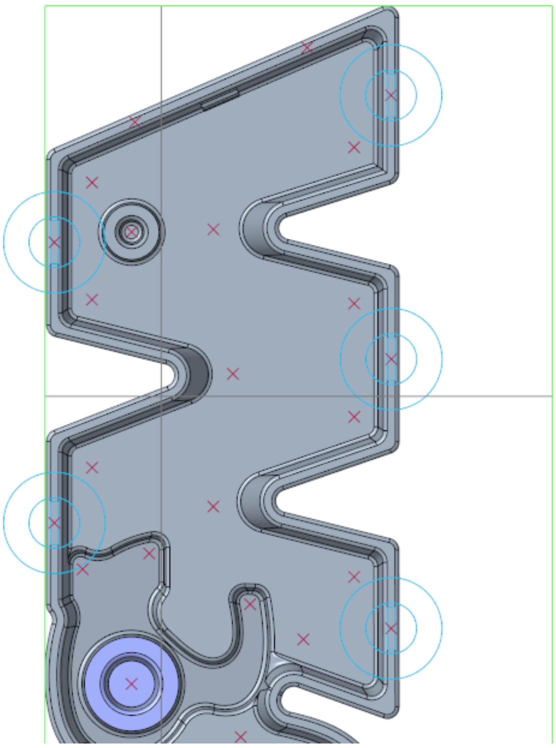
   The Selection Window opens.

2. In the graphics widow select a point from the multiple point featurePNTS_FLAT_EPIN_5-5X1MM_90-DEG.

   The point feature and a layout curve are assembled to the plate.

3. Enter 90 for **REF_ANGLE**.

4. Click **Preview** to update the layout curves.



5. Make sure **Trim to quilt/refmodel** is activated.

6. Click **OK** to finish the predefinition.

   The ejector pins are added to list.

17.9.9 Predefined Flat Ejector Pins with 23° Rotation on existing points

1. Click to predefine a new ejector pin on existing points.

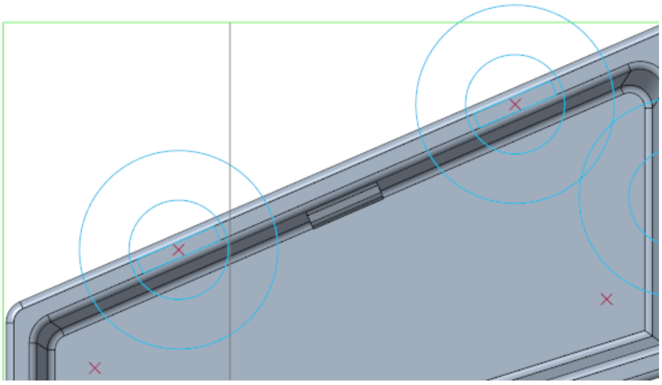
   The Selection Window opens.

2. In the graphics widow select a point from the multiple point feature PNTS_FLAT_EPIN_5-5X1MM_90-DEG.

   The point feature and a layout curve are assembled to the plate.

3. Enter 90 for **REF_ANGLE**.

4. Click Preview to update the layout curves.



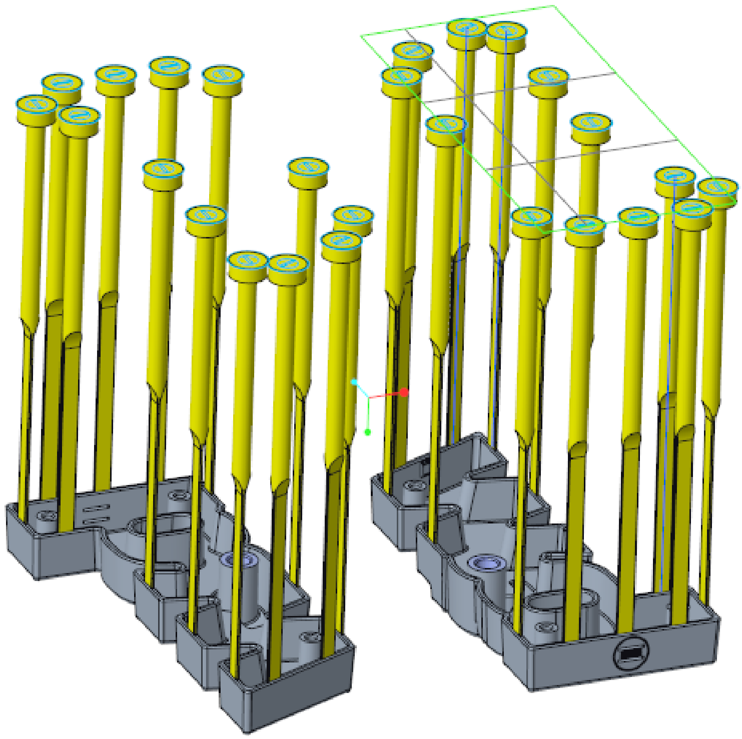
5. Make sure **Trim to quilt/refmodel** is activated.
6. Click **OK** to finish the predefinition.

   The ejector pins are added to list.

17.9.10 Assemble the Flat Ejector Pins on Existing Points

1. Click **Update Ejector Pins in 3D**

The ejector pins will be assembled and create the cut-outs in all interfering plates and components.



2. Leave the **Ejector Pin Designer** with **OK**.

17.10 About Predefining Ejector Pins with Layout Curves in Part Mode

To speed up the process of defining ejectors, you can use this feature to predefine ejector pins with layout curves directly in the article model.

As a result, you get a better view of where the ejector pins are placed on the article and whether their dimensions fit into the design.

You can then assemble all predefined ejector pins at one time within the mold base assembly.

1. Prepare the models where you need to predefine ejector pins:
   a. Click **EMX Components** ► **Ejector Pin** ►  **Identify ejector pin owner models**.
   b. Select one or several models where you need to predefine the ejector pins. This is usually the article model.
   c. Click **OK**. The **Select** dialog box closes.

   d. The selected models are prepared for predefining ejector pins.

2. Predefine the ejectors in the article model:

   a. Open one of the models you prepared in the previous step.
   b. Click **EMX Components ▸ Ejector Pin** to predefine an ejector pin.
   c. Select a supplier, the subtype of the ejector, and edit the dimensions or the options. For more details, see To Define Components on page 122
   d. Note that the assembly placement constraints for **Surface** and **Orientation Surface** are already selected and show the Select icons while the placement constraint for the **Point** is not selected and shows the To be Selected icon
   e. Select a point on the article to place the ejector pin.
   f. Note that under **Options**, the **Predefine Component** check box on page 120 is selected.
   g. Click to assemble the layout curve and to preview the real position and size of the ejector pin on the article
   h. Make as many changes as necessary and preview the results by clicking .
   i. Click **OK** to apply the changes and close the dialog box, or click **Cancel** to close the dialog box and discard all changes.
   j. Repeat these steps until you have defined all required ejector pins

3. The third step of the process is to assemble all predefined ejector pins at one time within the mold base assembly:

   a. Close the window of the article model and open and activate the window of the EMX mold base main assembly.
   b. Click **EMX Components ▸** **Assemble predefined components**.
   c. Select one or more models that have predefined ejector pins. EMX assembles the ejector pins with the predefined settings including components and cuts. For all ejector pins that were assembled successfully, the layout curve created in the previous step is removed.

17.11 To Predefine Ejector Pins in Part Mode with Layout Curves

Download 39 To Predefine Ejector Pins in Part Mode with Layout Curves to start from with this chapter.

17.11.1 Identify the Ejector Pin owner model

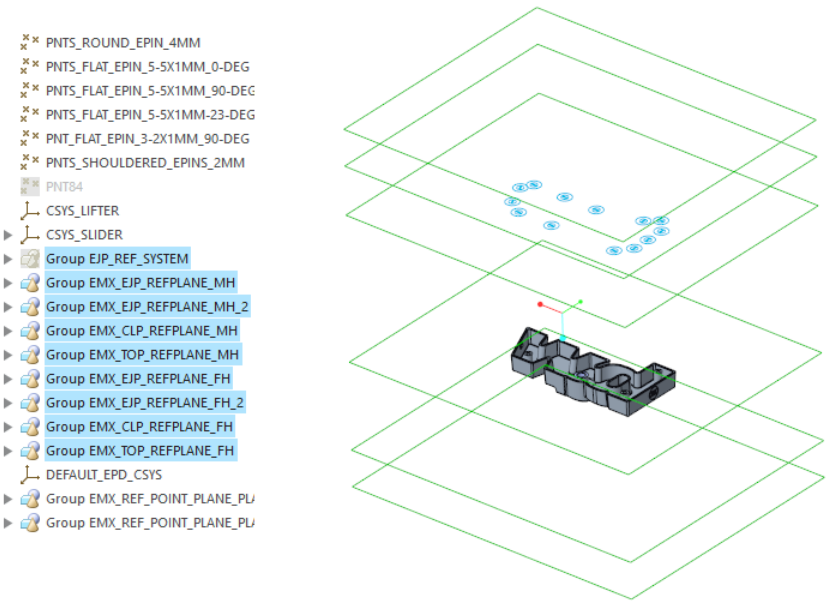
1. Open the `TUTORIAL.ASM`

   Before adding ejector pins in part mode it is necessary to copy the placement reference datum plane for ejector pins in the model. Otherwise the required length of the ejector pins cannot be determined.

2. Select **EMX Components ▸ Ejector Pin ▸ Identify ejector pin owner models** .

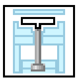
3. Select the model ARTICLE_REF.PRT

   EMX adds two orientation planes normal to the splitting plane and a copy of each clamping plane, ejector reference plane, both fixed half and moving half.



4. Click **OK**.
5. Open ARTICLE_REF.PRT .

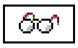
17.11.2 Predefine Ejectors in Part Mode

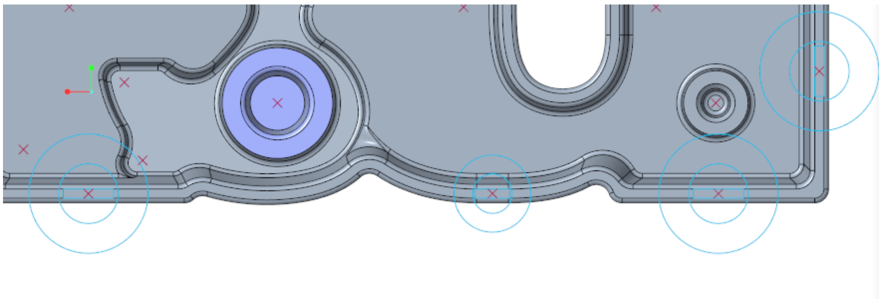
1. Select **EMX Part Mode ► Components ►**  **Ejector Pin** .

   The **Ejector Pin** dialog box shows up.
2. Select the **Supplier** knarr.
3. Select the **Type** 32141DLC.
4. Select **SIZE** 3.8x1.
5. Set the **REF_ANGLE** value to 90.
6. Define the references.

   a. As first Reference **(1) Point** select a point from the multiple point feature PNT_FLAT_EPIN_3-2X1MM_90-DEG.

      NOTE:
      Due to the previous Identify-Step the copy of the ejector retainer datum plane EMX_EJP_REFPLANE_MH is set as default reference. So the REF-distance can be measured like if you would work in assembly mode.
7. Make sure that the option **Pattern for all models** is enable.
8. Make sure **Trim to quilt/refmodel** is enabled.

9. Preview with 



   An additional predefined layout curves of the ejector pin can be seen now.

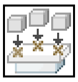
10. Click **OK** to assembled the predefined components reference groups to the model.
11. Close the `ASM_PALM_REF.PRT` and go back to the `TUTORIAL.ASM`.

17.11.3 Assemble Ejector Pins with “Assemble predefined components”

1. Open the `TUTOR_SHARED.ASM`.

   NOTE:
   These predefined components are always assembled to the currently opened Subassembly.

   The ejector pins will be assembled to `TUTOR_SHARED.ASM`.

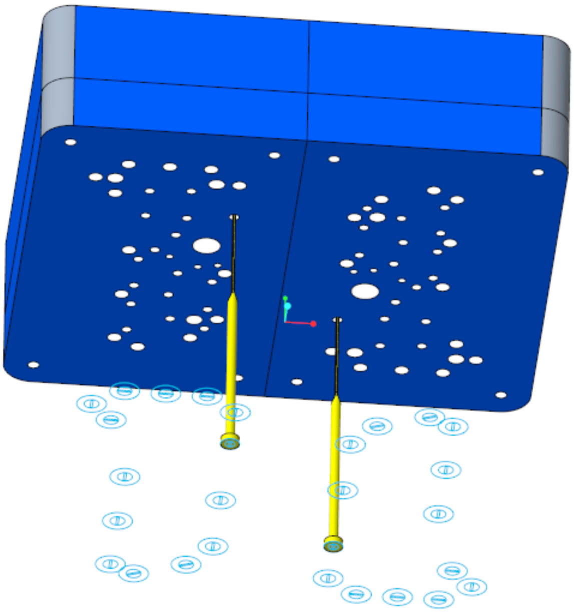
2. Click **EMX Components ► Component Handling ►**  **Assemble predefined components** .

   A selection window shows up and asks for the assembly from which the predefined components should be assembled.

3. Select `ARTICLE_REF.PRT`.

   All ejector pins will be assembled.

Cut-outs are only crated to the insert parts so far..



4. Close TUTOR_SHARED.ASM and return to the main assembly TUTORIAL.ASM.
5. Click **EMX Components ▸ Ejector Pin ▸ Add Ejector Bores** from the **Ejector Pin** overflow menu.

   All missing cut-outs are created to the mold base assembly

17.12 About Trimming Ejector Pin Heads

Use the **Trim Ejector Pin Heads** function in case several ejector pins interfere after assembling the ejector pins.

The **Trim Ejector Pin Heads** function can be started by clicking **EMX Components ▸ Ejector Pin ▸** **Trim Ejector Pin Heads**.

Select a pair of interfering ejector pins to assemble the trim udfs.

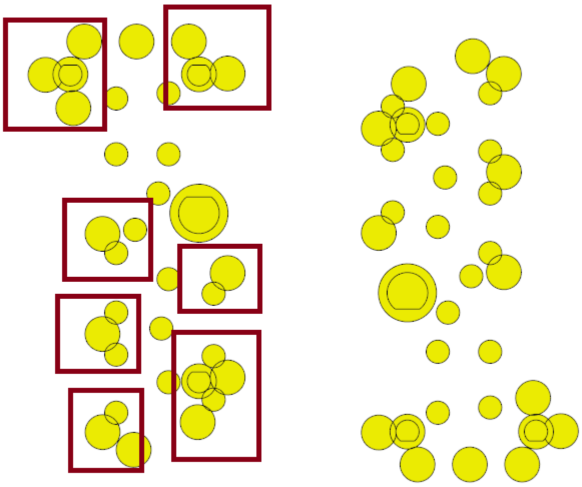
17.13 To Trim the Ejector Pin Heads

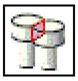
Download 40 To Trim the Ejector Pin Heads to start from with this chapter.

17.13.1 Trim the Ejector Pin Heads

1. Open the TUTOR_MH_EPINS.ASM

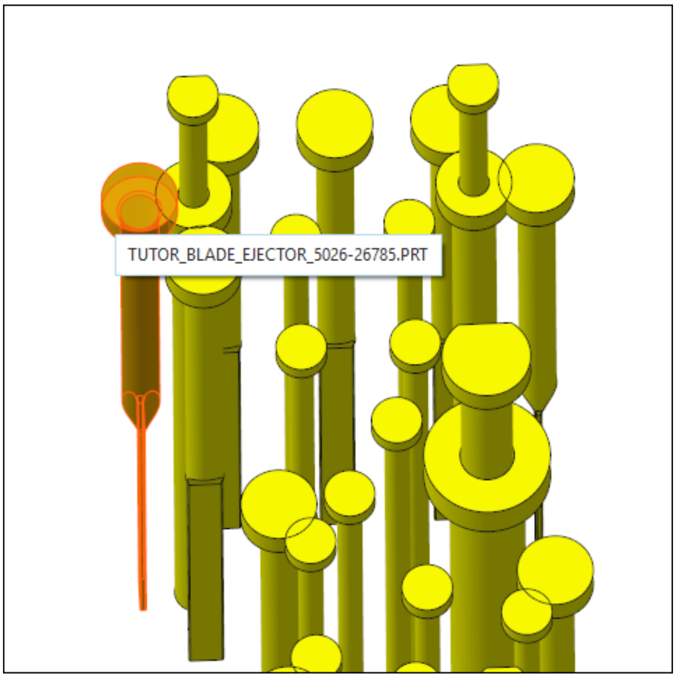
   Some ejector pin heads still interfere.



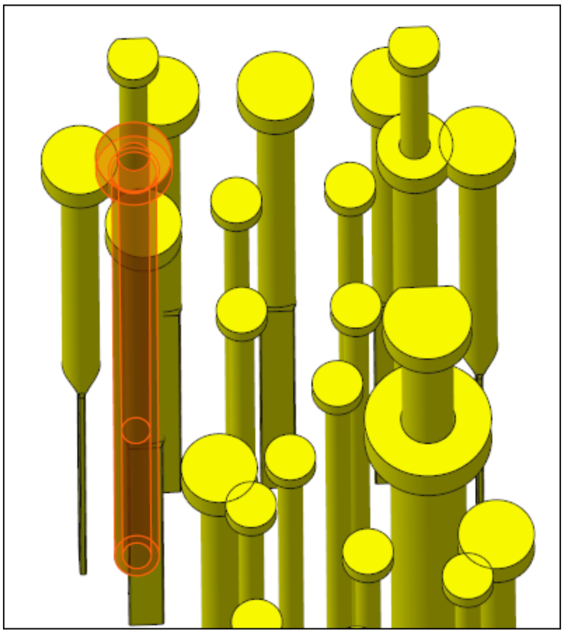
2. Select **EMX Components** ► **Ejector Pin** ►  **Trim Ejector Pin Heads** .

   The Selection Window opens.

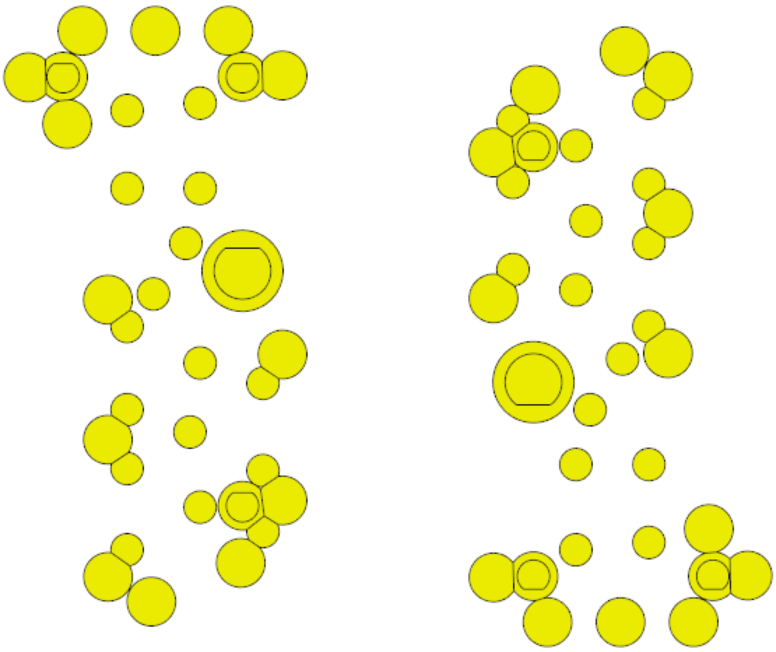
3. Select the first ejector pin of an interfering ejector pin pair.



4. Select the second ejector pin of an interfering ejector pin pair.



5. Continue to select all pairs of interfering ejector pins.



6. Leave selection loop with **Cancel**.
7. Close `TUTOR_MH_EPINS.ASM` to return to the main assembly.

17.14 About Updating Existing Ejector Pins

In can happen that design changes are required after the ejector pins are already assembled.

Certain design changes, i.e. changing the cavity plate thickness or changing the reference model shrinkage require an update of the ejector pin length.

The **Update Existing Ejector Pins** dialog box can be opened by clicking on **EMX Components ▶ Ejector Pin ▶ Update existing ejector pins**.

The dialog box contains a tree table with all assembled ejector pins and the related instances.

In the table columns one can find:

- The **Order Number**
- The **Quantity**
- The **Current Length** is measured at the time the ejector pin is assembled.
- The **Required Length** is measured at the time the **Update Existing Ejector Pin** dialog box is opened.

  The length values is measured for each ejector pin from the selected reference point to the reference surface.

  The values are different in case the mold base design has changed in a way that either the reference point or the reference surface have changed their position.
- The **Status**.

  In case the **Current Length** and the **Required Length** are different the update icon is displayed.

  In case no update is required the OK icon is displayed.

The update procedure can be started with **Update**.

For all ejector pins which are designated to be updated a new instance is selected which exceeds the **Required Length**.

The **Current Length** and the **Order Number** are updated.

17.15 Interlude: Update Existing Ejector Pins

Download 41 Interlude Update Existing Ejector Pins to start from with this chapter.

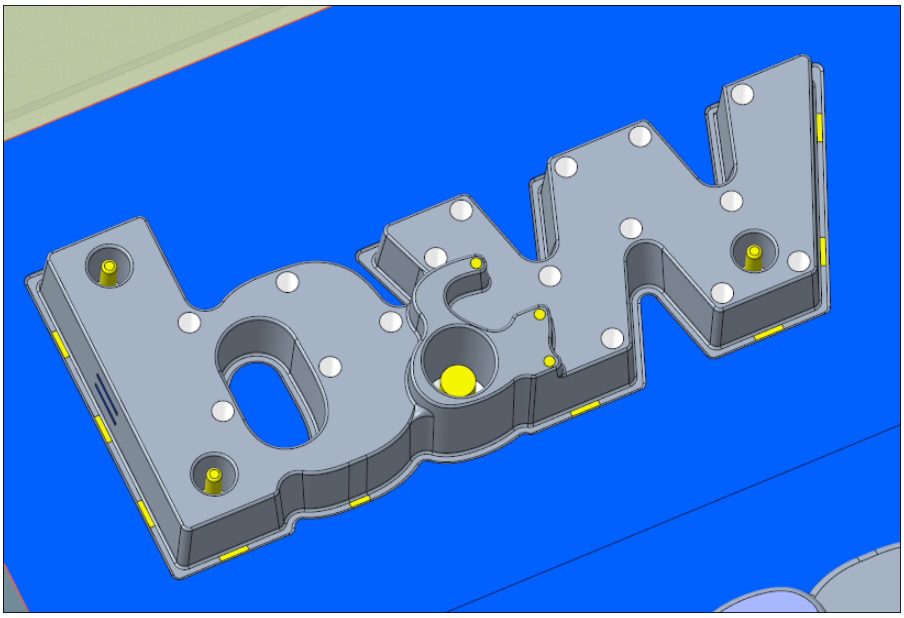
17.15.1 Modify the Cavity Plate Thickness on the Moving Half

1. Click **EMX Assembly ▶ Mold Base ▶ Assembly Definition**.

   The **Mold Base Definition** dialog box opens.
2. Double-click on the moving half cavity plate in the side view of **Mold Base Definition**.

   The **Plate** dialog box opens.

3. Change the **Thickness** to `86` .
4. Apply the modifications to the plate with **OK**.
5. Close the **Assembly Definition** with **Close**.
6. Click **EMX Assembly ► View ► Show ► Moving Half**.
7. Hide of the reference models.



The following problems can be discovered:

- Ejector pins with the option **Auto Length** are too short.

  The length value in the ejector pin model is not regenerating automatically.
- Ejector pin models with the option **Trim to quilt/refmodel** have a correct length.

  The **Solidify** and **Trim to geometry** feature regenerating,
- In both case the order number can be incorrect if the new required length exceeds the nominal length of the selected instance.

17.15.2 Update the Ejector Pins

1. Click **EMX Components ► Ejector Pin ► Update existing ejector pins**.

   The **Update Existing Ejector Pins** dialog box opens.

   NOTE:
   The difference between the **Required length** and the **Current length** is `10` mm, which is the difference between the old and the new cavity plate thickness.
2. Click **Update**.

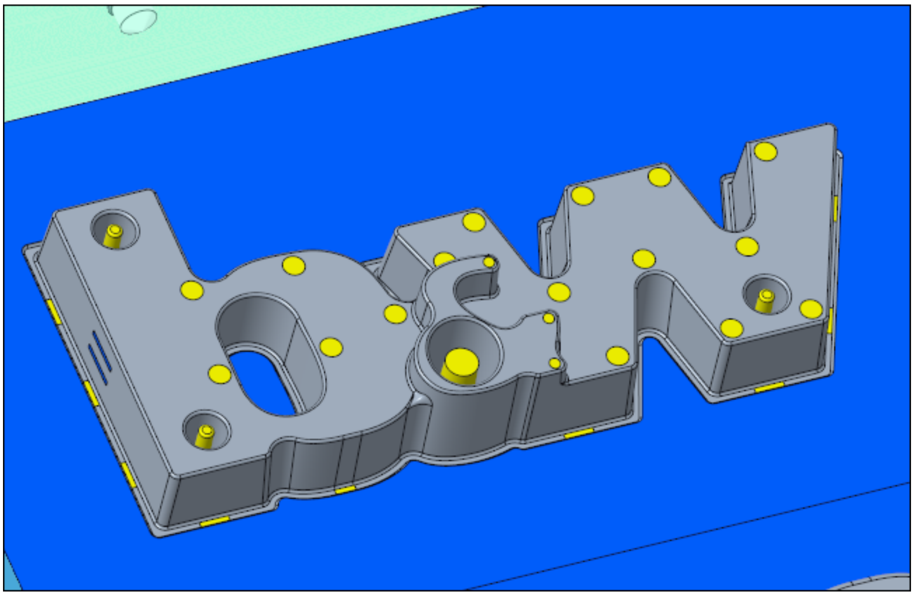
The update procedure is started and all ejector pins are regenerated.

NOTE:
Some ejector pins have new a **Order Number**.

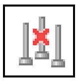
`37111/4-160` was enough for a **Current Length** of `157.541` mm.

`37111/4-200` is necessary to fit a **Required Length** of `167.541` mm.

3. Click **OK** to leave the dialog box.



17.16 About Deleting all Ejector Pins

1. Select **EMX Components ► Ejector Pin ►**  **Delete all Ejector Pins**. The **EMX Question** dialog box opens.
2. Click **Yes** to delete highlighted components and features. All ejector pins and cutouts are deleted.

Notes:

18

The Runner System

18.1 About the Component Library

You can define any component as a Creo Expert Moldbase Extension library component. If you integrate a Creo Parametric assembly into the EMX library, all components are handled using EMX operations. On the **EMX Components** tab, click **Assemble** to assemble a library component. The **Library Component** dialog box opens. Select the components required and click **OK**. A reference group is created with the component and all required cuts. All component information is stored as a reference group parameter.

18.2 About Assembling Library Components

1. Click **EMX Components ▸ Assemble**. The **Library Component** dialog box opens.
2. Select the name of the parent assembly.
3. The `LIBRARY_PATH` option sets the default root directory. Choose the unit of measure <unit> as **mm** or **inch**, then set the root directory to `/components/<unit>/library`.
4. Enter a search string and click to list all matches in the **Library Components** list, or select a directory from the browser window.
5. Select a component from the active directory. The **Component** tab is activated and the component is displayed in a preview window. The component dimensions appear in a table adjacent to the preview window.
6. Right-click a line in the table to open the **Measure** dialog box, double-click a cell in the **Value** column to edit the dimension value , or select a value from the list .

   NOTE:

   You cannot change values marked .
7. Enter a component name in the **Part Name** field. Select the check box to enter an alternate name. The component is displayed in the preview window.
8. The reference names in boldface must be set manually. Click the displayed references such as **Axis**, **Plane**, and so on and select the respective assembly placement constraints from the graphics window.
9. Select an instance from the table.
10. When you select the **Standard Part** check box at the bottom of the window, you cannot modify the parameter dimensions in the existing model. The new instance is substituted for the existing component. When you clear the **Standard Part** check box, and set new dimensions and parameters, the existing component is modified.
11. Select the **Add to BOM** check box to include the component in the bill of materials. When the component is an assembly, you can set subassemblies independently.
12. Select the following general options in the **Options** tab for library components:

- **Create Cut**—When you define a component with cut quilts, cut-by-quilt features are created in all models that interfere with the new component after it has been assembled.
- **Pattern Component**—When one of the assembly constraints is part of a pattern, the library component is placed on each instance of that pattern.

13. Click to preview the assembled component. A wire frame assembly appears in the mold base. The new component appears shaded so you can check the dimensions. If you redefine a component, the existing model is suppressed temporarily during preview.
14. Click **OK** to apply the changes and close the dialog box, or click **Cancel** to close the dialog box.

18.3 To Design the Runner

Download 42 To Design the Runner to start from with this chapter.

18.3.1 Create a Runner Assembly

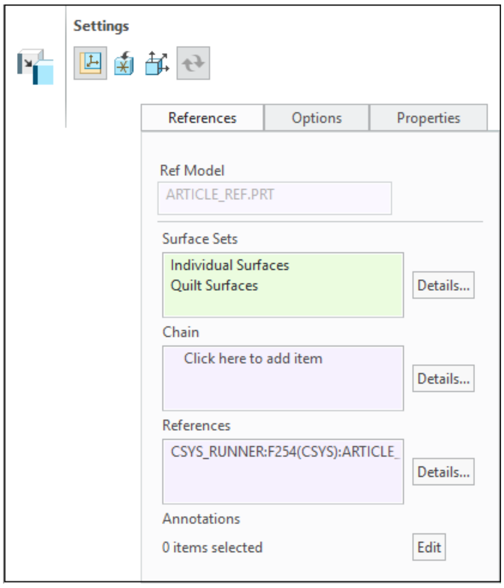
1. Click **EMX Assembly ► Structure ► Add Subassemblies**

   The **Subassembly** dialog box opens.
2. Select `TUTOR_SHARED` from the **Subassembly** pull-down list.
3. Enter `TUTOR_RUNNER` into the **Name** input panel.
4. Click **OK** to leave the **Subassembly** dialog box.

   A new EMX Subassembly will be added to the mold base assembly.
5. Open the `TUTOR_SHARED.ASM` assembly.

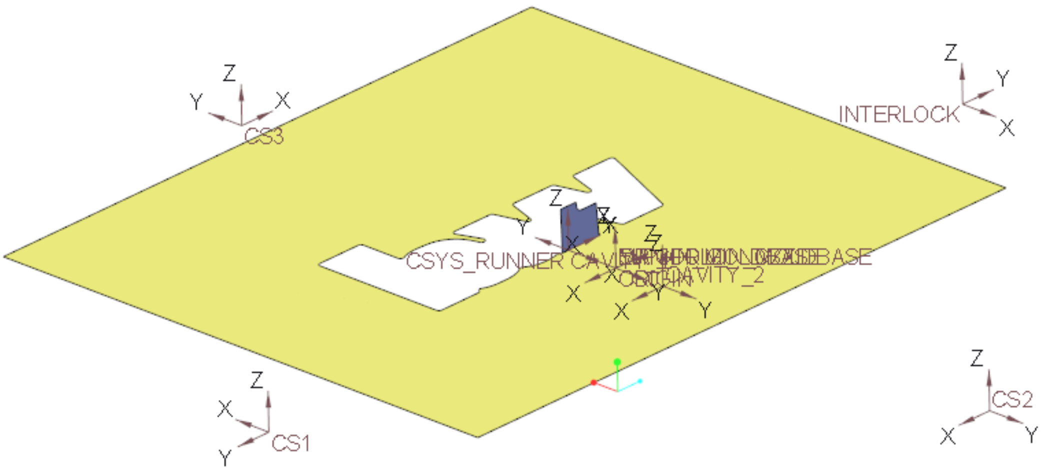
18.3.2 Create the Runner Quilt Surface

1. Active the `TUTOR_RUNNER.ASM` assembly.
2. Create a assembly dependent **Copy Geometry** feature which contains:



- The coordinate system `CSYS_RUNNER` from the `ARTICLE_REF.PRT`
- The relevant quilt surfaces from the article reference part. (You must be Show the Layer 00_BUW_DATUM for catch it)

3. Open the `TUTOR_RUNNER.ASM` assembly.
4. Make quilt surfaces are not hidden by layers.

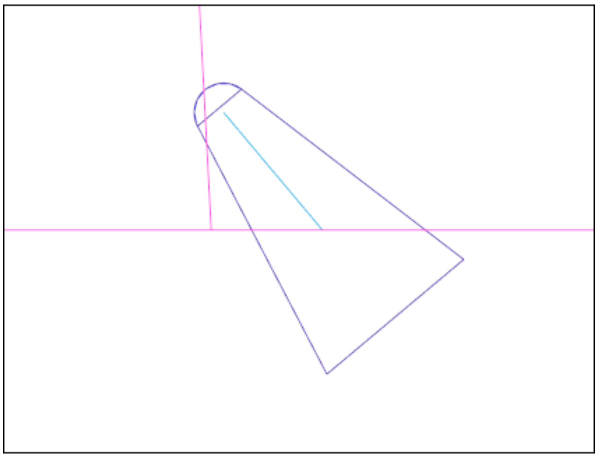


18.3.3 Assemble the Conic Gate

1. Click **EMX Components ► Library ►** **Assemble**

   The **Library Component** dialog box opens.
2. Select `runner` folder from the library tree.
3. Double-click on `gate_cone`.

   The **Component** tab is activated.
4. As first placement reference **(1) CSYS** select the `CSYS_RUNNER`.
5. Set gate diameter **DM** to `1.8` mm
6. Set **H** height value to `4` mm
7. Click on **Preview** to display the gate in the graphics window.

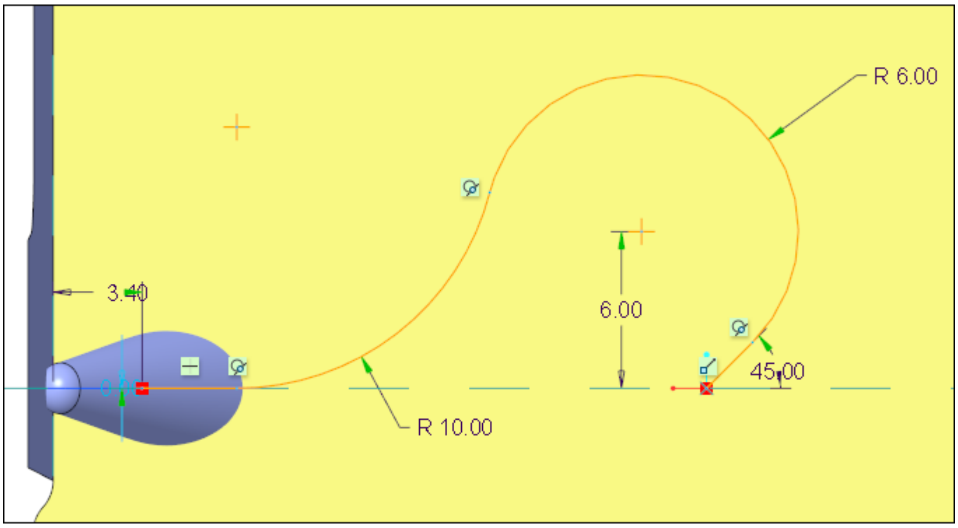


8. Click **OK** to assembled the gate cone part.

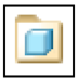
   Make sure all quilt surfaces are visible.

18.3.4 Create the Sprue Sketch

1. Create a **Sketch** on `MOLDBASE_X_Y` in the skeleton plane
2. Create the sketch as seen below.



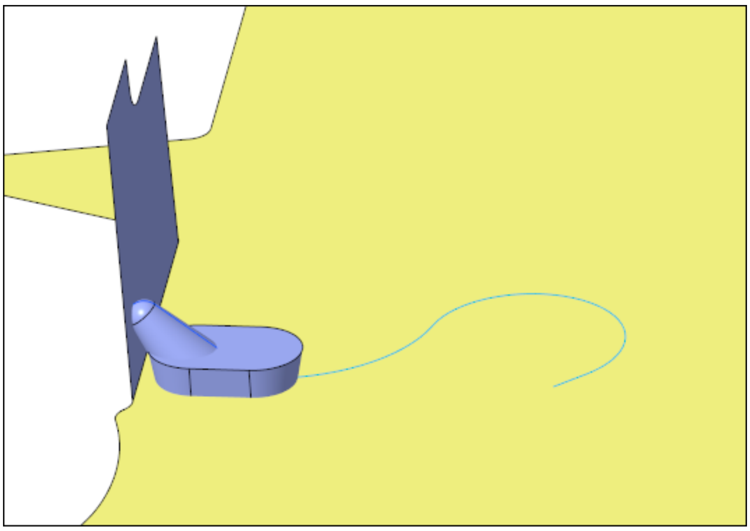
18.3.5 Create the Sprue Channels

1. Click **EMX Components ▸ Library ▸**  **Assemble**

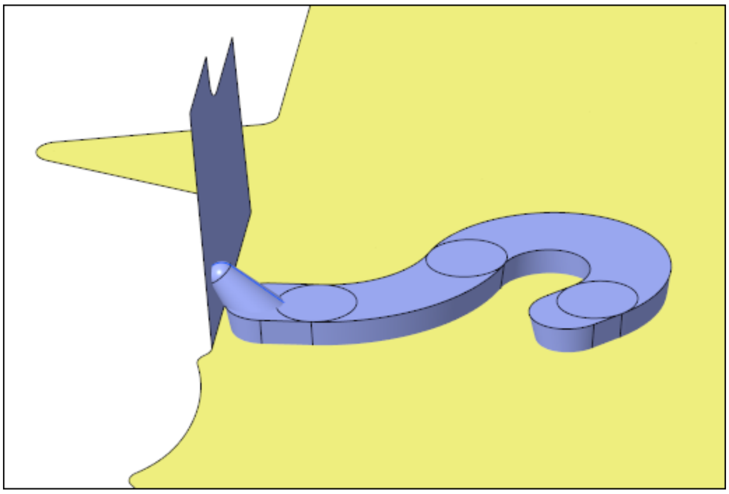
   The **Library Component** dialog box opens.
2. Select `runner` folder from the library tree.
3. Double-click on `trapezoid_es`.

   The **Component** tab is activated.
4. For **W** select a value 5 mm.
5. As first placement reference **(1) Curve** select a segment of the previously created sketch.
6. Click **OK** to assemble the sprue channel.

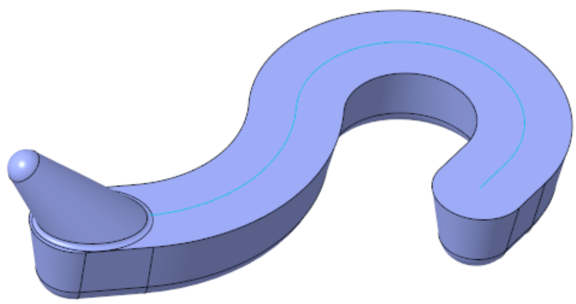
   NOTE:
   The `trapzeoid_es` is not a part or assembly. It is defined as UDF.



7. Repeat the assembly of the trapezoid UDF for all segments of the sketched sprue curve.



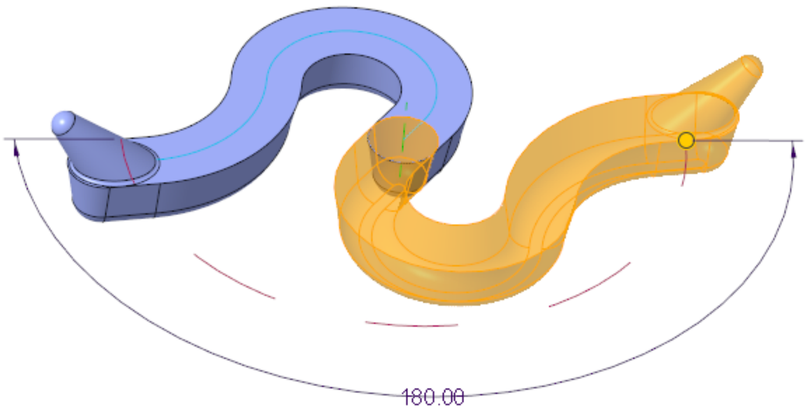
8. Trim and merge the created quilt to finally retrieve the sprue.
9. Create round to avoid sharp corners.



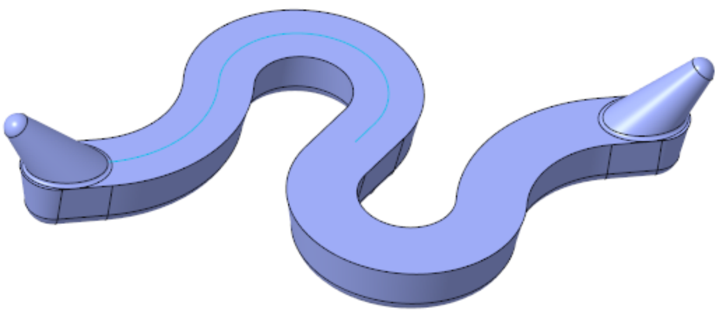
18.3.6 Mirror the Quilt

1. Select the quilt and use **Copy** and **Paste Special**
2. Select **Rotate** for **Settings**
3. Select the `AXIS_LOCATING` from the `TUTOR_SKELETON.PRT`
4. Set a value of `180` degrees.

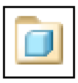
5. Select the **Options** tab and disable **Hide original geometry**



6. Click **OK** to finish the special copy operation.
7. Merge both quilt surfaces.



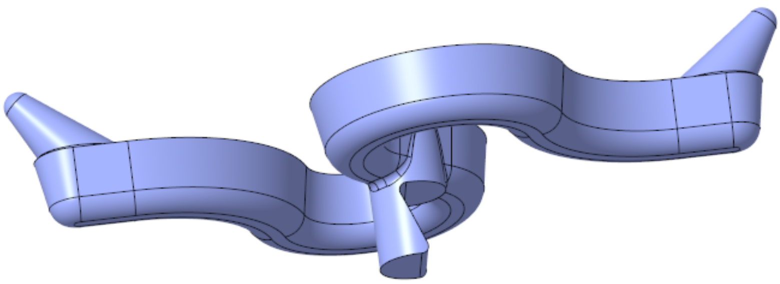
18.3.7 Create the Sprue Puller Pin Channel

1. Click **EMX Components ▸ Library ▸**  **Assemble**

   The **Library Component** dialog box opens.
2. Select `runner` folder from the library tree.
3. Double-click on `sprue_puller_channel`

   The **Component** tab is activated.
4. As first placement reference **(1) CSYS** select the `ORIGIN_MOLDBASE` from the `TUTOR_SKELETON.PRT` model.
5. Click **OK** to assemble the sprue puller pin channel.
6. Merge the quilts

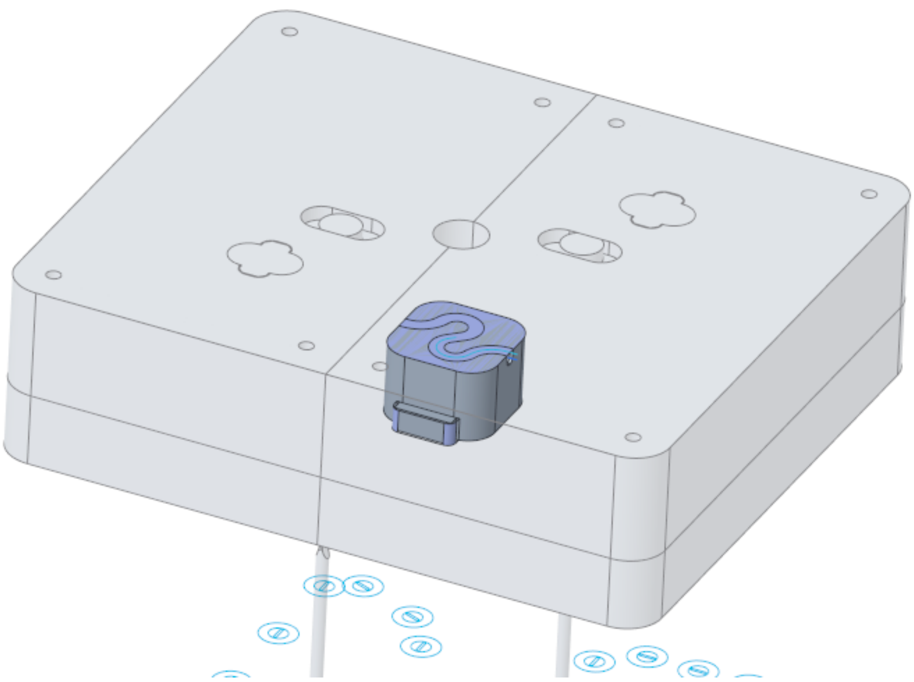
7. Create rounds to avoid sharp corners



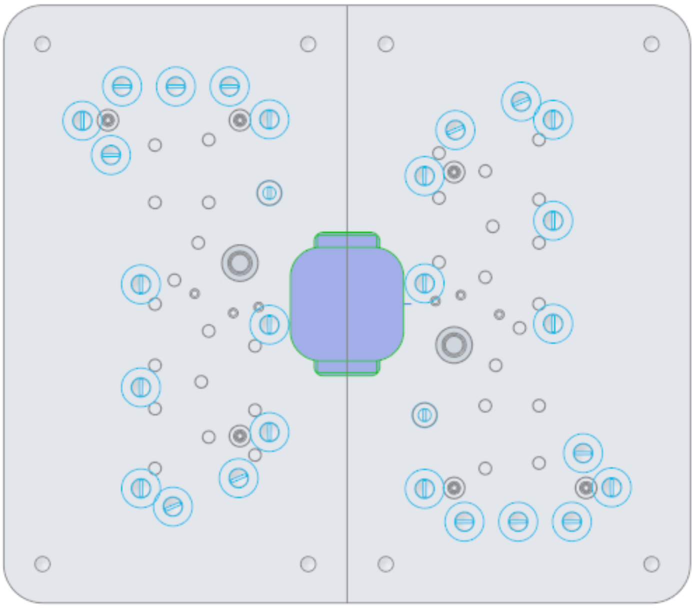
8. Close `TUTOR_RUNNER.ASM` to return to the `TUTOR_SHARED.ASM`

18.3.8 Design the Sprue Retainer Bush

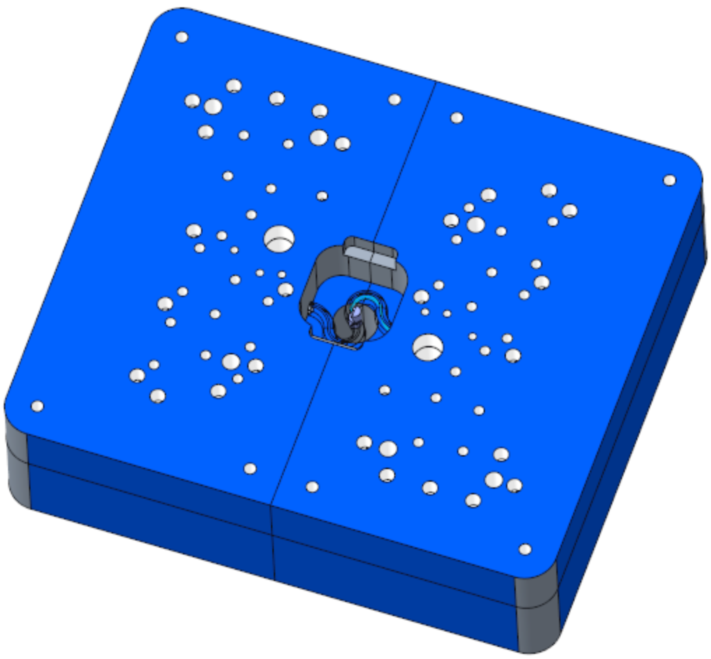
1. **Create** a new part `TUTOR_RETAIN_BUSH.PRT` and assemble it to the `ORIGIN_MOLDBASE`.
2. Design the part using default functionality.



The major outline in this example is `38x38 mm`

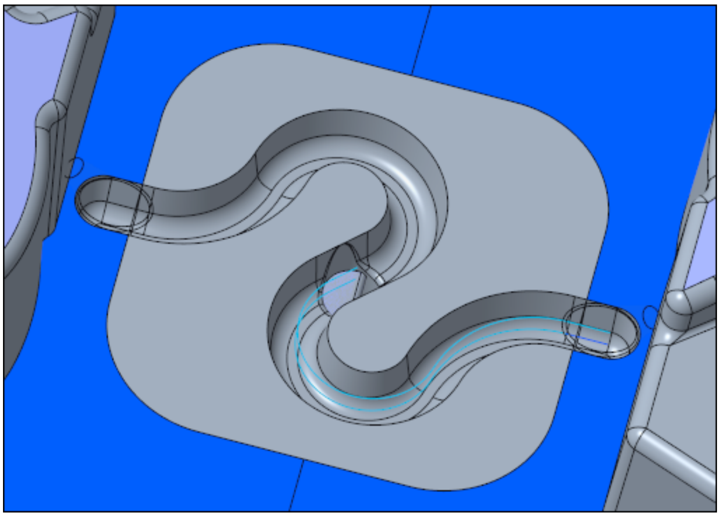


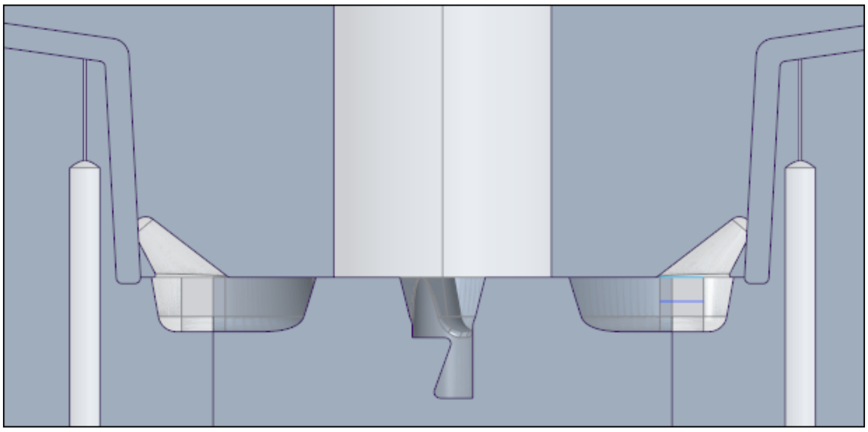
3. Create to quilt surface which can be used for **Solidify** operation
4. Create **Soldiify** feature in
   - `MOLD_VOL_ES_1.PRT`
   - `MOLD_VOL_ES_2.PRT`



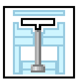
18.3.9 Create the Solidify Cuts for the Runner in Insert Parts

1. Create the **Solidify** cuts in using the runner quilt surface in
   - `TUTOR_RETAIN_BUSH.PRT`
   - `MOLD_VOL_ES_1.PRT`
   - `MOLD_VOL_ES_2.PRT`
   - `MOLD_VOL_IS_1.PRT`
   - `MOLD_VOL_IS_2.PRT`





18.3.10 Assemble the Sprue Puller Pin

1. Click **EMX Components** ►  **Ejector Pin**.

   The **Ejector Pin** dialog box opens.
2. Select `knarr` from the **Supplier** pull-down list.
3. Select `37111` from the **Types** pull-down list.
4. Set the **DM1** value to `5` mm.

5. Define the references
   a. For **(1) Point** select the point `PULLER_PIN` from the `TUTOR_SKELETON.PRT` model.

      The **LG1** value is automatically set to `160` mm.
6. Activate **Trim to quilt/refmodel** option.
7. Select  **Rotfix1** from the **Fix Rotation** pull-down list.
8. Make sure to set the **D1** value in the **UDF Dimensions** to `5.000` mm m the **Fix Rotation** pull-down list, otherwise the UDF can not be assembled
9. Close the **Ejector Pin** dialog box with **OK**.

   The pin is assembled and trimmed.

   Cut outs are added to the `TUTOR_RETAIN_BUSH.PRT`
10. Close the `TUTOR_SHARED.ASM` to return to the main assembly.
11. Click **EMX Components ► Ejector Pin ► Add Ejector Bores** from the overflow menu.

    The missing bores for the puller pin are added to the mold base assembly

19

Lifter Assemblies

19.1 About Lifters

Lifters are function units that allow demolding of undercuts without any intersection and without the usage of sliders.

The sideways moving direction under a certain angle during the ejection process releases the undercut geometry and the plastic part can drop out of the mold base with out any damage.

The movement of the lifter is driven by the ejector plate package, so its reach is more limited compared to the free movement of a slider.

Click **EMX Components ▸ Components ▸ Lifter** to define a new lifter.

Use the **Trim to quilt/refmodel** option to trim the lifter bar to either the reference model or a defined trim quilt..

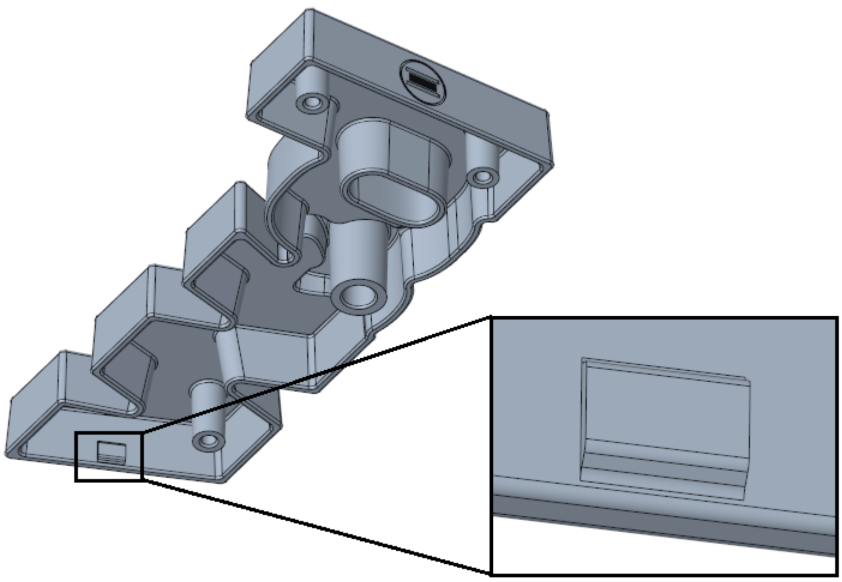
NOTE:
The quilt surfaces can be designated as trim surface using **EMX Assembly ▸ Preparation ▸ Identify trim faces**

19.2 To Create a Lifter

Download 43 To Create a Lifter to start from with this chapter.

19.2.1 Create a Lifter

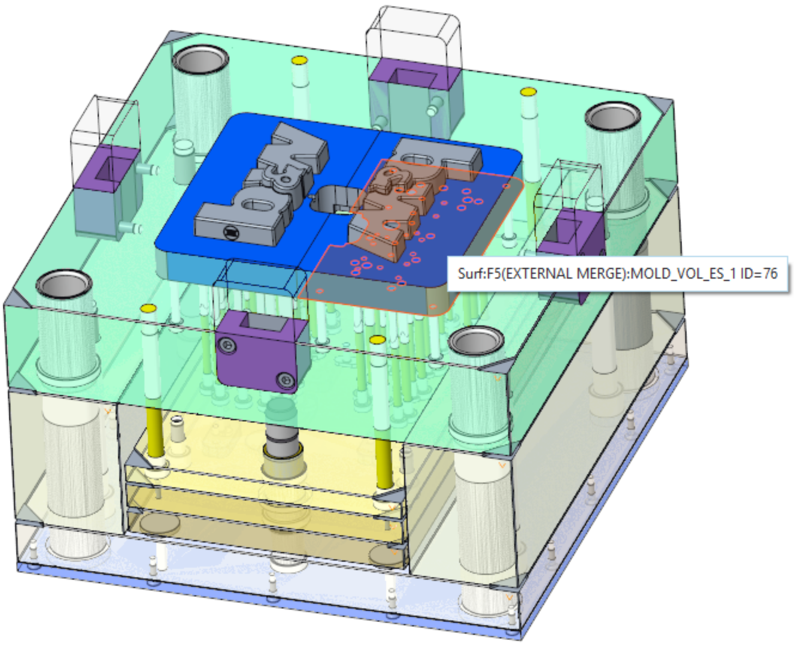
In this tutorial a lifter is designed to create the snap connection on the bottom side of the plastic part `ARTICLE_REF.PRT`.



1. Open the mold base `TUTORIAL.ASM`
2. Display the **Moving Half** of the mold base.

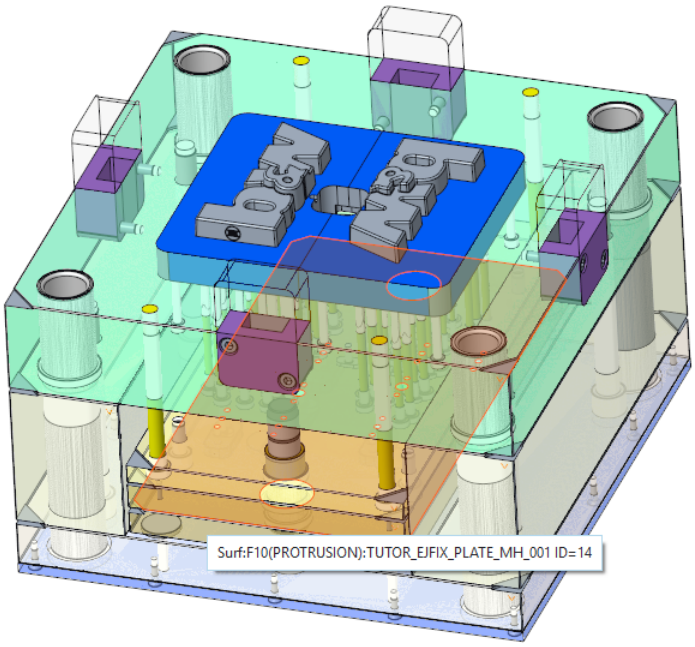
3. Click **EMX Components ► Components ► Lifter** .

   The **Lifter** dialog opens.
4. Select TUTOR_MH from the **Subassembly** pull-down menu.
5. Select **Supplier** meusburger.
6. Select the **Type** E3246_E3240.
7. Define the references.
   a. Click **(1) Csys** and select the CSYS_LIFTER coordinate system in the ARTICLE_REF.PRT.
   b. For reference **(6) Surface** select the bottom surface of the cavity insert part MOLD_VOL_ES_1.PRT

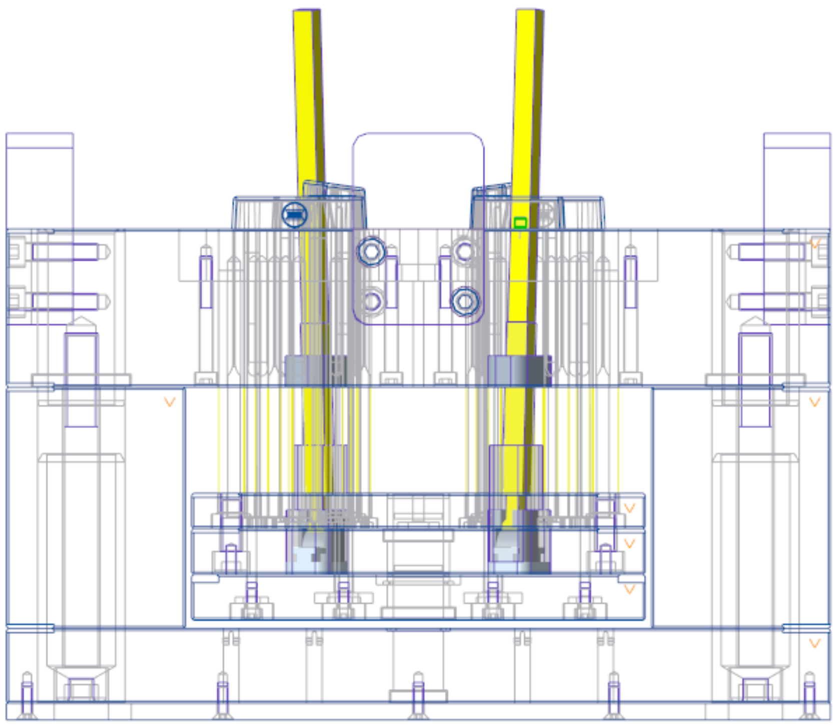


   c. Reference **(4) Plane Guide** and **(5) Plane Retainer** are set by default.
8. Click on **(5) Plane Retainer** to redefine this reference.

   Select the top surface of the ejector plate TUTOR_EJFIX_PLATE_MH.PRT.



9. Make sure that the option **Trim to quilt/refmodel** is enabled.
10. Enable the Option **Pattern for all models**.
11. Assemble the lifter with **Preview**



The position of the lifter is not correct and needs to be adapted.

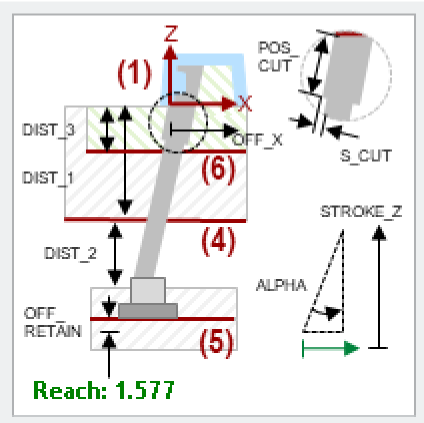
12. Set the **ALPHA** value to `6` ° to adjust the position.

13. Set the **STROKE_Z** value to `15` mm to adjust the position.

    NOTE:
    The resulting **Reach** is displayed in the Component image with green color.

    It is calculated depending on the angle **ALPHA** and the **STROKE_Z** of the lifter.

    The **Reach** value updates when the clicking on **Preview**.



14. Click **Preview** again.

    The Reach value is set to `1.577` mm

    The snap connection has a height of `1` mm.

    The resulting reach is larger then the required reach.

15. Set the **OFF_X** value to `-1.4` mm to adjust the position.

    This ensures the lifter bar position is correctly interfering with the reference mode

16. Set the **OFF_Z_RETAIN** value to `6` mm to adjust the position.

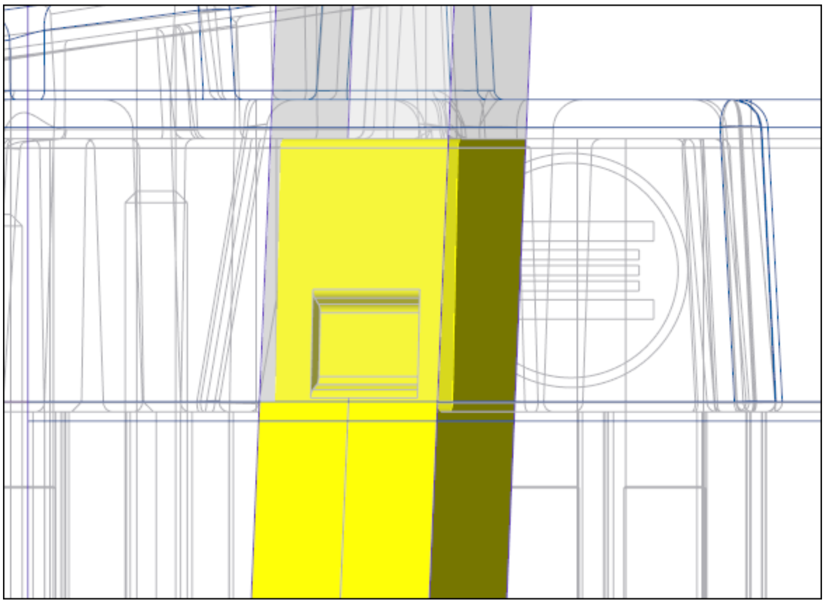
    This ensures the lifter retainer is not interfering with any ejector pins.

17. Set the **CUT_S** value to `2` mm.

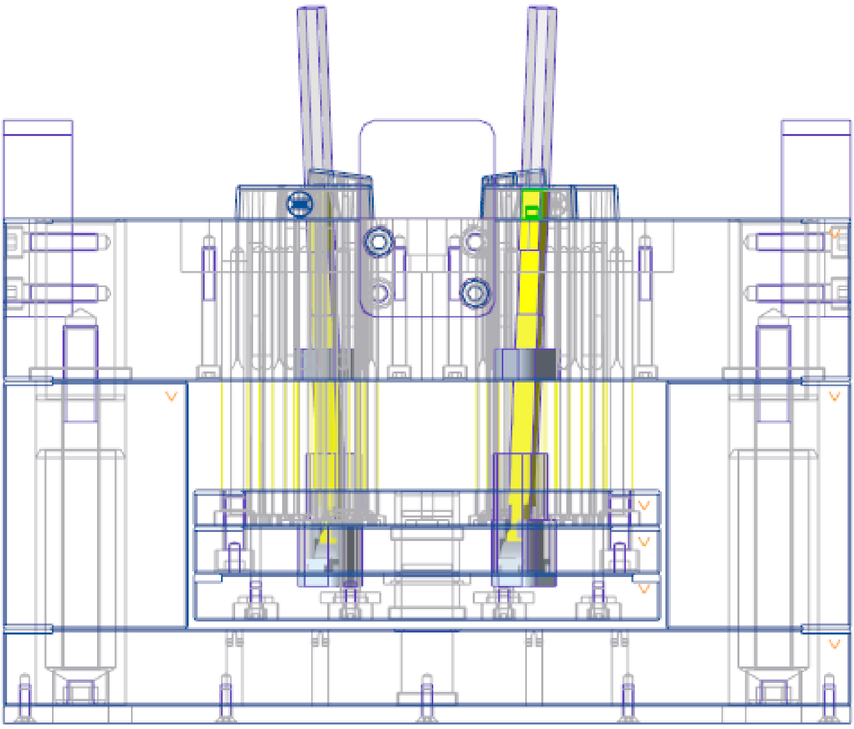
    This will create a step within the lifter bar. This step avoids damage to the ejector plates due to the high pressures during the injection molding procedure.

18. Assemble the lifter again with **Preview**.

    The resulting trimmed lifter bar is previewed.

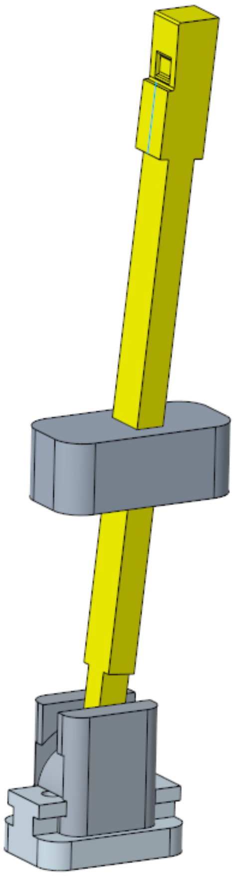


The position of the lifter is good, no other components are interfering.

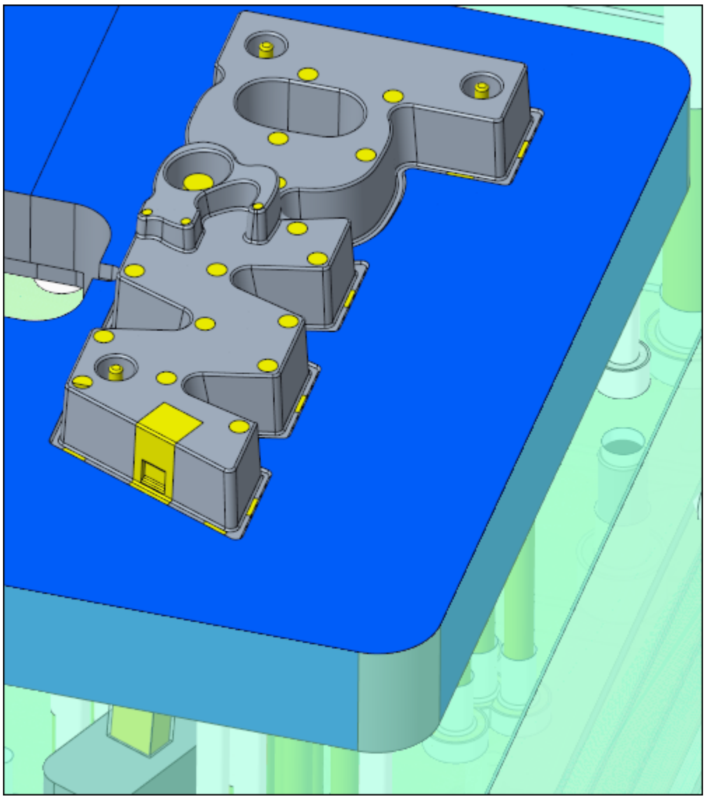


19. Finally assemble the lifter with **OK**.

    The trimmed lifter is assembled to the mold base and all cuts are created.



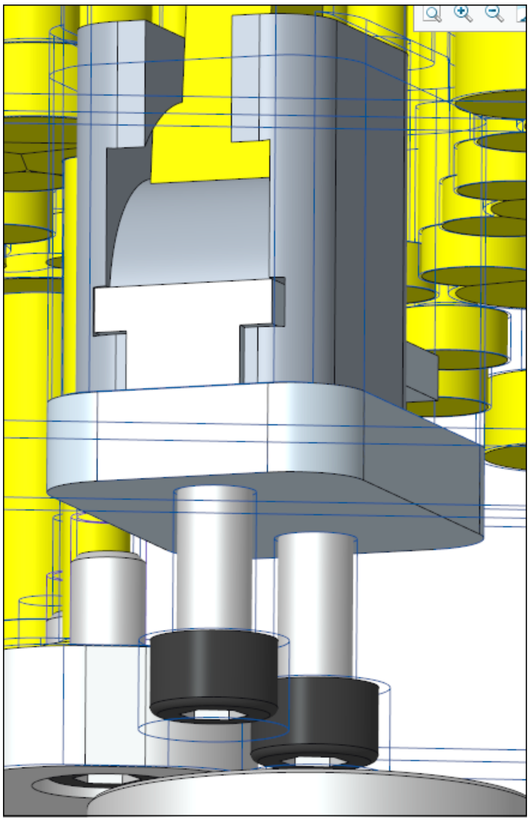
The lifter creates now the contour of the snap connector and releases it during the mold base opening process.



19.2.2 Create Screws for the Lifter

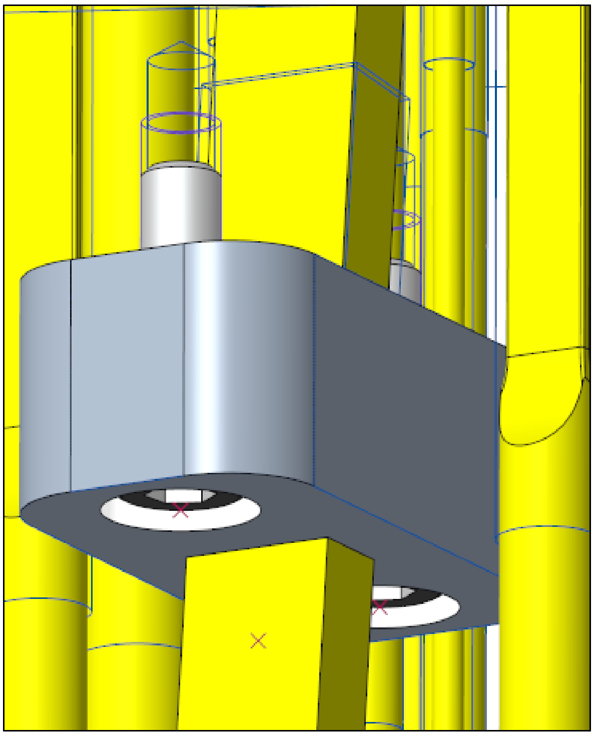
1. Assemble `E1200/5x20` screws to fixture the retainer to the `TUTOR_EJFIX_PLATE_MH_001.PRT`

   Make sure to assemble them with **Pattern for all models**.



2. Assemble `E1200/5x16` screws to fixture the guide to the `TUTOR_CAV_PLATE_MH_001.PRT`

   Make sure to assemble them with **Pattern for all models**.



Notes:

20

Slider Assemblies

20.1 About Sliders

Use slider sets to define the vertical movement of the mold base components when the mold base is opened or closed.

Sliders are elements used to achieve contours at the mold insert which cannot not be ejected without any intersection.

Undercut geometry can be achieved by implementing the geometry directly into the slider.

The sideways moving direction under a certain angle during the ejection process releases the undercut geometry and the plastic part can drop out of the mold base without any damage.

Click **EMX Components** ► **Components** ► **Slider** to define a slider.

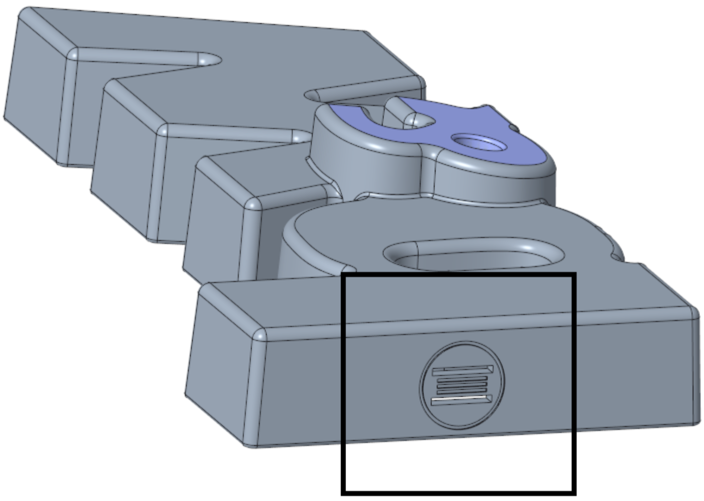
Select **Merge with CSYS parent** to automatically merge the cam of a slider unit with the assembly placement coordinate system.

20.2 To Create a Slider

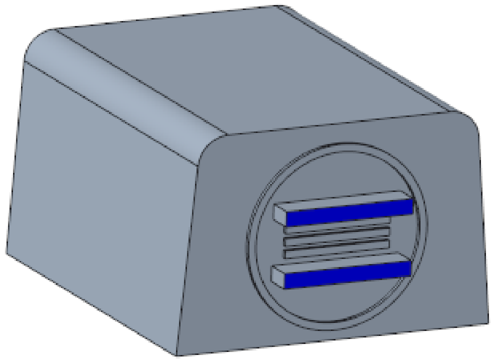
Download 44 To Create a Slider to start from with this chapter.

20.2.1 Assemble a Slider

In this tutorial a slider will be designed to create the two snap connection at the top of the plastic part `ASM_PALM_REF.PRT`.

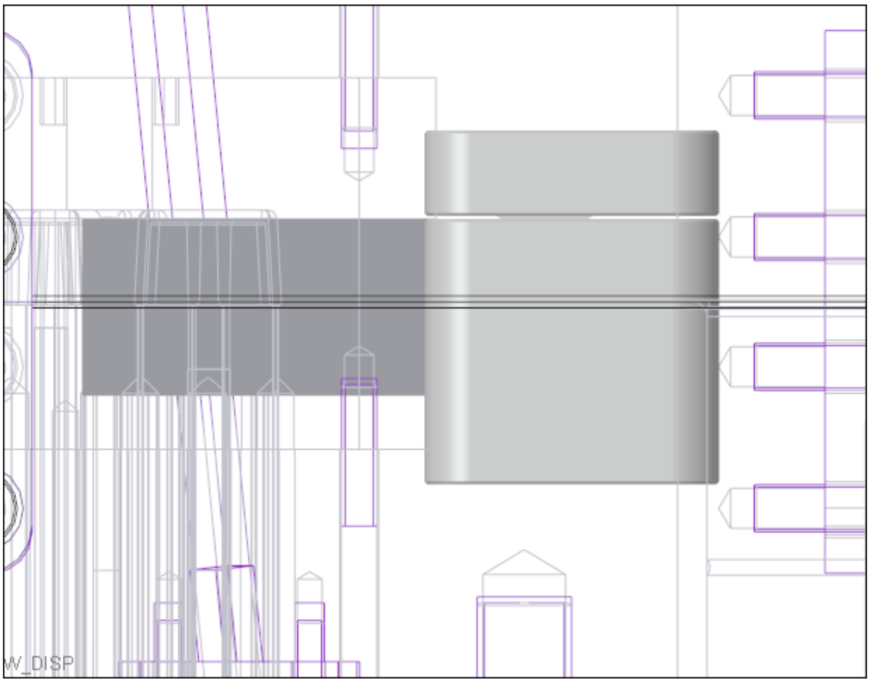


The cavity assembly `MFG_1.ASM` already contains a slider part `MOLD_VOL_SLIDER_1.PRT` which contains the contour geometry that needs to be moved sideways to prevent damage from plastic part during the ejection process.



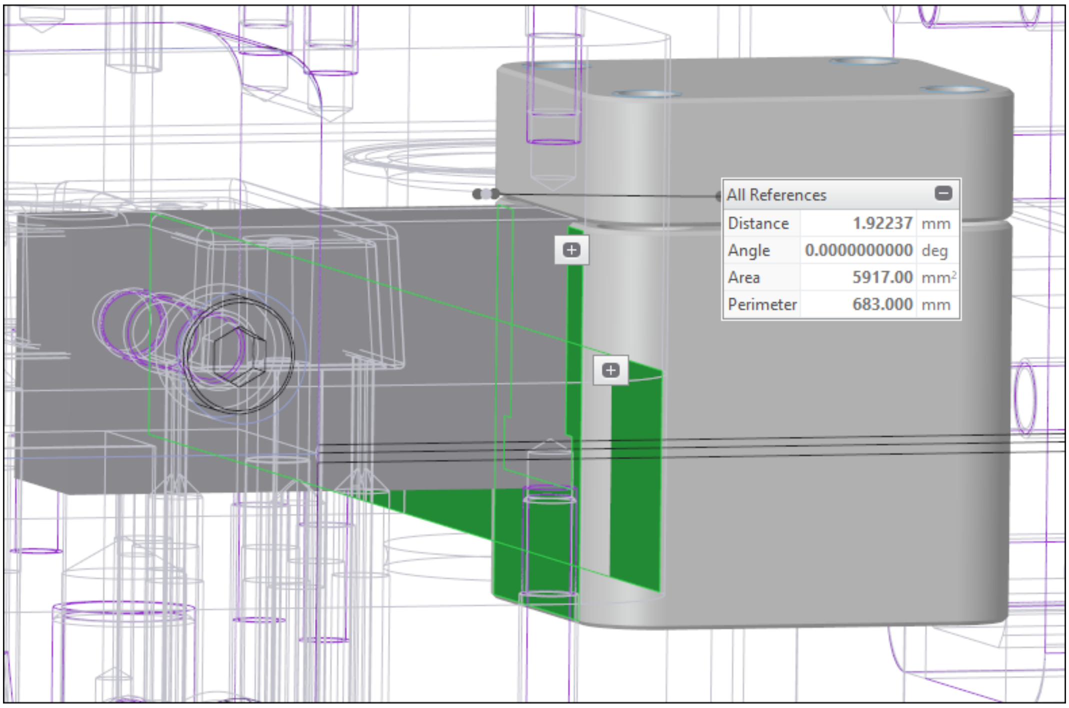
1. Open the mold base TUTORIAL.ASM
2. Select **EMX Components ► Components ► Slider**.
3. Select the TUTOR_SHARED from the **Subassembly** pull-down menu.
4. Select strack from the **Supplier** pull-down menu.
5. Select Z4294L from the **Type** pull-down menu.
6. Select 1 from the **TYPE_selector** in the **Nominal Values** table.
7. Define the references.
    a. Click **(1) Csys** and select the CSYS_SLIDER coordinate system in the ARTICLE_REF.PRT.
8. Make sure that the option **Remove tangent radii** is enabled.

    NOTE:
    This option is necessary as this slider contains a cut quilt with radii at the corners. Otherwise the cut geometry can not be created in case the sliders guide body is flush to the cavity insert parts.
9. Enable the Option **Pattern for all models**.
10. Assemble the slider with **Preview**



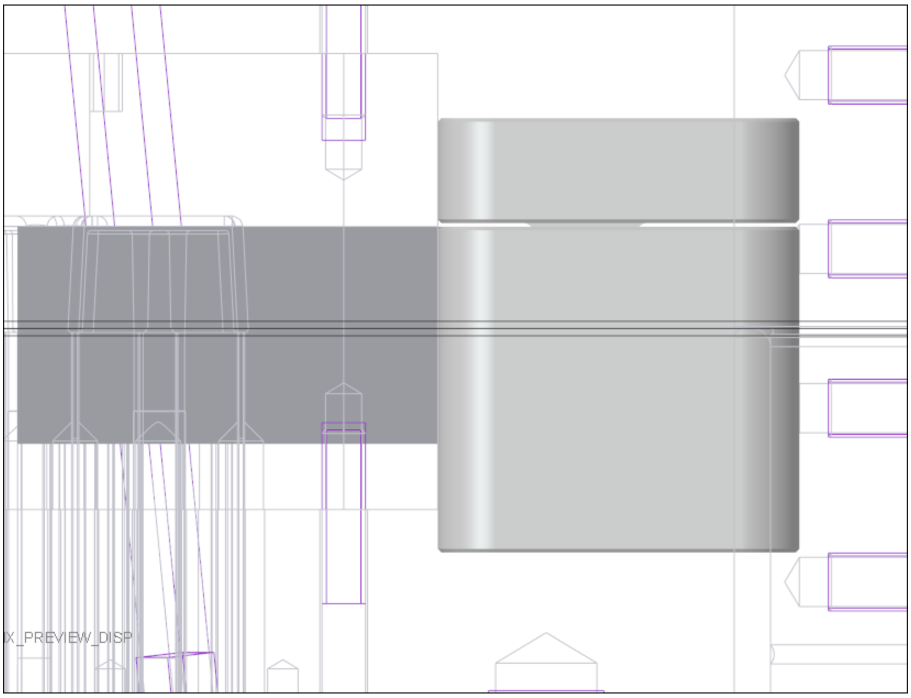
11. Measure the distance between the cavity insert side surface and the slider guide body.

    The measured distance is `1.92237` mm.



12. Set the **OFFSET_X** value to the recently `1.92237` mm to make sure the sliders is flush to the cavity insert part.

13. Assemble the slider again with **Preview**



14. Finally assemble the sliders with **OK**.

    Two sliders are assembled and cuts are created to the interfering parts.

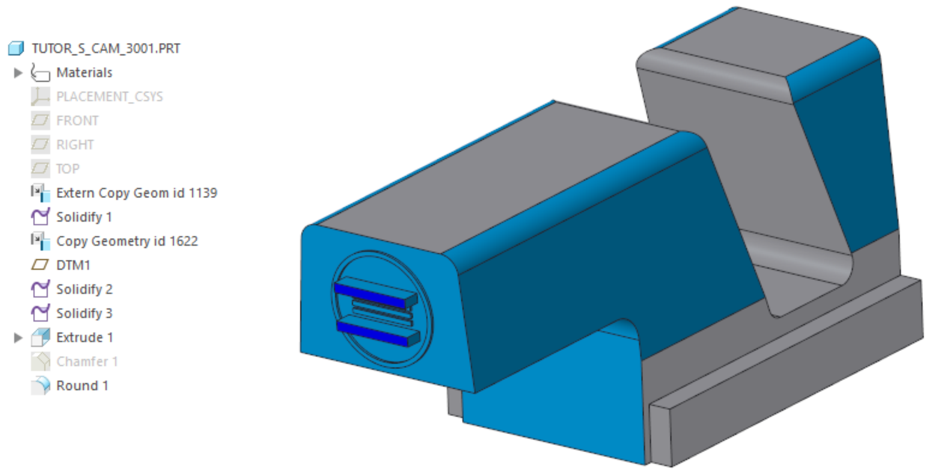
20.2.2 Finish the Slider Design

The cut quilt surface of the slider assembly was interfering with the `MOLD_VOL_SLIDER_1.PRT` part. Therefore a Solidify group was created in `MOLD_VOL_SLIDER_1.PRT`.

1. Delete the group `QUILT_CUT` in `MOLD_VOL_SLIDER_1.PRT`.

The geometry of the slider part `MOLD_VOL_SLIDER_1.PRT` need to be added to the newly created slider cam `TUTOR_S_CAM_3001.PRT`.

1. Activate the `TUTOR_S_CAM_3001.PRT` within the new slider assembly.
2. Create a assembly depended **Copy Geometry** feature of all solid surfaces of the `MOLD_VOL_SLIDER_1.PRT`
3. Open the `TUTOR_S_CAM_3001.PRT`.
4. Finish the slider design as seen in the image below using **Solidify** and **Extrude** features.

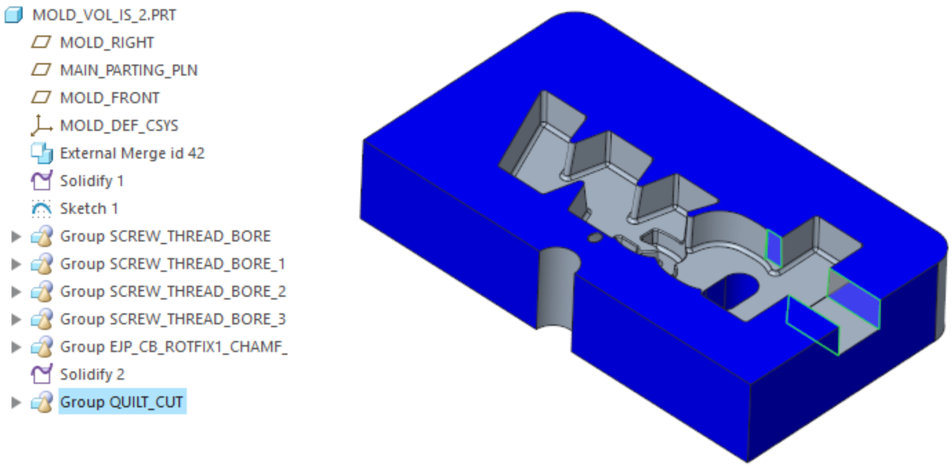


20.2.3 Remove Obsolete Quilt Cuts from Cavity Insert Parts

The cut quilt surface of the original slider assembly was interfering with the cavity insert parts.

These quilt cuts need to be removed.

1. Open `MOLD_VOL_IS_1.PRT`.
2. Delete the `QUILT_CUT` group `MOLD_VOL_SLIDER_1.PRT` which was created by the slider assembly

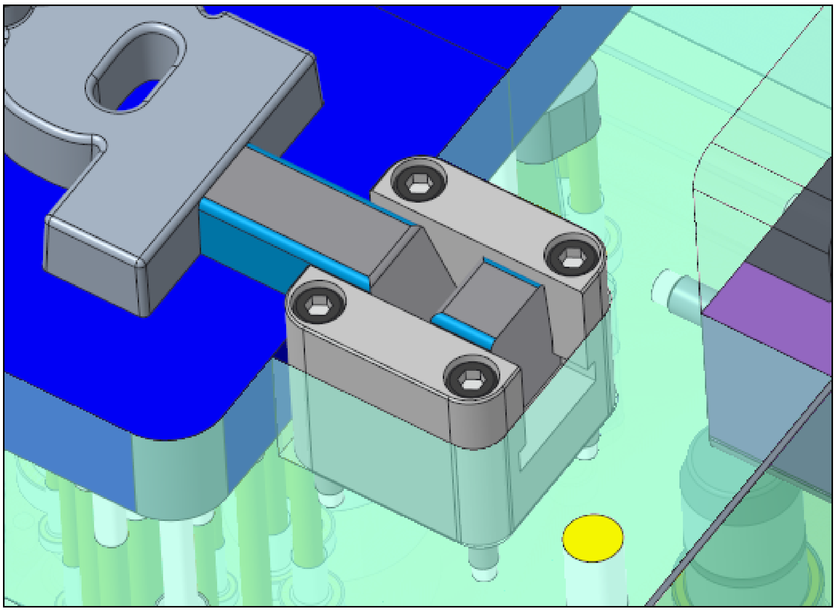


3. Repeat this steps for
   - `MOLD_VOL_IS_2.PRT`
   - `MOLD_VOL_ES_1.PRT`

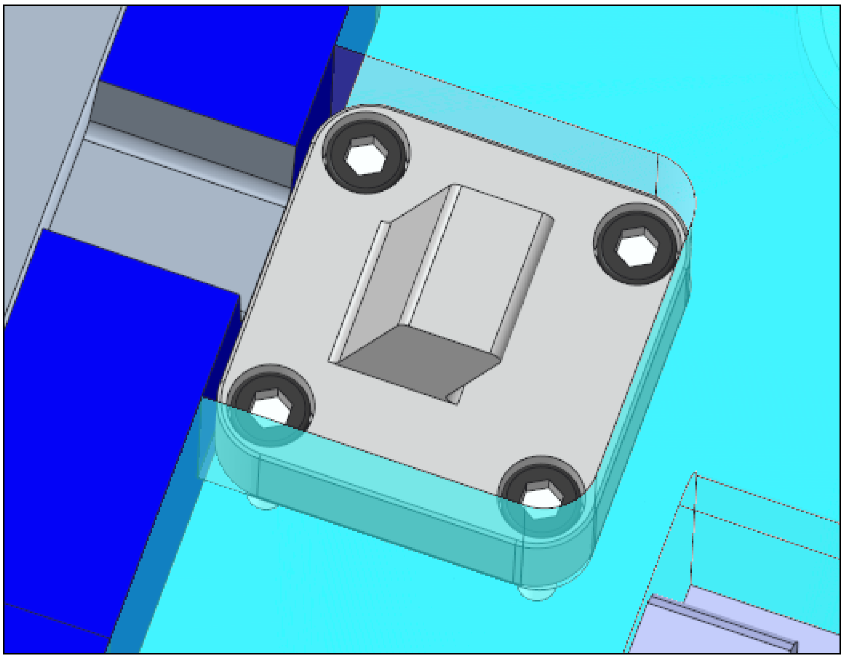
- `MOLD_VOL_ES_2.PRT`
- `MOLD_VOL_CORE_1.PRT`

20.2.4 Assemble Screws to Fixture the Slider ComponentsAsl

1. Assemble `E1200/6x50` screws to fixture the slider guide component to the moving half.
2. Make sure the Option **Pattern for all models** is active



3. Assemble `E1200/6x16` screws to fixture the slider bar component to the fixed half.
4. Make sure the Option **Pattern for all models** is active



Notes:

21

Support Pillars

21.1 About Support Pillars

Support pillars are used for extra support between the cavity plate and the clamping plate.

They are used to give more strength to the Moving Half of the mold base.

Click **EMX Components** ► **Components** ► **Support Pillar** to define a support pillar.

Select the **Toggle Direction** check box to change support pillar orientation.

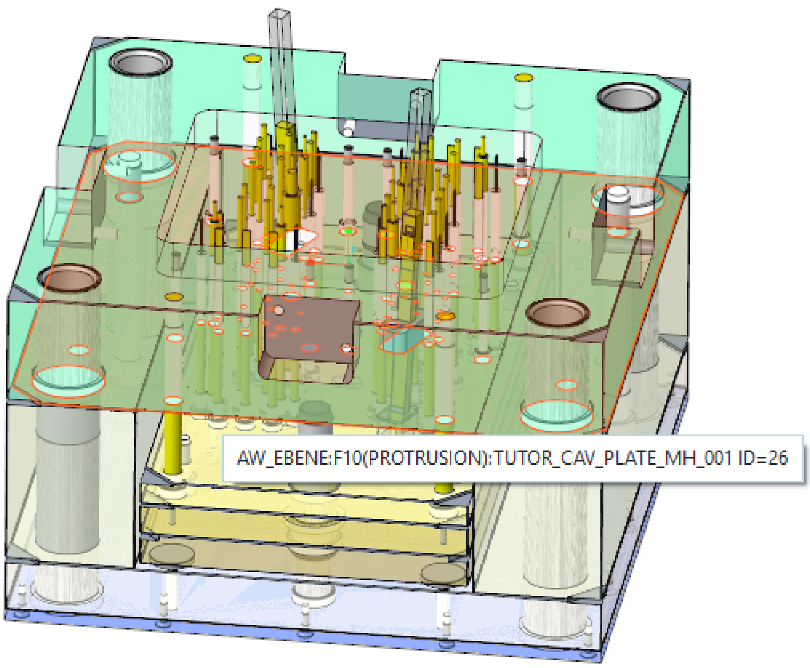
21.2 To Assemble Support Pillars on Mouse Pick Points

Download 45 To Assemble Support Pillars on Mouse Pick Points to start from with this chapter.

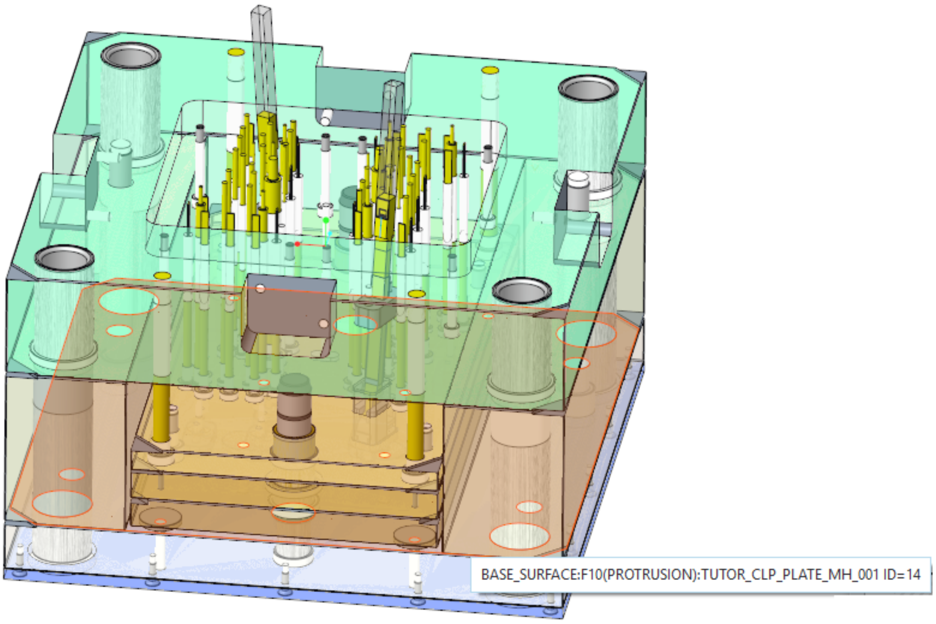
21.2.1 Assemble the Support Pillars

1. Open the `TUTOR_MH.ASM` assembly.
2. Click **EMX Components** ► **Components** ► **Support Pillar** .
3. Select `E1510` from the **Type** pull-down menu.
4. Select `40` mm for **DM1**.
5. Define the references.

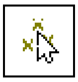
   a. As **(2) Surface** select the bottom side of the cavity plate `TUTOR_CAV_PLATE_MH_001.PRT`.



   b. As**(3) Second Surface** select the top side of the clamping plate .



The **REF1 Reference Distance** displays the offset between the plates.

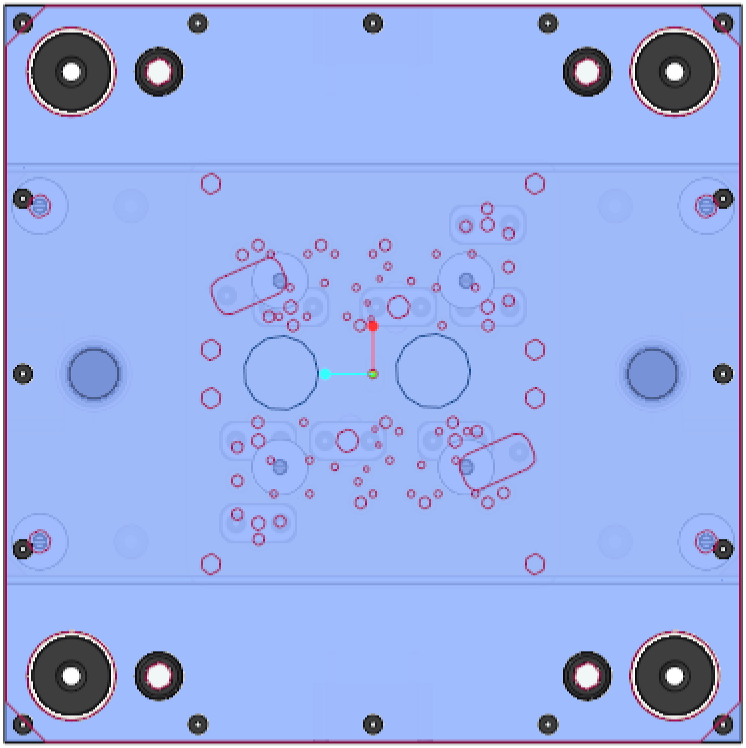
6. Click the  **Mouse Pick Placement** .

   The graphics window is reorientated to top view.

   The **Support Pillar** dialog box is disabled now unless you have finished the placement operation.

   A Circle with the **Diameter** value `40` mm is drawn at the mouse position in the 3D graphics window.

7. Add `2` points with the left mouse button.



8. Stop the placement loop by pressing middle-mouse button.

   New Points are created using the **(2) Surface** as placement plane.

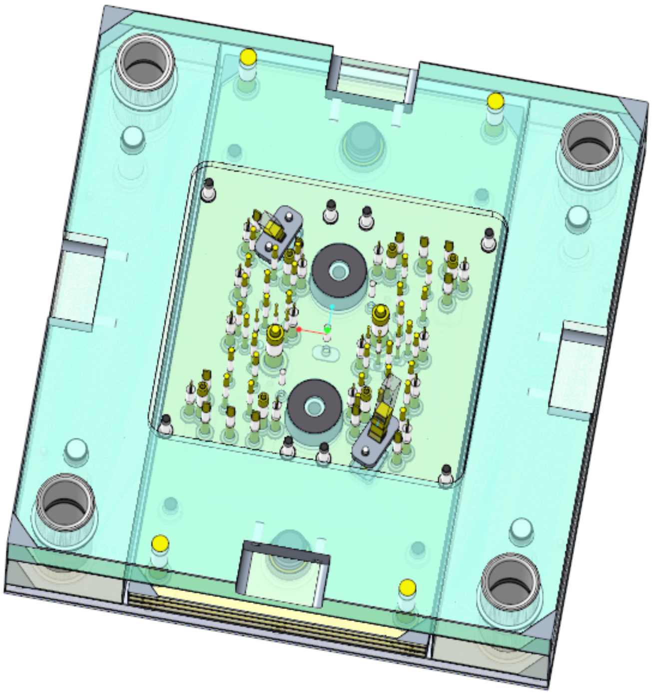
   The `MOLDBASE_X_Z` and `MOLDBASE_Y_Z` datum planes of the skeleton are used as placement reference.

   NOTE:
   This mouse pick placement can only be used for the 6 main sides of the mold base right now as the three skeleton datum planes are used as measurement references.

9. Make sure the option **Toggle Direction** is activated.
10. Close the **Support Pillar** dialog box with **OK**.

    Two new support pillars are created.

11. Modify the previously created **Datum Point** feature.
12. Set the **Offset References** values to `(50;0)` and `(-50;0)` mm



NOTE:
The NR_DIGITS_COMMON option is used to set the number of digits for this point offset dimensions.

The increment that is used while you drag the mouse is defined in **EMX Option** INCREMENT_POS.

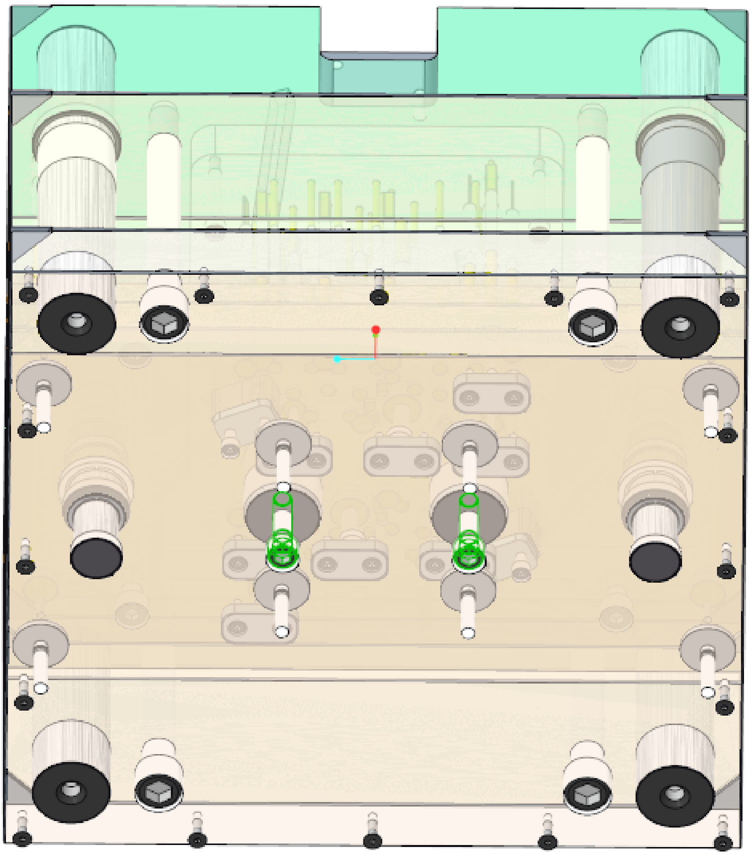
Use i.e. value 1 to round the mousepick-point offsets to full millimeter.

21.2.2 Assemble the Screws for the Support Pillars

1. Assemble screws E1200/10x40 by using the previously create datums as point reference.

As surface references select

- the bottom surface of the moving half clamping plate
- the bottom surface of the support pillars.



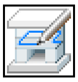
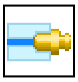
1. Close TUTOR_MH.ASM to return to the main assembly.

22

The Cooling System

22.1 About Cooling Components

Connectors, plugs, and o-rings are used as cooling components to complete the waterline inside a mold base. To define a cooling component, click the **EMX Components** tab and in the **Cooling Component** group, select a command described below:

-  **Waterline Designer**—Dynamically create cooling circuits.
-  **Cooling Component**—Defines a new cooling component.
-  **Show waterline**—Create a model that contains the full cooling circuits as surface copies.
-  **Assemble a waterline curve**—Assembles a waterline curve.
- **Change diameter of all cooling holes**—Modifies the diameter of all waterline holes.
- **Change diameter of selected cooling holes**—Modifies the diameter of single waterline holes.
- **Delete entire waterline**—Deletes all waterline components and holes.

From the **Cooling Component** dialog box, you can select the **Counterbore** and **Second Diameter** check boxes to assign a counterbore or a second diameter to all cooling bores. To define the depth of the bore, in the **Define depth by** list, select one of the following:

- **Enter value**—Enter a value.
- **Use model thickness**—Measure the model thickness and calculate bore depth using a UDF value.
- **Use curve length**—Measure the selected curve length as a placement reference and calculate bore depth using a UDF value.

NOTE:
While the Reference Groups for all types of components are located in the same assembly (like the component itself) cooling components can be fully placed in part mode. .Therefore the configuration option `ADD_COOLING_REF_IN_PART` is set to `YES` by default.

22.2 To Assemble Cooling Components on Curves

Download 46 To Assemble Cooling Components on Curves to start from with this chapter.

22.2.1 Sketch the Cooling System Curves

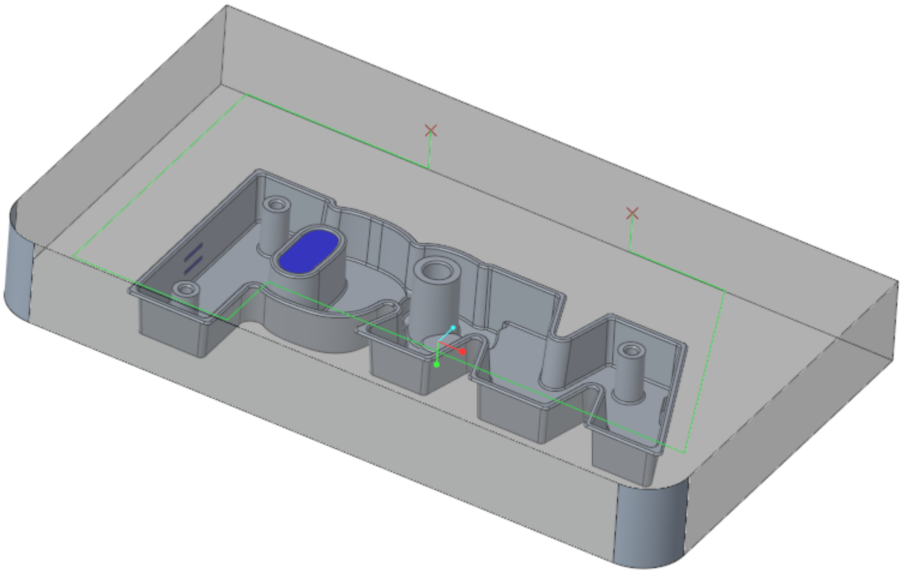
Depending on the requirements sketched curves can be used to define the cooling bore position.

In this tutorial a cooling is added to the cavity plates and the cavity insert parts.

1. Open the `MOLD_VOL_ES.PRT`.
2. Set the bottom side of the model to a transparent color.

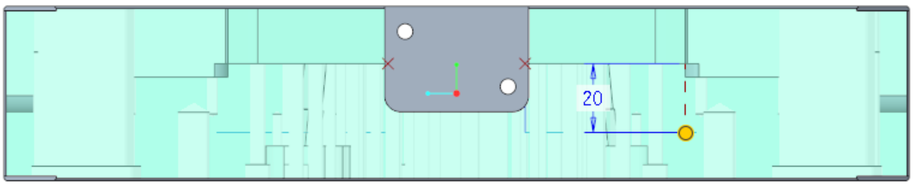
A sketch curve can be discovered within the cavity insert part.

This sketch curves can also be used in the duplicated cavity insert parts MOLD_VOL_ES_1.PRT and MOLD_VOL_ES2.PRT.

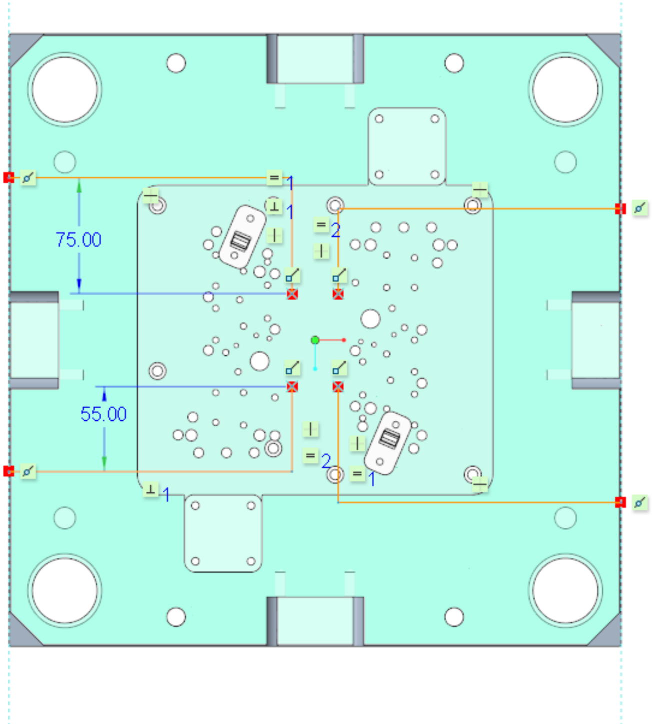


3. Close the MOLD_VOL_ES.PRT model and return to the main assembly.
4. Open the TUTOR_CAV_PLATE_MH_001.PRT.
5. Create a sketch within the cavity plate similar to the image below.

   As a placement reference a datum plane is used with an offset of 20 mm.



Make sure the sketch refers to the sketch created in the MOLD_VOL_ES.PRT.

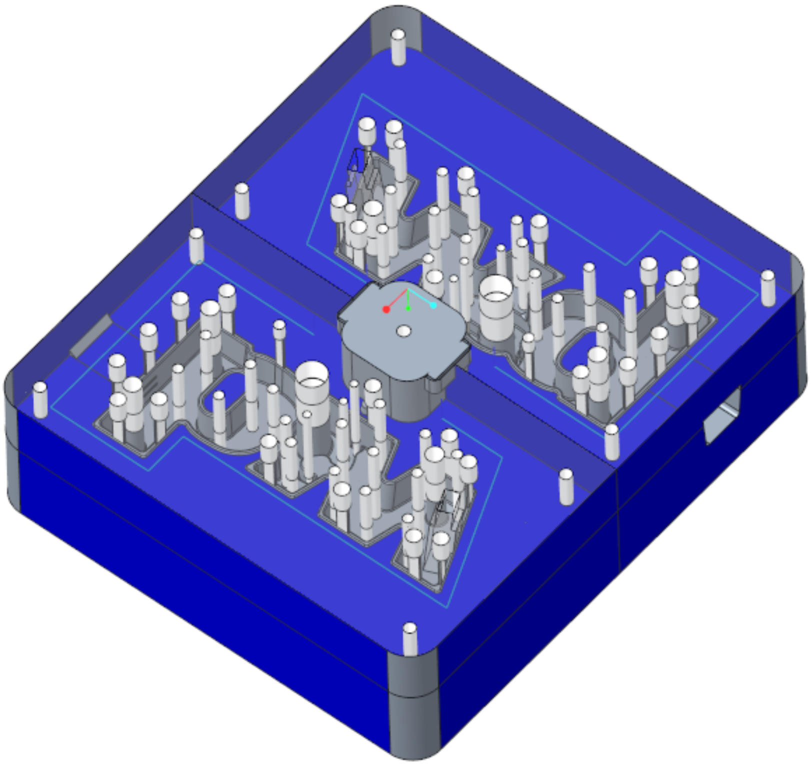


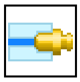
6. Close the TUTOR_CAV_PLATE_MH_001.PRT model and return to the main assembly.

22.2.2 Add Cooling Components on Sketched Curves

1. Open the subassembly TUTOR_SHARED.ASM.
2. Open the MOLD_VOL_ES_1.PRT and MOLD_VOL_ES_2.PRT
3. Set the bottom side of the models to a transparent color.

   This improves the visibility of the curves.

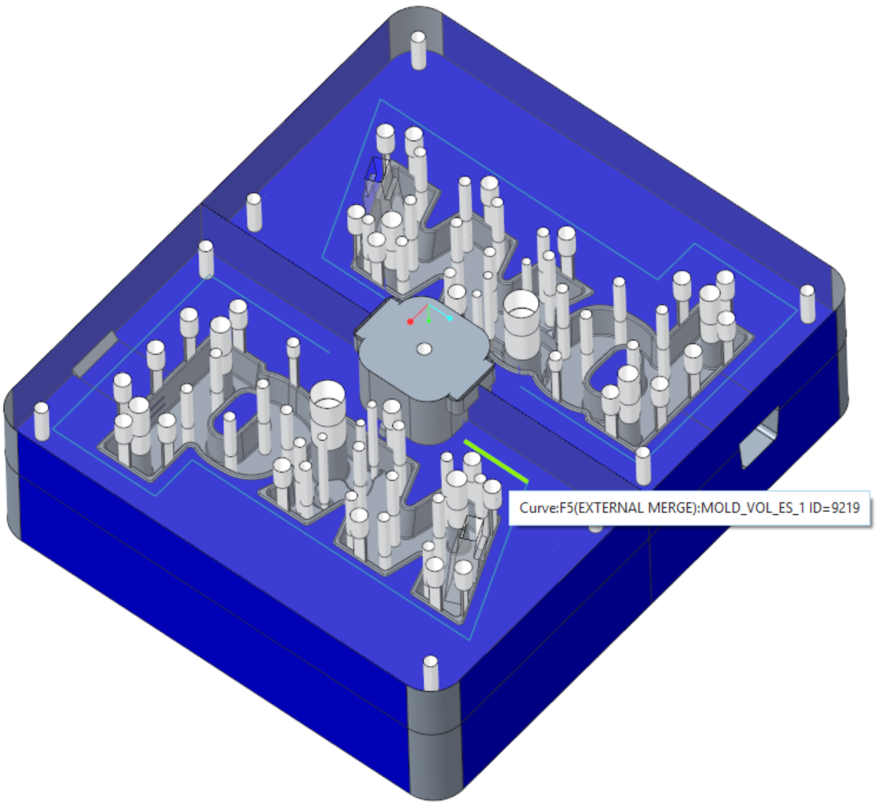


4. Select **EMX Components ► Cooling Component ►**  **Cooling Component**.

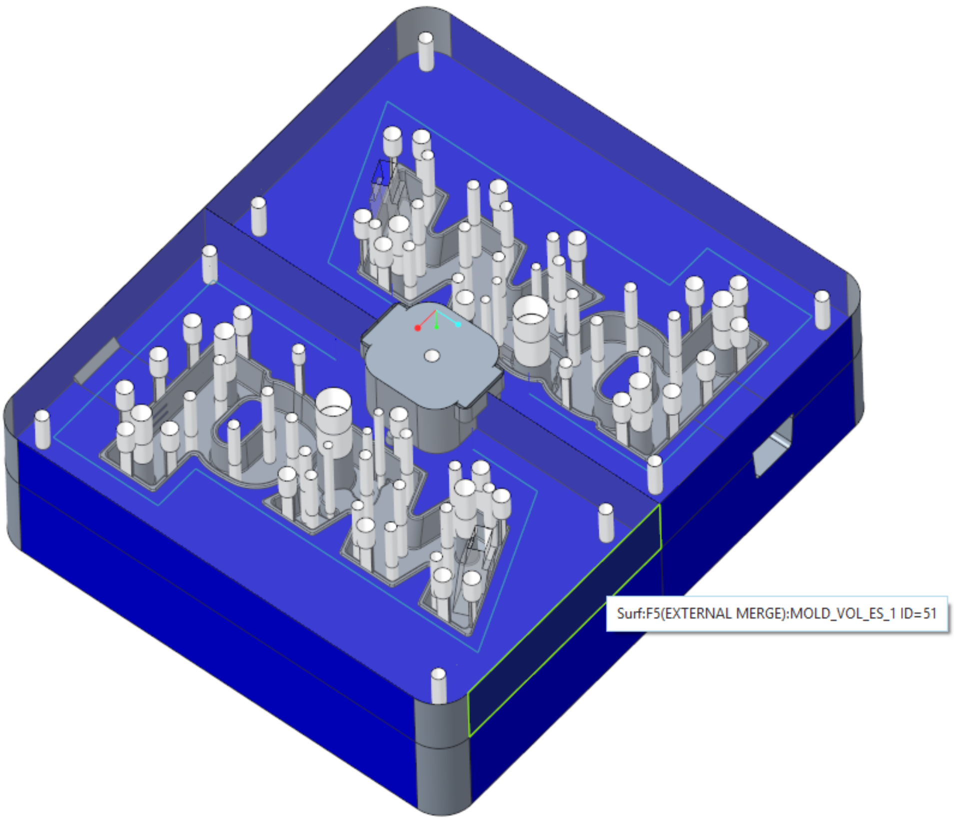
   The **Cooling Component** dialog box opens.

5. Select meusburger from the **Supplier** pull-down menu.
6. Select E2072 from the **Type** pull-down menu.
7. Select 12 mm for the **DM3** nominal value.
8. Define the references.

   a. Click **(1) Curve|Axis|Point** and select one of the curves created in the MOLD_VOL_ES.PRT



b. Click **(2) Surface** and select the side surface of the `MOLD_VOL_ES_1.PRT`



.

9. Click **OK** to assemble the cooling component.

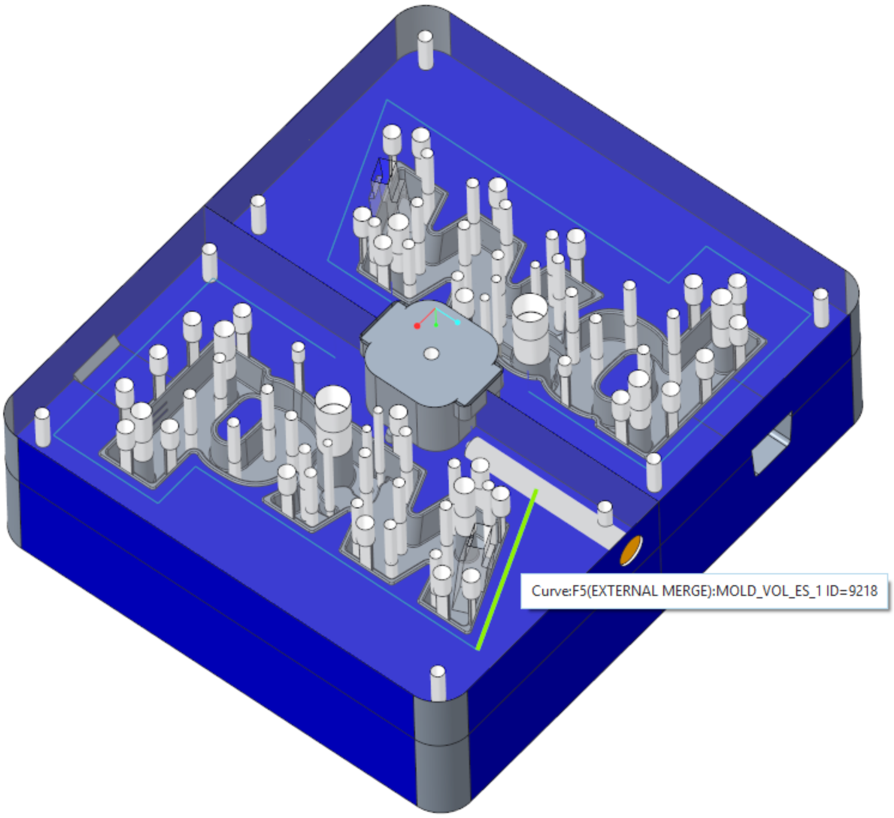
   The cooling component and the related cuts are created.

   NOTE:
   The Selection window stays active.

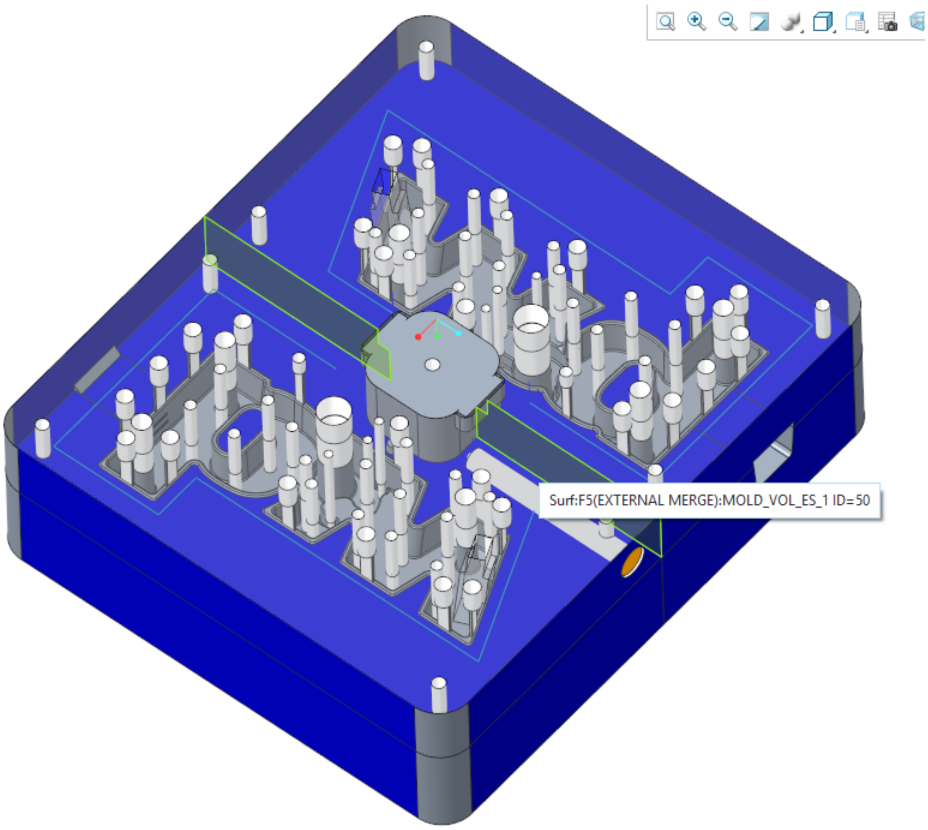
   You are now in a quick-placement loop.

10. To place the same cooling component again select

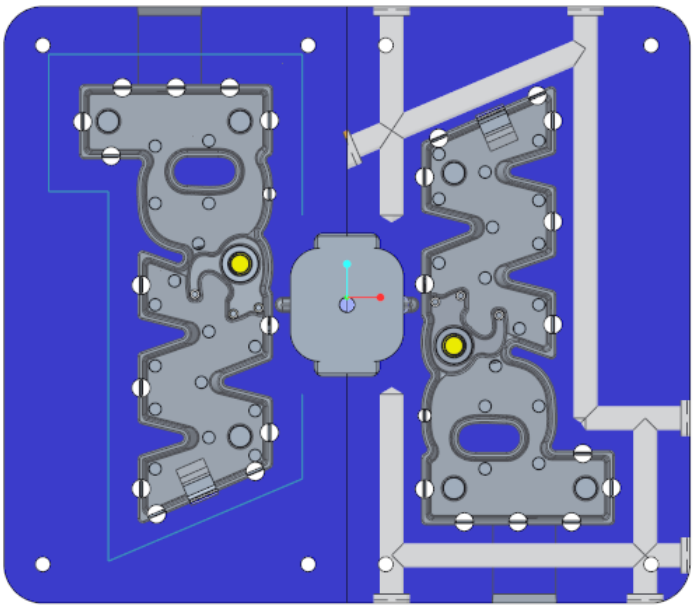
    - a curve



- a surface



11. Finish the cooling circuit.

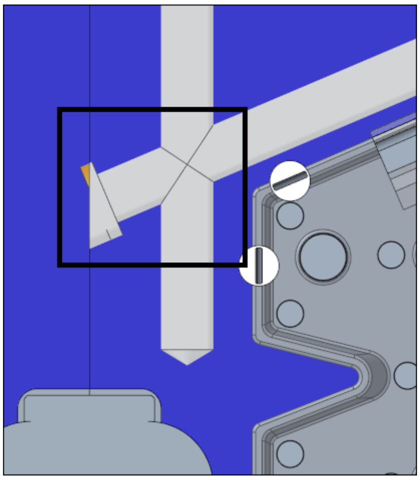


12. Leave the loop with middle mouse button and click **Cancel** to close the dialog box

22.2.3 Modify Counter Bore Depth of Cooling Component

Several cooling components need some modification to finish the placement

1. Select the cooling component which is placed on the angular curve either in the model tree or in the graphics window.

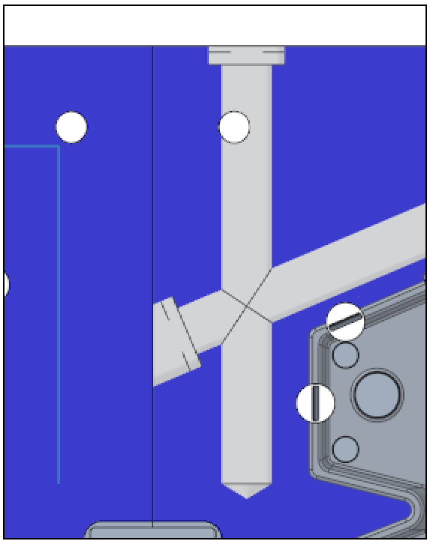


2. Click **Modify** from the mini toolbar or the right-Mouse menu.

   The **Cooling Component** dialog box opens.

3. In the UDF Dimension table set the **T3** value to 6 mm.
4. Click **OK** to adapt the modifications to the cooling component.

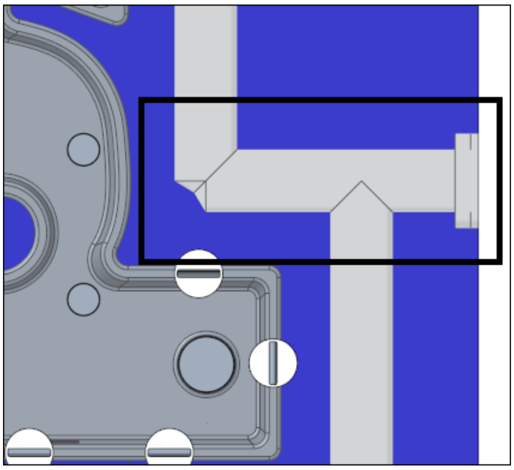
   The counter bore has now a depth of 6mm.



22.2.4 Modify Additional Bore Depth of Cooling Component

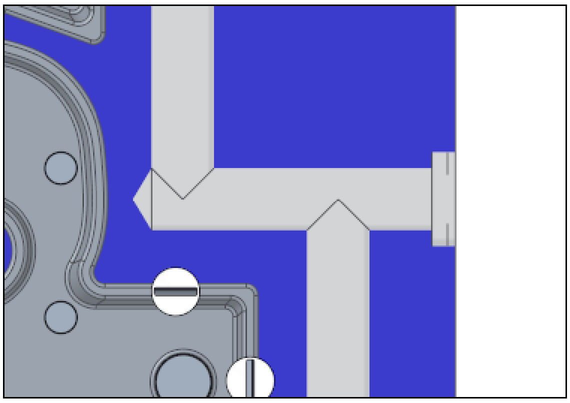
Some cooling components need an additional drill depth to ensure the cross-section of the cooling circuit always has the required diameter.

1. Select the cooling component displayed in the image below either in the model tree or in the graphics window.



2. Click  **Modify** from the mini toolbar or the right-Mouse menu.

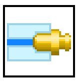
   The **Cooling Component** dialog box opens.
3. In the UDF Dimension table set the **T5** value to `4` mm.
4. Click **OK** to adapt the modifications to the cooling component.



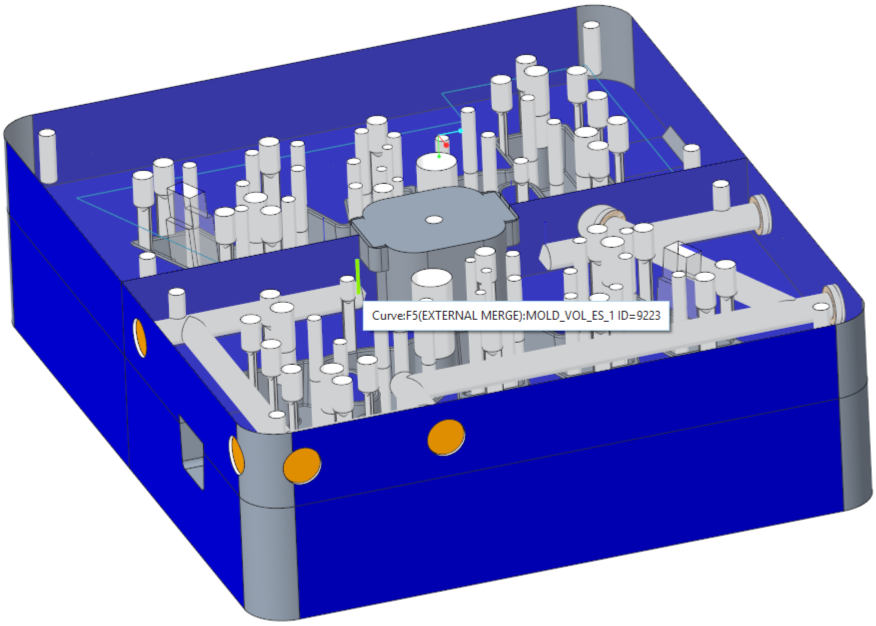
22.2.5 Add Vertical Holes on Sketched Curves

To finish the circuit two vertical holes need to be assembled.

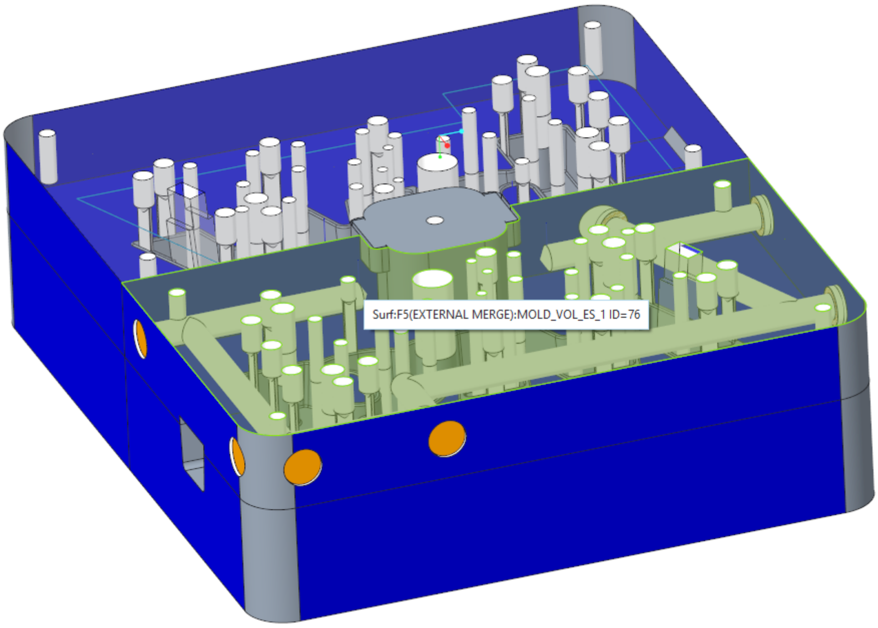
The vertical holes are the connections to the cavity plate on the moving half

1. Select **EMX Components ▸ Cooling Component ▸**  **Cooling Component**.

   The **Cooling Component** dialog box opens.
2. Select `meusburger` from the **Supplier** pull-down menu.
3. Select `Hole` from the **Type** pull-down menu.
4. Select `8` mm for the **NOM** diameter value.
5. In the UDF Dimension table set the **T5** value to `4` mm to ensure the additional bore depth
6. Define the references.
   a. Click **(1) Curve|Axis|Point** and select one of the vertical curves created in the `MOLD_VOL_ES.PRT`



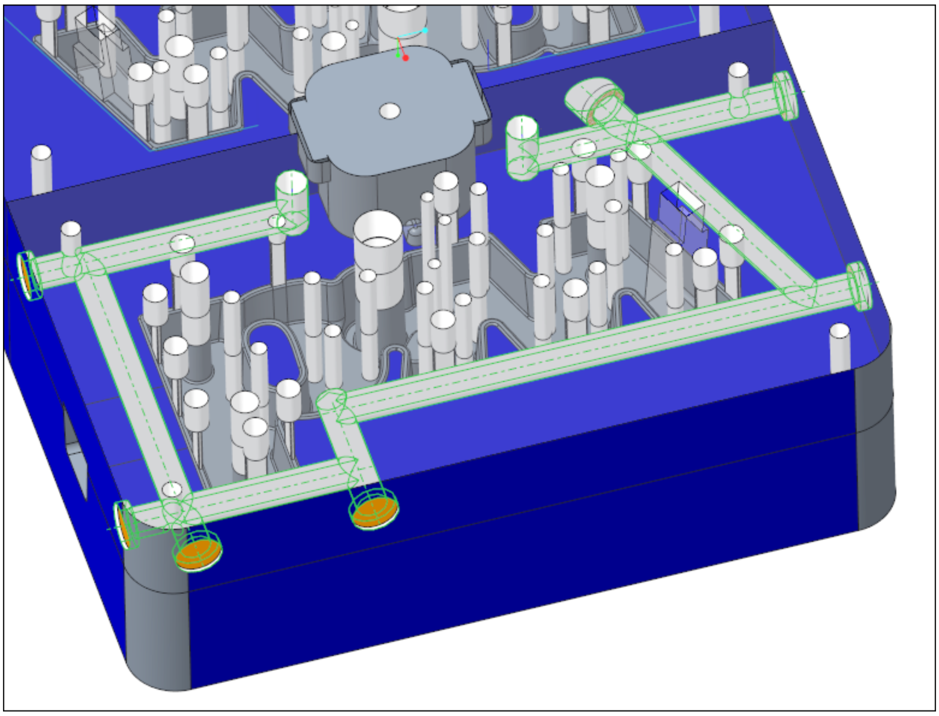
b. Click **(2) Surface** and select the bottom surface of the `MOLD_VOL_ES_1.PRT`



.

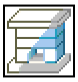
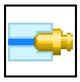
7. Assemble the second vertical hole in the quick selection loop.
8. Leave the loop with middle mouse button and click **Cancel** to close the dialog box.

   The cooling circuit in the cavity insert parts is finished.

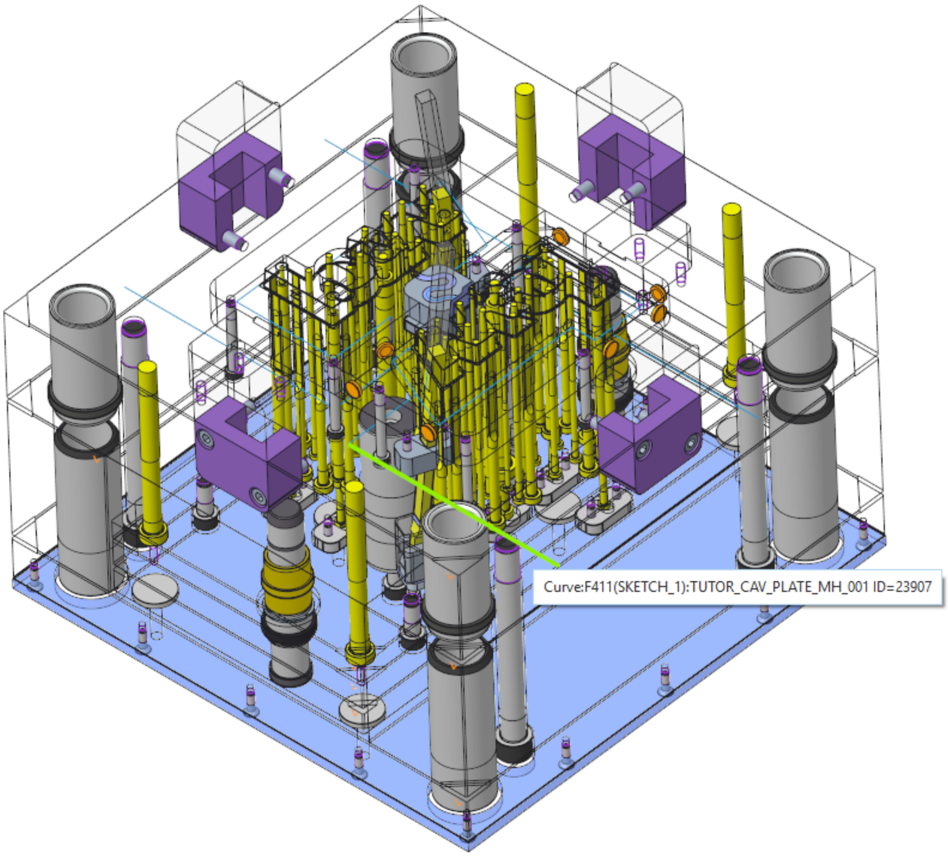


22.2.6 Assemble Related Cooling Channels to Cavity Plate on the Moving half

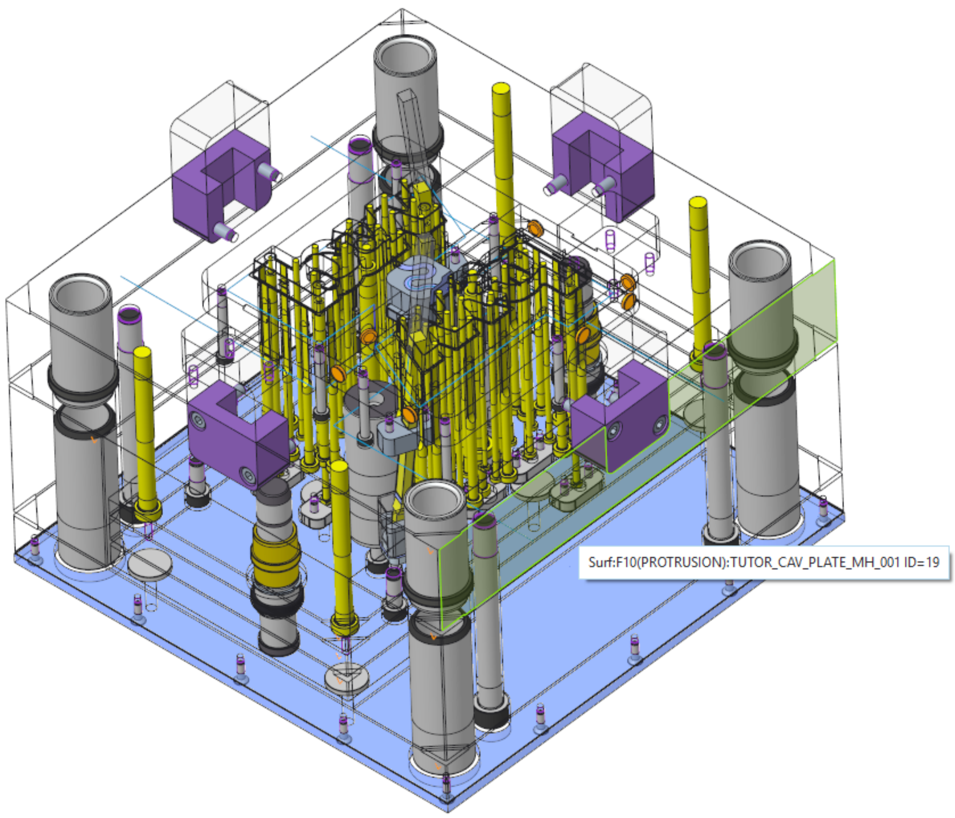
In a first step two cooling nipples for in and out flow are defined

1. Open the subassembly TUTORIAL.ASM.
2. Click **EMX Assembly** ► **View** ►  **Wireframe Style**.
3. Click **EMX Assembly** ► **View** ► **Show** ►  **Moving Half**.
4. Select **EMX Components** ► **Cooling Component** ►  **Cooling Component**.

   The **Cooling Component** dialog box opens.
5. Select TUTOR_MH from the **Subassembly** pull-down menu.
6. Select meusburger from the **Supplier** pull-down menu.
7. Select E2000 from the **Type** pull-down menu.
8. Select 9 mm for the **DM2** diameter value.
9. Define the references.
   a. Click **(1) Curve|Axis|Point** and select one of the curves created in the TUTOR_CAV_PLATE_MH_001.PRT



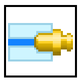
b. Click **(2) Surface** and select the front surface of the `TUTOR_CAV_PLATE_MH_001.PRT`



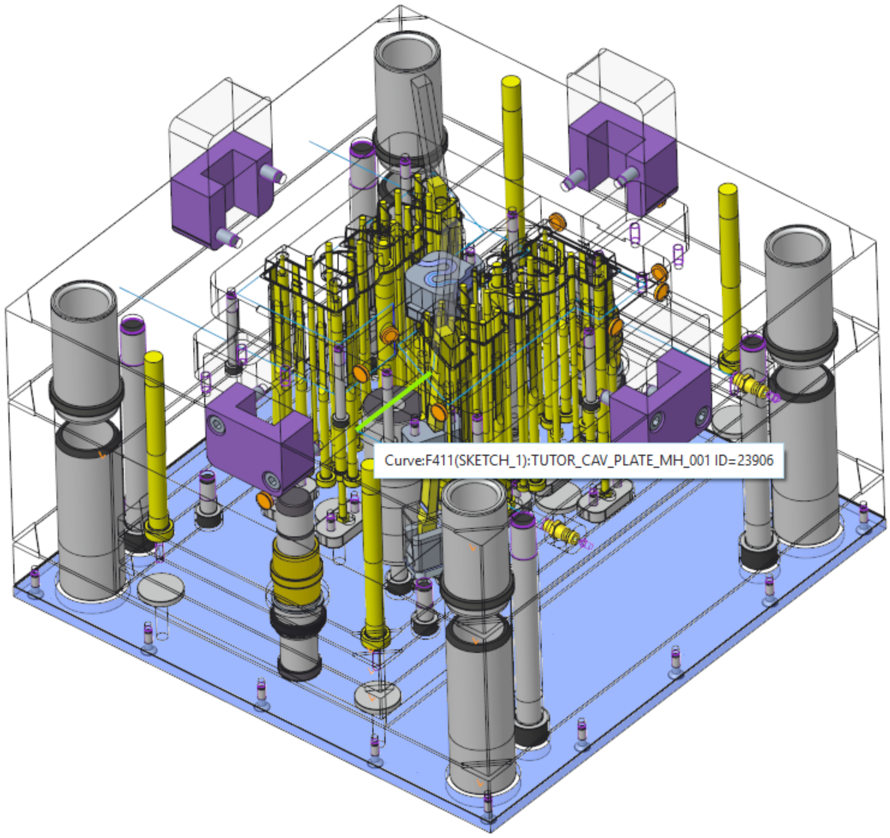
.

10. Click **OK** to assemble the component.
11. Repeat the steps for the second cooling nipple

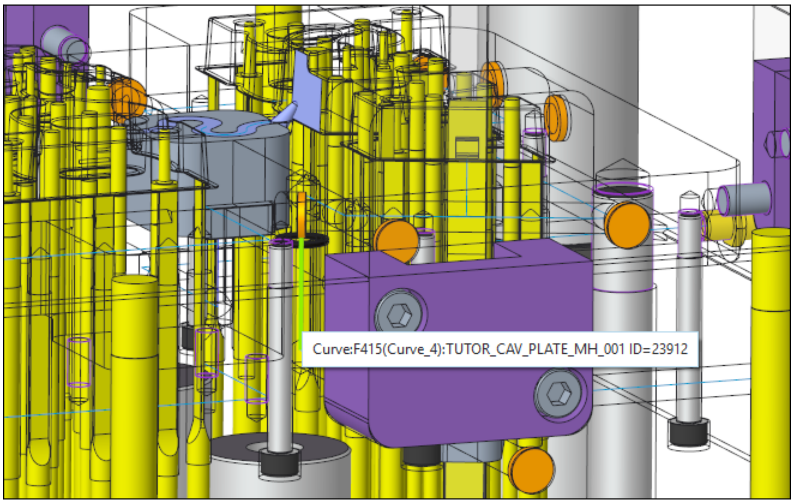
Assemble two additional seal plugs

1. Select **EMX Components ► Cooling Component ►**  **Cooling Component**.

   The **Cooling Component** dialog box opens.
2. Select `TUTOR_MH` from the **Subassembly** pull-down menu.
3. Select `meusburger` from the **Supplier** pull-down menu.
4. Select `E2072` from the **Type** pull-down menu.
5. Select 12 mm for the **DM3** diameter value.
6. Define the references.
   a. Click **(1) Curve|Axis|Point** and select one of the curves created in the `TUTOR_CAV_ PLATE_MH_001.PRT`



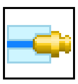
b. Click **(2) Surface** and select the left surface of the `TUTOR_CAV_PLATE_MH_001.PRT`



.

7. Click **OK** to assemble the component.
8. Repeat the steps for the second seal plug

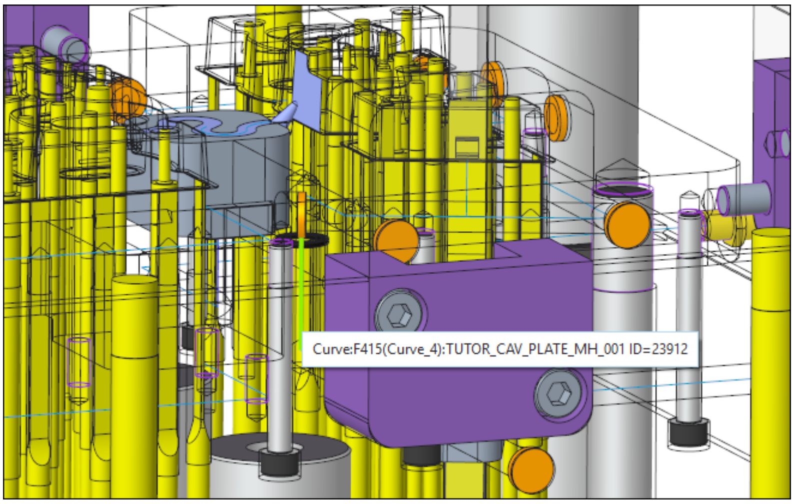
Assemble two additional o-rings to the vertical curves as connection to the cavity insert part

1. Select **EMX Components ▶ Cooling Component ▶**  **Cooling Component**.

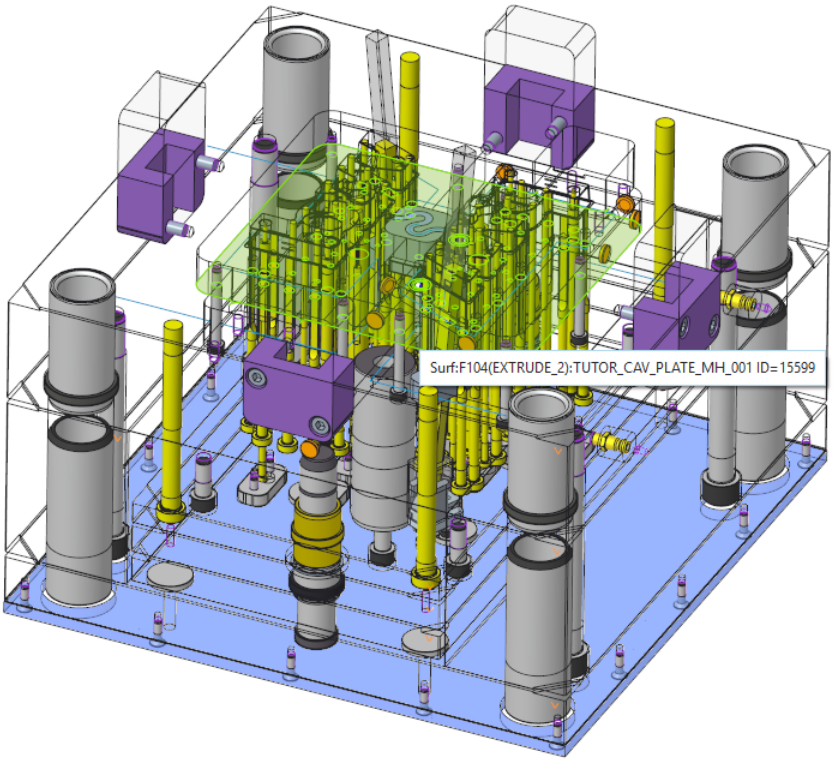
The **Cooling Component** dialog box opens.

2. Select TUTOR_MH from the **Subassembly** pull-down menu.
3. Select meusburger from the **Supplier** pull-down menu.
4. Select E2130 from the **Type** pull-down menu.
5. Select 11 mm for the **DM1** diameter value.
6. In the UDF Dimension table set the **T5** value to 4 mm to ensure the additional bore depth
7. Define the references.

   a. Click **(1) Curve|Axis|Point** and select one of the curves created in the TUTOR_CAV_PLATE_MH_001.PRT

   b. Click **(2) Surface** and select the left surface of the TUTOR_CAV_PLATE_MH_001.PRT



.

8. Click **OK** to assemble the component.
9. Repeat the steps for the second seal plug

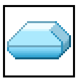
   All cooling components for the first cooling circuit are assembled.

22.3 To Use Mold Analysis to Check the Wall Thickness

Download 47 To Use Mold Analysis to Check the Wall Thickness to start from with this chapter.

22.3.1 Run an Mold Analysis for the Cavity Insert Part

The Mold Analysis for Creo Mold Design is a perfect tool to check the cooling circuit design.

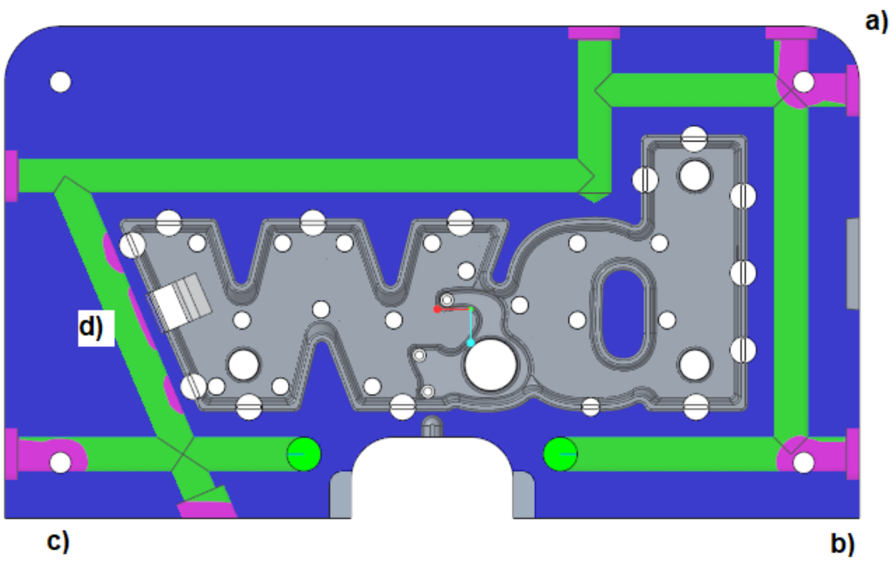
1. Open the `MOLD_VOL_ES_1.PRT`.
2. Click **Applications ▸**  **Mold/Cast**

   The Creo Mold Design ribbon is activated.
3. Click **Mold &Cast ▸ Analysis ▸**  **Mold Analysis**.

   the **Mold Analysis** dialog box opens.
4. For **Minimum Clearance** set 4 mm.

5. Click **Compute** to run the wall thickness check.

   The result is displayed in the graphics window.



Several issues are found:

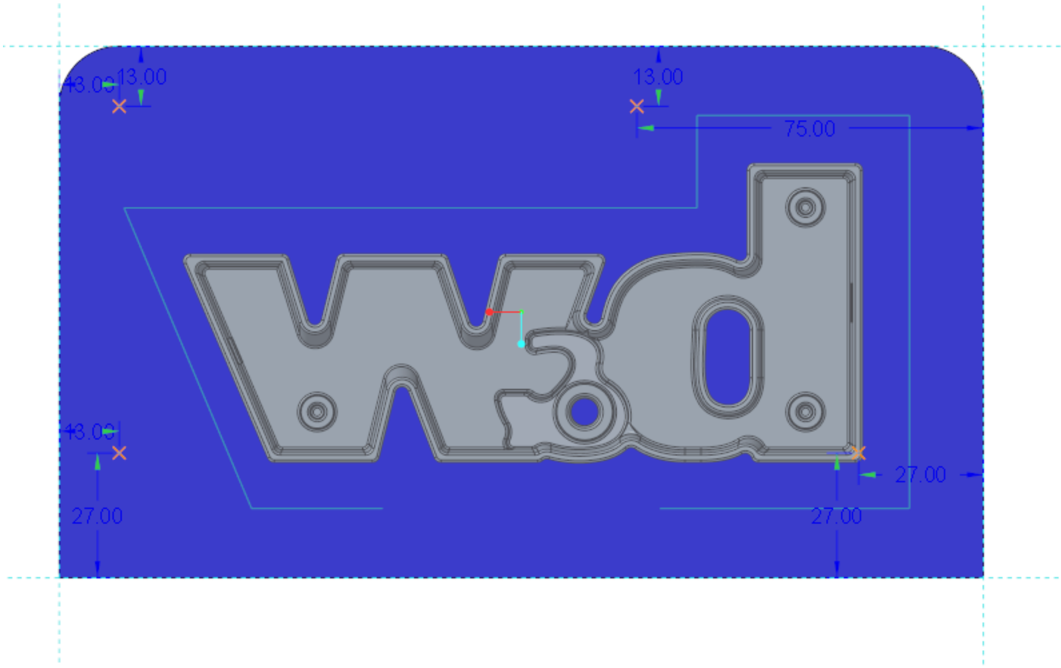
a. Interferences with screw hole
b. Interferences with screw hole
c. Interferences with screw hole
d. Cooling channel is too close to ejector pin bores

Design measures are require to fix these problems

6. Close the **Mold Analysis** dialog box with **Close.**
7. Click **Mold &Cast ► Close ► Close** to return to the Creo part mode.

22.3.2 Fix Issues discovered by Interference Check

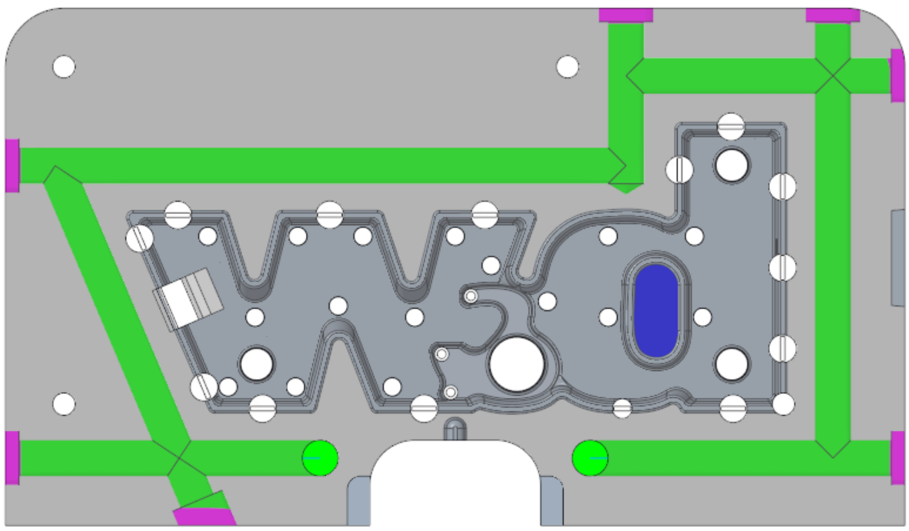
1. Modify the `Sketch 1` in the model tree to rearrange the screws.
2. Move the points to a position simillar to the image displayed below.



3. Open the `MOLD_VOL_ES.PRT` model
4. Modify the `Sketch 1` in the model tree to adapt the curves.

   Make sure the angular curve has an sufficient distance to the reference model geometry
5. Open the `TUTORORIAL.ASM`.
6. Regenerate the mold base assembly.
7. Return to the `MOLD_VOL_ES_1.PRT`.
8. Run the **Mold Analysis** again.

   The result is displayed in the graphics window.Several issues are found:



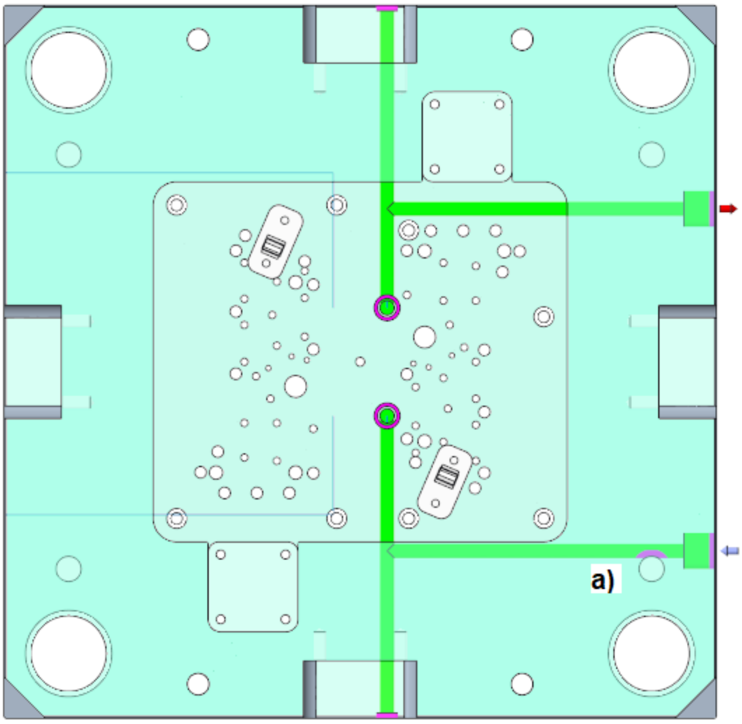
The previously discovered issues are resolved.

9. Return to the main assembly `TUTORIAL.ASM`

22.3.3 Run an Mold Analysis for the Cavity Plate in the Moving Half

1. Open the `TUTOR_CAV_PLATE_MH_001.PRT`.
2. Run a **Mold Analysis**.

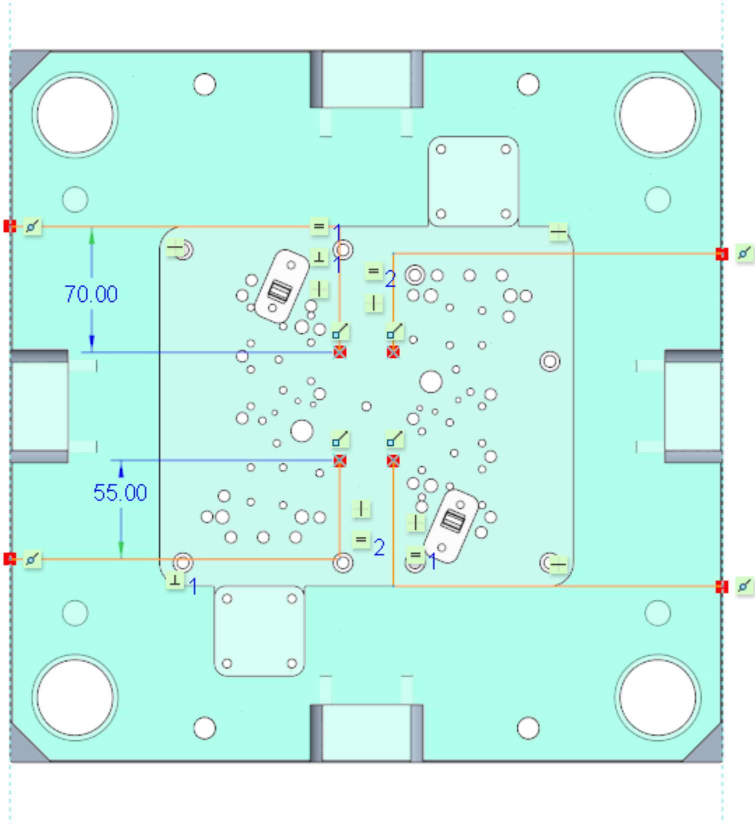
   The result is displayed in the graphics window.



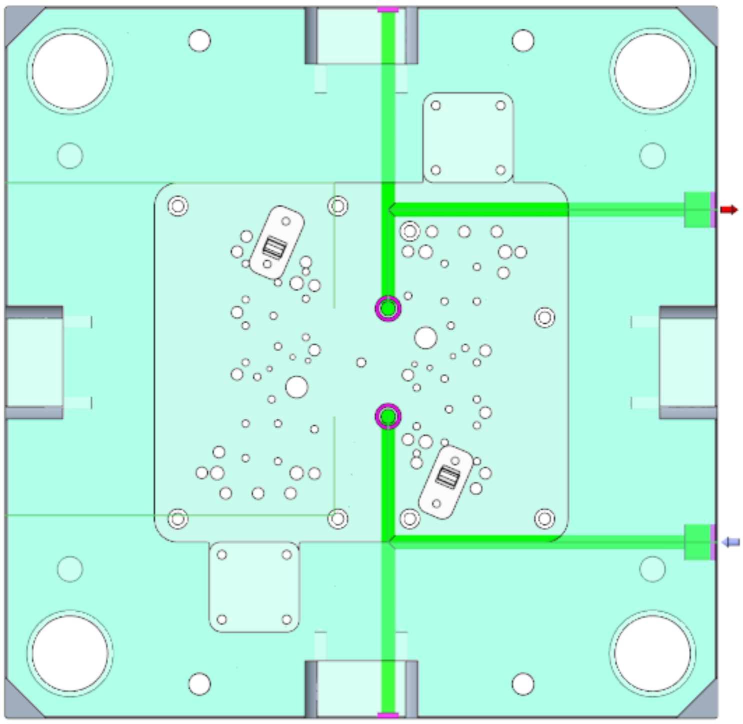
one issue is found:

a. The cooling channel interferes with the clamping screw

3. Modify the `Sketch 1` in the model tree to rearrange the curves.



4. Rerun the **Mold Analysis**.



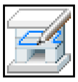
The previously discovered issue is resolved.

5. Return to the main assembly `TUTORIAL.ASM`

22.4 About Creating Cooling Circuits With the Waterline Designer

Use the **Waterline Designer** to speed up the process of defining waterline circuits.

It takes fewer clicks to create waterline leading curves and the entire system of cuts and cooling components.

1. Click **EMX Components** ►  **Waterline Designer**.

   The **Waterline Designer** and the **Select** dialog boxes open.
   The **Waterline Designer** dialog box contains the two tabs described below:

   - **Design**—Contains options for designing cooling circuits with the waterline.
   - **Options**—Contains filters and options to help you design the waterline.

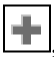
2. In the graphics window, select a list of the cavity plates where you need to define waterline circuits and then in the **Select** dialog box click **OK**.
3. The **Select Working Models** list is populated.

   NOTE:
   You can create cooling circuits within just one working model, or between plates in multiple working models.

4. In the **Design** tab, click the **Select Working Models** box, and select your active working model.

   The active working model is the model in which you can define, modify, or delete waterlines.

   In the **Select Working Models** area of this dialog box you can perform the actions described below:

   - To change the active working model click , select the model in the graphics window, and then click **OK** in the **Select** dialog box.
   - To add plates to the list of working models, click , and in the graphics window select the plate you want to add.

     The simplified representation is temporarily not visible.

     Click **OK** in the **Select** dialog box and the complete mold design appears.

5. Use the **Sketch Planes** table to manage your sketch planes.

   A sketch plane contains a Z position. The Z position of the drawing plane is measured from the coordinate system ORIGIN_MOLDBASE which must be present in the Creo Expert Moldbase Extension mold base assembly.

A sketch plane is valid only if a waterline is defined on it. Sketch planes without waterlines are automatically erased by the **Waterline Designer**.

- To add sketch planes click . By default the added sketch planes are placed in the middle of the model.
- To change the placement of a sketch plane, edit the value of the sketch plane in the **Z** column.

6. Use the **Cooling Circuits** table to add single waterlines to predefined cooling circuits. Each cooling circuit is defined by a color and a diameter.

- In the color column the ten predefined colors for waterlines appear.

  To make a cooling circuit active, double-click the row in the table.

  The **Waterline Designer** then adds new waterlines to the active cooling circuit.

  You can change and add waterline analysis colors by following either of the methods below:

  - Click **EMX Assembly ▶ Options** and then click the **Technology** tab.

    In the **Description** column, scroll to the `waterline_analysis_color` entries.

    In the **Hole Color** column, click a color to change to the color. Place the pointer in any `waterline_analysis_color` row and click to add new waterline analysis color and then edit the **Description** and **Hole Color** as required.

    The additional waterlines appear in the **Cooling Circuits** table.
  - From the <EMX_loadpoint> open the configuration folder. In the configuration folder open `bore_technology.cfg`.

    Scroll to the `waterline_analysis_color` entries and made additional entries, such as `waterline_analysis_color11`.

    The additional waterlines appear in the **Cooling Circuits** table.

- In the diameter column, the diameter of the cooling circuit appears.

  This value is the designated diameter of the cooling circuit.

  If you change this value all nominal waterline bores in the cooling circuit are adjusted.

  NOTE:
  All cooling bores in a cooling circuit should have the diameter value that is listed.

  In Creo Expert Moldbase Extension, the cooling bore diameter is usually defined by the selected component in the **Component** dialog box.

  You can manually change the value of the cooling bore later, but as a result, the component and the cooling bore diameter may not fit together.

  Therefore, you should avoid changing the diameter of the cooling bore manually for the components defined with the **Waterline Designer**.

7. Click the **Options** tab to set snapping rules and collision control to support your waterline creation.
    - Select or clear the check boxes under **Snap Rules** for support during your waterline creation process.

      The **Waterline Designer** recognizes all previously created waterlines and helps you to define new ones based on their positions.

      The **Waterline Designer** supports snapping to knots and waterlines.

      It also helps you to create waterlines that are orthogonal, symmetric, or directly on or to the ORIGIN_MOLDBASE.
    - If the **Collision Control** check box is selected, and the waterline intersects any cut within the plate when moving the pointer to select the second point, the waterline appears in red.

      If no collision is detected or the **Collision Control** check box is cleared, the waterline and the inlets appear in the color associated with the cooling circuit.

      In the **Stock allowance** box, you can change the value that represents the minimum allowed distance to the next hole in the plate.
8. In the **Design** tab, below the **Cooling Circuits** table, there are buttons for creating, modifying, and deleting waterlines and for assembling predefined components of waterlines.

   As you create waterlines, the waterlines for the currently selected working model appear in a list on the right panel of the **Waterline Designer** dialog box.

   For more information on the list of waterlines, go to step 9. The steps for creating, modifying, and deleting waterlines and for assembling predefined components of waterlines, are described below:
    - To create a horizontal waterline, follow the steps below:
        a. Click and then click anywhere within the plate boundaries to define the first point of the waterline.

           NOTE:
           The coordinates of your pointer position are visible in the plate with the origin on the `ORIGIN_MOLDBASE` coordinate system.

        b. In the graphics window, move the pointer within the plate boundaries to draw a cylinder-shaped waterline showing the real position and size of the waterline bore

        c. Click repeatedly while searching for the second point, to activate or deactivate the inlets for the waterline, one at a time.

        d. Click anywhere within the plate boundaries when the waterline is displayed as valid, to select the second point for the waterline. The waterline curve is assembled to the plate
    - To create a waterline on a sketched curve follow the steps below:
        a. Click and then select an existing straight sketched curve inside the current working model.

           A cylinder-shaped waterline appears at the selected curve.

b. Right-click repeatedly to activate or deactivate the inlets for the waterline, one at a time.

c. Click to finish the waterline.

NOTE:
Waterlines on existing curves update when regenerating, after the position of a sketched curves changes.

- To create a vertical waterline, follow the steps below:

a. Click  and then right-click anywhere within the plate boundaries.

A vertical waterline appears.

The cooling bore reaches to the top or to the bottom of the selected working model.

b. Right-click repeatedly to change the up or down direction.

If the waterline in both directions is not valid, the creation process is interrupted.

- To modify a waterline follow the steps below:

a. Click  to activate the modify mode for existing waterlines.

In this mode the waterline knots appear as white small squares on the waterline.

b. Place the pointer on one knot until the color of the knot turns red and then click the knot and move the pointer.

The selected knot follows the pointer within the plate.

c. Click again to place the selected knot at the current position of the pointer.

The waterline curve is updated to the new position.

Middle–click when you want to stop modifying.

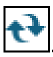
d. Click a waterline which is parallel to one of the main axes and then move the pointer.

The waterline moves in parallel to the previous direction.

Click again to relocate the waterline to the new position.

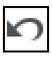
e. Right–click one of the knots that turned red to open the **Knot** dialog box.

In the **Knot** dialog box you can set each coordinate of the selected knot.

To update the values in the model, click .

NOTE:
In the **Options** tab you can select the **Shift all aligned knots** check box to shift all other knots of the defined waterlines which are on the same coordinate, with the selected knot.

Click **OK** to close the **Knot** dialog box.

If an incorrect coordinate was entered and you want to go back to the previous settings, click  and the previous coordinates are applied.

- To delete a waterline, follow the steps below:

  a. Click and then place the pointer on the waterline to be removed. The waterline is highlighted.

  b. Click the waterline to remove it. The waterline, the component, and the cooling bore cut out are deleted from the mold base.

- After you finish defining the waterlines in the plate, you can assemble all components including cooling bore cut outs for all defined waterlines.

  Click to assemble the cooling components and the cut outs after defining waterlines, and when applying modifications.

9. As waterlines are created, they appear in the right panel of the **Waterline Designer** dialog box.

   Review the bullets listed below for information on customizing your list of waterlines, and on the **Cooling Component** dialog box:

   - In the **Options** tab, under **Show Waterlines in List**, select or clear the check boxes to customize your list of waterlines.

     You can show all waterlines, waterlines in the active working model, or waterlines in the active cooling circuit.

   - In the list, a waterline is represented with icons for the start and end component. Depending on the type of component, you see a stylized preview.

   - Double-click the component icon , to open the **Cooling Component** dialog box where you can define the components and the associated cooling bore cut out.

   - Click **EMX Assembly ► Options** and set the configuration option `WATERLINE_DESIGNER_DEFAULT_COMPONENT`. By default, what you set is assigned at start and end of the waterline. A valid format for this EMX option is `unit|supplier|instance_name`, such as `mm|meusburger|E2074_8_10`

   - In the color column you can see the color of the associated cooling circuit.

   - The eye-shaped icons, or indicate whether a start or end component should be assembled.

10. When working with the **Waterline Designer**, consider the related EMX configuration options which are listed below:

    - `WATERLINE_DESIGNER_DEFAULT_COMPONENT`—Defines the component which is added to the waterlines by default.

      A valid option format is `unit|supplier|instance_name` such as `mm|meusburger|E2074_8_10`.

    - `DEFAULT_WATERLINE_DIA`—Defines which diameter is assigned to the cooling circuits by default.

      NOTE:
      Consider your mold base unit system. A diameter which is set to 8 in an INCH-Assembly may be larger than expected.

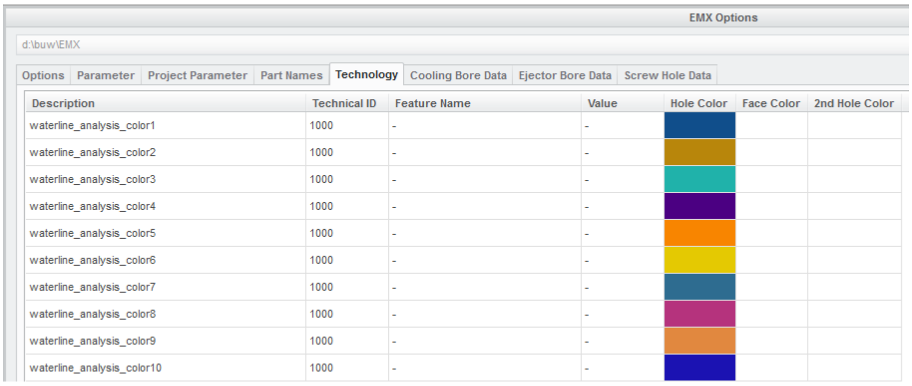
- `WATERLINE_DESIGNER_OVERMEASURE`—Defines the overmeasure value which is set by default.
- `WATERLINE_DESIGNER_DIA_LABEL`—Defines all diameter values available for selection in the diameter column of the **Cooling Circuits** table.

  A valid option format is `dia1|dia2|dia3` such as `8|10|12`.
- `WATERLINE_DESIGNER_XSEC_CREATE`—If set to `YES` the **Waterline Designer** creates cross-sections through all defined sketch planes in the selected working models when you click **OK** to close the **Waterline Designer** dialog box.

  You can then easily investigate your recently created cooling circuits. Note that these cross sections may be erased during future **Waterline Designer** sessions.

NOTE:
Click **EMX Assembly ► EMX Options** and then select the **Technology** tab.

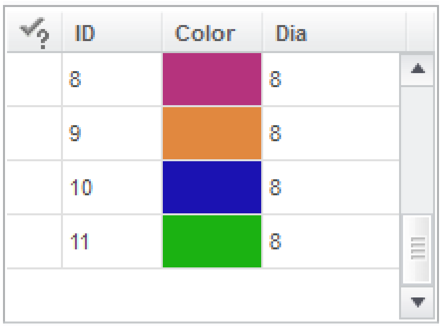
In the **Description** column, scroll to the waterline_analysis_color entries.



In the **Hole Color** column, click a line to change its color.

Place the pointer in any `waterline_analysis_color` row and click  to add new waterline analysis color and then edit the **Description** and **Hole Color** as required.

The additional waterlines appear in the **Cooling Circuits** table.

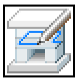


The new entry will be stored in the file <EMX_loadpoint>/configuration/bore_technology.cfg.

22.5 To Define Cooling Circuits Using the Waterline Designer

Download 48 To Define Cooling Circuits Using the Waterline Designer to start from with this chapter.

22.5.1 Start the Waterline Designer

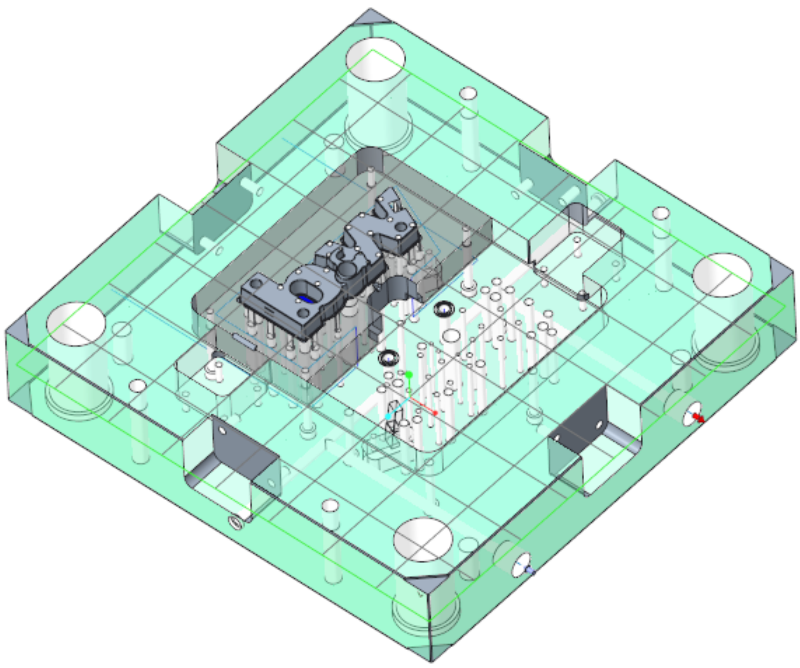
1. Open the assembly `TUTORIAL.ASM`
2. Click **EMX Components** ► **Cooling Components** ►  **Waterline Designer** .

   The **Waterline Designer** and the **Select** dialog boxes open.

   The **Waterline Designer** is still empty which means that the working models are not selected yet.
3. Select all cavity plates and models you want to define waterline circuits on.
4. Select the components:
   - `TUTOR_CAV_PLATE_MH001.PRT`
   - `MOLD_VOL_ES_2.PRT`
5. Click **OK** in the **Select** dialog box.

   The **Waterline Designer** dialog box opens.

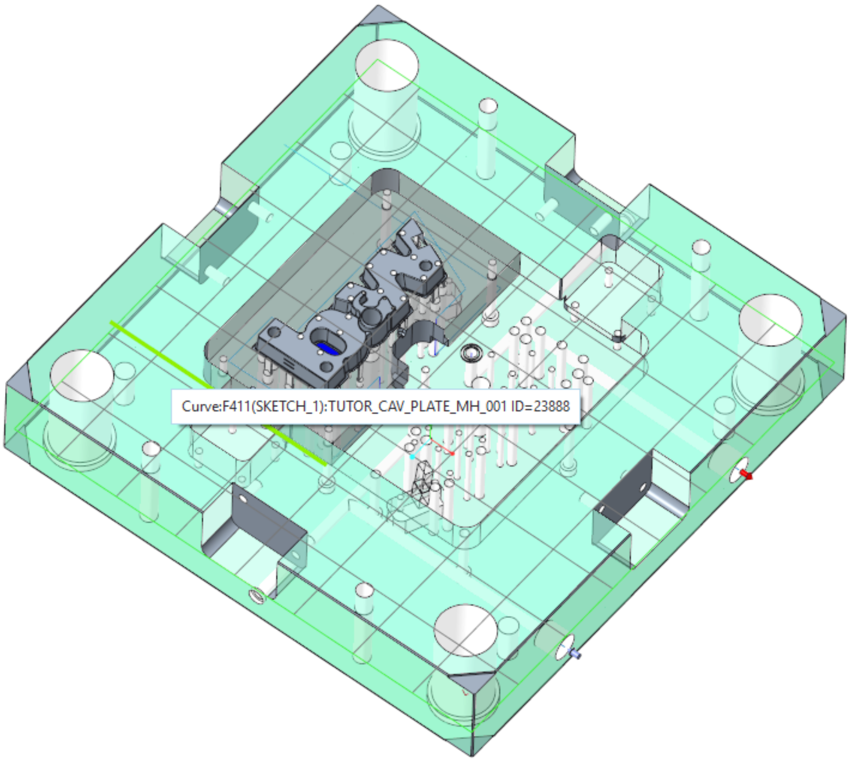
   A simplified representation is activated which only contains the listed working models.



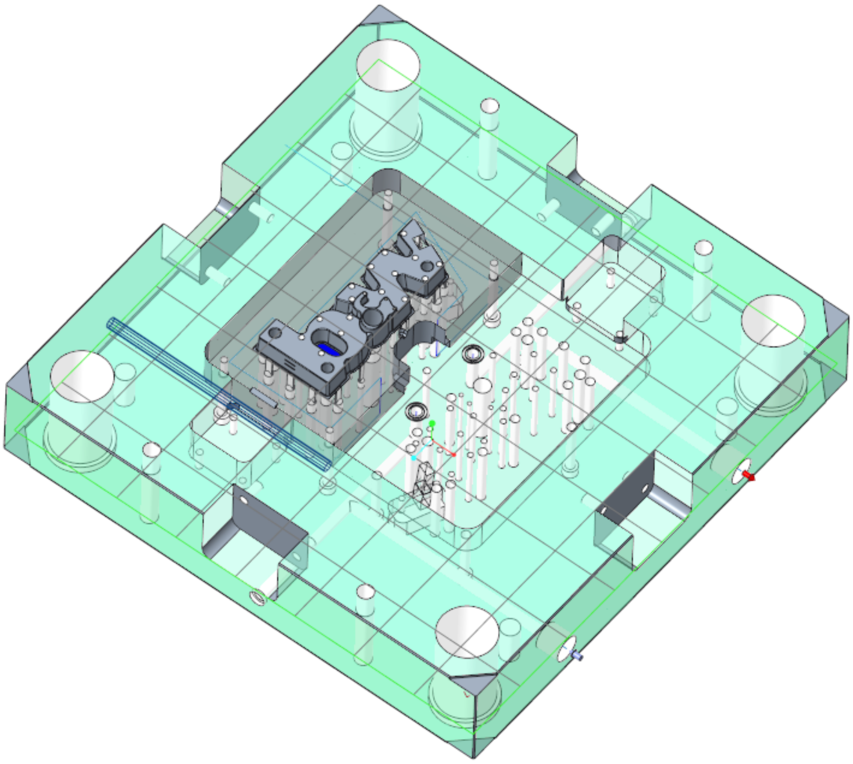
22.5.2 Define Waterline on Existing Curves Cavity Plate

1. Click  to define a waterline on an existing curve.

   The **Select** window appears.
2. Select the initial curve for the circuit with the left-mouse button.



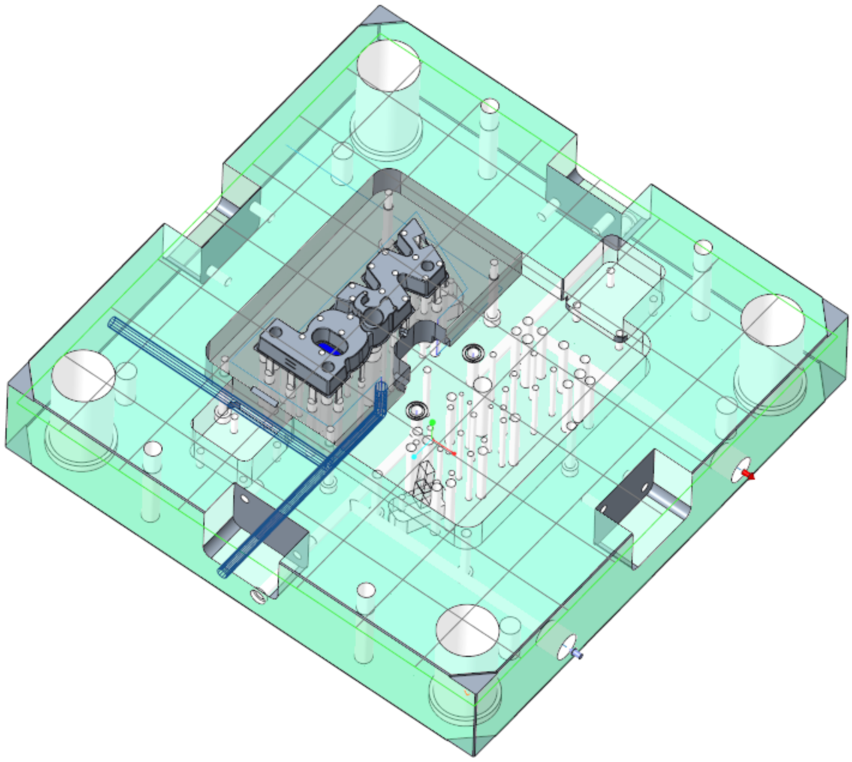
A waterline preview is displayed in the graphics window.



3. Click the right-mouse button to toggle the inlets.

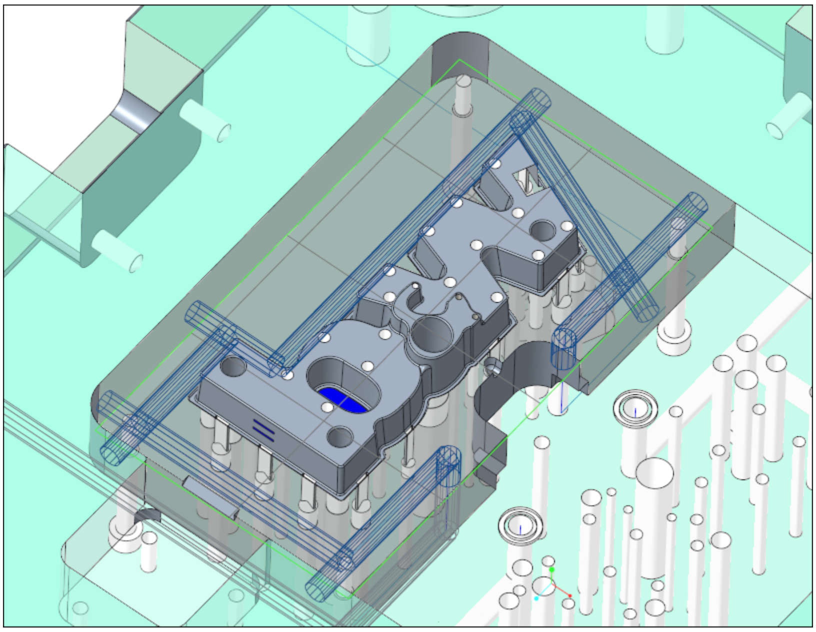
   Stop with toggling in case the inlet is set to the outer surface of the cavity plate as seen in the image above.

4. Click left-mouse button to finally place the waterline.
5. Repeat these steps for the next two curves.



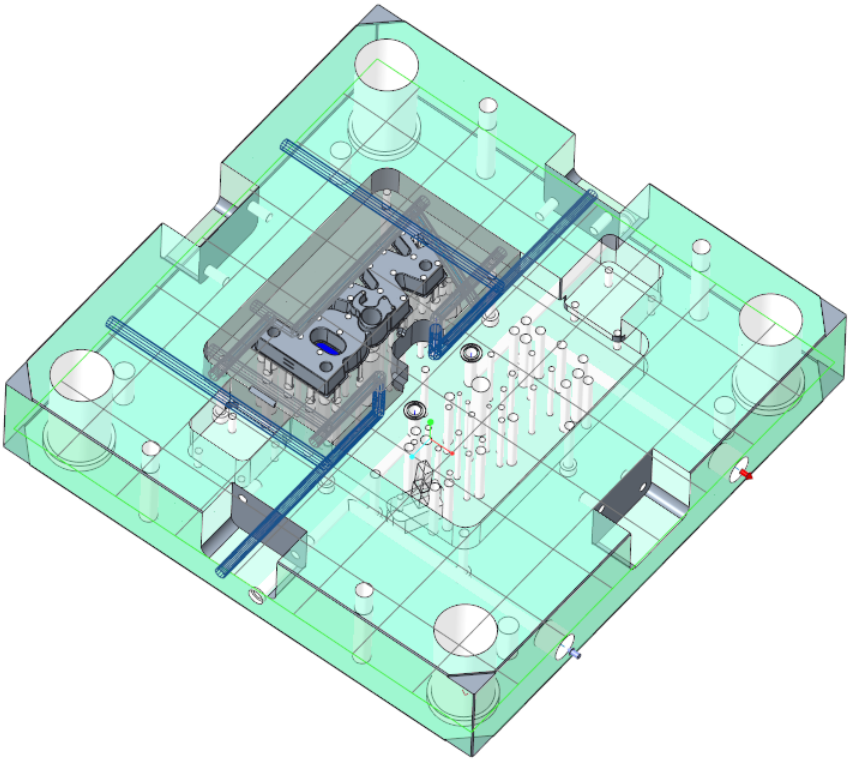
6. Leave the selection loop with **Cancel**
7. Select `MOLD_VOL_ES_2` in the **Select Working Models** pull-down menu to make it the active working model.
8. Click  again to define a waterline on an existing curve.

   The **Select** window appears.
9. Continue to place waterlines on the existing curves step by step within the `MOLD_VOL_ES_2.PRT`.



10. Leave the selection loop with **Cancel**
11. Select TUTOR_CAV_PLATE_MH001.PRT in the **Select Working Models** pull-down menu to make it the active working model.
12. Click  again to define a waterline on an existing curve.

    The **Select** window appears.
13. Continue to place waterlines on the existing curves step by step within the TUTOR_CAV_PLATE_MH001.PRT.



All waterlines are defined.

22.5.3 Work in the Waterline Table to Define Different Component Types

Modify the type of the first waterline to a cool nipple.

1. Double-Click on  in the first row of the waterline table.

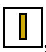
   The **Cooling Component** dialog box opens.
2. Select `E2000` in the **Types** pull-down menu.
3. Select `13.5` mm for **DM2** value.
4. Click **OK** to finalize the type definition.

   The new type is defined and the icon changes to , which stands for cooling nipple.

Modify the type of the second waterline to a seal plug.

1. Double-Click on  in the second row of the waterline table.

   The **Cooling Component** dialog box opens.
2. Select `E2072` in the **Types** pull-down menu.
3. Select `12` mm for **DM3** value.
4. Click **OK** to finalize the type definition.

   The new type is defined and the icon changes to , which stands for seal plug.

Modify the type of the second waterline to a o-ring.

1. Double-Click on in the second row of the waterline table.

   The **Cooling Component** dialog box opens.
2. Select E2130 in the **Types** pull-down menu.
3. Select 11 mm for **DM1** value.
4. Select 11 mm for **DM1** value.
5. Click **OK** to finalize the type definition.

   The new type is defined and the icon changes to , which stands for o-ring.

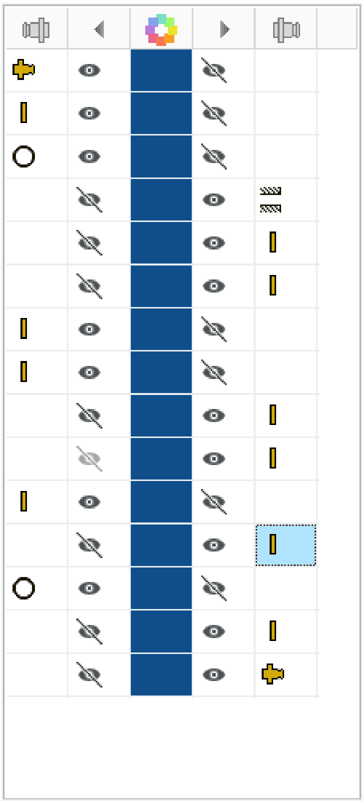
Modify the types using the right-mouse quick selection.

1. Right-Mouse Click on in the last row of the waterline table.
2. Click on the **E2000_13_14** button.

   The new type is defined and the icon changes to , which stands for cooling nipple.

   NOTE:
   These quick selection types can be set up using the configuration option WATERLINE_ DESIGNER_DEFAULT_COMPONENT

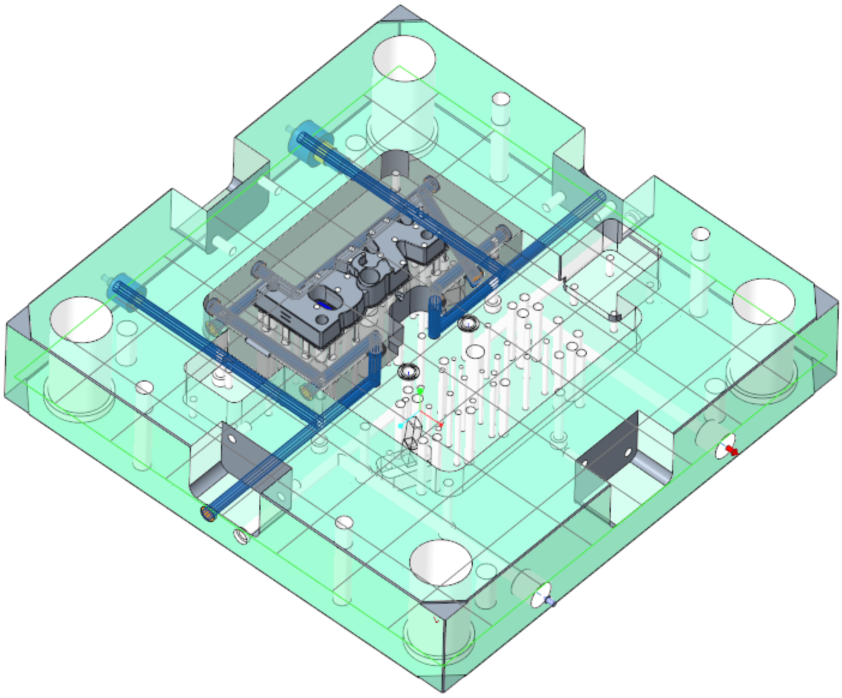
Finish the component type definition using for all waterlines in the table.



22.5.4 Assemble the Cooling Components

1. Click to assemble all cooling components.

   All components and cuts are created to the mold base.



2. Close the **Waterline Designer** dialog box with **OK**.

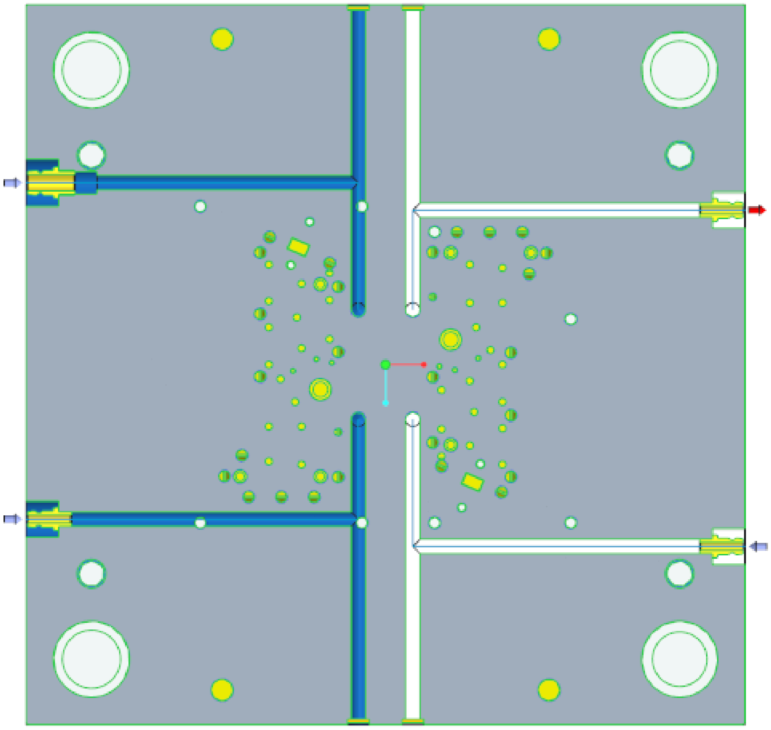
22.5.5 Check the Design Using the Automatically created Cross-Sections

Within the mold base assembly a set of Cross-Section is created

1. Activate the `WD_—55` cross section.

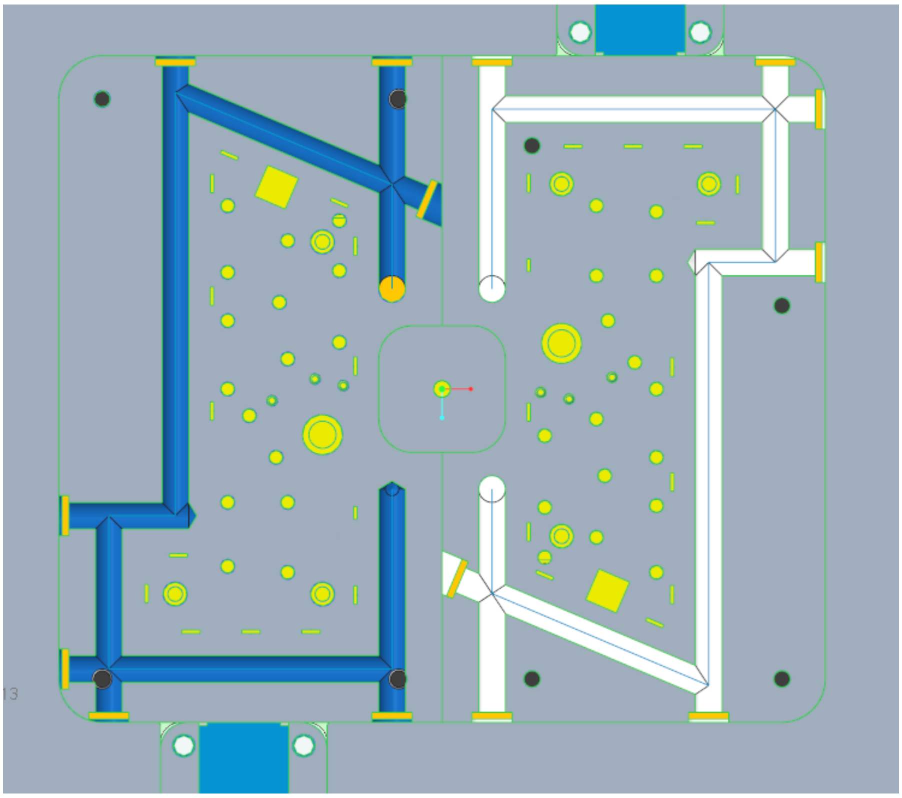
   The cooling channels still interfere with screw bore.

   The incoming cooling nipple is quite close to the clamping screw

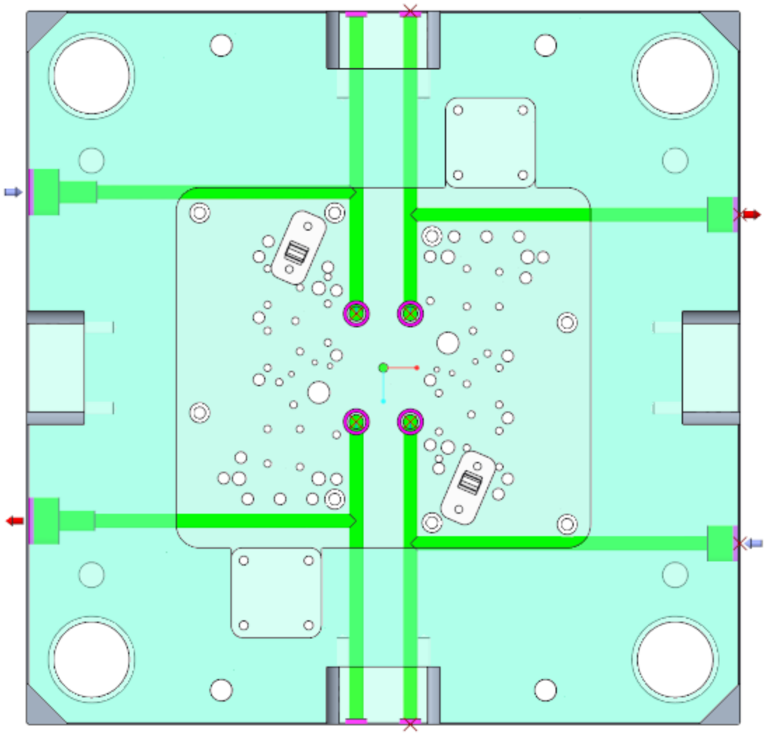


2. Activate the `WD_—13` cross section.

   The cooling channels still interfere with screw bore.



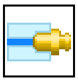
3. Run **Mold Analysis** to check the minimal wall thickness.
4. Modify the screw positions and curve sketches to fix the design issues



22.6 To Assemble Cooling Components On Points

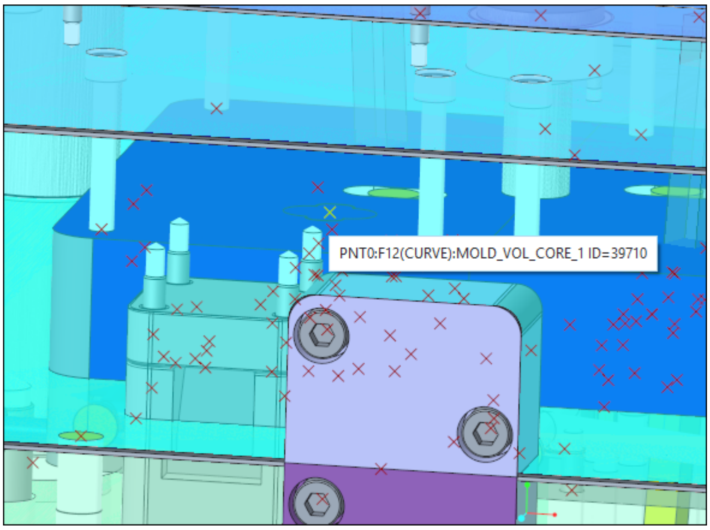
Download 49 To Assemble Cooling Components On Points to start from with this chapter.

22.6.1 Assemble Cooling Nozzle Cavity Core

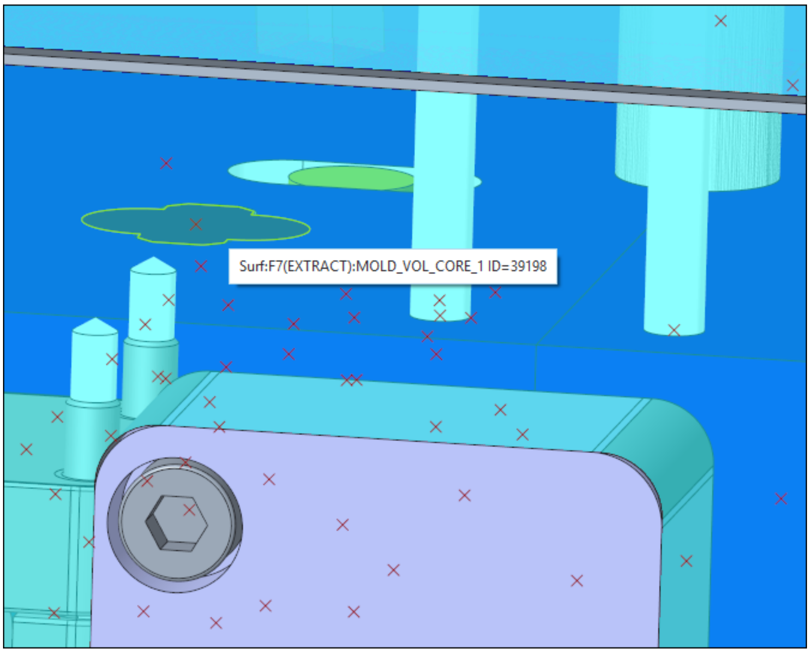
1. Open the `TUTORIAL.ASM`.
2. Select **EMX Components ▶ Cooling Component ▶**  **Cooling Component**.

   The **Cooling Component** dialog box opens.
3. Select `TUTOR_FH` from the **Subassembly** pull-down menu.
4. Select `meusburger` from the **Supplier** pull-down menu.
5. Select `E2110` from the **Type** pull-down menu.
6. Select `2.4` mm for the **DM2** diameter value.
7. Set `90` ° for the **ROT_ANGLE** value.
8. Define the references.
   a. Click **(1) Curve|Axis|Point**

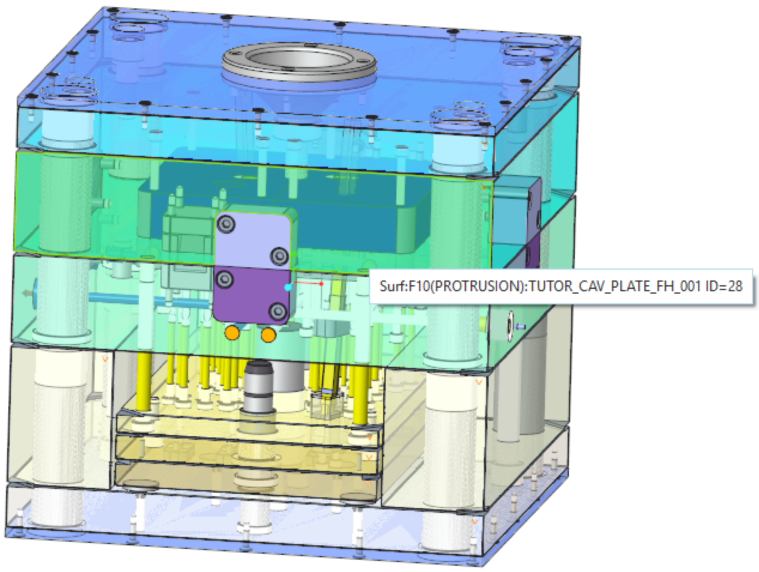
      Select the point from the sketch `COOL_NOZZLE` in the `MOLD_VOL_CORE_1.PRT`

   b. Click **(2) Surface** and select the top surface of the `MOLD_VOL_CORE_1.PRT`

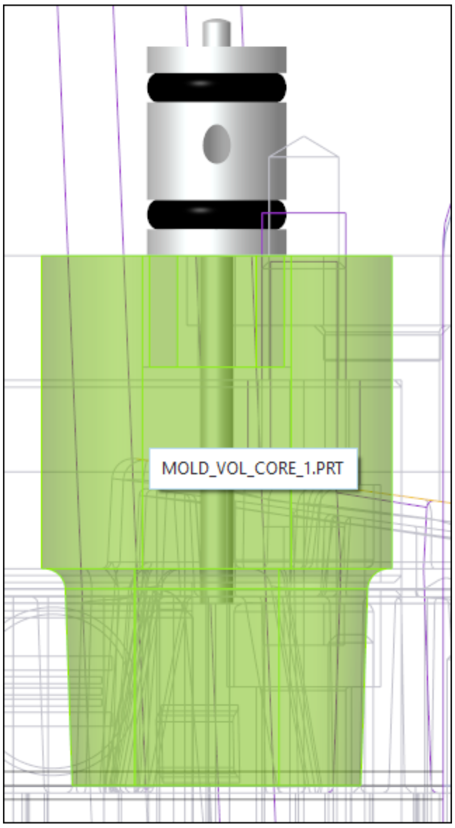


c. Click **(3) Orientation Surface** and select the side surface of the `TUTOR_CAV_PLATE_FH_001.PRT`



.

9. Set the **LG2** value to 25 mm
10. Click **Preview** to display the component



11. In the UDF Dimension set the **T5** value to `30` mm.
12. Click **OK** to assemble the cooling nozzle.

    The component is assembled to mold base and the cutouts are create in the cavity plate and the core model.
13. Select the `TUTOR_E2210_10_2_4.PRT` model either in the modeltree or the graphics window.
14. Click **Assemble again** to reassemble the exact same component, but with different assembly constraints.
15. Define the references.
    a. Click **(1) Curve|Axis|Point**

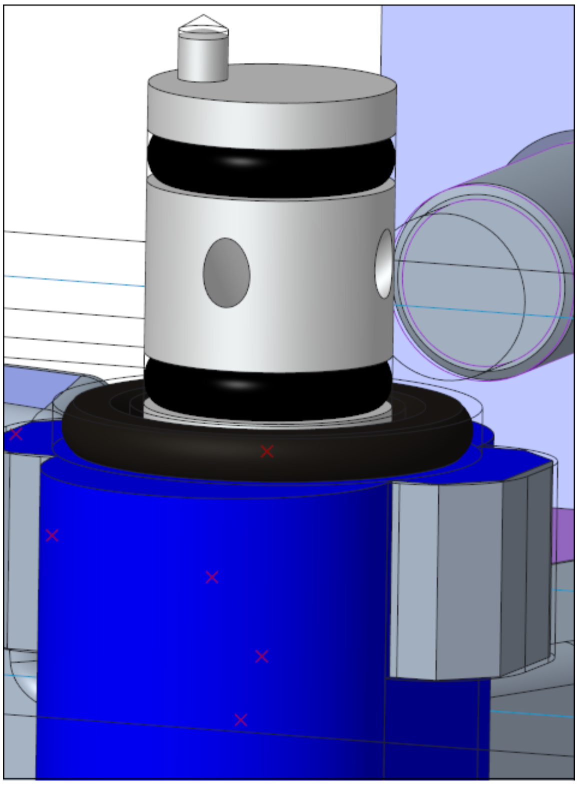
       Select the point from the sketch `COOL_NOZZLE` in the `MOLD_VOL_CORE_1.PRT`
    b. Click **(2) Surface** and select the top surface of the `MOLD_VOL_CORE_1.PRT` in the second `MFG_2.ASM`
    c. Click **(3) Orientation Surface** and select the frontsurface of the `TUTOR_CAV_PLATE_FH_001.PRT`
16. Click **Preview** to display the component.

    NOTE:
    Make sure the **ROT_ANGLE** is set correct that the flow in and flow out of the nozzle fits you cooling circuit.
17. Click **OK** to assemble the cooling nozzle.

The second nozzle is assembled to mold base and the cutouts are create in the cavity plate and the core model.

22.6.2 Assemble O-Rings for the Cooling Nozzles

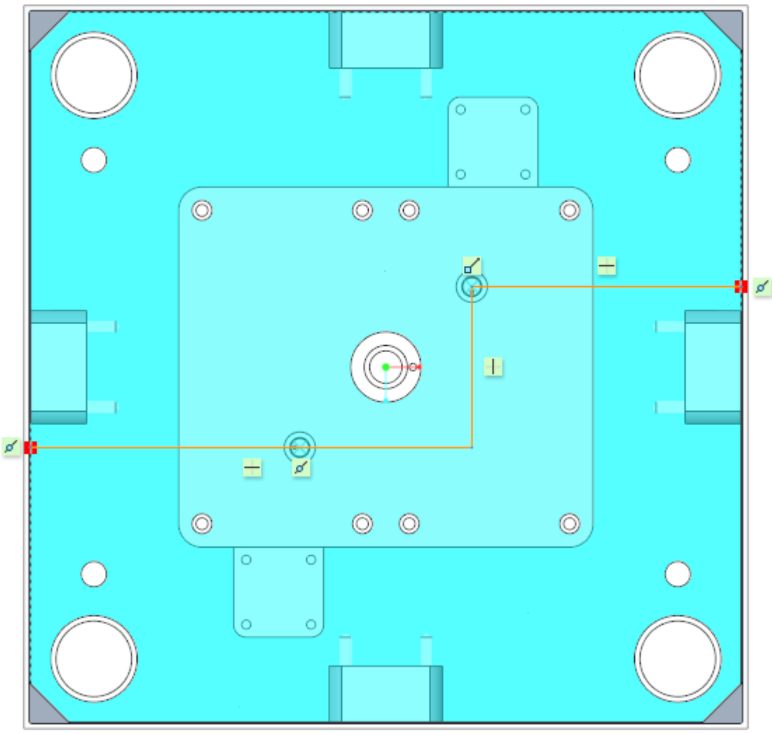
1. Assemble `Meusburger` O-Rings `E2130/11,8x2,4` around the nozzles.
2. Make sure the option **Without Core Bore** is active.



22.6.3 Create Sketch for Nozzle Cooling Circuit

1. Create a sketch within the `TUTOR_CAV_PLATE_FH_001.PRT` similar to the image below.

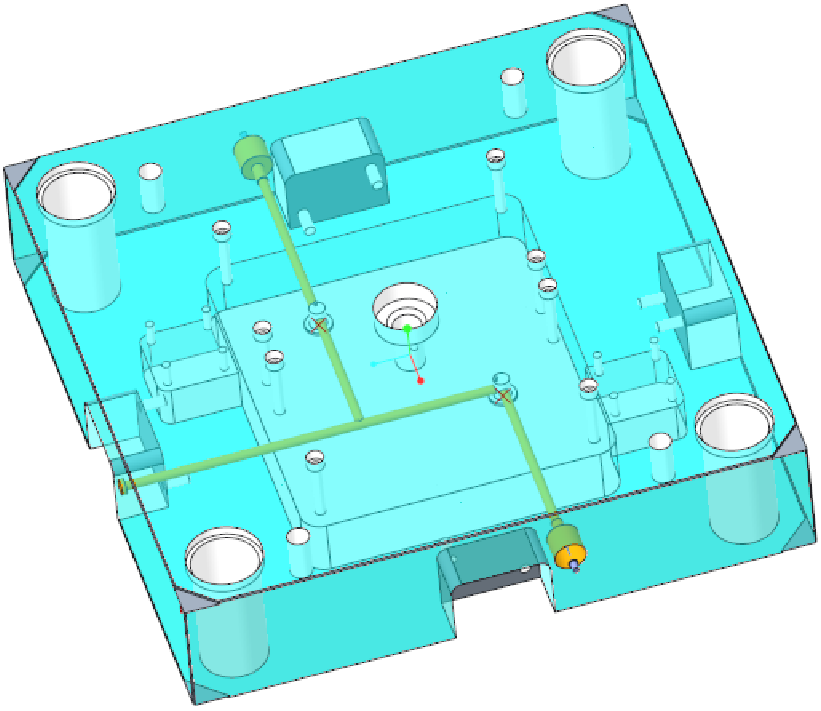
   Make sure to refer to the cooling component placement points, e.g. by using a **Copy Geometry**
   feature.



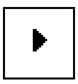
2. Create a second cooling circuit using the **Waterline Designer.**
3. Make sure to activate the second cooling circuit in the **Cooling Circuits** table.
4. Double-Click on the first column in the **#2** row.
5. Set the cooling circuit diameter to 6 mm.

	#		⌀
	1		8
✓	2		6
	3		8
	4		8
	5		8

6. Finish the cooling circuit by adding components on curves, defining correct type and assembling the components



22.7 About Extracting a Waterline Model

1. Click **EMX Components ► Cooling Component ►**  **Show waterline**. The **Show Waterline** dialog box opens.
2. In the **Waterline Model Name** box, enter the waterline model name.
3. Select the **Include Components** check box to copy all cooling component surfaces.
4. Select the **Include Reference Model** check box to add the reference model.
5. Select the **Include PTC Waterline Features** check box to include Creo Parametric waterline feature surfaces.
6. Select the **Create solids** check box to automatically create cylindrical solid models for each waterline.
7. Select the **Create MOLDFLOW curves** check box to automatically create curve features for each waterline.
8. Select the **Create a circuit for each part** check box to create an individual waterline model for each plate of the mold base
9. Click  to begin the analysis. If the waterline model does not exist in the assembly, a new model is created and assembled using the `configuration/templates/`

`waterline_template` template. All waterline features are copied to this model. If a waterline analysis model exists, the included surface copies are updated.

10. Click **Close** to close the dialog box.

NOTE:
The Waterline Model can either be a part or an assembly. In the **EMX Options** dialog box, set `USE_ASM_WATERLINE_MDL` to YES to create an assembly instead of part.

22.8 To Extract the Waterline Model

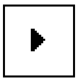
Download 50 To Extract the Waterline Model to start from with this chapter.

22.8.1 Finish the Cooling Circuits

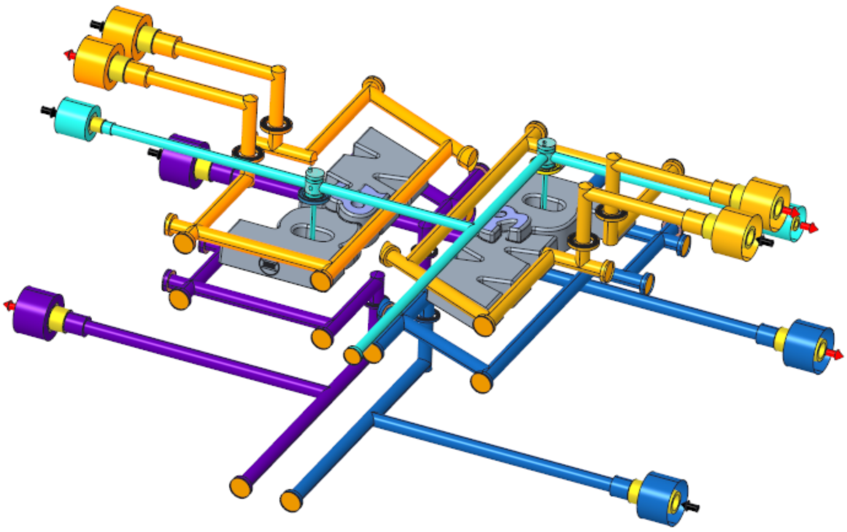
1. Before running the waterline analysis, finish the cooling circuits in the moving and the fixed half.

22.8.2 Run the Analysis with a Target Part

1. Click **EMX Components** ► **Cooling Component** ►  **Show waterline** .

   The **Show Waterline** dialog box opens.
2. Make sure the option **Include Components** is enabled.
3. Make sure the option **Include Reference Model** is enabled.
4. Click  to start the analysis.

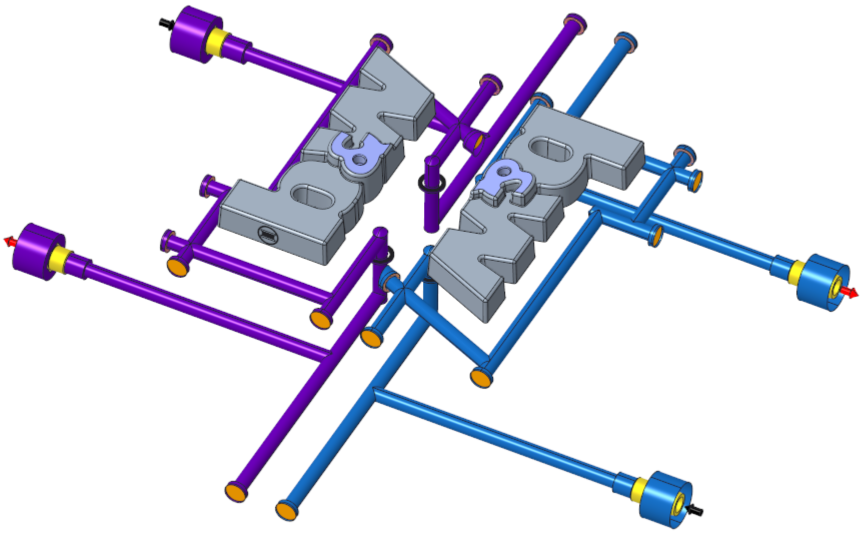
   A new model `TUTOR_WATERLINE_0` will be assembled and the surfaces of the cooling circuits will be copied.
5. Leave the dialog box with **Close**.
6. Open the model `TUTOR_WATERLINE_0.PRT` to check the result.

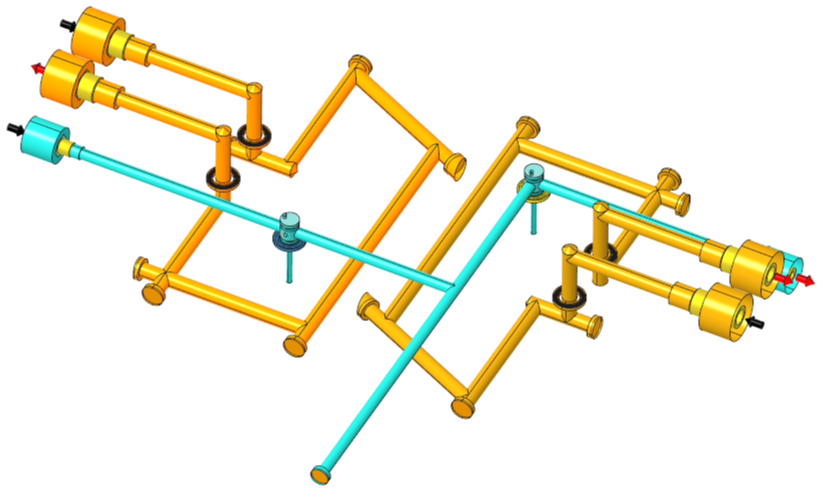


The assembly contains several parts:

- `WATERLINE_ES.PRT`: Contains all the waterline surfaces from the moving half
- `WATERLINE_IS.PRT`: Contains all the waterline surfaces from the fixed half
- A list of the insert reference models
- A list cooling components
- `WATERLINE_ANALSYIS.PRT`:A part that represents the copied waterlines as solids

It is possible to show either all cooling circuits from the fixed half or from the moving by using the predefined representations in the assembly.





7. Switch back to `TUTORIAL.ASM`.

22.9 About Assembling Waterline Curves

Use the **Waterline** dialog box to choose a standard curve and place it as a waterline on the model's points, axes, or curves.

1. Click **EMX Components** ► **Cooling Component** ►  **Assemble a waterline curve**. The **Waterline** dialog box opens.
2. Select **Assemble Model** or **Assemble UDF**.

3. Choose a predefined waterline from the list.
4. Select **Add waterline to the moving and fixed sides** to assemble the waterline to both sides of the mold base.
5. Click  and select the placement coordinate system.
6. Click **OK** to close the dialog box and assemble the model or UDF waterline, or click **Cancel** to exit.

23

Latch Lock Assemblies

23.1 About Latch Locks

You can use a latch lock to allow a non-standard component to move along the mold base opening axis. Click **EMX Components** and then in the **Components** group, click **Latch Lock** to define a new latch lock.

23.2 To Create a Latch Lock

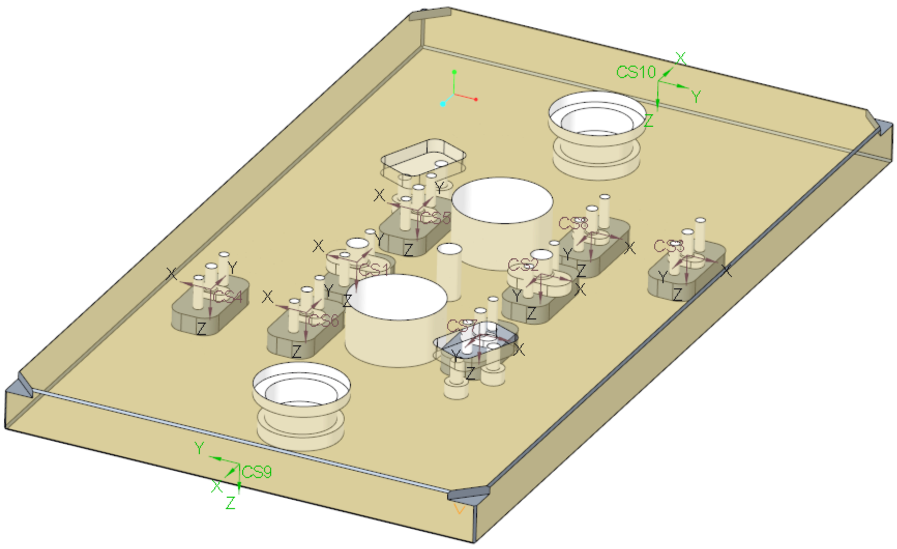
Download 51 To Create a Latch Lock to start from with this chapter.

23.2.1 Create and Pattern Coordinate System in Ejector Fix Plate

1. Open the fixed half assembly `TUTOR_MH.ASM`.
2. Open the ejector plate `TUTOR_EJFIX_PLATE_MH_001.PRT`.
3. Create a coordinate system as seen in the image below.

   Make sure the X direction is normal to the side surface of the plate.

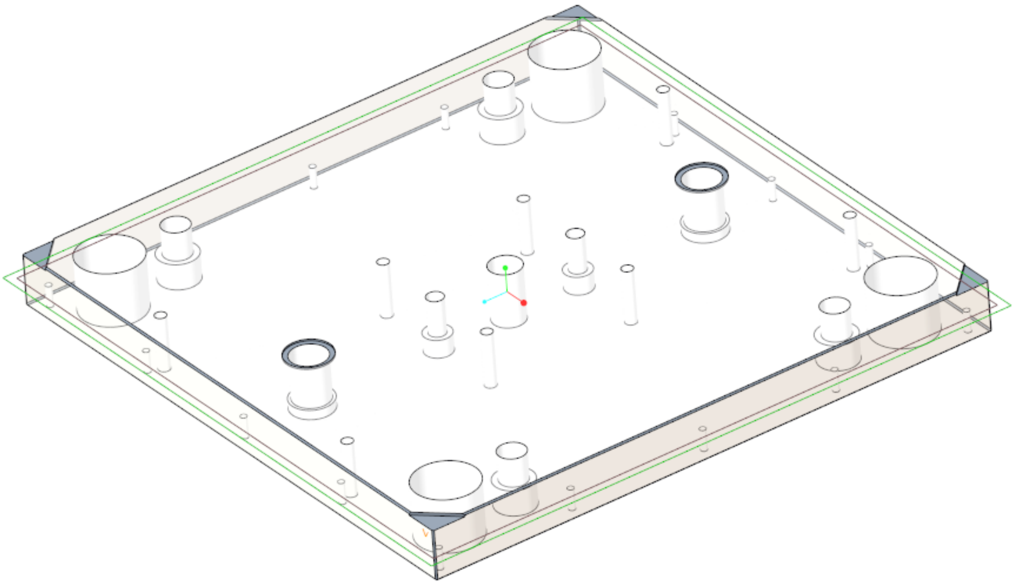
   Make sure Z direction is normal to the bottom surface of the plate.



4. Create an Axis pattern with `2` members and `180°` around the `A_Y` axis of the plate.

23.2.2 Create Positioning Plate for Steady in Clamping Plate

1. Open the ejector plate `TUTOR_EJFIX_PLATE_MH_001.PRT`.
2. Create a datum plane in the middle of the plate as seen in the image below.



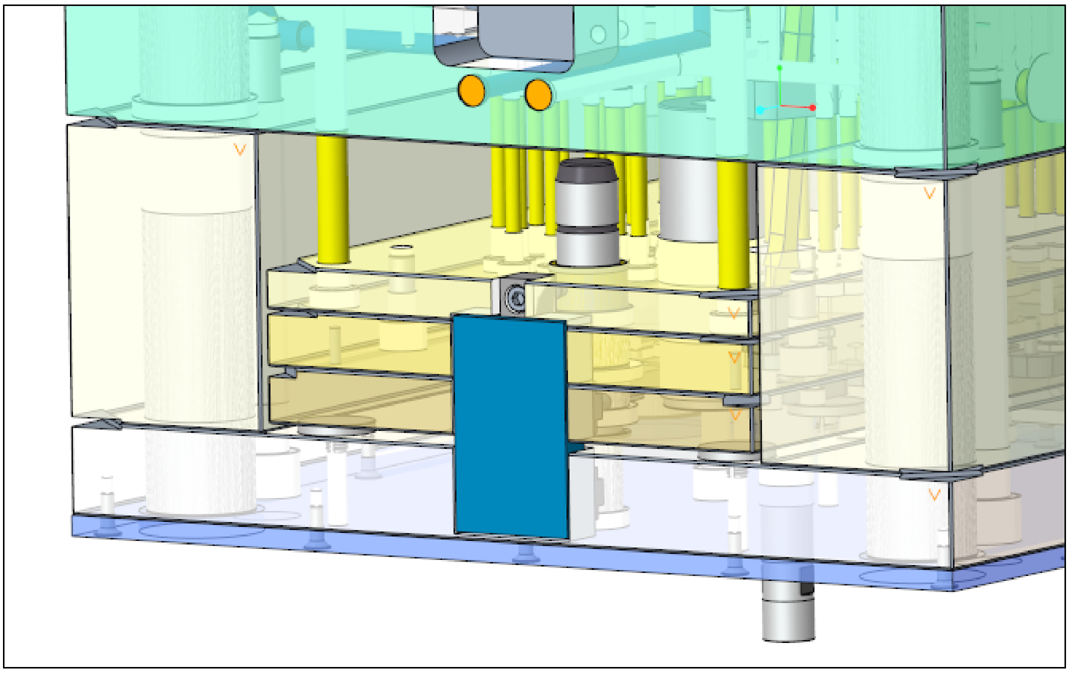
23.2.3 Assemble the latch lock

1. Click **EMX Components ▸ Components ▸** **Latchlock**

   The **Knockout** dialog box appears.

2. Select strack from the **supplier** pull down list.
3. Select Z4 from the **Types** pull down list.
4. As the first assembly reference **(1) CSYS** select the previously created and patterned coordinate system from the ejector plate TUTOR_EJFIX_PLATE_MH_001.PRT model.
5. As the second assembly reference **(4) pos_steady** select the previously created datum plane from the clamping plate TUTOR_CLP_PLATE_MH_001.PRT model.
6. As the third assembly reference **(5) cut_bar** select the top surface of the ejector base plate TUTOR_EJBASE_PLATE_MH_001.PRT model.
7. Set the **TYP** value to 11.
8. Set the **S1S2 — Stroke** value to 15 mm.
9. Set the **OFF_X** value to -22.3 mm.
10. Set the **CUT_BAR** value to 17 mm.
11. Enable the option **Pattern for all instances**.
12. Click **Preview** to verify the assembly constraints
13. Close the **Latchlock** dialog box with **OK**.

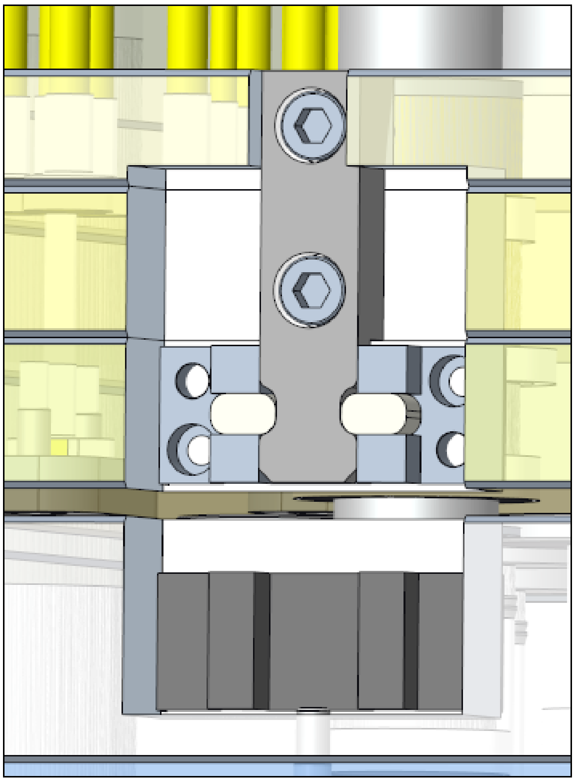
    The latch lock is assembled to the moving half. The latch lock is patterned twice.and. The cut-outs are added to the plates.



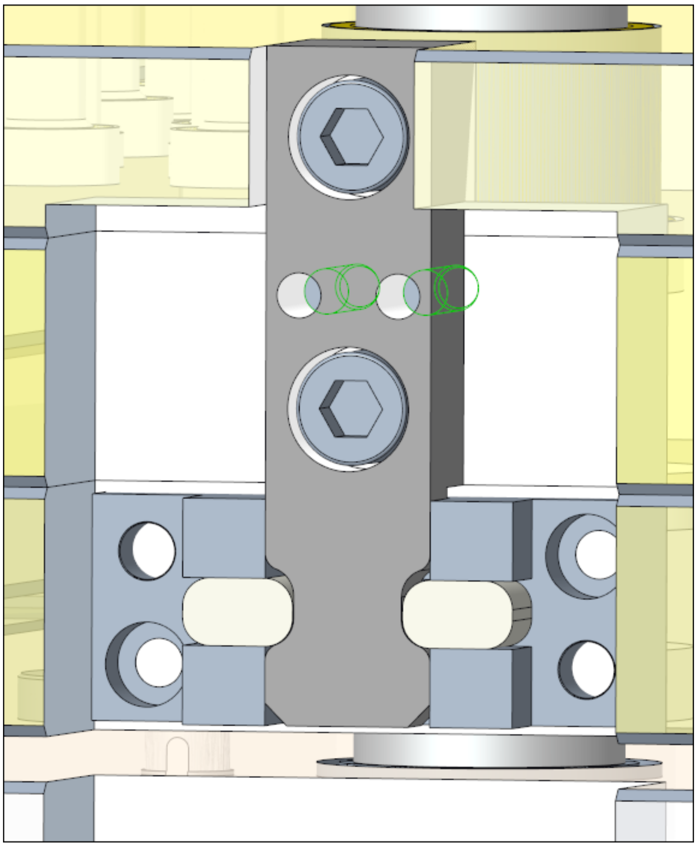
23.2.4 Finish the Latch lock Design

1. Fix the latch lock bar to the ejector base plate and ejector retainer plate using screws.

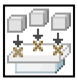
   Work with option **Pattern for all models**to make sure screws and pins are also assembled to the other pattern member.

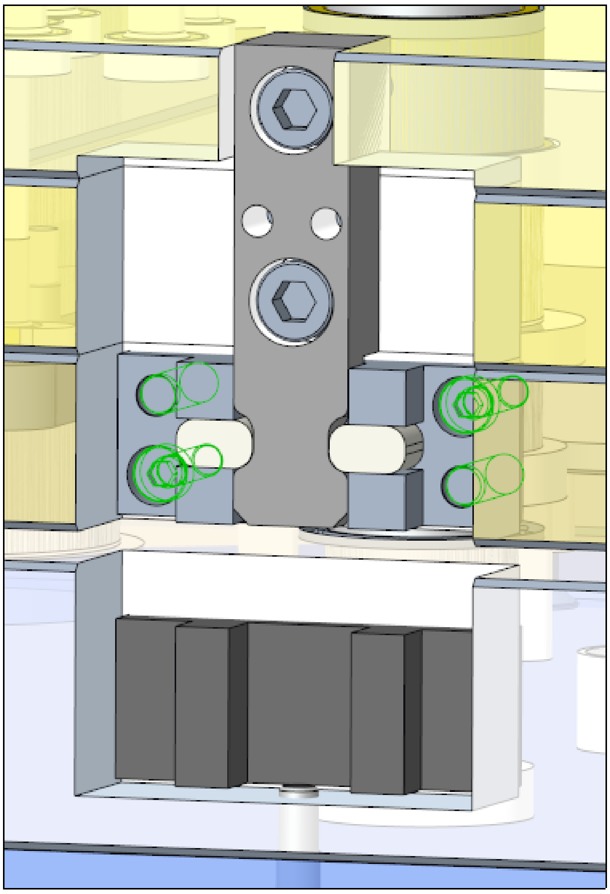
2. Ensure correct position of the latch bar using pins



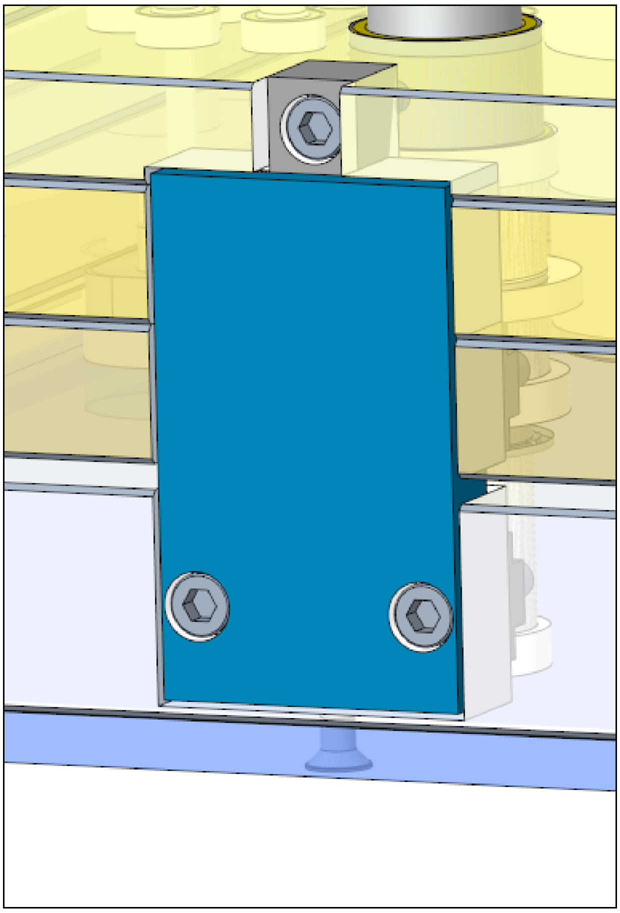
3. For the steady screws and pins are already predefined.

   Use  **Assemble Predefined Components** and select the latch lock assembly `TUTOR_LATCH_ASM_1001.ASM`.

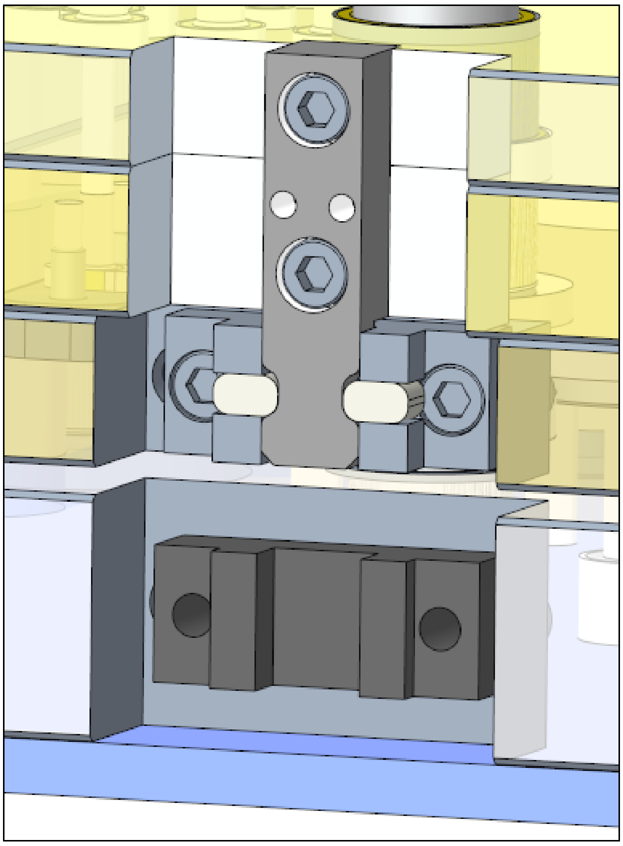
   Repeat this step for the other pattern member



4. Finally the latch lock control plate needs to be fixed to the clamping plate using screws. As placement reference axis from the `TUTOR_LATCH_STDY_1001.PRT` can be used.



5. Finish the cutouts inside the plates to reduce manufacturing costs,



24

Knockouts

To Assemble the Knock Out

24.1 To Assemble the Knock Out

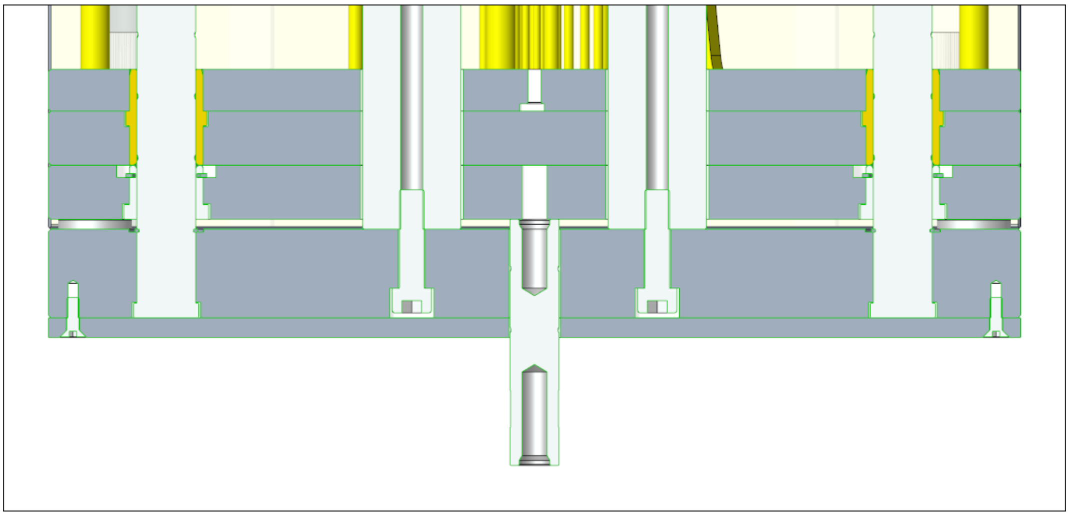
Download 52 To Assemble the Knock Out to start from with this chapter.

24.1.1 Assemble the Knockout

1. Open the fixed half assembly `TUTOR_MH.ASM`.
2. Click **EMX Components ▸ Components ▸ Knockout**

   The **Knockout** dialog box appears.
3. Select `E1050` from the **Types** pull down list.
4. Set the **DM1** value to `20`.
5. As the second assembly reference **(2) Surface** select the bottom surface of the ejector base plate`TUTOR_EJFIX_PLATE_MH_001.PRT` model.
6. Close the **Knockout** dialog box with **OK**.

   The knockout is assembled to the moving half and cut-outs are added to the plates.

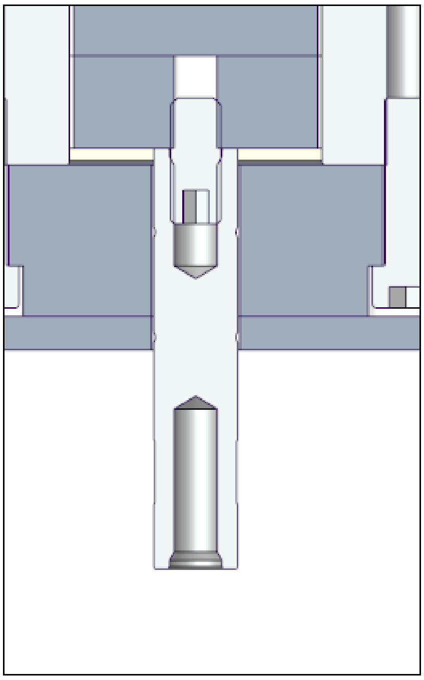


24.1.2 Fix Knockout using a Set Screw

1. Click **EMX Components ▸ Components ▸ Screw**

   The **Knockout** dialog box appears.
2. Select `E1230` from the **Types** pull down list.
3. Set the **DM1** value to `12` mm.
4. Set the **LG** value to `30` mm.
5. Set the **OFFSET** value to `18` mm.
6. As the second assembly reference **(2) Surface** select the bottom surface of the ejector base plate`TUTOR_EJFIX_PLATE_MH_001.PRT` model.
7. Select the option **No Cutout**.
8. Close the **Screw** dialog box with **OK**.

   The set screw is assembled to the moving half.



Notes:

25

Create the Moldbase Documentation

25.1 About the Bill of Materials

The bill of materials is based on part and component parameter definitions.

- Click **EMX Assembly ► Bill of Materials** or click **EMX Drawing Mode ► Bill of Materials**. The **Bill of Materials** dialog box opens. Use it to define part and component parameters before you start working on the design.

- To modify BOM parameters for a single part, click **EMX Assembly ► Edit BOM Parameters**. The **Edit BOM Entry** dialog box opens. Select a component in the current assembly to edit.

NOTE:
You can also define BOM parameters in the **EMX Options** dialog box.

25.2 About Editing BOM Parameters

1. Click **EMX Assembly ► Edit BOM Parameters**. The **Edit BOM Entry** dialog box opens.
2. Click a **Parameter Name** to unlock the parameter value. A locked parameter is indicated by the icon.
3. To edit the value, double-click the value in the **Value** box.
4. To reset a parameter to its default value, click again. The **Default Value** is automatically calculated using the rule defined in the **EMX Options** on page 358 dialog box.
5. To remove the component from the bill of materials, select the **Ignore this component in the BOM** check box.
6. Click **OK** to save changes and close the dialog box, or click **Cancel** to discard changes and close the dialog box.

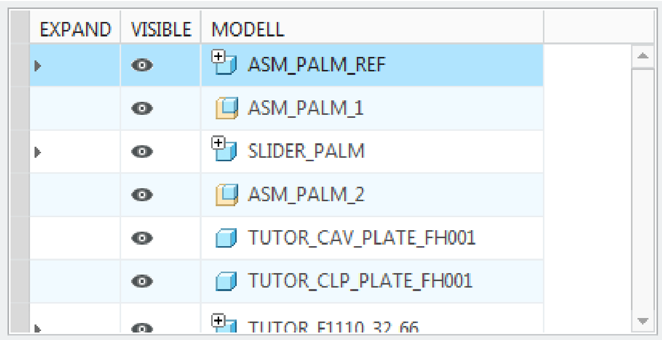
25.3 To Edit the Bill of Materials

25.3.1 The General Bill of Material Structure

The bill of material (BOM) generally displays information about parts and assemblies of the mold base as parameter values.

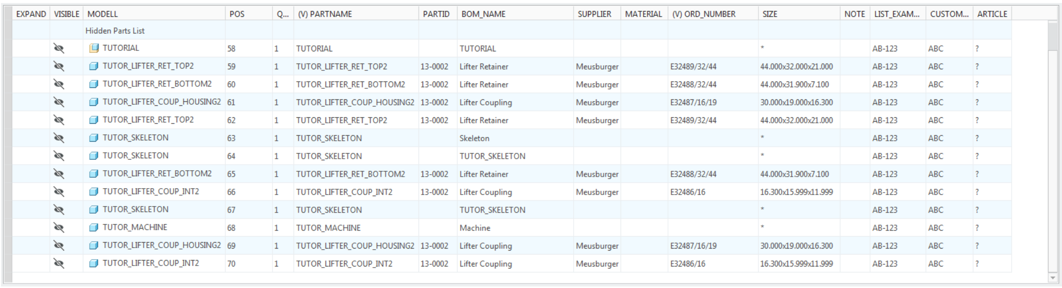
1. Select **EMX General ► Moldbase ► Bill of Material** to open the BOM.

While most of the parameters can be customized, the first 3 columns of the BOM table are not changeable.



EXPAND	VISIBLE	MODELL
		ASM_PALM_REF
		ASM_PALM_1
		SLIDER_PALM
		ASM_PALM_2
		TUTOR_CAV_PLATE_FH001
		TUTOR_CLP_PLATE_FH001
		TUTOR_F1110_32_66

- The **EXPAND** column can be used to expand summarized components.
- By switching the **VISIBLE** entry from to the component will be added to the **Hidden Area** at the bottom of the BOM. This can be useful to exclude parts like workpieces or subassemblies.



EXPAND	VISIBLE	MODELL	POS	Q...	(V) PARTNAME	PARTID	BOM_NAME	SUPPLIER	MATERIAL	(V) ORD_NUMBER	SIZE	NOTE	LIST_EXAM...	CUSTOM...	ARTICLE
		Hidden Parts List													
		TUTORIAL	58	1	TUTORIAL		TUTORIAL				*		AB-123	ABC	?
		TUTOR_LIFTER_RET_TOP2	59	1	TUTOR_LIFTER_RET_TOP2	13-0002	Lifter Retainer	Meusburger		E32489/32/44	44.000x32.000x21.000		AB-123	ABC	?
		TUTOR_LIFTER_RET_BOTTOM2	60	1	TUTOR_LIFTER_RET_BOTTOM2	13-0002	Lifter Retainer	Meusburger		E32488/32/44	44.000x31.900x7.100		AB-123	ABC	?
		TUTOR_LIFTER_COUP_HOUSING2	61	1	TUTOR_LIFTER_COUP_HOUSING2	13-0002	Lifter Coupling	Meusburger		E32487/16/19	30.000x19.000x16.300		AB-123	ABC	?
		TUTOR_LIFTER_RET_TOP2	62	1	TUTOR_LIFTER_RET_TOP2	13-0002	Lifter Retainer	Meusburger		E32489/32/44	44.000x32.000x21.000		AB-123	ABC	?
		TUTOR_SKELETON	63	1	TUTOR_SKELETON		Skeleton				*		AB-123	ABC	?
		TUTOR_SKELETON	64	1	TUTOR_SKELETON		TUTOR_SKELETON				*		AB-123	ABC	?
		TUTOR_LIFTER_RET_BOTTOM2	65	1	TUTOR_LIFTER_RET_BOTTOM2	13-0002	Lifter Retainer	Meusburger		E32488/32/44	44.000x31.900x7.100		AB-123	ABC	?
		TUTOR_LIFTER_COUP_INT2	66	1	TUTOR_LIFTER_COUP_INT2	13-0002	Lifter Coupling	Meusburger		E32486/16	16.300x15.999x11.999		AB-123	ABC	?
		TUTOR_SKELETON	67	1	TUTOR_SKELETON		TUTOR_SKELETON				*		AB-123	ABC	?
		TUTOR_MACHINE	68	1	TUTOR_MACHINE		Machine				*		AB-123	ABC	?
		TUTOR_LIFTER_COUP_HOUSING2	69	1	TUTOR_LIFTER_COUP_HOUSING2	13-0002	Lifter Coupling	Meusburger		E32487/16/19	30.000x19.000x16.300		AB-123	ABC	?
		TUTOR_LIFTER_COUP_INT2	70	1	TUTOR_LIFTER_COUP_INT2	13-0002	Lifter Coupling	Meusburger		E32486/16	16.300x15.999x11.999		AB-123	ABC	?

- In the **MODEL** column the component type and the component name are displayed.

  Parts and assemblies are displayed with different icons and .

  In case a component occurs multiple times in the mold base the parent is highlighted with a tiny .

25.3.2 Customize the Parameters for the BOM

The Bill of Material can be modified to the customers needs. Different parameters can be displayed and handled depending on the configuration.

1. Open the **EMX Options** and switch to the tab **Parameter**.

By default EMX offers a list of parameters as seen in the picture below.

Options | Parameter | Project Parameter | Part Names | Technology | Cooling Bore Data | Ejector Bore Data | Screw Hole Data

Parameter Name	Belongs to	Parameter Type	For...	Designate	Show in Table	Show in GUI	Use to summarize	Default Value
POS	COMP	INTEGER	10	YES	YES	YES	NO	&pos_id
QTY	COMP	INTEGER	10	YES	YES	YES	NO	&qty
PARTNAME	PART	STRING	40	YES	YES	YES	YES	&partname
PARTID	PART	STRING	20	YES	YES	YES	NO	&partid
BOM_NAME	PART	STRING	30	YES	YES	YES	NO	&bomname
SUPPLIER	PART	STRING	30	YES	YES	YES	NO	&supplier
MATERIAL	PART	STRING	30	YES	YES	YES	NO	&material
ORD_NUMBER	PART	STRING	30	YES	YES	YES	YES	&ord_number
SIZE	PART	STRING	20	YES	YES	YES	NO	&size
DRW_SHEET	COMP	STRING	12	YES	NO	YES	NO	&drw_page
REP_SHEET	COMP	STRING	12	YES	NO	YES	NO	&bom_page
NOTE	PART	STRING	20	YES	YES	YES	NO	&remark
SPECIAL	PART	STRING	20	NO	NO	NO	NO	&special
MODELED_BY	PART	STRING	20	NO	NO	YES	NO	&modeled_by
DATE	PART	STRING	10	NO	NO	YES	NO	&date
PROJECTNAME	PART	STRING	30	NO	NO	YES	NO	&projectname
PROJECTPREFIX	PART	STRING	20	NO	NO	NO	NO	&prefix
PROJECTPOSTFIX	PART	STRING	20	NO	NO	NO	NO	&postfix
SIDE	COMP	STRING	10	NO	NO	NO	NO	&side
SIM_GROUP	COMP	STRING	10	NO	NO	NO	NO	&sim_group
LIST_EXAMPLE	PART	STRING	20	YES	YES	YES	NO	AB-123

All parameters which are defined in the **Parameters** tab will be shown in the BOM if their entry **Show In GUI** is set to `YES`.

There are some important parameters and their **Default Values** that are explained here.

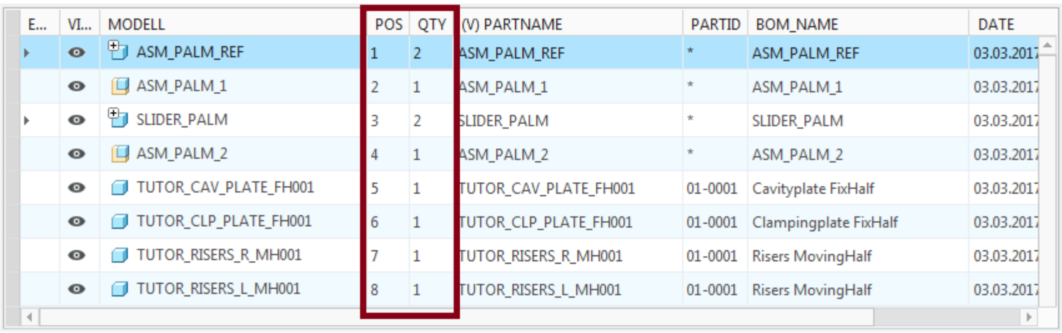
- **POS**: To this parameter the default value **&pos_id** is assigned. This default value describes the position of the component in the BOM list. If this parameter does not exist in the BOM it is not possible to display all components in the right order.

  Do not change this parameter entry manually!

- **QTY:** This parameter represents the quantity of the components and describes how often they are assembled to the mold base. It is important that the **Default Value &qty** is assigned. If this parameter does not exist the summarizing of components which occur multiple times in the mold base does not work.

  Do not change this parameter entry manually!

  NOTE:
  In case no **&qty** and no **&pos_id** parameter exists, the BOM can not work correctly. The parameter names can be different!



E...	VI...	MODELL	POS	QTY	(V) PARTNAME	PARTID	BOM_NAME	DATE
		ASM_PALM_REF	1	2	ASM_PALM_REF	*	ASM_PALM_REF	03.03.2017
		ASM_PALM_1	2	1	ASM_PALM_1	*	ASM_PALM_1	03.03.2017
		SLIDER_PALM	3	2	SLIDER_PALM	*	SLIDER_PALM	03.03.2017
		ASM_PALM_2	4	1	ASM_PALM_2	*	ASM_PALM_2	03.03.2017
		TUTOR_CAV_PLATE_FH001	5	1	TUTOR_CAV_PLATE_FH001	01-0001	Cavityplate FixHalf	03.03.2017
		TUTOR_CLP_PLATE_FH001	6	1	TUTOR_CLP_PLATE_FH001	01-0001	Clampingplate FixHalf	03.03.2017
		TUTOR_RISERS_R_MH001	7	1	TUTOR_RISERS_R_MH001	01-0001	Risers MovingHalf	03.03.2017
		TUTOR_RISERS_L_MH001	8	1	TUTOR_RISERS_L_MH001	01-0001	Risers MovingHalf	03.03.2017

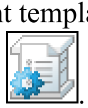
- **MATERIAL:** EMX does not only provide the material as a STRING-Parameter, it will also set a Creo Material to every model that is assembled by EMX. It is important that the **Default Value &material** is assigned.

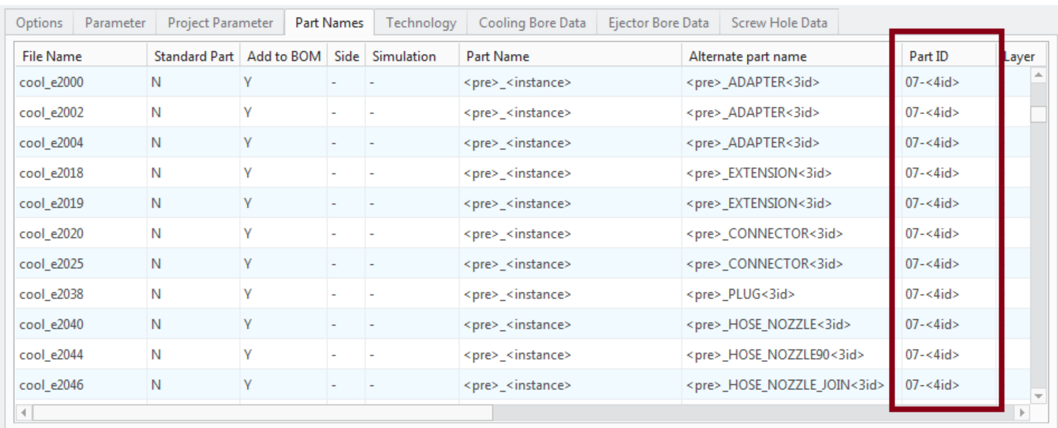
  NOTE:
  Make sure the required material files are located in the `MATERIAL_PATH` (in the **EMX Options**) with the correct `MATFILE_EXTENSION`.

For any other parameter customization take the default parameter settings as a reference. There are several other default value that can be assigned to parameters.

NOTE:
When you need to show the describing BOM_NAME in more than one languages (i.e. you share design on several countries) EMX allow usage of up to 5 additional BOM_NAMES from other languages. Set the **EMX Option** `ADD_LOCALIZED_BOM_NAME1` to a different language, i.e. `german`. When closing the **EMX Option** dialog box, EMX will read the partname-entries from the `<install_emx>/text/german` folder and adds a new column in partnames with German entries. Now additional Parameters with the rule `&bomname_loc1`, `&bomname_loc2` etc. can be added in **Parameter** configuration.

Following default values for EMX Parameters are available.

- **&partname**: Use this parameter default value to assign the name of the Creo part or assembly to the parameter.
- **&partid**: This default value assigns an unique ID string to the parameter. This ID string can be customized individually. For each component template a rule for the part id is defined in the **Part Names** tab of the **EMX Options** .

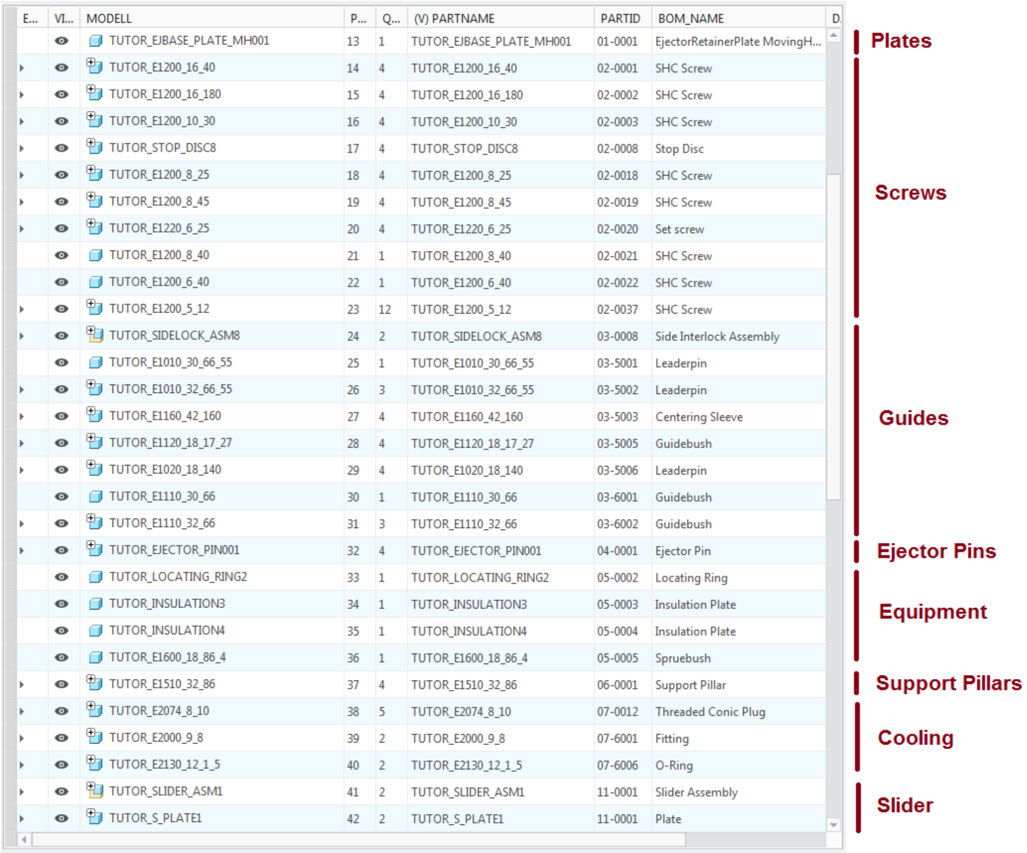


Options | Parameter | Project Parameter | Part Names | Technology | Cooling Bore Data | Ejector Bore Data | Screw Hole Data

File Name	Standard Part	Add to BOM	Side	Simulation	Part Name	Alternate part name	Part ID	Layer
cool_e2000	N	Y	-	-	<pre>_<instance>	<pre>_ADAPTER<3id>	07-<4id>	
cool_e2002	N	Y	-	-	<pre>_<instance>	<pre>_ADAPTER<3id>	07-<4id>	
cool_e2004	N	Y	-	-	<pre>_<instance>	<pre>_ADAPTER<3id>	07-<4id>	
cool_e2018	N	Y	-	-	<pre>_<instance>	<pre>_EXTENSION<3id>	07-<4id>	
cool_e2019	N	Y	-	-	<pre>_<instance>	<pre>_EXTENSION<3id>	07-<4id>	
cool_e2020	N	Y	-	-	<pre>_<instance>	<pre>_CONNECTOR<3id>	07-<4id>	
cool_e2025	N	Y	-	-	<pre>_<instance>	<pre>_CONNECTOR<3id>	07-<4id>	
cool_e2038	N	Y	-	-	<pre>_<instance>	<pre>_PLUG<3id>	07-<4id>	
cool_e2040	N	Y	-	-	<pre>_<instance>	<pre>_HOSE_NOZZLE<3id>	07-<4id>	
cool_e2044	N	Y	-	-	<pre>_<instance>	<pre>_HOSE_NOZZLE90<3id>	07-<4id>	
cool_e2046	N	Y	-	-	<pre>_<instance>	<pre>_HOSE_NOZZLE_JOIN<3id>	07-<4id>	

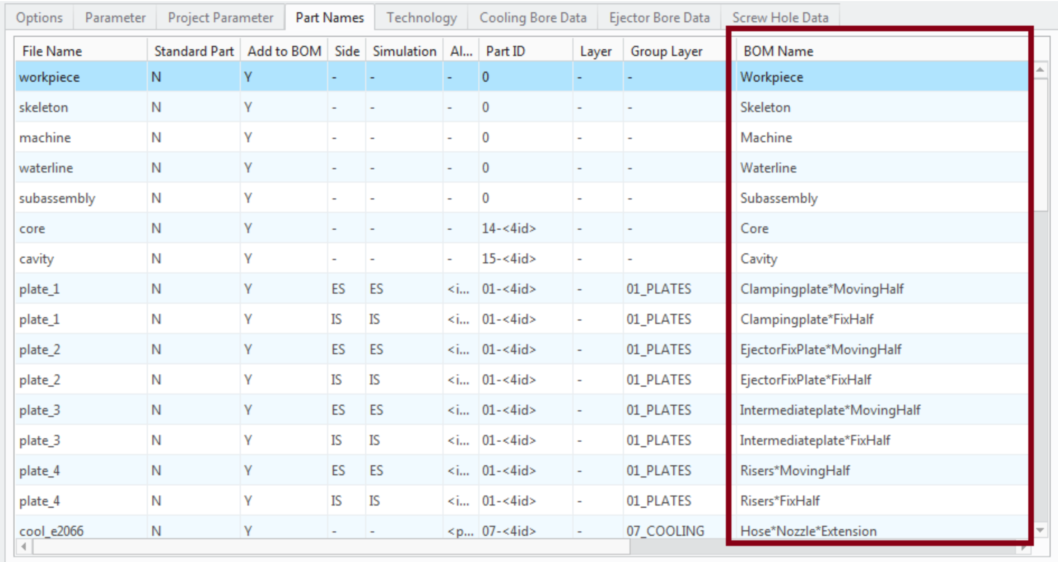
In the default configuration the part name ID has 7 digits. Beginning with a 2 digits number that represents the EMX component type (e.g. 07 stands for cooling components). The last 4 digits are a simple integer variable which increments with any newly assembled component of this type.

If the **EMX Option** `SORT_BOM_WITH_PARTID` is set to `YES`. EMX will use this part ID string to sort the BOM. This can be very handy because all components are then listed by type. Plates with 01, Screws with 02, Guides with 03, Ejector Pins with 04, Equipment with 05, Support Pillars with 06, Cooling Components with 07, Slider with 11 etc.



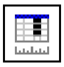
E...	VI...	MODELL	P...	Q...	(V) PARTNAME	PARTID	BOM_NAME	D
		TUTOR_EJBASE_PLATE_MH001	13	1	TUTOR_EJBASE_PLATE_MH001	01-0001	EjectorRetainerPlate MovingH...	
		TUTOR_E1200_16_40	14	4	TUTOR_E1200_16_40	02-0001	SHC Screw	
		TUTOR_E1200_16_180	15	4	TUTOR_E1200_16_180	02-0002	SHC Screw	
		TUTOR_E1200_10_30	16	4	TUTOR_E1200_10_30	02-0003	SHC Screw	
		TUTOR_STOP_DISC8	17	4	TUTOR_STOP_DISC8	02-0008	Stop Disc	
		TUTOR_E1200_8_25	18	4	TUTOR_E1200_8_25	02-0018	SHC Screw	
		TUTOR_E1200_8_45	19	4	TUTOR_E1200_8_45	02-0019	SHC Screw	
		TUTOR_E1220_6_25	20	4	TUTOR_E1220_6_25	02-0020	Set screw	
		TUTOR_E1200_8_40	21	1	TUTOR_E1200_8_40	02-0021	SHC Screw	
		TUTOR_E1200_6_40	22	1	TUTOR_E1200_6_40	02-0022	SHC Screw	
		TUTOR_E1200_5_12	23	12	TUTOR_E1200_5_12	02-0037	SHC Screw	
		TUTOR_SIDELOCK_ASM8	24	2	TUTOR_SIDELOCK_ASM8	03-0008	Side Interlock Assembly	
		TUTOR_E1010_30_66_55	25	1	TUTOR_E1010_30_66_55	03-5001	Leaderpin	
		TUTOR_E1010_32_66_55	26	3	TUTOR_E1010_32_66_55	03-5002	Leaderpin	
		TUTOR_E1160_42_160	27	4	TUTOR_E1160_42_160	03-5003	Centering Sleeve	
		TUTOR_E1120_18_17_27	28	4	TUTOR_E1120_18_17_27	03-5005	Guidebush	
		TUTOR_E1020_18_140	29	4	TUTOR_E1020_18_140	03-5006	Leaderpin	
		TUTOR_E1110_30_66	30	1	TUTOR_E1110_30_66	03-6001	Guidebush	
		TUTOR_E1110_32_66	31	3	TUTOR_E1110_32_66	03-6002	Guidebush	
		TUTOR_EJECTOR_PIN001	32	4	TUTOR_EJECTOR_PIN001	04-0001	Ejector Pin	
		TUTOR_LOCATING_RING2	33	1	TUTOR_LOCATING_RING2	05-0002	Locating Ring	
		TUTOR_INSULATION3	34	1	TUTOR_INSULATION3	05-0003	Insulation Plate	
		TUTOR_INSULATION4	35	1	TUTOR_INSULATION4	05-0004	Insulation Plate	
		TUTOR_E1600_18_86_4	36	1	TUTOR_E1600_18_86_4	05-0005	Spruebush	
		TUTOR_E1510_32_86	37	4	TUTOR_E1510_32_86	06-0001	Support Pillar	
		TUTOR_E2074_8_10	38	5	TUTOR_E2074_8_10	07-0012	Threaded Conic Plug	
		TUTOR_E2000_9_8	39	2	TUTOR_E2000_9_8	07-6001	Fitting	
		TUTOR_E2130_12_1_5	40	2	TUTOR_E2130_12_1_5	07-6006	O-Ring	
		TUTOR_SLIDER_ASM1	41	2	TUTOR_SLIDER_ASM1	11-0001	Slider Assembly	
		TUTOR_S_PLATE1	42	2	TUTOR_S_PLATE1	11-0001	Plate	

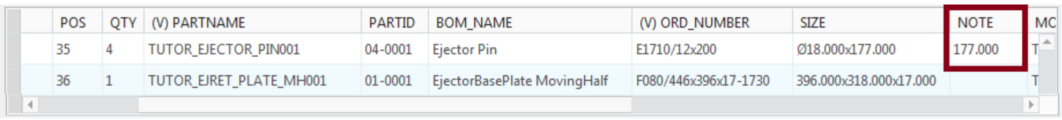
- **&bomname**: This default value assigns a descriptions which can also be customized in the **Part Names** tab of the **EMX Options**  to the parameter.



Options | Parameter | Project Parameter | Part Names | Technology | Cooling Bore Data | Ejector Bore Data | Screw Hole Data

File Name	Standard Part	Add to BOM	Side	Simulation	Al...	Part ID	Layer	Group Layer	BOM Name
workpiece	N	Y	-	-	-	0	-	-	Workpiece
skeleton	N	Y	-	-	-	0	-	-	Skeleton
machine	N	Y	-	-	-	0	-	-	Machine
waterline	N	Y	-	-	-	0	-	-	Waterline
subassembly	N	Y	-	-	-	0	-	-	Subassembly
core	N	Y	-	-	-	14-<4id>	-	-	Core
cavity	N	Y	-	-	-	15-<4id>	-	-	Cavity
plate_1	N	Y	ES	ES	<i...	01-<4id>	-	01_PLATES	Clampingplate*MovingHalf
plate_1	N	Y	IS	IS	<i...	01-<4id>	-	01_PLATES	Clampingplate*FixHalf
plate_2	N	Y	ES	ES	<i...	01-<4id>	-	01_PLATES	EjectorFixPlate*MovingHalf
plate_2	N	Y	IS	IS	<i...	01-<4id>	-	01_PLATES	EjectorFixPlate*FixHalf
plate_3	N	Y	ES	ES	<i...	01-<4id>	-	01_PLATES	Intermediateplate*MovingHalf
plate_3	N	Y	IS	IS	<i...	01-<4id>	-	01_PLATES	Intermediateplate*FixHalf
plate_4	N	Y	ES	ES	<i...	01-<4id>	-	01_PLATES	Risers*MovingHalf
plate_4	N	Y	IS	IS	<i...	01-<4id>	-	01_PLATES	Risers*FixHalf
cool_e2066	N	Y	-	-	<p...	07-<4id>	-	07_COOLING	Hose*Nozzle*Extension

- **&supplier**: Important default value, which assigns the supplier of a component to the parameter
- **&ord_number**: Important default value, which assigns the order number of a component to the parameter
- **&size**: EMX comes with an algorithm to calculate the outline of each part in the mold base. Click  to start this calculation. EMX writes all outline values to the size parameter.
- **&remark**: EMX uses this default value to write notes into the BOM. For ejector pins and baffle cooling components the trimmed length is written to this parameter. In this example the ordered length of the ejector pin is 200 mm. The trimmed length is 177 mm.



POS	QTY	(V) PARTNAME	PARTID	BOM_NAME	(V) ORD_NUMBER	SIZE	NOTE	MC
35	4	TUTOR_EJECTOR_PIN001	04-0001	Ejector Pin	E1710/12x200	Ø18.000x177.000	177.000	T
36	1	TUTOR_EJRET_PLATE_MH001	01-0001	EjectorBasePlate MovingHalf	F080/446x396x17-1730	396.000x318.000x17.000		T

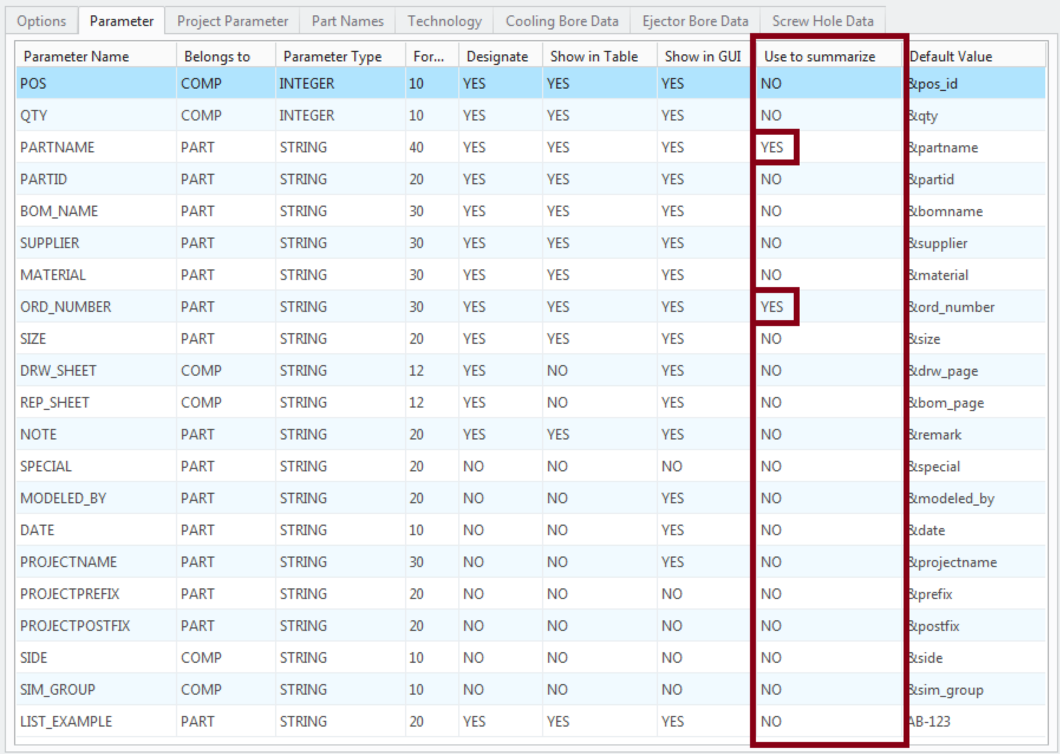
- **&modeled_by**: The designer defined in the **Project** dialog box is written to this parameter.
- **&date**: The date defined in the **Project** dialog box is written to this parameter.
- **&projectname**: The project name defined in the **Project** dialog box is written to this parameter.
- **&prefix**:The project prefix defined in the **Project** dialog box is written to this parameter.
- **&postfix**: The project post fix defined in the **Project** dialog box is written to this parameter.
- **&side**: The side fixed or moving half which the component is assigned to is written to this parameter.
- **&sim_group**: the simulation group for the opening simulation is assigned to this parameter.
- **&ptc_common_name**: The PTC_COMMON_NAME value is assigned to this parameter.

25.3.3 Summarize Equal Components in BOM

Define the criteria to sum up the components in the BOM.

1. Open the **EMX Options** and switch to the tab **Parameter**.

   In the **Parameter** tab the row **Use to summarize** can be found.



Options | Parameter | Project Parameter | Part Names | Technology | Cooling Bore Data | Ejector Bore Data | Screw Hole Data

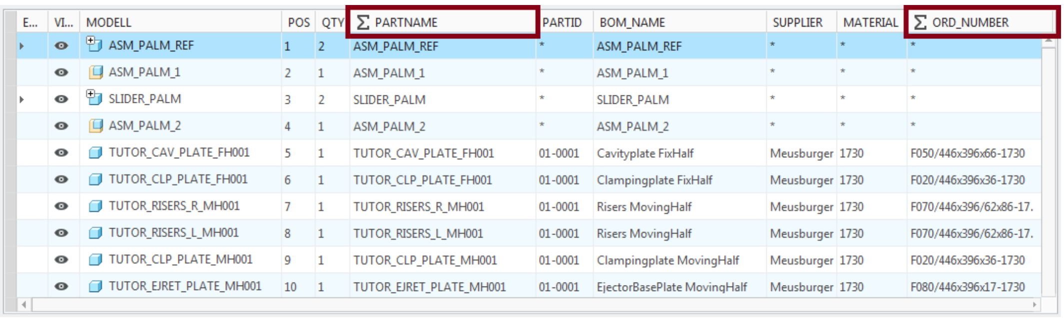
Parameter Name	Belongs to	Parameter Type	For...	Designate	Show in Table	Show in GUI	Use to summarize	Default Value
POS	COMP	INTEGER	10	YES	YES	YES	NO	&pos_id
QTY	COMP	INTEGER	10	YES	YES	YES	NO	&qty
PARTNAME	PART	STRING	40	YES	YES	YES	YES	&partname
PARTID	PART	STRING	20	YES	YES	YES	NO	&partid
BOM_NAME	PART	STRING	30	YES	YES	YES	NO	&bomname
SUPPLIER	PART	STRING	30	YES	YES	YES	NO	&supplier
MATERIAL	PART	STRING	30	YES	YES	YES	NO	&material
ORD_NUMBER	PART	STRING	30	YES	YES	YES	YES	&ord_number
SIZE	PART	STRING	20	YES	YES	YES	NO	&size
DRW_SHEET	COMP	STRING	12	YES	NO	YES	NO	&drw_page
REP_SHEET	COMP	STRING	12	YES	NO	YES	NO	&bom_page
NOTE	PART	STRING	20	YES	YES	YES	NO	&remark
SPECIAL	PART	STRING	20	NO	NO	NO	NO	&special
MODELED_BY	PART	STRING	20	NO	NO	YES	NO	&modeled_by
DATE	PART	STRING	10	NO	NO	YES	NO	&date
PROJECTNAME	PART	STRING	30	NO	NO	YES	NO	&projectname
PROJECTPREFIX	PART	STRING	20	NO	NO	NO	NO	&prefix
PROJECTPOSTFIX	PART	STRING	20	NO	NO	NO	NO	&postfix
SIDE	COMP	STRING	10	NO	NO	NO	NO	&side
SIM_GROUP	COMP	STRING	10	NO	NO	NO	NO	&sim_group
LIST_EXAMPLE	PART	STRING	20	YES	YES	YES	NO	AB-123

By default EMX uses the parameters **PARTNAME** with a default value of **&partname** and the parameter **ORD_NUMBER** with an assigned default value **&ord_number** to summarize the components.

This means: If a component has the same value in **PARTNAME** and in **ORD_NUMBER** EMX will summarize them in just one line of the BOM.

NOTE:

In the **BOM** the **Use to summarize** flag is displayed with a $\boxed{\Sigma}$ in front of the parameter name



E...	VI...	MODELL	POS	QTY	Σ PARTNAME	PARTID	BOM_NAME	SUPPLIER	MATERIAL	Σ ORD_NUMBER
		ASM_PALM_REF	1	2	ASM_PALM_REF	*	ASM_PALM_REF	*	*	*
		ASM_PALM_1	2	1	ASM_PALM_1	*	ASM_PALM_1	*	*	*
		SLIDER_PALM	3	2	SLIDER_PALM	*	SLIDER_PALM	*	*	*
		ASM_PALM_2	4	1	ASM_PALM_2	*	ASM_PALM_2	*	*	*
		TUTOR_CAV_PLATE_FH001	5	1	TUTOR_CAV_PLATE_FH001	01-0001	Cavityplate FixHalf	Meusburger	1730	F050/446x396x66-1730
		TUTOR_CLP_PLATE_FH001	6	1	TUTOR_CLP_PLATE_FH001	01-0001	Clampingplate FixHalf	Meusburger	1730	F020/446x396x36-1730
		TUTOR_RISERS_R_MH001	7	1	TUTOR_RISERS_R_MH001	01-0001	Risers MovingHalf	Meusburger	1730	F070/446x396/62x86-17.
		TUTOR_RISERS_L_MH001	8	1	TUTOR_RISERS_L_MH001	01-0001	Risers MovingHalf	Meusburger	1730	F070/446x396/62x86-17.
		TUTOR_CLP_PLATE_MH001	9	1	TUTOR_CLP_PLATE_MH001	01-0001	Clampingplate MovingHalf	Meusburger	1730	F020/446x396x36-1730
		TUTOR_EJRET_PLATE_MH001	10	1	TUTOR_EJRET_PLATE_MH001	01-0001	EjectorBasePlate MovingHalf	Meusburger	1730	F080/446x396x17-1730

25.3.4 Edit Parameters individually

1. Double-Click in the row of a summarized component. This will expand all single entries.

E...	VI...	MODELL	P...	Q...	∑ PARTNAME	PARTID	BOM_NAME	SUPPLIER	MATERIAL	∑ ORD_NUMBER
		TUTOR_E1200_16_180	15	4						
		TUTOR_E1200_16_180	15	1	TUTOR_E1200_16_180	02-0002	SHC Screw	Meusburger	12.9	E1200/16x180
		TUTOR_E1200_16_180	15	1	TUTOR_E1200_16_180	02-0002	SHC Screw	Meusburger	12.9	E1200/16x180
		TUTOR_E1200_16_180	15	1	TUTOR_E1200_16_180	02-0002	SHC Screw	Meusburger	12.9	E1200/16x180
		TUTOR_E1200_16_180	15	1	TUTOR_E1200_16_180	02-0002	SHC Screw	Meusburger	12.9	E1200/16x180

Parameters can be edited manually by Double-Click the certain entry in the BOM.

2. Edit the Parameter **ORD_NUMBER** of one of the components manually.

3. Click Refresh .Remember that the parameter **ORD_NUMBER** was used to summarize the components.

   Even if the **PARTNAME** parameter has still the same value, the **ORD_NUMBER** value is now different. As a result EMX will not summarize the component any more in the BOM.

E...	VI...	MODELL	P...	Q...	∑ PARTNAME	PARTID	BOM_NAME	SUPPLIER	MATERIAL	∑ ORD_NUMBER
		TUTOR_E1200_16_180	15	1	TUTOR_E1200_16_180	02-0002	SHC Screw	Meusburger	12.9	E1200/16x180_XYZ*
		TUTOR_E1200_16_180	16	3						
		TUTOR_E1200_16_180	16	1	TUTOR_E1200_16_180	02-0002	SHC Screw	Meusburger	12.9	E1200/16x180
		TUTOR_E1200_16_180	16	1	TUTOR_E1200_16_180	02-0002	SHC Screw	Meusburger	12.9	E1200/16x180
		TUTOR_E1200_16_180	16	1	TUTOR_E1200_16_180	02-0002	SHC Screw	Meusburger	12.9	E1200/16x180

NOTE:
A manually edited parameter is displayed with the *.

It is possible to edit all entries of the summarized components at once. Simply Double-Click the desired entry of the component row while it is collapsed.

25.3.5 Do not Show Parameters in BOM

1. Open the **EMX Options** and switch to the tab **Parameter**.

   The flag **Show in table** can be set for each parameter individually. If the flag is set to **NO** the parameter will not be displayed in the BOM.

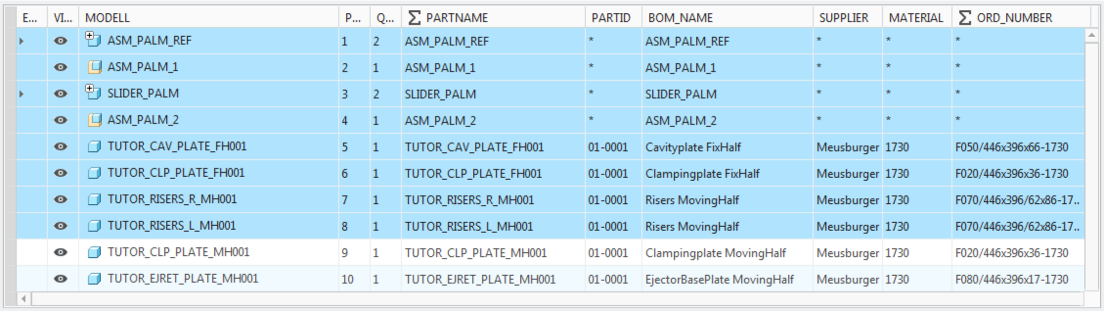
Options | Parameter | Project Parameter | Part Names | Technology | Cooling Bore Data | Ejector Bore Data | Screw Hole Data

Parameter Name	Belongs to	Parameter Type	Format	Designate	Show in Table	Show in GUI	Use to summarize	Default Value
POS	COMP	INTEGER	10	YES	YES	YES	NO	&pos_id
QTY	COMP	INTEGER	10	YES	YES	YES	NO	&qty
PARTNAME	PART	STRING	40	YES	YES	YES	YES	&partname
PARTID	PART	STRING	20	YES	YES	YES	NO	&partid
BOM_NAME	PART	STRING	30	YES	YES	YES	NO	&bomname
SUPPLIER	PART	STRING	30	YES	YES	YES	NO	&supplier
MATERIAL	PART	STRING	30	YES	YES	YES	NO	&material

25.3.6 Working with the BOM

After finishing the mold base the BOM can be edited by the designer via a variety of tools.

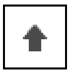
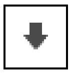
It is possible to select multiple components with **Ctrl** or **Shift**

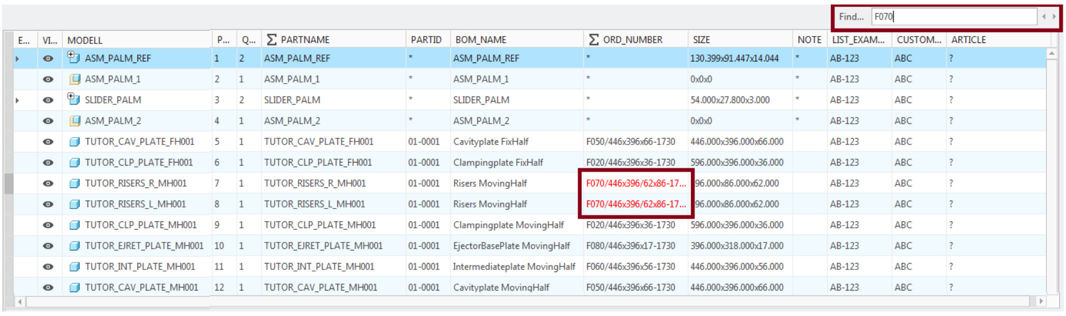


E...	VI...	MODELL	P...	Q...	Σ PARTNAME	PARTID	BOM_NAME	SUPPLIER	MATERIAL	Σ ORD_NUMBER
		ASM_PALM_REF	1	2	ASM_PALM_REF	*	ASM_PALM_REF	*	*	*
		ASM_PALM_1	2	1	ASM_PALM_1	*	ASM_PALM_1	*	*	*
		SLIDER_PALM	3	2	SLIDER_PALM	*	SLIDER_PALM	*	*	*
		ASM_PALM_2	4	1	ASM_PALM_2	*	ASM_PALM_2	*	*	*
		TUTOR_CAV_PLATE_FH001	5	1	TUTOR_CAV_PLATE_FH001	01-0001	Cavityplate FixHalf	Meusburger	1730	F050/446x396x66-1730
		TUTOR_CLP_PLATE_FH001	6	1	TUTOR_CLP_PLATE_FH001	01-0001	Clampingplate FixHalf	Meusburger	1730	F020/446x396x36-1730
		TUTOR_RISERS_R_MH001	7	1	TUTOR_RISERS_R_MH001	01-0001	Risers MovingHalf	Meusburger	1730	F070/446x396/62x86-17..
		TUTOR_RISERS_L_MH001	8	1	TUTOR_RISERS_L_MH001	01-0001	Risers MovingHalf	Meusburger	1730	F070/446x396/62x86-17..
		TUTOR_CLP_PLATE_MH001	9	1	TUTOR_CLP_PLATE_MH001	01-0001	Clampingplate MovingHalf	Meusburger	1730	F020/446x396x36-1730
		TUTOR_EJRET_PLATE_MH001	10	1	TUTOR_EJRET_PLATE_MH001	01-0001	EjectorBasePlate MovingHalf	Meusburger	1730	F080/446x396x17-1730

1. Select a couple of rows using the Shift key.
2. Use the **RMB** menu and **Copy-And-Paste** functionality to move them to another position in the BOM.

   NOTE:
   Use Ctrl +X and Ctr +V as shortcuts to Cut and Paste rows in the BOM.

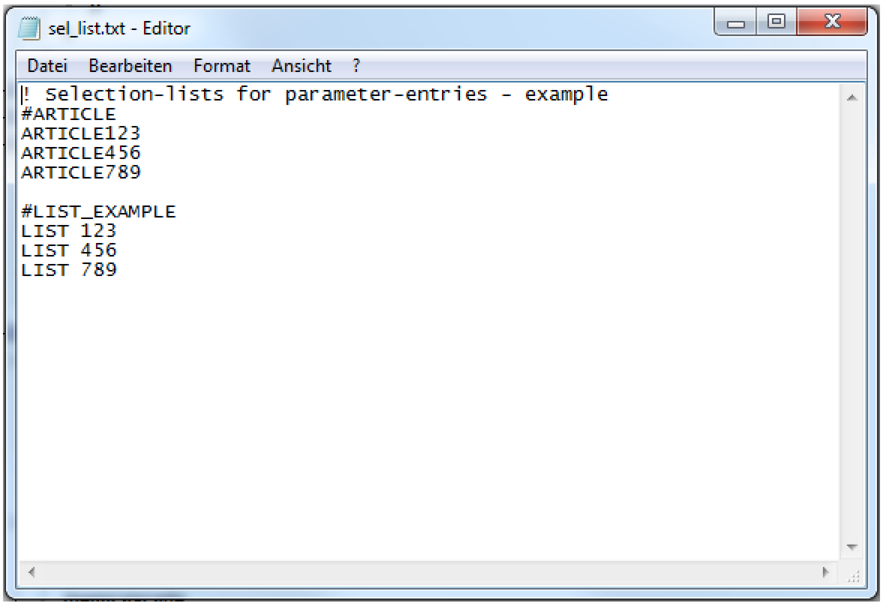
3. You can also move the selected components up or down step-by-step with  and 
   .
4. Toggle the **VISIBLE** flag for multiple components at the same time by using the **RMB** menu and the **Show/Hide items** feature.
5. Find the **Find** feature it is easy to find entries in the table.. All found entries are highlighted in red color.



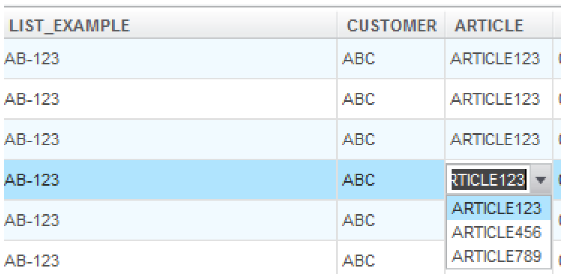
E...	VI...	MODELL	P...	Q...	Σ PARTNAME	PARTID	BOM_NAME	Σ ORD_NUMBER	SIZE	NOTE	LIST_EXAM...	CUSTOM...	ARTICLE
		ASM_PALM_REF	1	2	ASM_PALM_REF	*	ASM_PALM_REF	*	130.399x91.447x14.044	*	AB-123	ABC	?
		ASM_PALM_1	2	1	ASM_PALM_1	*	ASM_PALM_1	*	0x0x0	*	AB-123	ABC	?
		SLIDER_PALM	3	2	SLIDER_PALM	*	SLIDER_PALM	*	54.000x27.800x3.000	*	AB-123	ABC	?
		ASM_PALM_2	4	1	ASM_PALM_2	*	ASM_PALM_2	*	0x0x0	*	AB-123	ABC	?
		TUTOR_CAV_PLATE_FH001	5	1	TUTOR_CAV_PLATE_FH001	01-0001	Cavityplate FixHalf	F050/446x396x66-1730	446.000x396.000x66.000		AB-123	ABC	?
		TUTOR_CLP_PLATE_FH001	6	1	TUTOR_CLP_PLATE_FH001	01-0001	Clampingplate FixHalf	F020/446x396x36-1730	596.000x396.000x36.000		AB-123	ABC	?
		TUTOR_RISERS_R_MH001	7	1	TUTOR_RISERS_R_MH001	01-0001	Risers MovingHalf	F070/446x396/62x86-17...	96.000x86.000x62.000		AB-123	ABC	?
		TUTOR_RISERS_L_MH001	8	1	TUTOR_RISERS_L_MH001	01-0001	Risers MovingHalf	F070/446x396/62x86-17...	96.000x86.000x62.000		AB-123	ABC	?
		TUTOR_CLP_PLATE_MH001	9	1	TUTOR_CLP_PLATE_MH001	01-0001	Clampingplate MovingHalf	F020/446x396x36-1730	596.000x396.000x36.000		AB-123	ABC	?
		TUTOR_EJRET_PLATE_MH001	10	1	TUTOR_EJRET_PLATE_MH001	01-0001	EjectorBasePlate MovingHalf	F080/446x396x17-1730	396.000x318.000x17.000		AB-123	ABC	?
		TUTOR_INT_PLATE_MH001	11	1	TUTOR_INT_PLATE_MH001	01-0001	Intermediateplate MovingHalf	F060/446x396x56-1730	446.000x396.000x56.000		AB-123	ABC	?
		TUTOR_CAV_PLATE_MH001	12	1	TUTOR_CAV_PLATE_MH001	01-0001	Cavityplate MovingHalf	F050/446x396x66-1730	446.000x396.000x66.000		AB-123	ABC	?

25.3.7 Create a Quick Selection pull down menu with sel_list.txt

1. To define a list of values (like for ARTICLE or LIST_EXAMPLE), edit the file `/configuration/sel_list.txt` as follows:

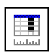


2. Identify the Parameter with a "#"-symbol. All following lines are the values that will be displayed.

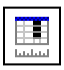


25.3.8 Recalculate Component Size

The Size of a component can be assigned to a parameter with the default value **&size**. By Default EMX adds the parameter **SIZE** to all components.



The size is computed by an algorithm which uses the outline of the model.

With  this calculation process can be started again.

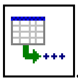
NOTE:
If the **EMX Option** `RECALC_SIZE_WHEN_OPEN_BOM` is set to `YES` the size of all components will be calculated automatically when the **BOM** dialog box opens. Be aware that this can take same time. All models that have a parameter included specified with `EXCL_SIZE_CALC_PARAM` will not be calculated. Use this for standard parts to speed up the functionality.

When the option `RECALC_SIZE_WHEN_OPEN_BOM` is set to `YES`, the **Recalculate Size** button is not visible.

25.3.9 Export neutral 3D model files from BOM

With EMX you can export neutral 3D file formats as STEP and IGES from the BOM dialog.

1. Select a component in the list and specify the file format 

.
2. Export the file with .The neutral 3D file will be written to the working directory.

25.3.10 Export neutral 2D drawing files from BOM

Drawings can also be export in the neutral file format PDF and DXF.

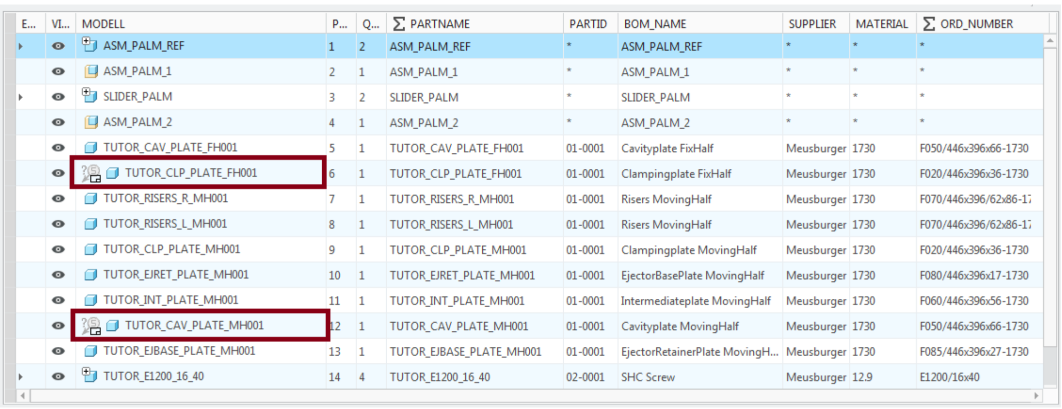
This can only be done in case a drawing exists.

NOTE:
Set the **EMX Option** `SAVE_DRAWINGS` to `YES` to automatically save drawings of plates and components defined in the **Component Dialog**.

Additionally the export can only work properly if the **EMX Option** `CHECK_DWG_WHEN_OPEN_BOM` is set to `YES`.

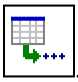
By default this is set to `NO` as checking for drawings can take very long for large assemblies.

1. Set this option to `YES`.
2. Return to the **BOM** dialog.In case a drawing for the component was found in the working directory a symbol will be displayed in the **MODEL** column.



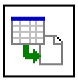
E...	VI...	MODELL	P...	Q...	Σ PARTNAME	PARTID	BOM_NAME	SUPPLIER	MATERIAL	Σ ORD_NUMBER
		ASM_PALM_REF	1	2	ASM_PALM_REF	*	ASM_PALM_REF	*	*	*
		ASM_PALM_1	2	1	ASM_PALM_1	*	ASM_PALM_1	*	*	*
		SLIDER_PALM	3	2	SLIDER_PALM	*	SLIDER_PALM	*	*	*
		ASM_PALM_2	4	1	ASM_PALM_2	*	ASM_PALM_2	*	*	*
		TUTOR_CAV_PLATE_FH001	5	1	TUTOR_CAV_PLATE_FH001	01-0001	Cavityplate FixHalf	Meusburger	1730	F050/446x396x66-1730
		TUTOR_CLP_PLATE_FH001	6	1	TUTOR_CLP_PLATE_FH001	01-0001	Clampingplate FixHalf	Meusburger	1730	F020/446x396x36-1730
		TUTOR_RISERS_R_MH001	7	1	TUTOR_RISERS_R_MH001	01-0001	Risers MovingHalf	Meusburger	1730	F070/446x396/62x86-17
		TUTOR_RISERS_L_MH001	8	1	TUTOR_RISERS_L_MH001	01-0001	Risers MovingHalf	Meusburger	1730	F070/446x396/62x86-17
		TUTOR_CLP_PLATE_MH001	9	1	TUTOR_CLP_PLATE_MH001	01-0001	Clampingplate MovingHalf	Meusburger	1730	F020/446x396x36-1730
		TUTOR_EJRET_PLATE_MH001	10	1	TUTOR_EJRET_PLATE_MH001	01-0001	EjectorBasePlate MovingHalf	Meusburger	1730	F080/446x396x17-1730
		TUTOR_INT_PLATE_MH001	11	1	TUTOR_INT_PLATE_MH001	01-0001	Intermediateplate MovingHalf	Meusburger	1730	F060/446x396x56-1730
		TUTOR_CAV_PLATE_MH001	12	1	TUTOR_CAV_PLATE_MH001	01-0001	Cavityplate MovingHalf	Meusburger	1730	F050/446x396x66-1730
		TUTOR_EJBASE_PLATE_MH001	13	1	TUTOR_EJBASE_PLATE_MH001	01-0001	EjectorRetainerPlate MovingH...	Meusburger	1730	F085/446x396x27-1730
		TUTOR_E1200_16_40	14	4	TUTOR_E1200_16_40	02-0001	SHC Screw	Meusburger	12.9	E1200/16x40

3. Select one of these components and specify the desired file format.

4. Export the file with .The PDF file will be written to the working directory.

25.3.11 Export BOM to a text file

The content of the BOM can be exported to a specified text file.

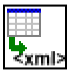
1. Click  to export the text file to the working directory

   NOTE:
   With the **EMX Option** `BOM_FILENAME` the desired name of this file can be set.

   There is also an opportunity to use a VBA macro to format the content before exporting it to Excel. For further details read chapter Write BOM to Excel via Macro on page 314.

25.3.12 Export BOM to a xml file

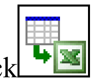
The content of the BOM can be exported to a specified xml file.

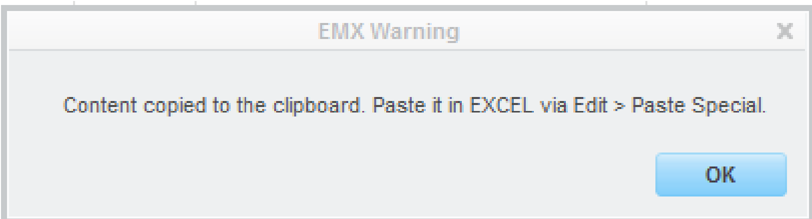
1. Click  to export the text file to the working directory

   NOTE:
   With the **EMX Option** `BOM_FILENAME` the desired name of this file can be set.

25.3.13 Copy/Paste content to Excel

The BOM content can be copied and pasted to Excel.

1. Click to export the content to the Clipboard.



2. Paste the content to an Excel worksheet.

25.3.14 Exclude all assemblies from the BOM

Usually the assemblies of a Creo Moldbase do not appear in the BOM. They are used for structural reasons only. Right now models like `ASM_PALM_1.ASM` or `TUTOR_STOP_ASM2` are still visible in the BOM. Instead of changing their visibility status manually, you can set the **EMX Option** `EMX_HIDE_ASSEMBLIES_IN_BOM` to `YES`.

25.3.15 Include IDs as prefix in component names

If the **EMX Option** `PART_RENAME_FORMAT` is set to a wildcard including <pos_id> and/or <partid>, two extra functions will appear in the BOM dialog box.

With the selected models will be renamed to the new name defined by this wildcard.

With this renaming is set back.

25.4 About the Bill of Materials Dialog Box in Drawing Mode

From the **Bill of Material** dialog box, you can place BOM balloons in the current drawing.

Use the following commands from the shortcut menu to add and delete balloons:

- Right–click an item and select **Add Balloon Note**. The **Bill of Materials** dialog box is minimized, and item is selected in the drawing sheet. Click one time, define the attachment position, and click a second time to define the balloon position.
- Click **Remove Balloon Note** to remove all balloons from the selected item.

NOTE:

- You can place multiple balloons for each BOM item.
- You can select multiple items. The balloons are placed in a loop

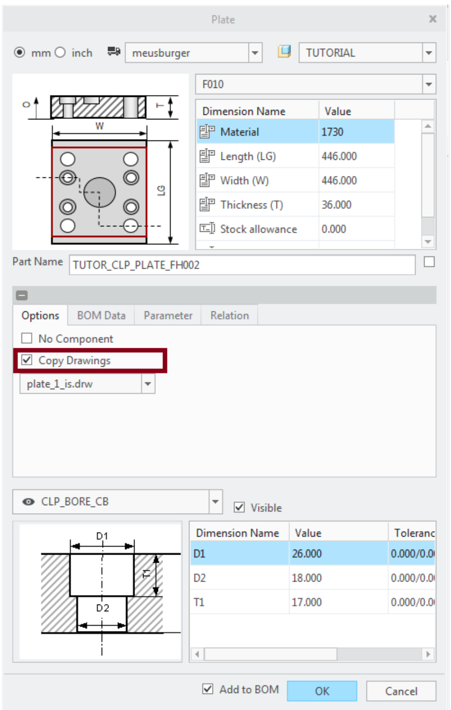
25.5 To Work with Drawing Support Features

25.5.1 Add a format to all plate drawing templates

Beside the BOM the drawing is still one of the most important sources of information for mold manufacturing. EMX offers drawing templates for almost all template models. They are located in the installation directory of EMX.

Beside the assembly drawing that is created when create/modify an EMX project, additional drawings might be required for individual components such as plates, ejector pins, etc. EMX provides drawing-templates for most EMX components. They are located in the `<install_emx>/components/mm/<comptype>` folder.

In the **Plate** dialog box, the check button to copy the drawing is activated by default. All drawings that have the plate-template name included are listed in the drawing pull-down.



The available plate types (with ES for moving half and IS for fix half) are:

plate_1	Clamping plate
plate_2	Corepin Retainer Plate
plate_3	Support Plate
plate_4 / plate_4_left	Rail (one or two models)
plate_5	Cavity Plate
plate_6	Ejector Retainer Plate
plate_7	Ejector Base Plate
plate_8	Stripper Plate

1. Add your drawing formats to this templates BEFORE starting your design. Use the function **Administrator Tools ► Add Drawing Formats** to support this task.

   If you want to use EMX drawings it is recommended to set the **EMX Option** `SAVE_DRAWINGS` to `YES`. It is *not* possible to create a plate drawing automatically if you forgot it during the initial plate definition.

   For other EMX components, you can decide wether the check button **Copy Drawing** in the Component dialog box is set or not in the **EMX Option** `EMX_CHECK_DWG_TYPES`. Simply list the component types for which the checkbutton should be active, the default is `ejector|slider|lifter|latchlock`.

2. Use the option `SET_EMX_DTL` to decide wether the dtl-settings from file `<install_emx>/configuration/symbols_dtl/<unit>/emx.dtl` should be applied to a drawing or not.

25.5.2 Quick Access to drawings when working in assembly mode

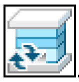
1. Select **EMX Assembly ► Documentation ► Open Drawing of selected model**  to open the drawing from the current working directory or from session directly.
2. Select a component with LMB. If a drawing with the same name exist, it will be opened.

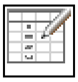
25.5.3 Balloon placement with EMX

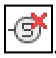
1. Open the assembly`TUTORIAL.ASM`.

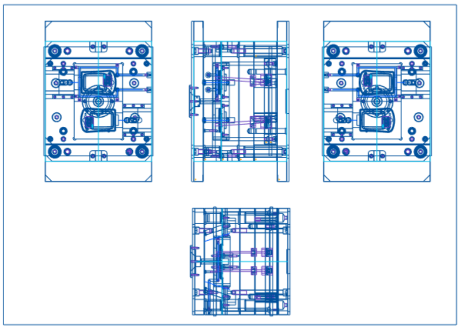
   Use **EMX Assembly ► Documentation ► Open Drawing of current model**  to open the drawing from session directly.

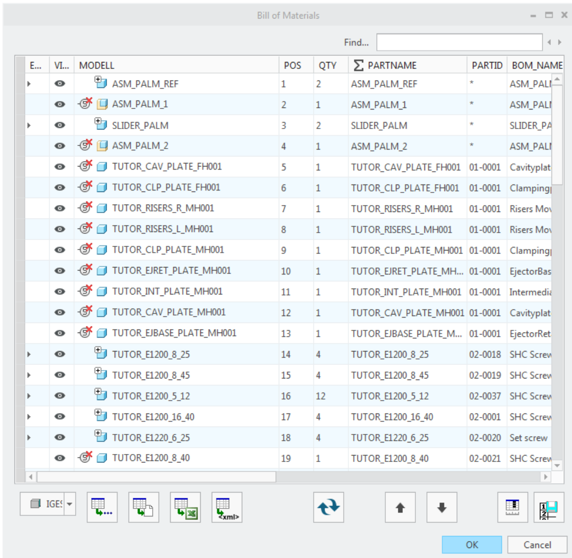
   NOTE:

   In case no drawing exists use **EMX Assembly ► Project ► Modify** . The **Project** Dialog box appears. Select the check button **Copy Drawings** and close the dialog with **OK**. EMX will create a drawing.

2. Select **Bill of Materials**  to open the EMX **Bill of Materials** dialog box.

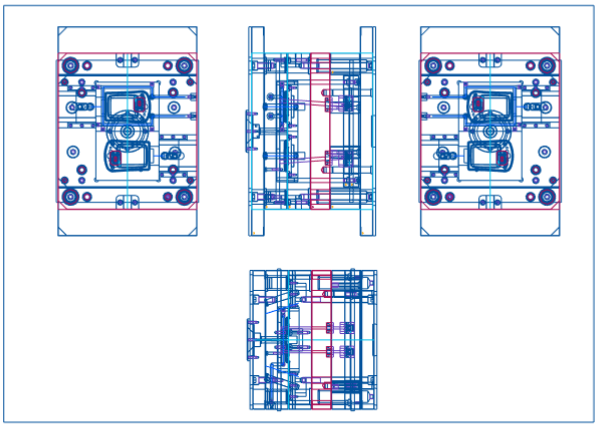
   A symbol in the table indicates wether the balloon for a component exist  or not .



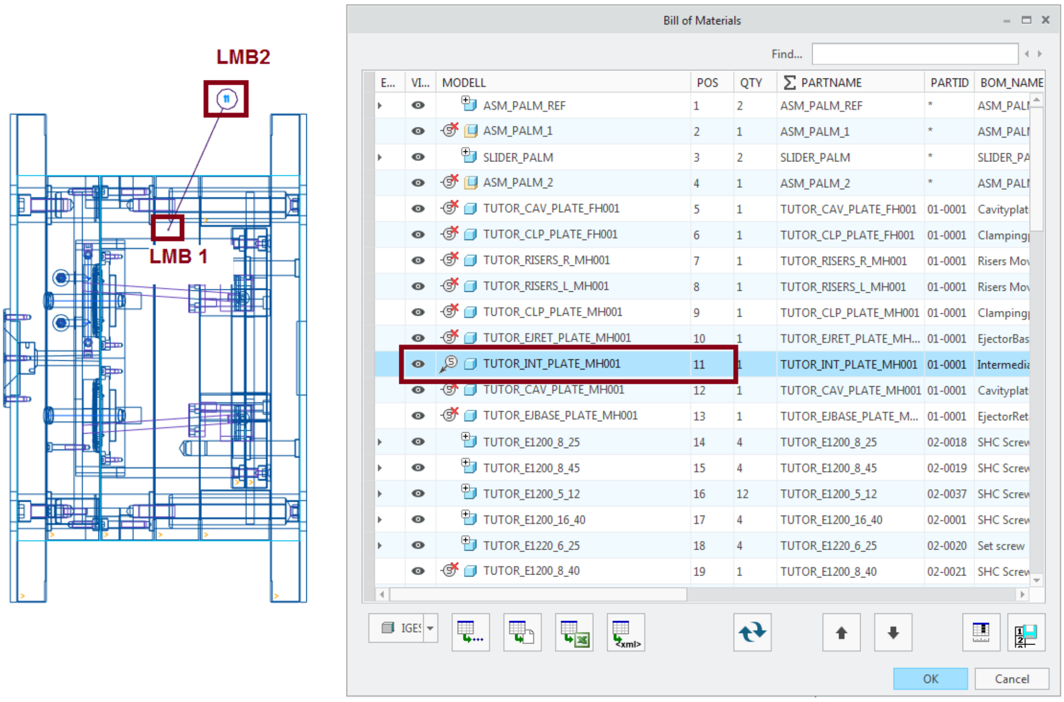


3. Click RMB on this symbol in the line of TUTOR_INT_PLATE_MH001 and Select **Add BOM Number**.

   The model is highlighted in all views of the current sheet and the **Bill of Materials** is hidden.



4. Click with LMB the position of the arrow end (LMB 1)and click again for the balloon position (LMB 2).



5. Click RMB again on the symbol again in the BOM table and remove the balloon again with **Remove BOM Number** .

   NOTE:
   By default the Parameter **&pos_id** is displayed in the balloon. If the **EMX Option** `USE_PARTID_IN_BALLOON` is set to `YES`, the parameter with rule **&part_no** is displayed in the balloon.

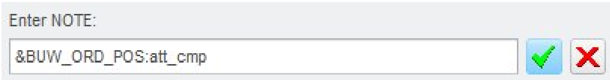
   If `SORT_BOM_WITH_PARTID` is set to `YES`, EMX will sort the BOM always by the content of the parameter with rule **&part_no**.

   Additional relevant **EMX Options** are `BALLOON_ARROW_END_TYPE` and `BALLOON_SIZE`. `PLACE_BALLOON_ON_FACE` is by default set to `YES`.

25.5.4 Balloon placement with Creo

Creo offers a flexible way to place balloons. Based on the BOM-Table with repeat region on sheet 3 of the EMX assembly drawing the balloons can be placed with **Table ► Ballons ► Balloon Note**.

It is possible to define the content in the balloon rather flexible. The easiest way is to use the Component Parameter BUW_ORD_POS in the balloon note.



Background Information

The symbols and their diameter-table are located in `<install_emx>/configuration/symbols_dtl/<unit>`. In `ejpin_symbols.cfg` the nominal diameter for ejector pins and their related symbol is listed. With option `SUPPRESS_EJP_NOTE` set to `YES` the display of the extra note can be avoided.

25.6 About Opening Drawings

You can use one of the following commands to quickly open drawing files.

- Click **EMX Assembly ► Documentation ►**  **Open drawing of current model** to open a drawing with the same name as the current model.
- Click **EMX Part Mode ► EMX Tools ►**  **Open drawing of current model** to open a drawing with the same name as the current model.
- Click **EMX Assembly ► Documentation ►**  **Open drawing of selected model** and select a component in the current assembly.
- Click **EMX Part Mode ► EMX Tools ►**  **Open drawing of selected model** and select a component in the current assembly.

NOTE:
If there is no drawing, nothing happens.

25.7 To Write BOM to Excel via Macro

In most cases a formatted EXCEL-file for the BOM is needed. This chapter explains how to set up a special Macro for EXCEL. The EMX BOM content can be imported automatically to an EXCEL-template.

25.7.1 Use the template emx_bom.xls

The setting of the option `BOM_FILENAME` drives the output format of the button **Copy content to windows clipboard** .

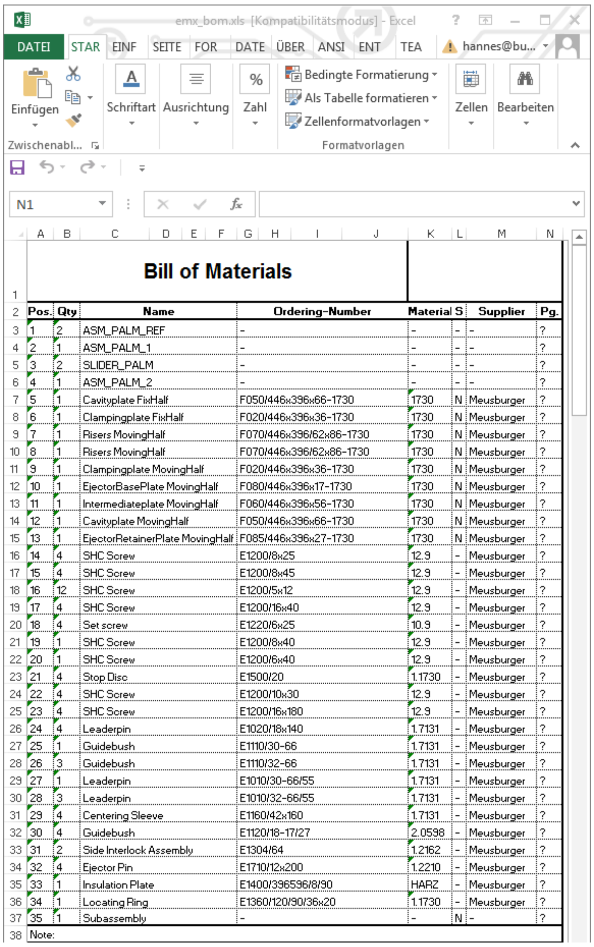
1. Set the **EMX Option** `BOM_FILENAME` to `C:\TMP\buw_excel.txt`

   It is important to use an absolute file path in this case. With this setting, the **Export to EXCEL-command** will create two files in `C:\TMP`.
2. Make sure the folder `C:\TMP` exists.
3. Export the BOM with .

   `buw_project_excel.txt` has five lines with the first five Project Parameters defined in EMX Options.

   `buw_excel.txt` is a listing of all components in the BOM with their parameters &pos_id, &partid, &qty, &bomname, &supplier, &ord_number, &material, &remark, &special and &dwg_page.

After exporting these two files EMX will open EXCEL automatically using the template `/configuration/templates/emx_bom.xls`. This EXCEL-file has an Import Macro that will be started automatically when open the file. Due to the fix content of the two export files, the macro can import and resort the parameters into the correct sheets (Blatt 1–Blatt3) of the template.



Bill of Materials

Pos.	Qty	Name	Ordering-Number	Material	S	Supplier	Pg.
1	2	ASM_PALM_REF	-	-	-	-	?
2	1	ASM_PALM_1	-	-	-	-	?
3	2	SLIDER_PALM	-	-	-	-	?
4	1	ASM_PALM_2	-	-	-	-	?
5	1	Cavityplate FixHalf	F050/446x396x66-1730	1730	N	Meusburger	?
6	1	Clampingplate FixHalf	F020/446x396x36-1730	1730	N	Meusburger	?
7	1	Risers MovingHalf	F070/446x396/62x86-1730	1730	N	Meusburger	?
8	1	Risers MovingHalf	F070/446x396/62x86-1730	1730	N	Meusburger	?
9	1	Clampingplate MovingHalf	F020/446x396x36-1730	1730	N	Meusburger	?
10	1	EjectorBasePlate MovingHalf	F080/446x396x17-1730	1730	N	Meusburger	?
11	1	Intermediateplate MovingHalf	F060/446x396x56-1730	1730	N	Meusburger	?
12	1	Cavityplate MovingHalf	F050/446x396x66-1730	1730	N	Meusburger	?
13	1	EjectorRetainerPlate MovingHalf	F085/446x396x27-1730	1730	N	Meusburger	?
14	4	SHC Screw	E1200/8x25	12.9	-	Meusburger	?
15	4	SHC Screw	E1200/8x45	12.9	-	Meusburger	?
16	12	SHC Screw	E1200/5x12	12.9	-	Meusburger	?
17	4	SHC Screw	E1200/16x40	12.9	-	Meusburger	?
18	4	Set screw	E1220/6x25	10.9	-	Meusburger	?
19	1	SHC Screw	E1200/8x40	12.9	-	Meusburger	?
20	1	SHC Screw	E1200/6x40	12.9	-	Meusburger	?
21	4	Stop Disc	E1500/20	1.1730	-	Meusburger	?
22	4	SHC Screw	E1200/10x30	12.9	-	Meusburger	?
23	4	SHC Screw	E1200/16x180	12.9	-	Meusburger	?
24	4	Leaderpin	E1020/18x140	1.7131	-	Meusburger	?
25	1	Guidebush	E1110/30-66	1.7131	-	Meusburger	?
26	3	Guidebush	E1110/32-66	1.7131	-	Meusburger	?
27	1	Leaderpin	E1010/30-66/55	1.7131	-	Meusburger	?
28	3	Leaderpin	E1010/32-66/55	1.7131	-	Meusburger	?
29	4	Centering Sleeve	E1160/42x160	1.7131	-	Meusburger	?
30	4	Guidebush	E1120/18-17/27	2.0598	-	Meusburger	?
31	2	Side Interlock Assembly	E1304/64	1.2162	-	Meusburger	?
32	4	Ejector Pin	E1710/12x200	1.2210	-	Meusburger	?
33	1	Insulation Plate	E1400/396596/8/90	HARZ	-	Meusburger	?
34	1	Locating Ring	E1360/120/90/36x20	1.1730	-	Meusburger	?
35	1	Subassembly	-	-	N	-	?

Note:

25.7.2 Use the template emx_bom_adv.xls

While the method mentioned above can only handle certain parameters, EMX offers a more flexible way to define the content of the EXCEL-template.

1. Set the **EMX Option** `BOM_FILENAME` to `C:\TMP\emx_bom_adv.txt` to use this more flexible template `configuration/templates/emx_bom_adv.xls`.
2. Open the EXCEL template <emx_install>\configuration\templates\emx_bom_adv.xls

   Before using the xls-file, you need to configure the column names and (optional) the substructure.

   Check if the **Source File Name** is equal to the previously set **EMX Option** `BOM_FILENAME`, in this case `C:\TMP\emx_bom_adv.txt`.

Source File Name
C:\TMP\emx_bom_adv.txt

3. In row number 2 (**EMX_PARAMETER_NAMES**) add the exact parameter names used in EMX Options.

   In row number 3 (**CUSTOMER_COLUMN_NAMES**) add the according column names used in the **BOM** sheet.

   In this example six EMX parameters are used: MODEL, POS, QTY, BOM_NAME, SUPPLIER, MATERIAL

EXPAND	VISIBLE	MODELL	POS	QTY	BOM_NAME	SUPPLIER	MATERIAL
		TUTOR_E2074_8_10	26	5	Threaded Conic Plug	Meusburger	2.0401
		TUTOR_E2000_9_8	27	2	Fitting	Meusburger	2.0401

   In the **BOM** sheet of the columns look like this:

	A	B	C	D	E	F	G
1	Bill Of Material						
2							
3	Pos	Model name	Qty	Bom Name	Supplier	Material	
4							

   In the **System** sheet of the excel the parameters need to be set up accordingly:

	A	B	C	D	E	F	G
1	Column mapping	GROUP_COLUMN	POS_COLUMN				
2	EMX_PARAMETER_NAMES	Model	POS	QTY	BOM_NAME	SUPPLIER	MATERIAL
3	CUSTOMER_COLUMN_NAMES	Model name	Pos	Qty	Bom Name	Supplier	Material

   NOTE:
   There are two special parameter columns **GROUP_COLUMN** and **POS_COLUMN**.

   - Without a correct **POS_COLUMN** the macro does not work. It is important, that the EMX Parameter with the default value **&posid** is used here.
   - **GROUP_COLUMN** specifies the column in which the group titles appear.

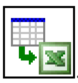
4. It is possible to add extra comment lines in the BOM.

   Add the desired group titles in row 9 (**GROUP_TITLE**). It is possible to set up as much comments as you want. In the row underneath (**STARTING_POS_NR**) a position id needs to be defined which describes the starting position

   Set up four titles as shown below and make sure **GROUP_MODELS** option is set to YES in row 6.

Options				
GROUP_MODELS	YES			
Group Titles				
GROUP_TITLE	Title_1	Title_2	Title_3	Title_4
STARTING_POS_NR	0	20	40	60

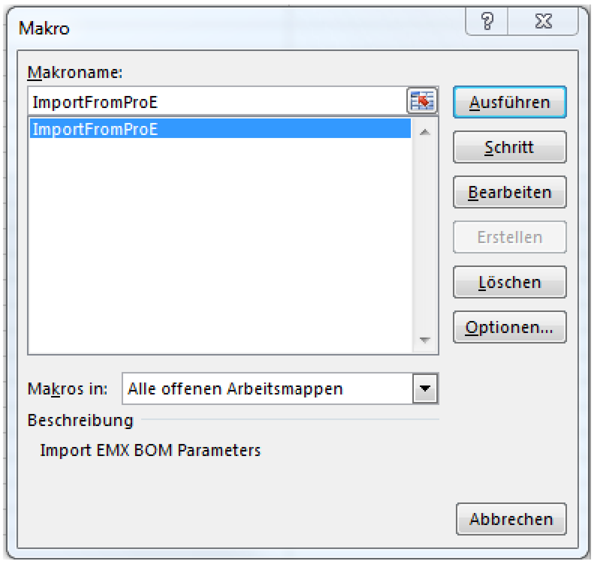
5. Save the `emx_bom_adv.xls` and close EXCEL.

6. Go back to Creo and export the BOM with .

In case macros in Excel are deactivated active content.

7. Run the Import of EMX Parameters with **Developer Tools ► Macro** 

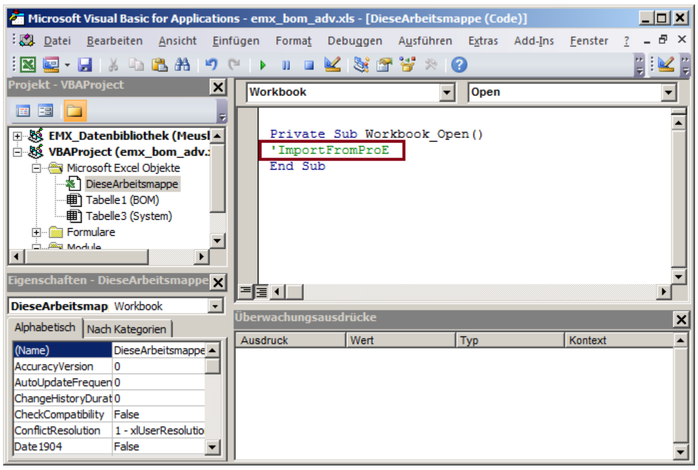
► **ImportFrom ProE**



A new EXCEL file will be created named `emx_bom_adv_output.xls` and written to the current working directory.

	A	B	C	D	E	F
1				**Bill Of Material**		
2						
3	**Pos**	**Model name**	**Qty**	**Bom Name**	**Supplier**	**Material**
4		Title_1				
5	1	TUTOR_STOP_DISC8	4	Stop Disc	Meusburger	1,173
6	2	TUTOR_SLIDER_ASM1	2	Slider Assembly	Meusburger	-
7	3	TUTOR_SIDELOCK_ASM8	2	Side Interlock Assembly	Meusburger	12,162
8	4	TUTOR_S_PLATE1	2	Plate	Meusburger	-
9	5	TUTOR_S_LGIB_R1	2	Lgib	Meusburger	-
10	6	TUTOR_S_LGIB_L1	2	Lgib	Meusburger	-
11	7	TUTOR_S_CAM1	2	Plate	Meusburger	-
12	8	TUTOR_S_ANGLEPIN1	2	Anglepin	Meusburger	-
13	9	TUTOR_RISERS_R_MH001	1	Risers MovingHalf	Meusburger	1730
14	10	TUTOR_RISERS_L_MH001	1	Risers MovingHalf	Meusburger	1730
15	11	TUTOR_MH	1	Subassembly	-	-
16	12	TUTOR_LOCATING_RING2	1	Locating Ring	Meusburger	1,173
17	13	TUTOR_LIFTER_GUIDE2	2	Lifter Guide	Meusburger	
18	14	TUTOR_LIFTER_BAR2	2	Lifterbar	Meusburger	
19	15	TUTOR_LIFTER_ASM2	2	Lifter Assembly	Meusburger	-
20	16	TUTOR_INT_PLATE_MH001	1	Intermediateplate MovingHalf	Meusburger	1730
21	17	TUTOR_INSULATION4	1	Insulation Plate	Meusburger	HARZ
22	18	TUTOR_INSULATION3	1	Insulation Plate	Meusburger	HARZ
23	19	TUTOR_GUIDE_E1304M008	2	E13041_64	Meusburger	
24		Title_2				
25	20	TUTOR_GUIDE_E1304F008	2	E13042_64	Meusburger	
26	21	TUTOR_FH	1	Subassembly	-	-
27	22	TUTOR_EJRET_PLATE_MH001	1	EjectorBasePlate MovingHalf	Meusburger	1730
28	23	TUTOR_EJECTOR_PIN001	4	Ejector Pin	Meusburger	1,221
29	24	TUTOR_EJBASE_PLATE_MH001	1	EjectorRetainerPlate MovingHalf	Meusburger	1730
30	25	TUTOR_E2130_12_1_5	2	O-Ring	Meusburger	Viton
31	26	TUTOR_E2074_8_10	5	Threaded Conic Plug	Meusburger	20,401
32	27	TUTOR_E2000_9_8	2	Fitting	Meusburger	20,401
33	28	TUTOR_E1600_18_86_4	1	Spruebush	Meusburger	12,826
34	29	TUTOR_E1510_32_86	4	Support Pillar	Meusburger	1,173
35	30	TUTOR_E1220_6_25	4	Set screw	Meusburger	10,9
36	31	TUTOR_E1200_8_45	4	SHC Screw	Meusburger	12,9
37	32	TUTOR_E1200_8_40	1	SHC Screw	Meusburger	12,9
38	33	TUTOR_E1200_8_25	4	SHC Screw	Meusburger	12,9
39	34	TUTOR_E1200_6_40	1	SHC Screw	Meusburger	12,9
40	35	TUTOR_E1200_5_12	12	SHC Screw	Meusburger	12,9
41	36	TUTOR_E1200_16_40	4	SHC Screw	Meusburger	12,9
42	37	TUTOR_E1200_16_180	4	SHC Screw	Meusburger	12,9
43	38	TUTOR_E1200_10_30	4	SHC Screw	Meusburger	12,9
44	39	TUTOR_E1160_42_160	4	Centering Sleeve	Meusburger	17,131
45		Title_3				
46	40	TUTOR_E1120_18_17_27	4	Guidebush	Meusburger	20,598
47	41	TUTOR_E1110_32_66	3	Guidebush	Meusburger	17,131
48	42	TUTOR_E1110_30_66	1	Guidebush	Meusburger	17,131
49	43	TUTOR_E1020_18_140	4	Leaderpin	Meusburger	17,131
50	44	TUTOR_E1010_32_66_55	3	Leaderpin	Meusburger	17,131
51	45	TUTOR_E1010_30_66_55	1	Leaderpin	Meusburger	17,131
52	46	TUTOR_CLP_PLATE_MH001	1	Clampingplate MovingHalf	Meusburger	1730
53	47	TUTOR_CLP_PLATE_FH001	1	Clampingplate FixHalf	Meusburger	1730
54	48	TUTOR_CAV_PLATE_MH001	1	Cavityplate MovingHalf	Meusburger	1730
55	49	TUTOR_CAV_PLATE_FH001	1	Cavityplate FixHalf	Meusburger	1730
56	50	SLIDER_PALM	2	SLIDER_PALM		
57	51	MH_PALM_2	1	Core		
58	52	MH_PALM_1	1	Core		
59	53	FH_PALM_2	1	Cavity		
60	54	FH_PALM_1	1	Cavity		
61	55	ASM_PALM_REF	2	ASM_PALM_REF		
62	56	ASM_PALM_2	1	ASM_PALM_2		
63	57	ASM_PALM_1	1	ASM_PALM_1		

If you want the Import Macro run automatically when open the file, uncomment this line `ImportFromProE` in the Visual Basic Macro



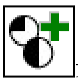
25.8 About Applying and Modifying Ejector Pin Symbols

1. Open a drawing that contains models with ejector pin cutouts.
2. Click **EMX Drawing Mode** ►  **Add drawing symbols**.
3. Select a view of the model. All the ejector pin cutouts in the model are identified, and diameter-dependent symbols are added or updated.

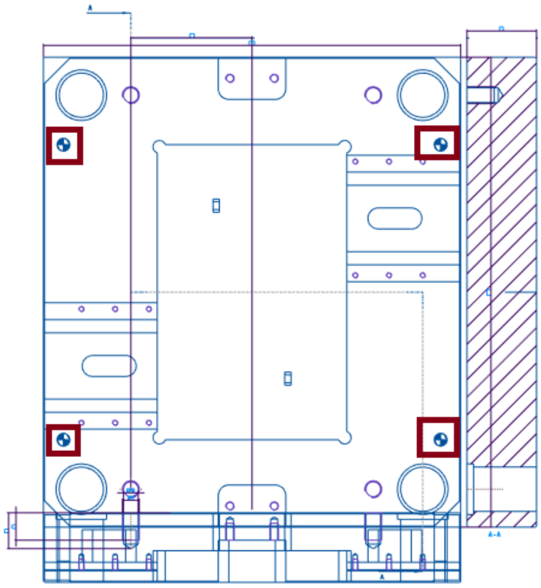
NOTE:
These symbols can be modified. They are located in `<installdir>/configuration/symbols_dtl/<unit>`. The cutout diameter is automatically compared with the `epin_symbols.cfg` file setting to determine the correct symbol to use.

25.9 To Add Ejector Pin Symbols to Drawing

EMX can place certain symbols on plate drawings to indicate ejector pin holes. The symbol size is adapted automatically to the real diameter of the ejector pin bore.

1. Open drawing `TUTOR_EJRET_PLATE_MH001.DRW`.
2. Select **Add drawing symbols**  from the **EMX Drawing Mode** tab.
3. Select the TOP_VIEW of the plate.

EMX add size-specific symbols to each ejector pin cutout and adds a note with the Ejector ID



Notes:

26

About Working with Mold Base Definition Templates

26.1 About Saving An Assembly Recipe

1. Click in the **Mold Base Definition** dialog box to open the **Save EMX Assembly** dialog box. The current mold base appears in the preview window.
2. Select a supplier from the **Supplier** list.
3. Enter a name for the mold base assembly in the box below the **Stored Assemblies** list.
4. Click to save the current mold base assembly.
5. Click **OK** to save the assembly and return to the **Mold Base Definition** dialog box, or click **Cancel** to close the operation without saving changes.

26.2 About Loading an Assembly Recipe

Mold base definitions and supplier default mold bases are stored in the assembly database.

1. Click in the **Mold Base Definition** dialog box. The **Load EMX Assembly** dialog box opens.
2. Select a supplier from the **Supplier** list.
3. Select a mold base from the **Stored Assemblies** list. The selected mold base appears in the preview window, and the mold base name appears in the box under the list.
4. Select the **Keep size and model data** check box to retain the current mold base size and cavity layout after the new mold base is loaded.
5. Click to remove the active mold base from the list.
6. Click to load the active mold base. A preview appears in the **Mold Base Definition** dialog box, but no changes are generated in the assembly.
7. Click **OK** to apply changes to the assembly and return to the **Mold Base Definition** dialog box, or click **Cancel** to undo all changes and return to the **Mold Base Definition** dialog box.

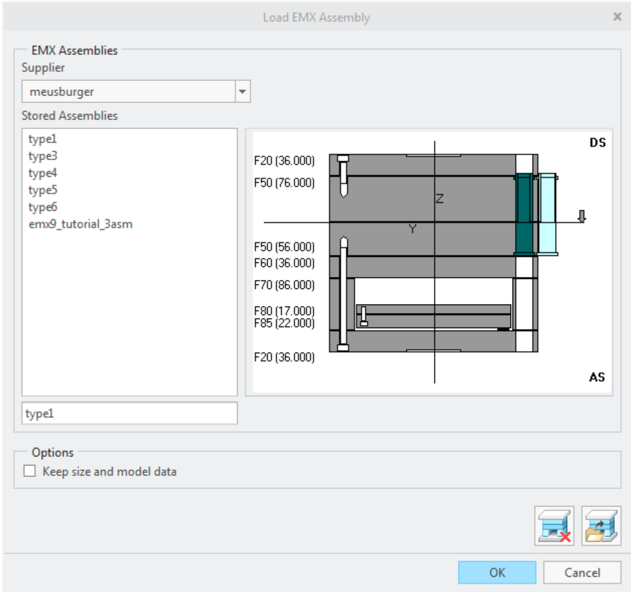
26.3 To Load and Save Mold Base Recipes

26.3.1 Load a Mold Base Recipe

It does not make sense to set up the mold base for each project step by step. This is the reason why EMX supports "recipes", that predefines sets of plates and components for the mold base.

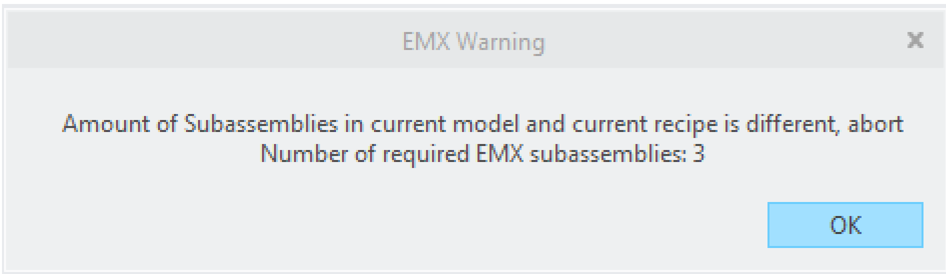
1. Create new **EMX Project** and open the **Mold Base Definition** dialog.
2. Click **Load assembly definition from file** .

The **Load EMX Assembly** dialog box provides a list of predefined mold bases for each supplier.



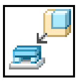
3. Select the file `emx9_tutorial_3asm` and click **Load** 

   A EMX warning message appears.



   The current EMX project does not contain the correct number of EMX subassemblies. In the given recipe `emx9_tutorial_3asm` three subassemblies are defined. The blank EMX project has zero subassemblies. Due to that EMX is not able to assemble all defined components from the recipe to the correct subassemblies.

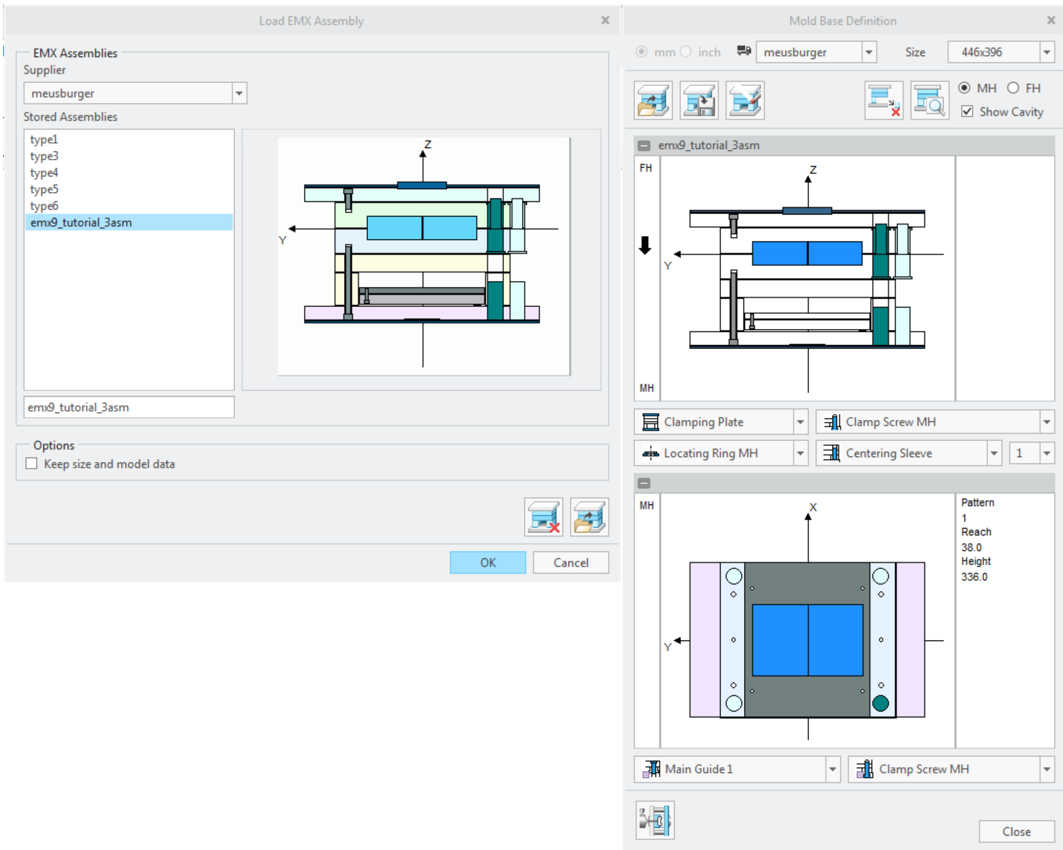
4. Leave the EMX Warning with **OK**.
5. Close the **Load EMX Assembly** dialog box with **Cancel**.
6. Close the **Mold Base Definition** dialog box with **Close** .
7. Add 3 subassemblies to the EMX project with**EMX Assembly ► Add subassemblies**

8. Reopen the **Load EMX Assembly** dialog box and selected `emx9_tutorial_3asm`.

   Click **Load** 

9. The entire mold base is previewed in the **Mold Base Definition** dialog box.



In this case the size and cavity-settings in the stored recipe are identical with the current mold base size and cavity layout you have defined.

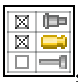
If you are working on a mold base with a different size, activate the button **Keep size and model data**. In that case EMX will try to convert all components to the mold base size you have defined before retrieving the recipe.

10. Click **OK** to load all plates and models.

The complete mold base is created in the 3D-model. The previously defined side interlocks and ejector guidance components had been removed as they are not included in the recipe. The cavity cutout has a radius instead of the mouse ears defined before.

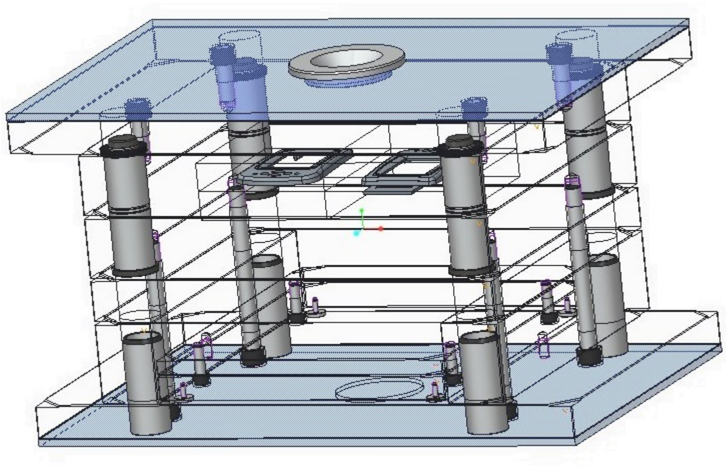
NOTE:
All components are loaded to the correct subassembly.

11. Close the **Mold Base Definition** dialog box with **Close**.

12. Open the **Component Status** , activate all types and click OK.

13. Set the Style `VIS0001` in the Creo **View Manager**.

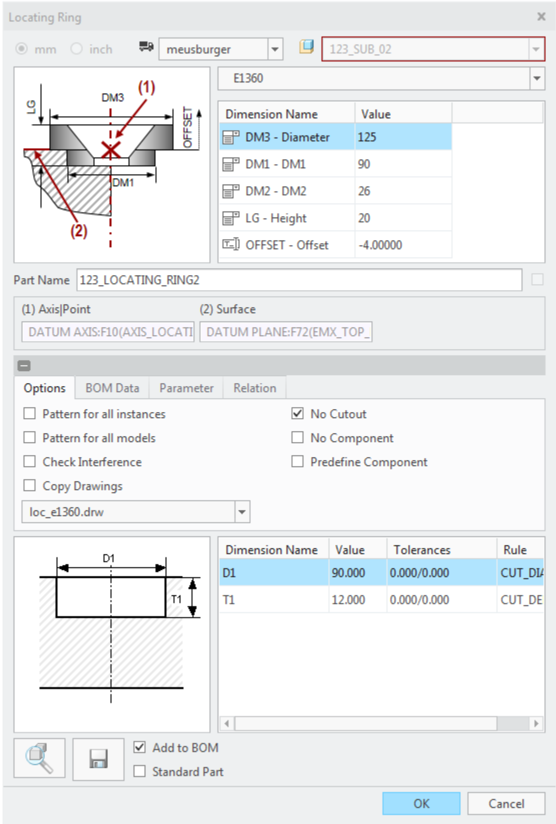
This automatically created rule based style is added when starting an EMX project. Its name is defined in the **EMX Option** `NAME_COMP_DISPLAY`.



If you like to modify details of some components, you can also edit them directly in the graphics window. As an example we modify the properties of the FH locating ring.

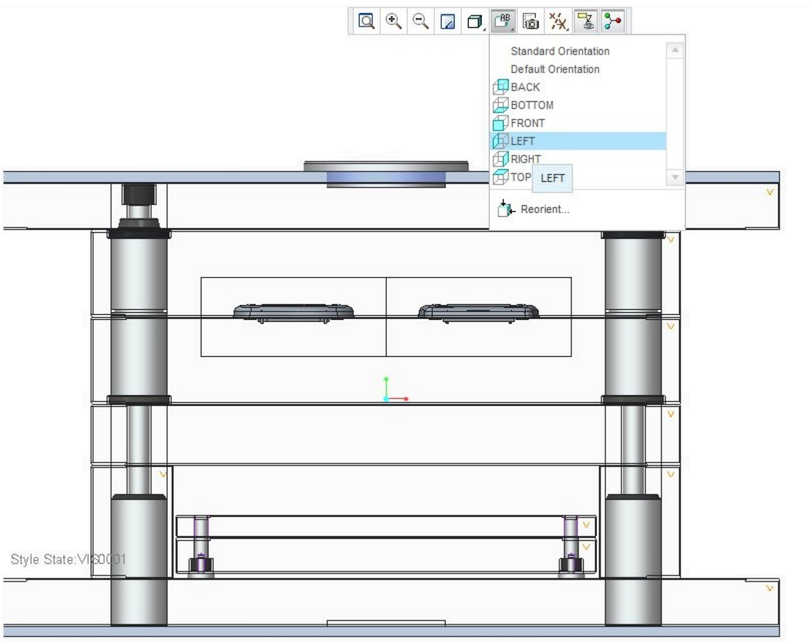
14. Select the Locating Ring `TUTOR_LOCATING_RING2.PRT` in the 3D graphics window and press RMB.
15. Select **Modify Component** from the RMB menu. The **Locating Ring** dialog box is opened again.
16. Select `125` for diameter **DM3**.

17. Check the **No Cutout** button (the cutout is defined in plate F020 anyhow).



18. Close the dialog box with **OK**.

The locating ring is set to the new size.

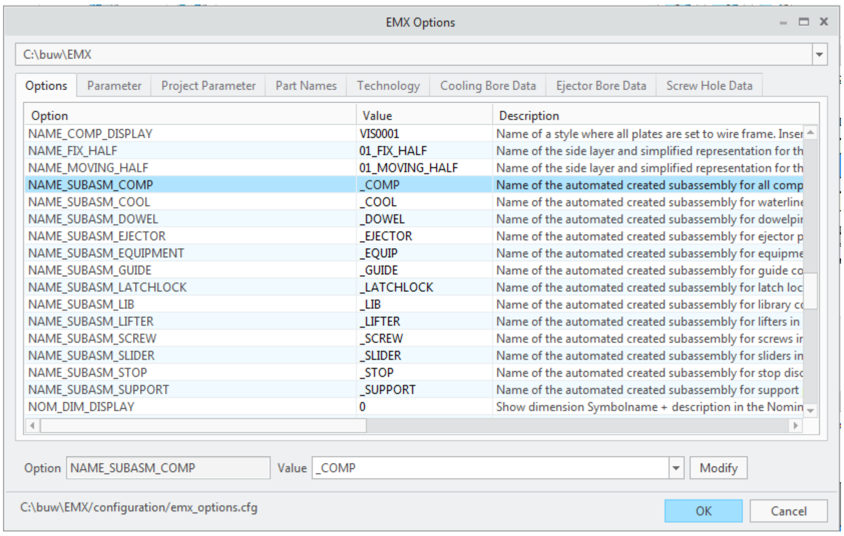


NOTE:
As you can see, all components defined in the Mold Base Definition dialog box can be redefined also outside the dialog box in direct editing mode.

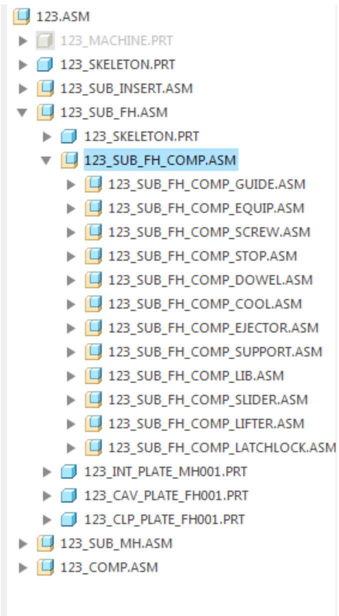
Background Information

The **Component Status** works different if the **EMX Option** `NAME_SUBASM_COMP` is set:

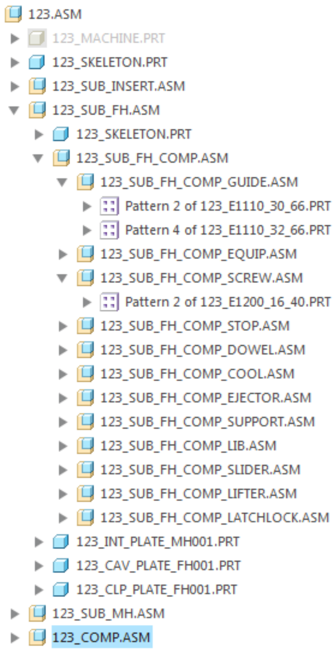
Here is an example on how an assembly looks like when `NAME_SUBASM_COMP` is not `"-"`:



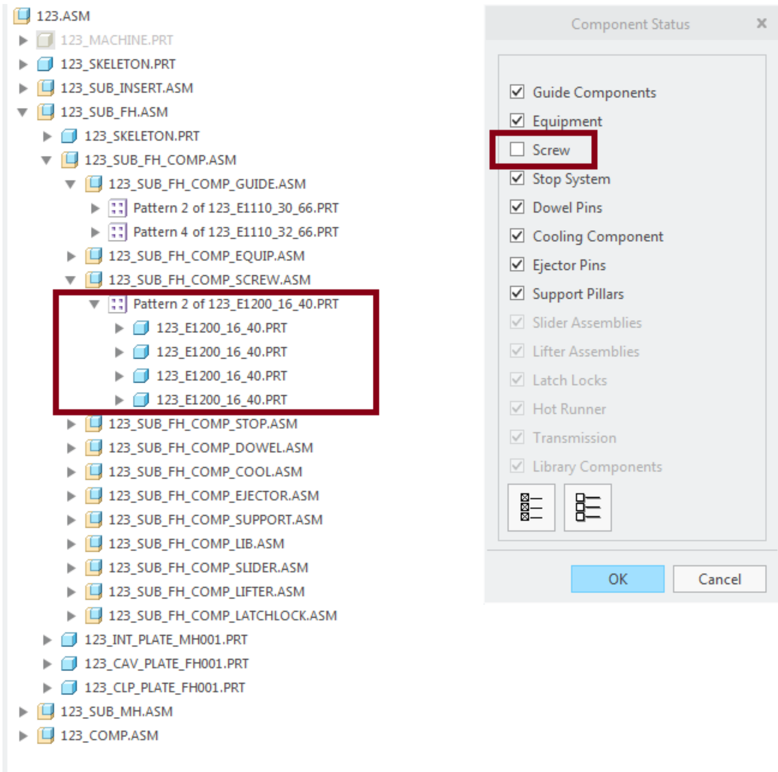
After creating the project and three sub assemblies have been added:



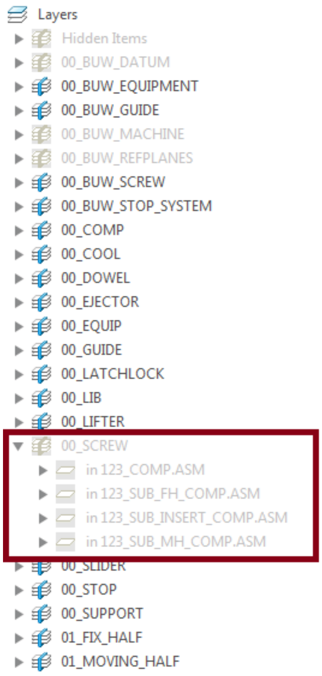
...and after retrieving the recipe `emx_tutorial_3asm` - template:



If you toggle the Component Status to hidden for i.e. screws, the models remain in the model tree



but the layer `00_SCREW` in the layer tree is set to hidden in all assemblies.



Notes:

27

Additional Mold Base Defintion Features

27.1 About Modifying the Mold Base Size

If the mold base size needs to be modified while components have already be assembled, these existing clamping screws, leader pins, bushings etc. will not automatically be updated.

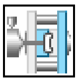
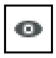
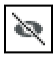
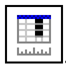
To adjust the components automatically to the new mold base size set configuration Option `UPDATE_MOLDBASE_COMPS` to `YES`.

Additionally existing ejector pins and support pillars will be updated if `UPDATE_ADDITIONAL_COMPS` is set to `YES`.

27.2 To Modify the Mold Base Size

27.3 About Machine Parameters

Use the **Machine** dialog box to define machine-dependent parameters. Changes made in this dialog box appear in the **Mold Base Definition** preview window.

1. Click  on the **Mold Base Definition** dialog box. The **Machine** dialog box opens.
2. Select a supplier from the list.
3. Set the unit of measure (**mm** or **inch**).
4. To show machine features (machine bars, plates and minimum/maximum opening ranges), select the **Show Machine** check box.
5. Select a machine from the list. The selected machine appears in the preview window. Machine dimensions appear in the table. To change a dimension value, double-click a cell in the **Value** column.
6. Values for the **Machine Bar Pattern** and the **Knockout Pattern** appear in the **Quantity**, **Pattern Size**, and **Pattern Diameter** boxes.
7. Double-click a cell in the table to change a value.  or  indicates visibility status.
8. Select the **Automatically Recalculate** check box to automatically recalculate the machine bar and knockout patterns after editing. To manually recalculate the pattern, click .
9. Click **OK** to close the dialog box and to update the assembly, or click **Cancel** to close the dialog box, and return to the **Mold Base Definition** dialog box.

   NOTE:
   If the **Show Machine** check box is selected in the **Machine** dialog box, the Machine layer is set to `VISIBLE` in the assembly.

27.4 About Reseting the Current Assembly

1. Click in the Mold Base definition dialog.
2. When prompted to delete all plates and components, click **Yes**. All components are deleted in the 3D assembly.

Notes:

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Analysis Tools and Features

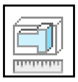
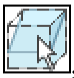
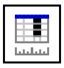
28.1 About Creo Expert Moldbase Extension Tools

To streamline design creation, Creo Expert Moldbase Extension provides various tools.

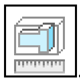
For additional tools click **EMX Assembly ▸ EMX Tools** and then select a tool described below:

- **Model Size**—Determines the exact bounding box of a solid part. The result of this analysis is saved as a model parameter.
- **Measuring Points**—Defines the measurement positions of solid parts. This is available when working with a part in EMX. Click **EMX Part Mode ▸ EMX Tools**.
- **Set Technology Colors**—Opens the **Predefined Colors** dialog box.
- **Calculate Clamping Force**—Determines the projected area of all reference models and runner features.
- **Set Accuracy**—Modifies the accuracy of one or more assembly components.
- **Export CSYS Position**—Saves the `x`, `y` and `z` positions of a coordinate system that is identified with a specific feature parameter to a file.

28.2 About Determine the Model Size

1. Click **EMX Assembly ▸ EMX Tools ▸**  **Model Size** or click **EMX Part Mode ▸ Model Size** to measure the model outline in part or assembly mode. The **Model Size** dialog box opens.
2. Click , available if the current model is an assembly, to select a model. The name of the selected model appears in the **Selected Model** box. If the current model is a part, the name is automatically set.
3. Click  to select a coordinate system. The name of the coordinate system appears in the **Selected CSYS** box.
4. Click  to run an analysis.
   - The bounding box outline and volume appears in the **Calculated Size** box.
   - The string format is `A x B x LG`.
   - The rotational symmetric part outline string is formatted as `DM x LG`.
   - The gray box displays the volume of the bounding box.
   - The number of digits is defined by the option `NR_DIGITS_COMMON`.
   - The preview box shows the top and side projections on which the analysis result is based.
5. Use **Convert outline result to** to convert the result into INCH or MM.
6. Click **OK** to save the model outline as a parameter and close the dialog box, or click **Cancel** to close the dialog box without saving the results.

28.3 To Determine the Model Size

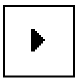
1. Select **EMX Assembly ► EMX Tools ► Model Size** .
2. Select a model from the assembly.
3. Click  and select `TUTOR_LOCATING_RING2.PRT` .By default the models origin is used to determine the x-, y- and z-axis of the outline determination. You can also click  and select any other coordinate system of the model if you want to define other defaults.
4. Click  and select the `ORIGIN_MOLDBASE` as reference coordinate system. The **Outline string** prints the result.

   NOTE:
   For rotational symmetric parts the format is "diameter x thickness", otherwise "length x width x height".
   Make sure the coordinate system has the correct orientation.

   The entry right beside size shows the volume.

   NOTE:
   The numbers of digits used for printing the result is defined by the **EMX Option** `NR_DIGITS_BOM`.
5. Close the **Model Size** dialog box with **OK**.

28.4 About Setting Accuracy

1. Set the following Creo Parametric configuration options:
   - `enable_absolute_accuracy` to `YES`.
   - `accuracy_lower_bound` to an appropriate value.
2. Click **EMX Assembly ► EMX Tools ► Set Accuracy** to set the accuracy of single components or an entire assembly. The **Set Accuracy** dialog box opens.
3. Enter a value in the **Accuracy** box.
4. Select or clear the **Absolute** or **Relative** check boxes.
5. Click  to select components. The system attempts to set component accuracy to the given value and type.
6. Click **Close** to close the dialog box.

28.5 To Set Accuracy

28.5.1 Set Accuracy of the whole Mold Base

EMX offers an easy way to set the accuracy for single parts or the whole assembly to the same value.

1. Open the **Set Accuracy** dialog boy with **EMX Assembly ▶ EMX Tools ▶ Set Accuracy**.
2. Set you desired value, either **absolute** or **relative**.
3. Click and select the mold base `TUTORIAL.ASM` in the model tree. EMX will set the accuracy for all components of the assembly.

Background information

The default for Accuracy Type and Accuracy Value can be predefined with the **EMX Options** `ACCURACY_TYPE` and `ACCURACY_VALUE`.

28.6 About Exporting CSYS Positions

1. Click **EMX Assembly ▶ EMX Tools ▶ Export CSYS Position** to save parameter-specific coordinate system positions and orientations to a file. The **Export CSYS Position** dialog box opens.
2. Enter a string parameter in the **Parameter** box.
3. Enter a file name in the **File Name** box.
4. Click to search all coordinate systems that include a `STRING` parameter with the given name. The orientation and the `x`, `y` and `z` positions are saved in the file specified by **File Name**.
5. Click **Close** to close the dialog box.

28.7 About Exporting Measurement Points

1. Click **EMX Part Mode ▶ EMX Tools ▶ Measuring Points** to place measuring points on Creo Parametric solid parts. The **Measuring Points** dialog box opens.
2. Click to select the **Reference CSYS** in the 3D model.
3. The **Measuring point list** table shows existing measuring points. Each line is represented by one point. Select the symbol in the **ID** column to create a 3D note. The **X**, **Y**, and **Z** columns show the measuring-point position on the solid part. The **(X)**, **(Y)**, and **(Z)** columns show the offset position of `0.5` * probe diameter normal to the solid face.
4. Enter the **Probe Diameter**. This represents the OMM machine ball diameter and will be used to calculate the offset between the original points on the solid and the required measurement position.

5. Enter the **Grid Width**. This value is used to calculate a grid of x and y positions when a new set of points is created on a solid part with .
6. Click to remove the selected entry from the point list.
7. Click to select a position on the model. The position and the related offset point are added as new line to the point list.
8. Click to save the point list to the file `QMM_FILENAME`.
9. Click **OK** to create or update a point in the assembly that represents the position of all measurement points.
10. Click **Cancel** to close the dialog box without saving.

28.8 About Setting the Technology Colors

1. Click **EMX Assembly ▸ Administrator Tools ▸** **Set Technology Colors**. The **Predefined Colors** dialog box opens.
2. Click the required item in the table.
3. Click to select one or more surfaces. Selection of several surfaces by defining a rectangular box is supported. The selected color is applied to the selected surfaces.
4. Click **Close** to close the dialog box.

28.9 To Set Up Technology Colors

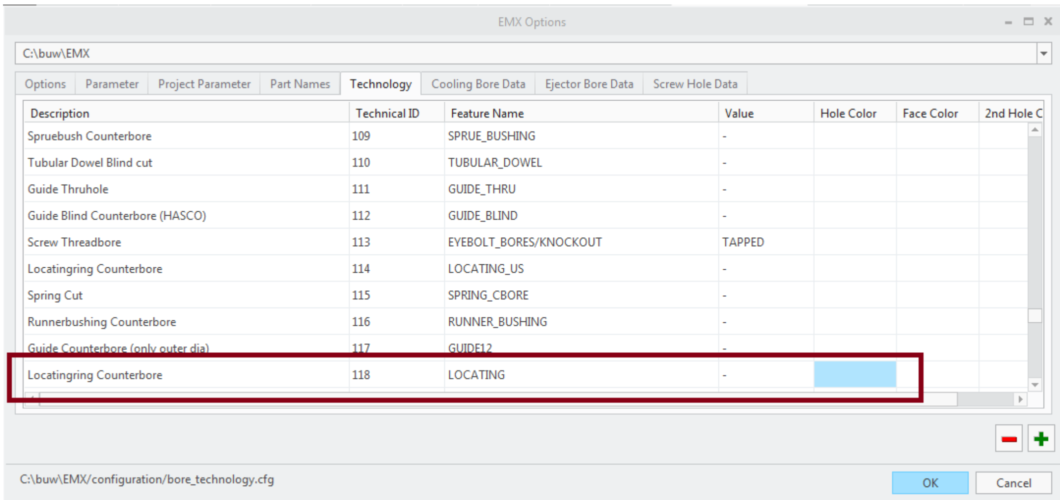
28.9.1 Define the colors in EMX Options

1. Open **EMX Options ▸ Technology**.

   This lists shows two types of features:

   Plate features are the suppressed features in the plate templates. Their name is written in capital letters in the **Feature Name** column. One example is the LOCATING-feature in the clamping plate templates. The value define in the Value column will be written to an additional feature parameter. Its name is defined in **EMX Option** `TECH_PARAM`. It will be written to the feature when `ADD_TECH_PARAM` is `YES`.

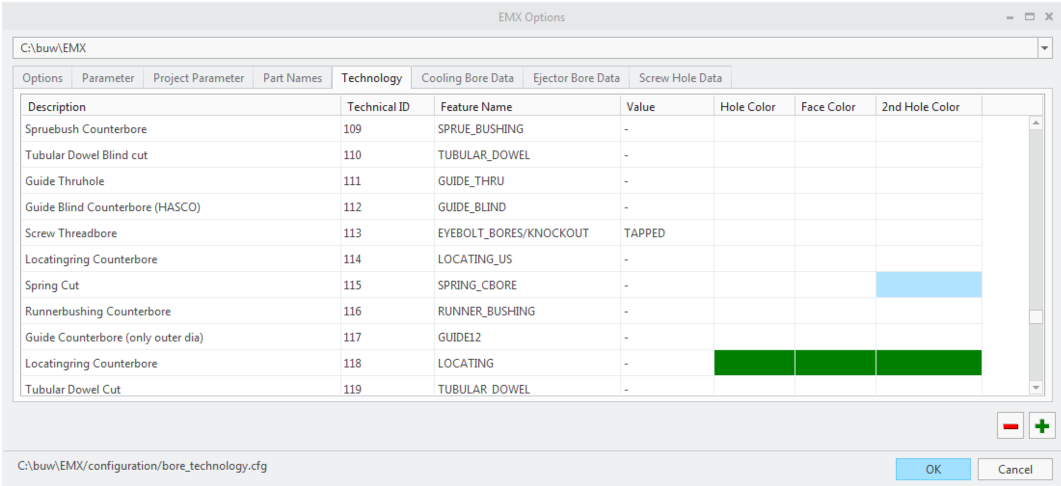
2. Double click in the **Hole Color** column.



If the **EMX Option** `ADVANCED_TECH_COLOR` is set to `YES`, a certain list of predefine colors is displayed.

If the option is set to `NO`, the regular Windows color selection dialog box is shown.

3. Select the desired color and click **Close**.
4. Repeat this steps in the columns **Face Color** and **2nd Hole Color**.



EMX Options

C:\buw\EMX

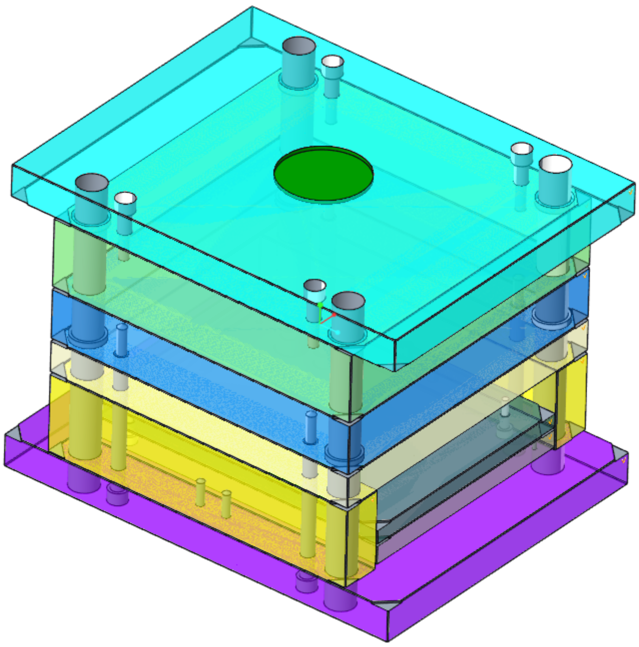
Options | Parameter | Project Parameter | Part Names | Technology | Cooling Bore Data | Ejector Bore Data | Screw Hole Data

Description	Technical ID	Feature Name	Value	Hole Color	Face Color	2nd Hole Color
Spruebush Counterbore	109	SPRUE_BUSHING	-			
Tubular Dowel Blind cut	110	TUBULAR_DOWEL	-			
Guide Thruhole	111	GUIDE_THRU	-			
Guide Blind Counterbore (HASCO)	112	GUIDE_BLIND	-			
Screw Threadbore	113	EYEBOLT_BORES/KNOCKOUT	TAPPED			
Locatingring Counterbore	114	LOCATING_US	-			
Spring Cut	115	SPRING_CBORE	-			
Runnerbushing Counterbore	116	RUNNER_BUSHING	-			
Guide Counterbore (only outer dia)	117	GUIDE12	-			
Locatingring Counterbore	118	LOCATING	-			
Tubular Dowel Cut	119	TUBULAR_DOWEL	-			

C:\buw\EMX/configuration/bore_technology.cfg

OK Cancel

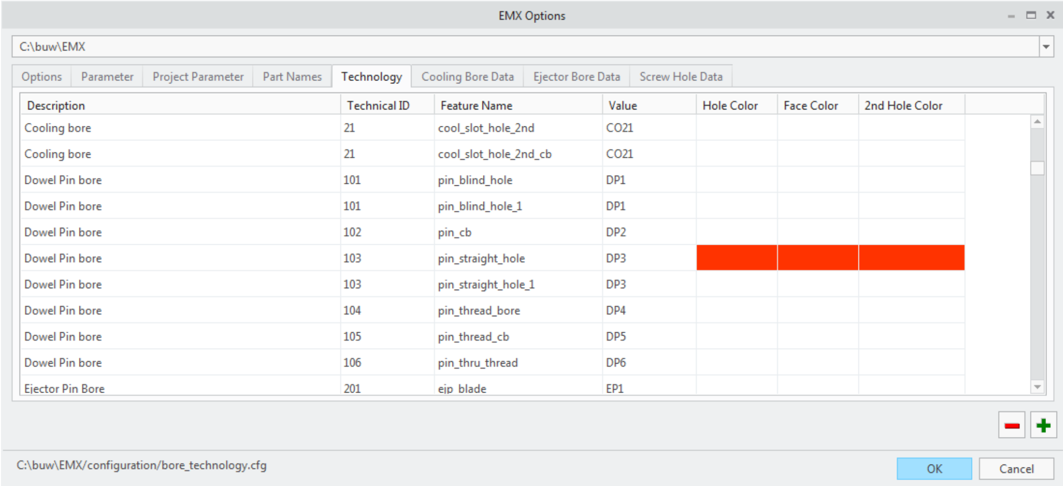
This color setting is used when **EMX Option** `ADD_TECH_COLOR` is set to `YES`. If you edit a plate now that has the feature LOCATING included, its faces are colored automatically with this color when closing the **Plate** dialog box.



Beside these plate features, all UDFs that create cutouts on the fly can be configured the same way. So ejector pin cutouts, screw holes etc. can be automatically colored in the predefined colors. EMX enables only three colors per cutout, the first cylindrical cut, the first plane face and the second cylindrical cut (i.e. for a counterbore).

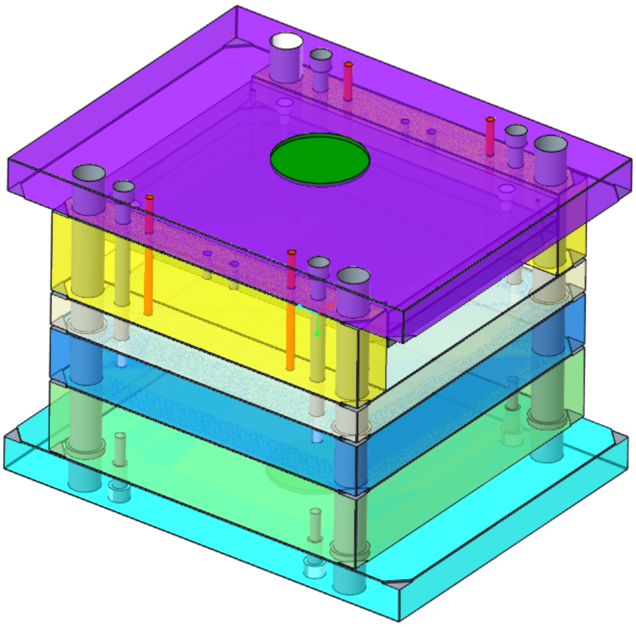
For each UDF in the components-folders an extra coloring can be defined.

1. Open **EMX Options** again and modify the colors for dowel-pin cutouts like this.



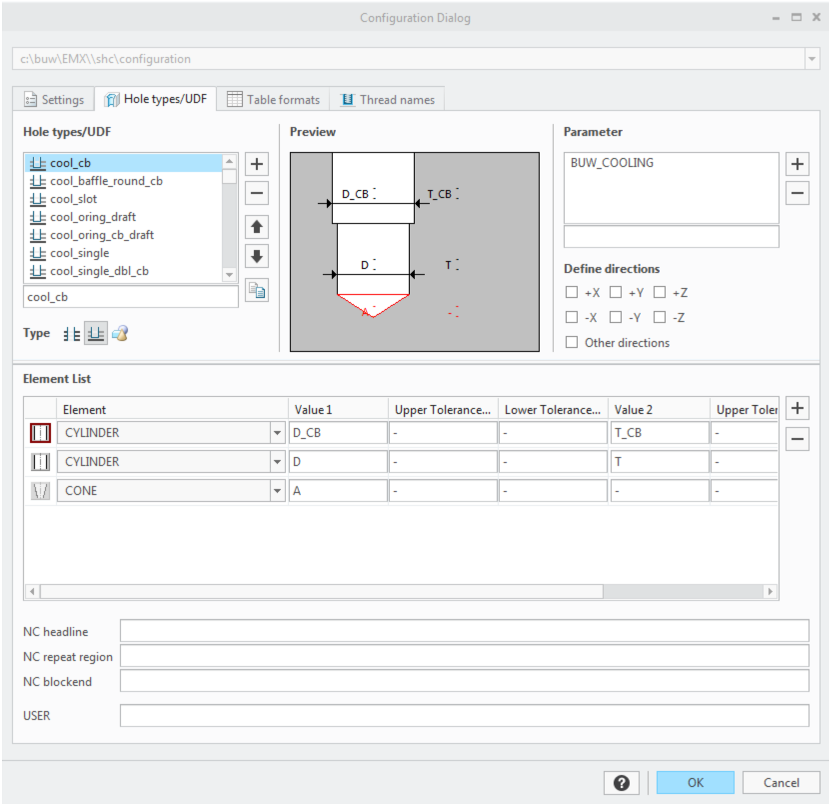
Description	Technical ID	Feature Name	Value	Hole Color	Face Color	2nd Hole Color
Cooling bore	21	cool_slot_hole_2nd	CO21			
Cooling bore	21	cool_slot_hole_2nd_cb	CO21			
Dowel Pin bore	101	pin_blind_hole	DP1			
Dowel Pin bore	101	pin_blind_hole_1	DP1			
Dowel Pin bore	102	pin_cb	DP2			
Dowel Pin bore	103	pin_straight_hole	DP3			
Dowel Pin bore	103	pin_straight_hole_1	DP3			
Dowel Pin bore	104	pin_thread_bore	DP4			
Dowel Pin bore	105	pin_thread_cb	DP5			
Dowel Pin bore	106	pin_thru_thread	DP6			
Ejector Pin Bore	201	ejp_blade	EP1			

If you display the Features in the modeltree its very easy to find the according UDF-names used by EMX. Here its the PIN_STRAIGHT_HOLE-feature. When adding new cutouts now, they will appear in yellow.



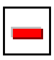
28.9.2 Using more flexible coloring with SmartHolechart.

Due to the limitation of EMX Technology color setting, users that need even more flexibility should use the coloring function from SmartHolechart. For each individual base feature of a complex bore an individual color can be set in the **Hole types/UDF** tab of the **Configuration Dialog**. See the SmartHolechart documentation for more details.



28.10 About Calculating The Clamping Force

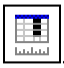
Use this **Calculate Clamping Force** to assess the compatibility of a mold base with a specific injection machine.

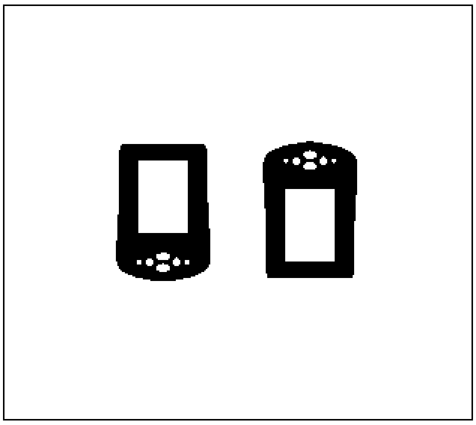
1. Click **EMX Assembly ► EMX Tools ► Calculate Clamping Force**. The **Clamping Force** dialog box opens. The machine defined in the **Mold Base Definition** dialog box is displayed with values for possible injection pressure and clamping force.
2. Enter a value for the injection pressure (less than the possible value). The top view of the mold base is displayed in the drawing area.
3. The color of the reference models and runner features changes to gray. The **Projected Area** and the **Clamping Force** are automatically calculated.
4. Click  and select additional features in the graphics window. All selected features appear in the **Additional Runner Features** list.
5. Click  to remove a selected line from the list.
6. Click **OK** to close the dialog box.

28.11 To Calculate the Clamping Force

28.11.1 Calculate Clamping Force for Insert Parts only

EMX offers the possibility to calculate the necessary clamping force depending on the projection of the reference model.

1. Select **EMX Assembly ▶ EMX Tools ▶ Calculate clamping force**
2. Insert the **Injection Pressure** value. If a machine was defined in **Mold Base Definition**, the **Machine Clamping Force** can be found in the top-right corner.
3. Start a Calculation with .

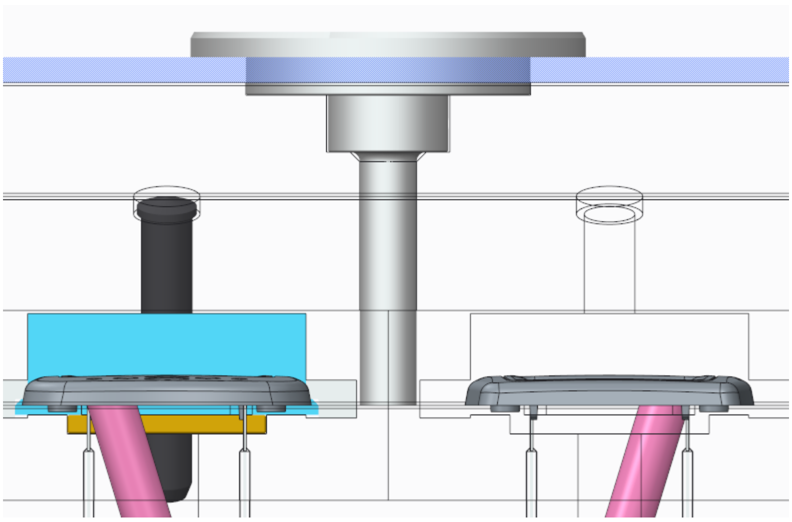


EMX displays the projection in the **Drawing Area**.

As a result you will get the **Projected Area** and the **Req. Clamping Force**.

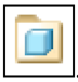
28.11.2 Assemble a Runner

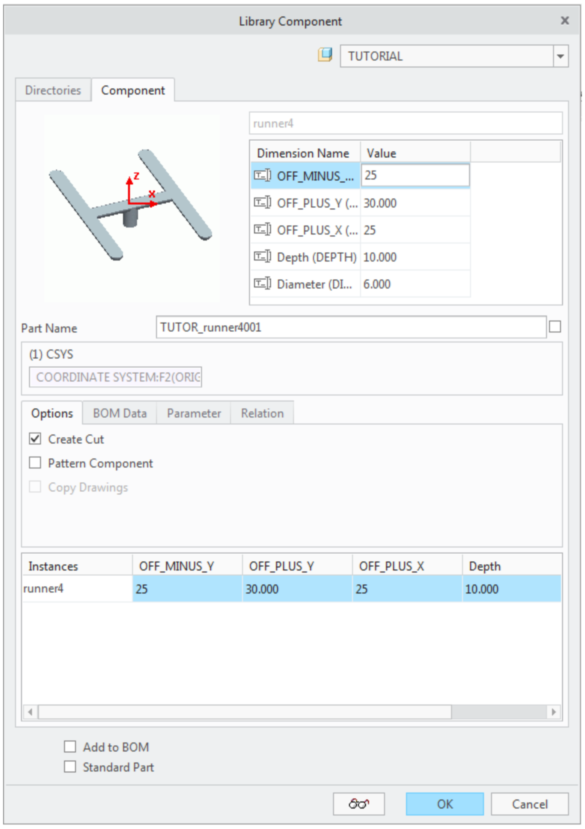
It is possible to add runners to the projection. Before you can assemble the runner make sure you have the correct sprue bushing assembled.



The Sprue Bushing should reach down to the separation plane of the mold base.

Runner can be found in the **Component Library**. You can define as many different runners as you want. Look for this in chapter To Add a Part to the Library on page 388

1. Select **EMX Components ► Library ► Assemble** .
2. Select the directory **runner** from the directory tree.
3. Select part **runner2** with Double-Click.
4. Set the **OFF_PLUS_Y** value to `25`.
5. Set the **OFF_MINUS_Y** value to `25`.
6. As assembly reference for runners **(1) CSYS** the coordinate system `ORIGIN_MOLDBASE` is predefined.

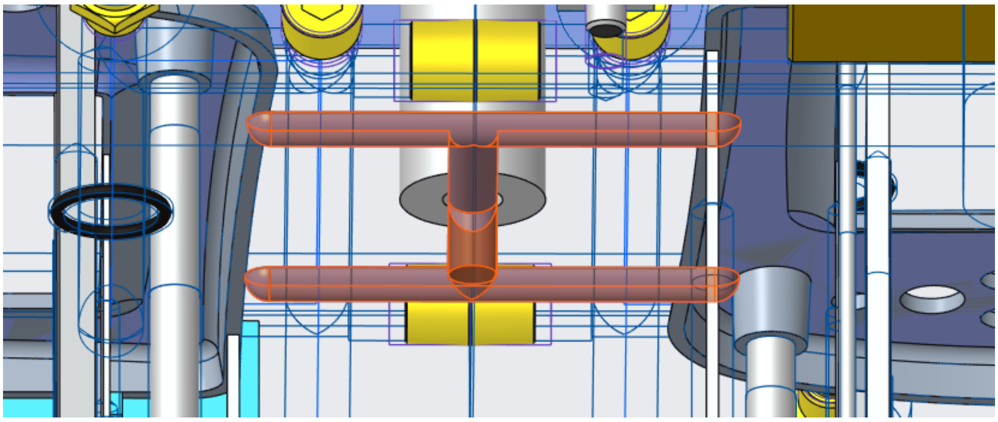


7. Leave the **Library** dialog box with **OK**.

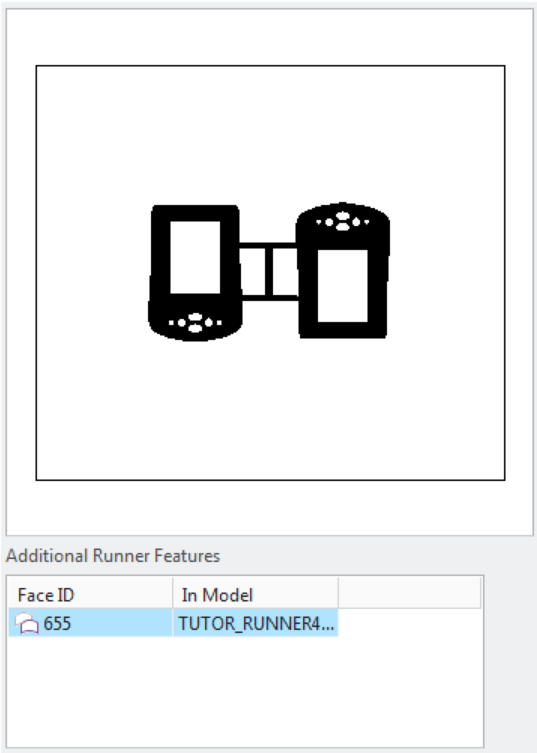
   The runner is assembled to the `TUTORIAL.ASM` and cut outs are added to the moving half insert parts `MH_PALM_1.PRT` and `MH_PALM_2.PRT`.

28.11.3 Calculate Clamping Force for Insert Parts and Runners

1. Click **EMX Assembly ► EMX Tools ► Calculate clamping force** .
2. Click  to add a runner to the projection.
3. Select the quilt of the previously created runner in the graphic window.



The result can be seen in the **Calculate Clamping Force** dialog box. The runner is displayed in the **Drawing Area** as well.



28.11.4 Calculate the MOULDING plastic part

While opening this Clamping Force dialog box, a second action can be run by EMX that is related to the Projected Area calculation.

1. Set the **EMX Option** `ENABLE_MOULDING_REPAIR` to `YES`.

   When creating a MOUDLING model with Creo Pro/TOOLDESIGN, all ejector pin cutouts added by EMX will generate a wrong geometric result in this model. The ejector pin cuts are added to the moulding volume. This cause troubles when calculate the required amount of plastic.

2. Click **EMX General ► EMX Tools ► Calculate clamping force** again.

Now you are prompted to select the MOULDING model that was designed previously. EMX will find all ejector pin cutouts and remove them from the MOULDING.

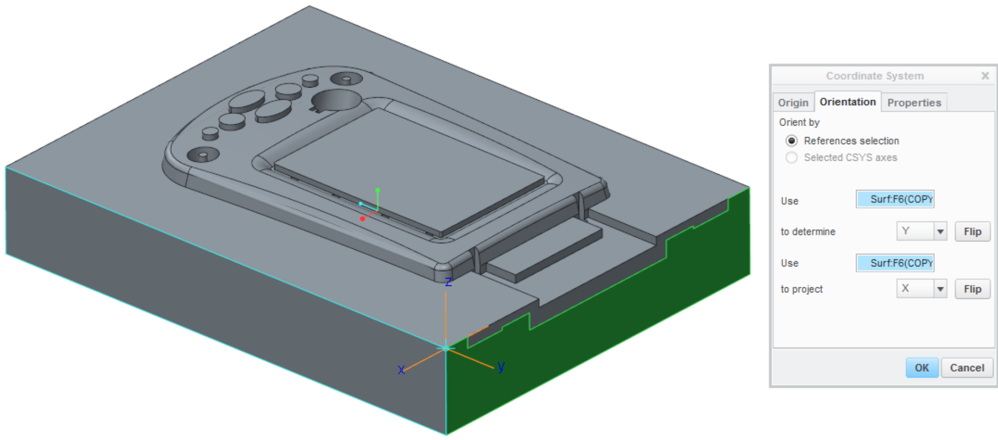
28.12 To Apply a Set of Measurement Points for 3D Measurement

28.12.1 Create Reference CSYS

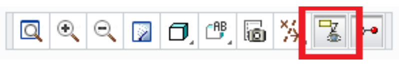
EMX supports the usage of 3D measurement for Quality Assurance. To define the required position for the probe ball of the measurement machine you need to know its center position normal to the face where the "real" point to be measured is located. EMX can calculate this position for you.

1. Open the mold base `TUTORIAL.ASM`.
2. Open the model `MH_PALM_1.PRT`.
3. Create a coordinate system as shown below.

   This coordinate system will be used as reference for the set of measurement points. The direction of the z-axis is important if you like to add a full set of points in one step.



4. Make sure **Show 3D Notes** is enabled.



28.12.2 Add Single Measurement Points

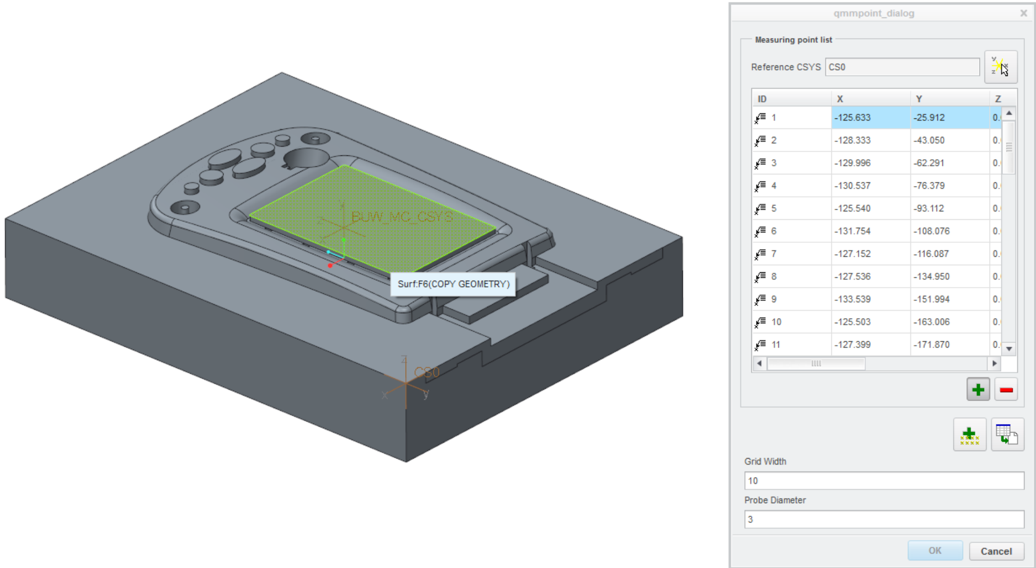
1. Select **EMX Part Mode ► EMX Tools ► Measurement Points** .

   The **Measuring Points** dialog opens.

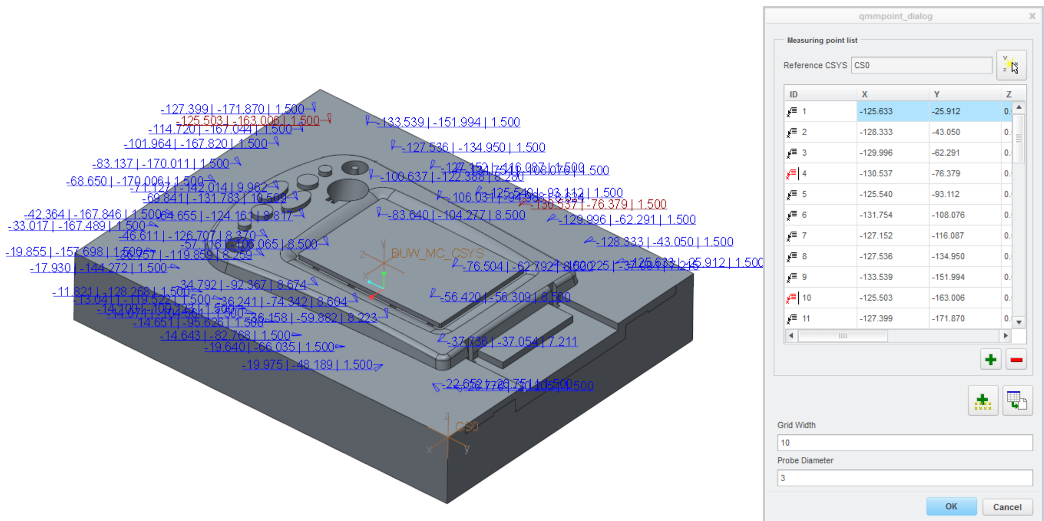
2. Select  to identify the reference coordinate system.
3. Select the previously created `CS0`.
4. Click  and pick the points to be measured on the models top face.With each pick a note is displayed with the real x,y,z position regarding `CS0` and the required probe ball

position. The position is also listed in the Point list of the **Measurement Points** dialog box.

NOTE:
Make sure the **Probe diameter** is set to the correct value as this is used to calculate the offset points.



5. Cancel the input loop with **MMB**.If you select a line from the points list its related note gets highlighted in the graphics window.



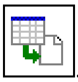
6. Close the **Measurement Points** dialog box with **OK**.A point feature is added to the current model MH_PALM_1. Each entry of the **Measurement points** list is represented by an extra point.

28.12.3 Export List of Points

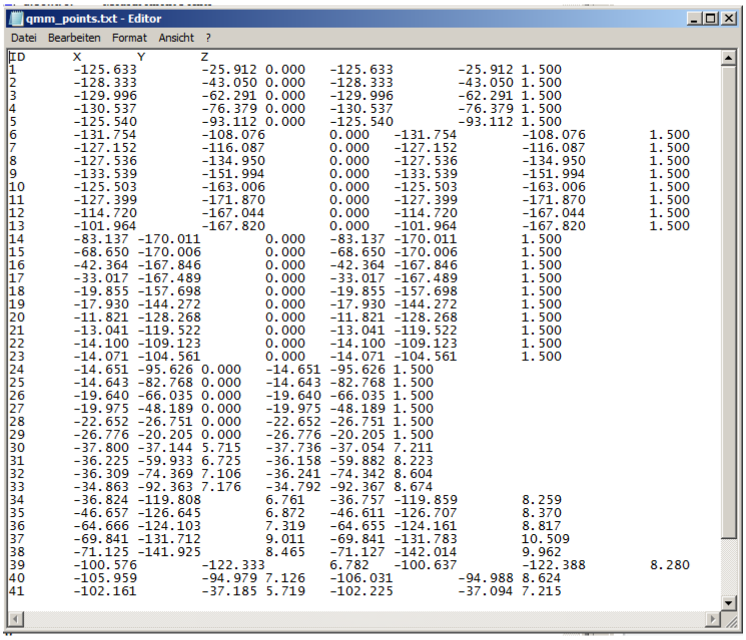
1. Select **EMX Part Mode ► EMX Tools ► Measurement Points** .

EMX will read the stored positions of all points again and displays them in the **Measurement points** list.

2. Select any line to highlight the according point in 3D.

3. Click . The content of the list is written to file specified in **EMX Option** `QMM_FILENAME`.

   By default the file will be stored in the working directory.



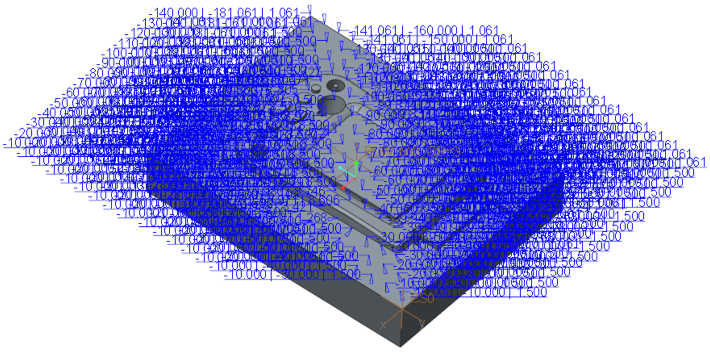

```
ID        X        Y        Z
1 -125.633 -25.912 0.000 -125.633 -25.912 1.500
2 -128.333 -43.050 0.000 -128.333 -43.050 1.500
3 -129.996 -62.291 0.000 -129.996 -62.291 1.500
4 -130.537 -76.379 0.000 -130.537 -76.379 1.500
5 -125.540 -93.112 0.000 -125.540 -93.112 1.500
6 -131.754 -108.076 0.000 -131.754 -108.076 1.500
7 -127.152 -116.087 0.000 -127.152 -116.087 1.500
8 -127.536 -134.950 0.000 -127.536 -134.950 1.500
9 -133.539 -151.994 0.000 -133.539 -151.994 1.500
10 -125.503 -163.006 0.000 -125.503 -163.006 1.500
11 -127.399 -171.870 0.000 -127.399 -171.870 1.500
12 -114.720 -167.044 0.000 -114.720 -167.044 1.500
13 -101.964 -167.820 0.000 -101.964 -167.820 1.500
14 -83.137 -170.011 0.000 -83.137 -170.011 1.500
15 -68.650 -170.006 0.000 -68.650 -170.006 1.500
16 -42.364 -167.846 0.000 -42.364 -167.846 1.500
17 -33.017 -167.489 0.000 -33.017 -167.489 1.500
18 -19.855 -157.698 0.000 -19.855 -157.698 1.500
19 -17.930 -144.272 0.000 -17.930 -144.272 1.500
20 -11.821 -128.268 0.000 -11.821 -128.268 1.500
21 -13.041 -119.522 0.000 -13.041 -119.522 1.500
22 -14.100 -109.123 0.000 -14.100 -109.123 1.500
23 -14.071 -104.561 0.000 -14.071 -104.561 1.500
24 -14.651 -95.626 0.000 -14.651 -95.626 1.500
25 -14.643 -82.768 0.000 -14.643 -82.768 1.500
26 -19.640 -66.035 0.000 -19.640 -66.035 1.500
27 -19.975 -48.189 0.000 -19.975 -48.189 1.500
28 -22.652 -26.751 0.000 -22.652 -26.751 1.500
29 -26.776 -20.205 0.000 -26.776 -20.205 1.500
30 -37.800 -37.144 5.715 -37.736 -37.054 7.211
31 -36.225 -59.933 6.725 -36.158 -59.882 8.223
32 -36.309 -74.369 7.106 -36.241 -74.342 8.604
33 -34.863 -92.363 7.176 -34.792 -92.367 8.674
34 -36.824 -119.808 6.761 -36.757 -119.859 8.259
35 -46.657 -126.645 6.872 -46.611 -126.707 8.370
36 -64.666 -124.103 7.319 -64.655 -124.161 8.817
37 -69.841 -131.712 9.011 -69.841 -131.783 10.509
38 -71.125 -141.925 8.465 -71.127 -142.014 9.962
39 -100.576 -122.333 6.782 -100.637 -122.388 8.280
40 -105.959 -94.979 7.126 -106.031 -94.988 8.624
41 -102.161 -37.185 5.719 -102.225 -37.094 7.215
```

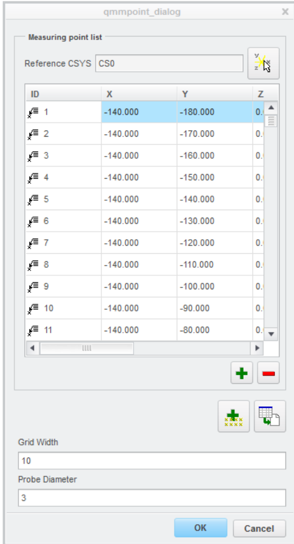
28.12.4 Create a Grid of Measurement Points

1. Delete the previously created point feature and the annotations.
2. Select **EMX Part Mode ► EMX Tools ► Measurement Points** again.

3. Set the **Grid Width** of the point pattern to `10`.

4. Start the creation process of the points pattern with A set of **Measurement Points** with a constant distance of `10` mm is added to the `MH_PALM_1.PRT`.





28.13 To Export EDM Positions

28.13.1 Set the search options

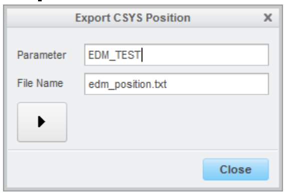
EMX offers a function to print an easy form of burnsheet where position and orientation of electrodes can be found.

1. Select **EMX Tools ► Export CSYS Position**.
2. Set **EMX Options** `EMX_CSYS_PARAMETER` to`EDM_TEST` .
3. Set **EMX Options** `EDM_CSYS_RESULT_FILE` to`edm_position.txt` .

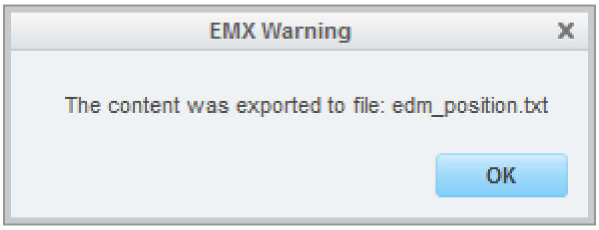
   The result with all coordinate system positions will be stored in this file.

28.13.2 Run the Export

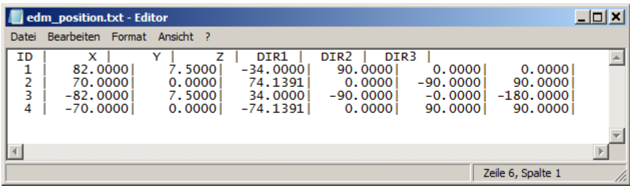
1. To demonstrate the functionality, open the assembly `TUTOR_INSERT.ASM`.
2. Add an INTEGER-parameter `EDM_TEST` to the CS0–coordinate systems in the models `ASM_PALM_REF` and `SLIDER_PALM`. A very convenient way to add a parameter to a lot of features is this:
3. Show it as an extra column in the model tree. Then click in this column, and you can easily set up the desired Parameter. EMX will only check if a parameter with the name `EDM_TEST` exists.

4. Start the function with **EMX Tools ► Export CSYS Position**. The predefined Options are 

set as default but can be overwritten.
5. Click the Run-button.EMX search through the entire assembly for coordinate systems marked with the parameter EDM_TEST. Once this is done, this dialog shows up

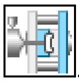
   The resulting file in the current working directory looks like this

ID	X	Y	Z	DIR1	DIR2	DIR3
1	82.0000	7.5000	-34.0000	90.0000	0.0000	0.0000
2	70.0000	0.0000	74.1391	0.0000	-90.0000	90.0000
3	-82.0000	7.5000	34.0000	-90.0000	-0.0000	-180.0000
4	-70.0000	0.0000	-74.1391	0.0000	90.0000	90.0000

28.14 About he Machine Representation

The EMX machine data is hosted in <install_emx90>/components/<unit>/machines subdirectories.

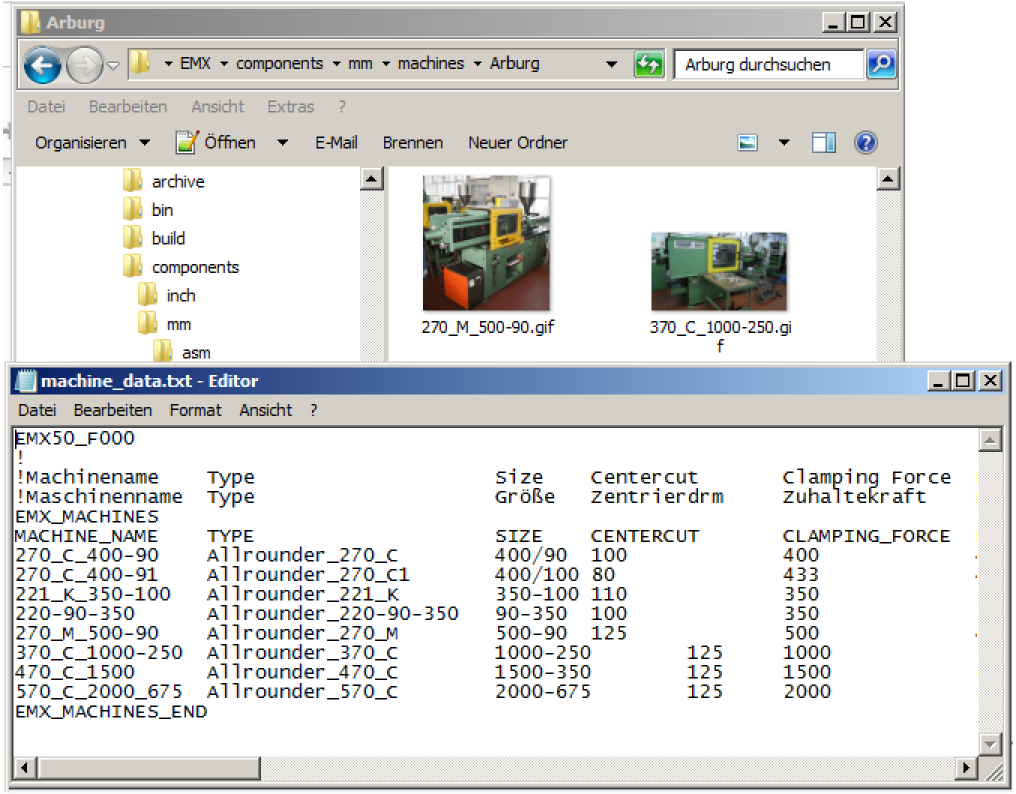
1. Open the **Mold Base Definition** dialog box.
2. In **Mold Base Definition** dialog box click .

   The **Machine** dialog box is opened.

   If no file <install_emx50>/components/<unit>/machines/supplier.txt exist, all sub-directories will be listed in the supplier list of the **Machine** dialog box.

   All data hosted in the machine_data.txt file is provided in the GUI.

The **MACHINE_NAME** can be selected from the machine list. EMX will display any gif-image that has the name of the machine in the **Machine** dialog box (i.e. 270_c_400-90.gif).



All further columns are listed in the dimension list so you can select a machine by any of its dimensions.

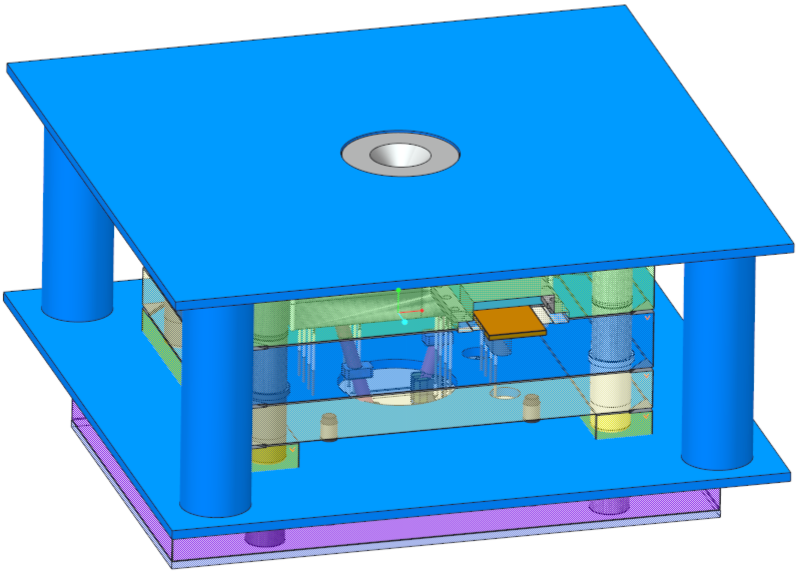
NOTE:
The columns `HOLM_xxx` and `KNOCKOUT_xxx` columns have a special meaning as they are used to calculate the machine bar pattern and the knockout pattern.

3. Select Machine **Supplier** `Arburg` **Type** `470_C_1500`.
4. Click **Show machine** in top right corner. The preview of the chosen machine is shown in **Mold Base Definition** dialog box:

   a. In the side view the minimum and maximum machine opening is displayed using three plates.

   b. In the top view the machine bar pattern and the knockout pattern are displayed.

5. Close the **Machine** dialog box with **OK**.The layer with the model `TUTOR_MACHINE.PRT` is displayed in `TUTORIAL.ASM` so you can check if the machine is big enough to host your mold base.



Notes:

29

Configuration Options - Admin Tutorial

About Configuration Options

29.1 About the EMX Options Dialog Box

Click **EMX Assembly ▸** **Options** to open the **EMX Options** dialog box. Use the tabs to set Creo Expert Moldbase Extension parameters and defaults. Any changes made to parameters using this dialog box are immediately applied. The following tabs are available:

- **Options**—Edit common Creo Expert Moldbase Extension settings.
- **Parameter**—Edit common BOM parameter definitions.
- **Project Parameter**—Edit project-specific BOM parameter definitions.
- **Part Names**—Edit predefined model and BOM names, ordering numbers and other settings.
- **Technology**—Edit Creo Expert Moldbase Extension cutout type colors and parameters.
- **Cooling Bore Data**—Edit cooling bore dimension defaults.
- **Ejector Bore Data**—Edit ejector pin bore dimension defaults.
- **Screw Hole Data**—Edit screw bore dimension defaults.

All configuration files are located in <installdir>/configuration directory. If the EMX_USER_CONFIG_PATH environment variable exists, the system automatically looks for it in the path defined by EMX_USER_CONFIG_PATH/configuration instead.

29.2 To Set Creo Expert Moldbase Extension Configuration Options

1. Click the **Options** tab in the **EMX Options** dialog box to access Creo Expert Moldbase Extension configuration options. When you select an option from the list, its name and value are copied to the **Option** and **Value** boxes.
2. Select or type a value for the option.

29.3 Common Configuration Options

ADD_COOLING_REF_IN_PART

NO: Assemble Refgroups for cooling components in the chosen subassembly.

YES: Assemble Refgroups for cooling components in the part that is owner of the placement surface.

Default: NO.

ARCHIV_PATH

Path to the archive used to create component instances. Default: <installpath>/archive.

COMPANY_ID

Special company identifier used to activate certain hidden options. Default: -.

CUT_PLATES_ONLY

NO: All components that interfere with a new component will be cut using the CUTQUILT.

YES: Only EMX plates that interfere with a new component will be cut using the CUTQUILT.

Default: NO.

DEFAULT_EMXNAME

Default value for the EMX Projectname. Default: 123.

DEFAULT_PREFIX

Default value for the project prefix. Default: 123.

DEFAULT_POSTFIX

Default value for the project postfix. Press ENTER to hide the input panel for the postfix. Default: 456.

DEFAULT_WATERLINE_DIA

The default D5-diameter for all cooling bores. Set it to 0 (zero) to ignore this option.

Default: 8.

DELETE_FILES_AFTER_DELETE

NO: The part files are left on the disc after parts have been deleted in Creo Parametric.

YES: When deleting a part the respective part-files will be erased from the hard-disc.

Default: NO.

DELETE_FILES_AFTER_DISASSEMBLE

NO: The part files are left on the disc after parts have been disassembled in Creo.

YES: During part-disassembly also the files are erased from the hard-disc.

Default: NO.

ENABLE_MULTI_EJP

Sets the default value for the check box, **Individual ejector models on each point** in the **Ejector Pin** dialog box.

NO: No Action.

YES: When defining ejectors on a multiple point feature with different height, place individual ejectors on all points.

Default: NO.

ENABLE_SUBASM_PREFIX

NO: Subassemblies must have same Project Prefix like the main assembly.

YES: Allow the user to define individual prefixes for subassemblies.

Default: NO.

LIBRARY_PATH

Root path to the EMX library. Default: <installpath>/components.

LIBRARY_THUMB_SIZE

Size in pixels of the thumbnail pictures in the **Library Component** dialog box.

Default: 100.

MATERIAL_PATH

Root path to the directory where Creo Parametric Material files are located.

Default: -.

PDM_SYSTEM

0: No PDM system.

1: Retrieve standard parts from the SEARCH_PATH variable.

2: Retrieve standard parts from Windchill.

Default: 0.

PRE_HIGHLIGHT_BEFORE_DELETE

NO: Do not highlight dependent components before delete-prompt - faster than using YES.

YES: Highlight all features and components that are dependent before delete prompt. Can be slow.

Default: YES.

PROMPT_FOR_CUTTING_PARTS

NO: All components that interfere with a new component will be cut using the CUTQUILT.

YES: Creo Expert Moldbase Extension displays a dialog box for the user to select which components should be cut.

Default: NO.

SAVE_LOCAL_PARAM_IN_ASM

NO: All project parameters are stored in the file project_parameter.cfgin the local working directory (4.1 behavior).

YES: Project parameters are stored in the Creo Expert Moldbase Extension main assembly. It is better to use this for PDM-systems.

Default: NO.

SAVE_SIZE_OF_PART_IN_EMPTY_ORDNUMBER

NO: No Action.

YES: The calculated outline of parts is written into order number in case the order number string is empty.

Default: NO.

WINDCHILL_AUTONUMBER

NO: Parts and assemblies are renamed with partnames.cfg rules

YES: Parts and assemblies are renamed with auto number from Windchill in case Windchill is active.

Default: NO.

29.4 Assembly Options

ALLOW_SIZE_EDIT

NO: No Action.

YES: In the **Mold Base Definition** dialog box, edit the size manually. You can set up the mold base without restriction.

NOTE:
All pattern values must be defined manually

Default: NO.

ASSEMBLY_UNIT

0: Sets mm (millimeters) as default unit in the **Project** dialog box.

1: Sets inch (inches) as default unit in the **Project** dialog box.

Default: 0.

BALLOON_ARROW_END_TYPE

Set to AUTOMATIC, ARROWHEAD, DOT, FILLED_DOT ,DOUBLE_ARROW, SLASH, INTEGRAL, BOX, FILLED_BOX, NONE, TARGET, HALF_ARROW, TRIANGLE

Default: FILLED_DOT

BALLOON_SIZE

Absolute size of BOM-balloon

Default: 1

CREATE_ASM_DRAWING

NO: Do not create an assembly drawing.

YES: Create an assembly drawing.

Default: YES.

CREATE_ASM_REPORT

NO: Do not create an assembly report.

YES: Create an assembly report.

Default: YES.

CREATE_GROUP_LAYER

NO: No action.

YES: Create a group layer and place each part-layers on it if the group layer is not preceded by a hyphen (-) in configuration/partnames.cfg.

Default: NO

CREATE_LAYER_PER_PART

NO: No action.

YES: Create a layer for each part if the layer name is not preceded by a hyphen (-) in configuration/partnames.cfg.

Default: NO

DELAY_MOLDBASE_REGEN

NO: Creo Expert Moldbase Extension will immediately regenerate the mold base after any modification in the mold base definition.

YES: User must regenerate the EMX assembly manually in the mold base definition.

Default: NO.

EJP_DESIGNER_DEFAULT_COMPONENT

Instance name for the component to use as the default in the Ejector Pin Designer.

Default: mm|meusburger|E1725_2_8_0_5_80.

EMX_HIDE_ASSEMBLIES_IN_BOM

`NO`: Does not automatically hide assemblies in the BOM.

`YES`: Always hides assemblies in the BOM.

Default: `NO`.

KEEP_SIZE_DEF_SETTING

`NO`: Clear the **Keep size and model data** check box in the **Load EMX Assembly** dialog box.

`YES`: Select the **Keep size and model data** check box in the **Load EMX Assembly** dialog box.

Default: `NO`

MATFILE_EXTENSION

`mat`: Use `*.mat` files from MATERIAL_PATH.

`mtl`: Use `*.mtl` files from MATERIAL_PATH.

— : Do not consider content of MATERIAL_PATH.

Default: `mat`

NAME_COMP_DISPLAY

Name of a style where all plates are set to wireframe. Insert '–' if EMX should not create this style.

Default: `VIS0001`.

NAME_FIX_HALF

Names the side layer and simplified representation of the fixed half.

Default: `01_FIX_HALF`.

NAME_MOVING_HALF

Names the side layer and simplified representation of the moving half.

Default: `01_MOVING_HALF`.

NAME_SUBASM_COMP

Name of the automated created subassembly for all components. Use '–' if you do not want to create this subassembly.

Default: `-`.

NAME_SUBASM_COOL

Name of the automated created subassembly for waterline components in the assembly `NAME_SUBASM_COMP`. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_DOWEL

Name of the automated created subassembly for dowel pins in the assembly NAME_SUBASM_COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_EJECTOR

Name of the automated created subassembly for ejector pins in the assembly NAME_SUBASM_COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_EQUIPMENT

Name of the automated created subassembly for equipment in the assembly NAME_SUBASM_COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_GUIDE

Name of the automated created subassembly for guide components in the assembly NAME_SUBASM_COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_LIB

Name of the automated created subassembly for library_components in the assembly NAME_SUBASM_COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_SCREW

Name of the automated created subassembly for screws in the assembly NAME_SUBASM_COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_STOP

Name of the automated created subassembly for stop disc and stop pins in the assembly NAME_SUBASM_COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_SUPPORT

Name of the automated created subassembly for support pillars in the assembly NAME_ SUBASM_COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_SLIDER

Name of the automated created subassembly for sliders in the assembly NAME_SUBASM_ COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_LIFTER

Name of the automated created subassembly for lifters in the assembly NAME_SUBASM_ COMP. Use '–' if you do not want to create this subassembly.

Default: –.

NAME_SUBASM_LATCHLOCK

Name of the automated created subassembly latchlocks in the assembly NAME_SUBASM_ COMP. Use '–' if you do not want to create this subassembly.

Default: –.

CONFIG_ASM_LABEL_PARAM

Name of a parameter which can be defined at the EMX subassemblies. In the parameter exist, its value appears in the EMX subassembly lists instead of the subassembly name.

Default: – (hyphen).

PREVIEW_VIS_MODE

disabled: Do not modify display and style in preview mode for components.

shade: Set display to shade when preview a component.

wireframe: Set display to wireframe when preview a component.

hiddenvis: Set display to hiddenvis when preview a component.

hiddeninvis: Set display to hiddeninvis when preview a component.

shadewithedges: Set display to shadewithedges when preview a component.

shadewithreflect: Set display to shadewithreflect when preview a component.

Default: hiddenvis

UPDATE_ADDITIONAL_COMPS

NO: No action.

YES: If a plate is modified, added or removed, ejector pins and support pillars in the 3D mold base are updated.

Default: NO.

UPDATE_CUTS_IN_REDEFINE

NO: When redefine a library component EMX only updates existing cuts

YES: When redefine a library component always remove existing cuts and place new ones

Default: NO

UPDATE_MOLDBASE_COMPS

NO: No action.

YES: If a plate is modified, added or removed mold base components are updated.

Default: NO.

CONFIG_USE_LAYER_FOR_SIDE_VIEWS

NO:

YES:

Default: NO.

USE_P_PLATE_STACK

NO: Cutouts for mold base screws and guide components are defined as plate features.

YES: None of the mold base plates has predefined cuts for screws and guide components. These cuts are generated by the components.

Default: NO.

USE_RULEBASED_LAYER_CONCEPT

NO: Use default EMX layers for new EMX Projects.

YES: Use rule-based layers and combined states defined in template files `configuration\templates\rule_layers` for new EMX Projects.

Default: NO.

WATERLINE_DESIGNER_DEFAULT_COMPONENT

Instance name for the component to use as the default in the Waterline Designer.

Default: `mm|meusburger|E2074_8_10`.

WATERLINE_DESIGNER_SHOW_Z_POS_IN_WORKMDL_COORDS

YES: Show drawing height in coordinates of current working model with Waterline Designer.

NO: Show drawing height in ORIGIN_MOLDBASE coordinates.

Default: YES

WATERLINE_DESIGNER_DIA_LABEL

Defines all diameter values available for selection in the diameter column of the **Cooling Circuits** table. A valid option format is dia1|dia2|dia3.

Default: 8|10|12.

WATERLINE_DESIGNER_OVERMEASURE

The default overmeasure up to the next cut for all cooling bores.

Default: 3.

WATERLINE_DESIGNER_XSEC_CREATE

YES: Automatically creates a cross section through sketch planes with the Waterline Designer.

NO: Does not a create a cross section through sketch planes with the Waterline Designer.

Default: YES.

29.5 Component Options

ADD_TECH_COLOR

NO: No action.

YES: Set technology color in EMX UDFs.

Default: YES.

ADD_TECH_PARAM

NO: No action.

YES: Add technology parameter in EMX UDFs.

Default: YES.

AUTO_ASSEMBLE_SCREWSPINS

NO: No Action.

YES: Assemble predefined components on a library part automatically.

Default: NO.

AUTO_REASSEMBLE

NO: No action.

YES: Creo Expert Moldbase Extension will prompt the placement references for a cooling component in a loop after the first placement. Stop component placement with Middle Mouse Button.

Default: NO.

AUTO_REASSEMBLE_ALL_TYPES

NO: No action.

YES: Creo Expert Moldbase Extension will prompt the placement references for any component in a loop after the first placement. Stop component placement with Middle Mouse Button.

Default: NO.

AUTO_SAVE_LAST_COMP

NO: No action.

YES: Creo Expert Moldbase Extension will save the parameters of the last component to reuse it when open the component dialog box again.

Default: NO.

COOLING_ADDITIONAL_BORE_DEPTH_RULE

Rule for calculating the additional bore depth rule to apply to all cooling components defined on curves such as 0.5*BORE_REF_DIA.

Default: — (hyphen)

EJP_NOTE_PRE_STRING

Text to add before the real ejector pin length in the &remark-parameter.

Default: -.

EJP_NOTE_POST_STRING

Text to add after the real ejector pin length in the &remark-parameter.

Default: -.

EMX_CHECK_DWG_TYPES

List all component types such as guide|plates|slider|library, that have the **Copy Drawings** check box selected by default, when the drawing exists in same folder. Set to - to disable drawings.

Default: ejector|slider|lifter|latchlock.

EMX_TOL_MODE

Tolerance display of EMX components and UDFs.

-1: Do not modify tolerance display of EMX components and user-defined features (UDFs).

0: Set all dimensions to nominal.

1: Set all dimensions to plus-minus.

2: Set all dimensions to limits.

3: Set all dimensions to plus-minus-sym.

Default: 0.

EMX_NO_CUT_PARAM

Name of a INTEGER-Parameter. If this Parameter exists at a part with value 1,
Creo Expert Moldbase Extension will not cut this part.

Default: “-”.

EXCL_SIZE_CALC_PARAM

Name of an integer parameter that directs EMX not to calculate size from the owner model. Use - to ignore this functionality.

Default: -.

INCREMENT_POS

Increment that should be used for point offset when creating points picked by mouse.

Default: 0.01.

MIN_SCREW_LENGTH_RULE

Rule for calculating the minimum screw length once the references have been chosen.

Default: 1.5*diameter+REF1+OFFSET

NOM_DIM_DISPLAY

0: Show dimension Symbolname + description in the Nominal Dimension list of the component dialog box

1: Show only dimension Symbolname in the Nominal Dimension list of the component dialog box

2: Show only description in the Nominal Dimension list of the component dialog box

Default: 0

NR_DIGITS_BOM

Set the number of decimal places to format the SIZE and ORDER NUMBER Parameter String. Set this parameter to 0 to use NR_DIGITS_COMMON for all formatting.

Default: 3.

NR_DIGITS_COMMON

Number of decimal places for all dimensions in EMX components.

Default: 3.

PATTERN_QUILT_CUTS

NO: Run interference check and add cuts for patterned components individually.

YES: Run interference check for patterned components only one time and create a reference pattern for the cuts.

Default: YES.

PARAM_PLATE_LENGTH

Parameter name to save the plate length to the plate.

Default: –

PARAM_PLATE_WIDTH

Parameter name to save the plate width to the plate.

Default: –

PARAM_PLATE_THICKNESS

Parameter name to save the plate thickness to the plate.

Default: –

PARAM_VOLUME

Parameter name to save the volume of the Model Size dialog box.

Default: –

SAVE_DRAWINGS

NO: No action.

YES: Save drawings of all EMX components automatically.

Default: NO.

SET_EMX_DTL

NO: No action.

YES: Set drawing options /configuration /symbols_dtl/ <unit>emx.dtl for each drawing.

Default: NO.

SHOW_PRYSLOT

NO: Hide PRYSLOT-features in all plates in any case.

YES: Show PRYSLOT-features in all plates even when they are defined in a different way in the feature_data.txt file.

Default: YES.

USE_INSTANCE_IN_REFNAME

NO: Rename references using the format "<component-type>_<id>".

YES: Rename references using the format "<component-type>_<instancename>_<id>".

Default: NO.

USE_ONE_MDL_RAIL

Sets the default of the **Both rails in one model** check box, in the **Plate** dialog box.

NO: Rails are represented by two individual models.

YES: Both rails use the same Creo Parametric model.

Default: NO.

USE_SHORT_NAME_TRIM_EJP

NO: Add the geomitem-id to the name of multiple, trimmed ejector pins.

YES: Use a short id for the name of multiple, trimmed ejector pins

Default: NO.

29.6 Parameter Options

ADD_LOCALIZED_BOM_NAME1

-: Do not add additional localized ordering names in partnames.cfg.
chinese_cn: Add traditional Chinese ordering names in partnames.cfg
chinese_tw: Add modern Chinese ordering names in partnames.cfg
french: Add French ordering names in partnames.cfg
german: Add German ordering names in partnames.cfg
italian: Add Italian ordering names in partnames.cfg
japanese: Add Japanese ordering names in partnames.cfg
korean: Add Korean ordering names in partnames.cfg
spanish: Add Spanish ordering names in partnames.cfg
usascii: Add English ordering names in partnames.cfg.
Default: -.

ADD_LOCALIZED_BOM_NAME2

-: Do not add additional localized ordering names in partnames.cfg.

chinese_cn: Add traditional Chinese ordering names in partnames.cfg
chinese_tw: Add modern Chinese ordering names in partnames.cfg
french: Add French ordering names in partnames.cfg
german: Add German ordering names in partnames.cfg
italian: Add Italian ordering names in partnames.cfg
japanese: Add Japanese ordering names in partnames.cfg
korean: Add Korean ordering names in partnames.cfg
spanish: Add Spanish ordering names in partnames.cfg
usascii: Add English ordering names in partnames.cfg.
Default: -.

ADD_LOCALIZED_BOM_NAME3

-: Do not add additional localized ordering names in partnames.cfg.
chinese_cn: Add traditional Chinese ordering names in partnames.cfg
chinese_tw: Add modern Chinese ordering names in partnames.cfg
french: Add French ordering names in partnames.cfg
german: Add German ordering names in partnames.cfg
italian: Add Italian ordering names in partnames.cfg
japanese: Add Japanese ordering names in partnames.cfg
korean: Add Korean ordering names in partnames.cfg
spanish: Add Spanish ordering names in partnames.cfg
usascii: Add English ordering names in partnames.cfg.
Default: -.

ADD_LOCALIZED_BOM_NAME4

-: Do not add additional localized ordering names in partnames.cfg.
chinese_cn: Add traditional Chinese ordering names in partnames.cfg
chinese_tw: Add modern Chinese ordering names in partnames.cfg
french: Add French ordering names in partnames.cfg
german: Add German ordering names in partnames.cfg
italian: Add Italian ordering names in partnames.cfg
japanese: Add Japanese ordering names in partnames.cfg
korean: Add Korean ordering names in partnames.cfg
spanish: Add Spanish ordering names in partnames.cfg
usascii: Add English ordering names in partnames.cfg.
Default: -.

ADD_LOCALIZED_BOM_NAME5

-: Do not add additional localized ordering names in partnames.cfg.
chinese_cn: Add traditional Chinese ordering names in partnames.cfg
chinese_tw: Add modern Chinese ordering names in partnames.cfg
french: Add French ordering names in partnames.cfg
german: Add German ordering names in partnames.cfg
italian: Add Italian ordering names in partnames.cfg
japanese: Add Japanese ordering names in partnames.cfg
korean: Add Korean ordering names in partnames.cfg
spanish: Add Spanish ordering names in partnames.cfg

usascii: Add English ordering names in partnames.cfg.
Default: -.

BOM_FILENAME

Full path to the BOM ASCII file that can be exported from the BOM dialog box.

Default: emx_bom.txt.

CHECK_DRAWING_WHEN_OPEN_BOM

NO: No action.

YES: Check if a drawing exists for each part when opening the BOM.

Default: NO.

CHECK_RELATIONS_WHEN_OPEN_BOM

- (hyphen) : When opening the BOM do not check relations defined in EMX rel-files. No action.

FOR_PARTS_AND_ASMS: When opening the BOM, check and set relations from EMX rel-files for parts and assemblies.

FOR_PARTS_ONLY: When opening the BOM, check and set relations from EMX rel-files for parts only.

FOR_ASMS_ONLY: When opening the BOM, check and set relations from EMX rel-files for assemblies only.

Default: NO.

CREATE_LOCAL_PARAMS

NO: No action.

YES: Defines additional project-specific parameters in the current working directory.

Default: YES.

DATE_FORMAT

Date format. d = day; m = month; y = year.

Default: dd-mm-yyyy.

PART_RENAME_FORMAT

Format string for renaming parts using POS or PARTID in BOM dialog box. Use '-' to deactivate this function. Use <name>, <pos_id> and <partid> as placeholders.

Default: -.

PLACE_BALLOON_ON_FACE

NO: Balloons in drawings are attached to the view only.

YES: Balloons in drawings are attached to a selected face.

Default: YES.

RECALC_MASS_PROPS

NO: No action.

YES: Recalculate the point of gravity when closing the mold base definition dialog box.

Default: YES.

RECALC_SIZE_WHEN_OPEN_BOM

NO: No action.

YES: Determine the solid outline of every component when opening the EMX BOM dialog box.

Default: YES.

SORT_BOM_WITH_PARTID

NO: When open BOM dialog box sort components by &POS_ID

YES: When open BOM dialog box sort components by &PART_ID

Default: NO.

SUPPRESS_EJP_NOTE

NO: Display the ejector ID as additional note on the drawing when placing ejector symbols.

YES: Do not display the ejector ID as additional note on the drawing when placing ejector symbols.

Default: NO.

TECH_PARAM

Name of technology parameter.

Default: TECH.

USE_PARTID_IN_BALLOON

NO: Display the content of the parameter &POS_ID in BOM balloons.

YES: Display the content of the parameter &PART_ID in BOM balloons.

Default: NO.

CONFIG_BOM_BALLOON_CUSTOM_PARAM

Name of a parameter that appears in the BOM Balloons instead of the position ID.

Default: – (hyphen).

29.7 Analysis Options

ACCURACY_VALUE

Accuracy value set with the Set Accuracy command.

Default: 0.001.

ACCURACY_TYPE

0: Set the accuracy type to absolute when using the Set Accuracy function.

1: Set the accuracy type to relative when using the Set Accuracy function.

Default: 0.

ADD_TRIM_FACE_IN_CORE

NO: No action.

YES: Add a solid surface copy to all solid models classified as "Reference model" that can be used as ejector trim face.

Default: NO.

ADVANCED_TECH_COLOR

NO: Regular technology color handling.

YES: Allow usage of HASCO standards for surface coloring.

Default: YES.

EDM_CSYS_PARAMETER

Name of the parameter used in the Print EDM Position function.

Default: EDM_ORIGIN.

EDM_CSYS_RESULT_FILE

Name of the result file generated by the Export CSYS Position command.

Default: edm_position.txt.

ENABLE_MOULDING_REPAIR

NO: No action

YES: Repaint ejector cutouts in molding part when calculating the clamping force

Default: NO

QMM_FILENAME

Name of the file used with the Measurement Points function.

Default: qmm_points.txt.

SIZE_FORMAT

L1xL2xL3: Sort dims by size from largest to smallest.

XxYxZ: Sort dims by x, y, and z.

Default: L1xL2xL3.

TEST_MODE

NO: No action.

YES: Save additional debug information in the Creo Parametric trail file.

Default: NO.

USE_ASM_WATERLINE_MDL

NO: Use partfile for waterline analysis and copy all faces into it.

YES: Use assembly for waterline analysis and assemble refmodel and cooling comps.

Default: NO.

29.8 About Common Parameters

To set Bill of Materials parameters, click **EMX ► Options**. The **EMX Options** dialog box opens. Select the **Parameter** tab. This tab lists all parameters associated with each Creo Expert Moldbase Extension assembled component. Double-click a parameter to modify its value.

The **Parameter** tab contains the following columns:

- **Parameter Name**—Displays the parameter name.
- **Belongs to** —Displays the type of parameter (PART or COMP) to create.
- **Parameter Type**—Displays the parameter type (STRING, INTEGER or DOUBLE).
- **Format** —Displays the drawing/report table and the ASCII file output format. Enter the number of characters to use for STRING and INTEGER, or the number of characters or digits for DOUBLE parameters.
- **Designate**—Indicates whether the parameter is for INTRALINK.
- **Show in Table**—Indicates whether the parameter is for BOM.

- **Show in GUI**—Indicates whether the parameter can be edited in an Creo Expert Moldbase Extension dialog box.
- **Use to summarize**—Indicates whether the parameter can be used to compare identical components in the **Bill of Materials** dialog box.
- **Default Value**—Displays the default parameter value. Click to display a default value list.
- Click to add a new parameter. When a parameter is selected, it is copied to a new line.
- Click to delete the selected parameter.

29.9 Default Parameter Configuration Options

To set internal parameter values, enter these parameter configuration options in the **Value** column of the **Parameter** definition table.

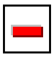
Rule	Definition
`&pos_id`	Unique position #
`&qty`	Quantity of model instances used in the Creo Expert Moldbase Extension assembly
`&partname`	Model name of a component
`&partid`	Part ID to presort the BOM
`&bomname`	Part name for BOM
`&bomname_loc1`	1st localized part name for BOM
`&bomname_loc2`	2nd localized part name for BOM
`&bomname_loc3`	3rd localized part name for BOM
`&bomname_loc4`	4th localized part name for BOM

Rule	Definition
&bomname_loc5	5th localized part name for BOM
&supplier	Supplier
&material	Material
&ord_number	Order number
&size	Model size
&drw_page	Page number in the drawing where the model is shown i view
&bom_page	Page number in the report where the model is listed in tl BOM table
&remark	Remark
&special	Identifies this component is a standard part or not
&modeled_by	Name of the designer
&date	Current date
&projectname	Name of the Creo Expert Moldbase Extension project
&prefix	Project prefix
&postfix	Project suffix
&side	Fixed half or Moving half
&sim_group	Name of the simulation group to which the component belongs
&user_number	Content of the Column USER_NUMBER in the parts coı data.txt file
&sap_number	Name of the simulation group to which the component belongs
&weight	Component weight that will be recalculated when the B dialog box is opened
&price	Content of the Column PRICE in the parts comp_data txt file

29.10 About Project Parameters

Project parameters are project-specific parameters that are stored in the project working directory. The default values for these parameters are defined when a new Creo Expert Moldbase Extension project is created.

When you click the **Project Parameter** tab, the following table appears:

- **Parameter Name**—Shows the parameter name.
- **Belongs to**—Shows the parameter as a part or component. Set the value to `PART` or `COMP`.
- **Parameter Type**—Shows the parameter type. Valid types are `STRING`, `INTEGER`, or `DOUBLE`.
- **Format**—Shows the Drawing/report table and the ASCII file output format. Enter the number of characters used for `STRING` and `INTEGER`, or the number of characters or digits for `DOUBLE` parameters.
- **Designate**—Determines if a parameter is automatically designated for INTRALINK.
- **Show in Table**—Determines if this parameter is used in the BOM.
- **Show in GUI**—Sets the parameter as editable or not editable in an EMX dialog box.
- The **Use to summarize** column shows whether this parameter is used to compare identical components in the BOM dialog box or not.
- **Default Value**—Shows the default parameter value. Click to see default parameters supported by EMX list.
- Click  to add a new parameter. When a parameter is selected, it is copied to a new line.
- Click  to delete the selected parameter.

29.11 About Part Names

When you assemble a component, you must add data to uniquely identify each part. Click the **Part Names** tab in the **EMX Options** dialog box to set default values for most Bill of Materials (BOM) parameters.

- **File Name**—Links the file name to the component template. When the component template name and the file name are the same, these values in this line are used.
- **Standard Part**—Sets component usage as standard (not modifiable) or non-standard (modifiable).
- **Add to BOM**—Sets the default value in the **Component** dialog box. When set to `N`, the component is suppressed in the BOM.
- **Side**—Determines if a component is renamed when assembled on the Moving or Fixed Half of the mold base. Enter `ES` for the moving half or `IS` for fixed half. If the names on both halves are identical, enter `-`.
- **Simulation**—Sets the default simulation group to use for this template.
- **Part Name**—Uses a wildcard-based formatted name for the component.
- **Alternate part name**—Defines an alternative name using the same wildcards.

- **Part ID**—Defines an ID used to sort the BOM so you can list parts in an order that you define.
- **Group Layer**—Sets the group layer name.
- **Layer**—Shows the layer name to use.
- **BOM Name**—Describes the component to print in the BOM. To use multi-language entries, set the configuration option ADD_LOCALIZED_BOM_NAME1 to ADD_LOCALIZED_BOM_NAME5. You can set up to five additional languages.
- **Order Number**—Defines the order number format. By default, order number is set to a value from the database represented by the wildcard <ord_number. Overwrite this setting as needed.

29.12 About Part Name Wildcard Characters

Use the following wildcards when defining part names, part IDs, layers, and order numbers in the **Part Names** tab of the **EMX Options** dialog box:

- <ord_number>
- <instance>
- <sap_number>
- <pre> is replaced by the project prefix defined in the **Project** dialog box.
- <post> is replaced by the project suffix defined in the **Project** dialog box.
- <xid> where x represents the number of characters used to print the component specific id. For example if you set x to equal 3, than the result is 001, 002, and so on.
- <typ> is replaced by the supplier-dependent type name for a component. For example Z41 may be the name for an ejector pin.

Use any component dimension as a wildcard for ordering numbers. Add the dimension symbol name in angled brackets (<>). For example, the ordering number for dowelpin1 is represented by the dimension symbols.

29.13 About Technology Data

Click the **Technology** tab in the **EMX Options** dialog box to set feature colors using customizable color codes, or to define feature parameters with feature-dependent values.

- **Description**—Feature or UDF description.
- **Technical ID**—Internal identifier.
- **Feature Name**—Unique geometry-feature name.
- **Value**—Technology parameter value.
- **Hole Color**—Hexadecimal color code for the first (larger) feature or UDF cylinder faces.
- **Face Color**—Hexadecimal color code for the first feature or UDF flat face (counterbore).
- **2nd Hole Color**—Hexadecimal color code for conic or cylindrical features or UDF faces.

29.14 About Cooling Bore Data

You can modify the default cooling component bore parameters. When you define a cooling component, its main nominal dimension is compared with the data on the **Cooling Bore Data** tab. You can also set defaults for other dimensions such as bore diameter, counterbore dimensions, and so on.

- COOL_NOM—Nominal dimension used to find the correct data set.
- COOL_CORE_DIA—Core diameter.
- COOL_CORE_DEPTH—Core depth.
- COOL_THREAD_DIA—Thread diameter.
- COOL_THREAD_DEPTH—Thread depth.
- COOL_BORE_DIA—Bore diameter.
- COOL_CB_DIA—Counterbore diameter.
- COOL_CB_DEPTH—Counterbore depth.
- COOL_HALF_ANGLE—Blind hole half angle.

29.15 About Ejector Bore Data

When you place an ejector pin, a set of cutouts is automatically made in several plates. These holes and counterbores should comply with company-specific standards. The standard values are defined on the **Ejector Bore Data** tab of the **EMX Options** dialog box:

- EJP_LOWER_B—Lower limit of the ejector pins nominal diameter.
- EJP_UPPER_B—Upper limit of the ejector pins nominal diameter.
- EJP_LL—Minimum length of the ejector pin guided section in the core-plate.
- EJP_LD—Add this value to the guided bore diameter add-on (fit H7).
- EJP_FD—Add this value to the diameter add-on of the counterbore in the core-plate.
- EJP_BD—Add this value to the diameter add-on of all thru holes.
- EJP_ND—Add this value to the diameter add-on for the smaller diameter in the head counterbore.
- EJP_CD—Add this value to the head counterbore diameter assigned in the ejector pin dialog box.
- EJP_CL—Add this value to the head-counterbore depth assigned in the ejector pin dialog box.
- EJP_FL—Default length for Rotation fix-cutout (use absolute value or <x>*D for multiple nominal diameter).
- EJP_FW—Default width for Rotation fix-cutout (use absolute value or <x>*D for multiple nominal diameter).

29.16 About Screw Hole Data

Screw cutouts follow standards similar to those for cooling holes and ejector holes. You can set twelve hole dimensions on the **Screw Hole Data** tab:

- SCR_TYPE_ID—Screw types for which these values must be used. (`ANY` applies settings to all screw types.)
- SCR_NOM—Screw nominal dimension. Use to identify the correct line of this list.
- SCR_CB_DIA—Counterbore diameter.
- SCR_CB_DEPTH—Counterbore depth.
- SCR_DIA_F—Thru holes diameter, fine.
- SCR_DIA_M—Thru holes diameter, middle.
- SCR_DIA_R—Thru holes diameter, rough.
- SCR_CORE_DIA—Thread bore core diameter.
- SCR_THREAD_DEPTH—Minimum thread depth.
- SCR_CORE_DEPTH—Thread bore core depth.
- SCR_TOL—Fit (number, no set-id, default values are H7.)
- SCR_PITCH—Thread per inch or mm = Pitch.
- SCR_CB_DIA_WASHER—Counterbore diameter if using spring washer.
- SCR_CB_DEPTH_WASHER—Counterbore depth if using spring washer.

30

The Component Editor - Admin Tutorial

30.1 About the Component Editor

Click **EMX Assembly ► Administrator Tools ►**  **Component Editor** to open the **Component Editor** dialog box. Use options in this dialog box to maintain the EMX component and library parts database.

30.2 About the Data Structure

The data structure in Creo Expert Moldbase Extension 10.0 and later is different from the data structure in earlier releases. In Creo Expert Moldbase Extension 10.0 and later, there is a separate `*.dat` file for each component. The `*.dat` file contains all information required for the following:

- The component data such as assembly constraints, nominal values, and instances
- The entire cut-information such as UDFs and UDF options

The `comp_data.txt` file contains a listing of available component types so that you can sort the order in the selection list and change the description text.

The advantages of using `.dat` files are listed below:

- Less content for each file, making it easier to manipulate and organize.
- Individual files make it easy to transfer components from one custom configuration to another.
- The UDF section can be changed for each component individually.

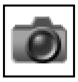
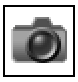
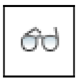
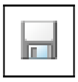
You can migrate comp_data.txt and udf_data.txt files to *.dat files or continue to use the data structure from previous releases.

30.2.1 Data Structure in Creo Expert Moldbase Extension 9.0 and Earlier

The data structure in Creo Expert Moldbase Extension 9.0 and earlier is also supported. In Creo Expert Moldbase Extension 9.0 and earlier, the component data of component types such as the cooling and locating component types for suppliers, are listed in `comp_data.txt` files. Additionally, the user–defined feature (UDF) information for each component is hosted for all component types in the `udf_data.txt` file. From the **Component Editor** dialog box, EMX administrators have direct access to the `comp_data file.txt`, but not to the `udf_data.txt` information. Therefore, manual editing of `udf_data.txt` in Creo Expert Moldbase Extension 9.0 and earlier is required. As the result of rising demands in new components and UDF flexibility, the following occurs in Creo Expert Moldbase Extension 9.0 and earlier:

- The `comp_data.txt` and `udf_data.txt` files are more complex.
- There is a higher–level of effort for EMX administrators to edit components.

30.3 About Setting BOM Data in Component Editor

1. Click **EMX ► Administrator Tools ►**  **Component Editor** to open the **Component Editor** dialog box.
2. Click **Open model or *.dat file** and then browse to and double-click the `*.dat.` file of a library component or any standard EMX part. The path to the currently active library component or EMX component appears.
3. View the list of instances in the file.
4. Enter the required component details.
5. Click  to update the preview image of the current model.
6. The **Model Type** (**Part** or **Assembly**) is automatically set when a model is in session.
7. Enter the remaining settings on the following tabs:
   - **BOM Data**
   - **Assembly References**
   - **Nominal Dimensions**
   - **UDFs**
   - **Options**
   - **Features**
   - **Instances**
   - **Subcomponents**
8. Click the **BOM Data** tab and enter the BOM related data.
   - Select the **Standard Part** check box to prevent changes to the model and define the default model behavior.
   - Select **Add to BOM** to show the value in the component definition.
   - Select **Add/Modify entry in part name configuration** to add or update the BOM data in the `partnames` configuration file.
9. Select **Archive all instances** to create new models for each instance of the current library component or standard part. Use the `ARCHIV_PATH` configuration option to set the path to the archive.
10. Click  to take a snapshot of the current window.
11. Click  to preview the all of the component data in a dialog box.
12. Click  to save component data.
13. Click **Close** to close the dialog box.

30.4 About the Options, Features, and Subcomponents Tabs in the Component Editor Dialog Box

30.4.1 Options Tab

In the **Options** tab you can define customized options for an EMX component. A fully-defined option contains the items listed below:

- Option display name—Translated string
- Option name— Keyword used by the software
- Default value—The default value for the option
- Options type—Select from the list or click `Yes` or `No`.

Options are not completely customizable. For different component types there are several valid options such as `COUNTERBORE` and `SECONDBORE` for screws, `TRIM` or `AUTO_LENGTH` for cooling comps, and `ROTFIX` for ejector pins.

30.4.2 Features Tab

From the **Features** tab you can suppress and resume certain features in models depending on a defined option and its value. A valid feature definition requires the following:

- Feature name—A list of features by names in the model is available in the option menu
- Option name—From the **Options**
- Value—The value for the option which needs to be set to make the feature visible.

30.4.3 Subcomponents Tab

When you define an assembly as a library component, you need to provide EMX with the location of the definitions of the subcomponents in the EMX components directory. Valid subcomponents require the information listed below:

- Name of the component
- Unit (mm or inch)
- Directory such as screw, dowel pin, lifter, and so in the components folder
- Supplier such as Hasco
- File name (name of the template part)
- Type name. EMX searches for this name under Instances. In the instances section for each assembly instance, a valid subcomponent instance must be defined.
- Side value defines the view to which the component is added.

A valid assembly contains multiple parts. Subassemblies are not possible.

30.5 About Setting Assembly References in Component Editor

1. Click **EMX ► Administrator Tools ► Component Editor** to open the **Component Editor** dialog box.
2. Click **Assembly References**.
3. Click a line to activate the assembly reference.
4. Set default constraint set values:
   - —Axis or point, datum plane, and orient plane.
   - —Axis or point and datum plane.
   - —Axis or point, or curve and datum plane.
   - —Axis or point on curve, datum plane, and orient plane.
   - —Coordinate system.
   - —Datum plane.
   - —Prints the internal Creo Parametric identifiers of the selected geometry. This ID is used to identify the component assembly reference.
5. Click to add a new constraint to the list using the current edit area values, or click to remove the active constraint.

30.6 About Setting Nominal Dimensions in Component Editor

1. Click **EMX ► Administrator Tools ► Component Editor** to open the **Component Editor** dialog box.
2. Select the **Nominal Dimensions** tab.
3. Select a row and edit the dimensions.
4. Click to add a new nominal dimension or click to remove an active nominal dimension.

30.7 About Setting Cuts And Threads In the UDF Tab of the ComponentEditor

1. Click **EMX Assembly ▶ Component Editor**. The **Component Editor** dialog box opens.
2. Select the **Cuts and Threads** tab.
3. Select a UDF to create cutouts for the component. For library components this is `QUILT_CUT`.
4. The predefined threads in the current model are displayed in the list of predefined threads. The UDF `auto_libthread.gph` represents the information required to add a tapped bore to models that interfere with this UDF after the component is placed in the assembly.
5. The selected cut quilts are displayed in the cut quilt list. Ensure that the quilts used for cutouts are closed.
6. Click to add a new quilt or thread, or click to remove the selected cut quilt or thread from the list.

30.8 About Setting Instances in Component Editor

Component instances are saved in a family table.

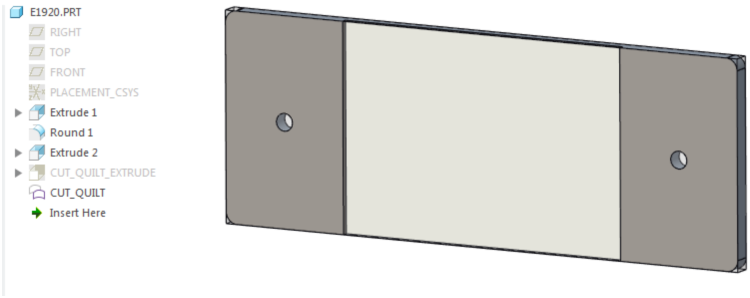
1. Click **EMX ▶ Administrator Tools ▶ Component Editor** to open the **Component Editor** dialog box.
2. Select the **Instances** tab.
3. Double-click an instance to edit it.
4. Click to copy all instances information to the clipboard for editing externally, such as editing in an EXCEL file.
5. Click to paste the externally edited instances information back to the component editor. You can edit the cell values and insert new columns.
6. Click to add a new instance. When you select a row, the information is copied to the new instance.
7. Click to remove the selected instance from the list.

30.9 To Add a Part to the Library

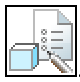
Download 00_library_part to start with this chapter.

30.9.1 Load the new library component

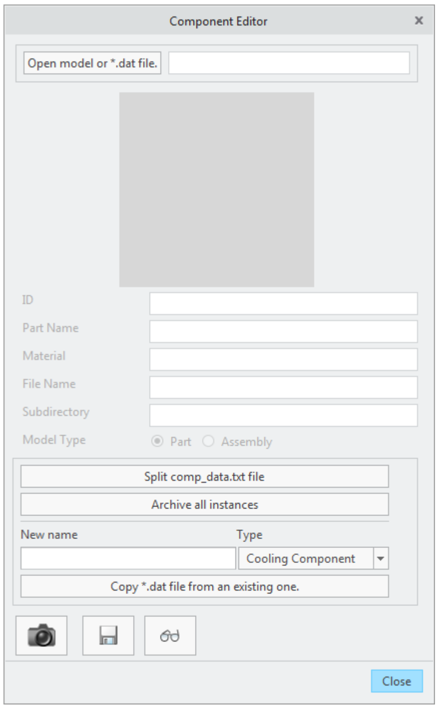
1. Erase all models from the Creo session.
2. Copy the file E1920.PRT from <install_tutorial>/models/library_prt into an new folder<install_emx90>/components/mm/library/tutorial.
3. Open the model E1920.PRT from this directory.



30.9.2 Create an Image and Define the Component BOM Data

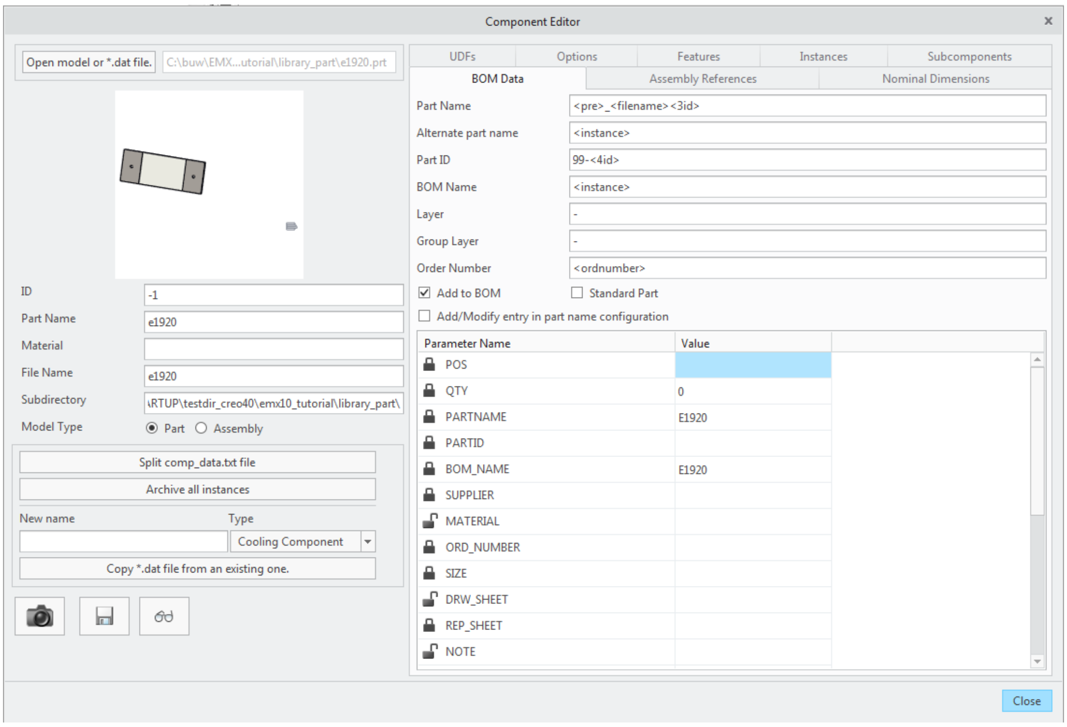
1. Start **EMX Part Mode ► Administrator Tools ► Component Editor** .

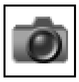
   The **Component Editor** dialog box opens.



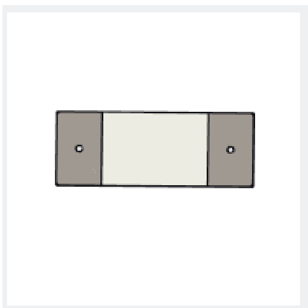
2. .Click **Open model or *.dat file**

The **File Open** dialog box appears. Select `e1920.prt` from current working directory. The dialog is filled with default library component information.



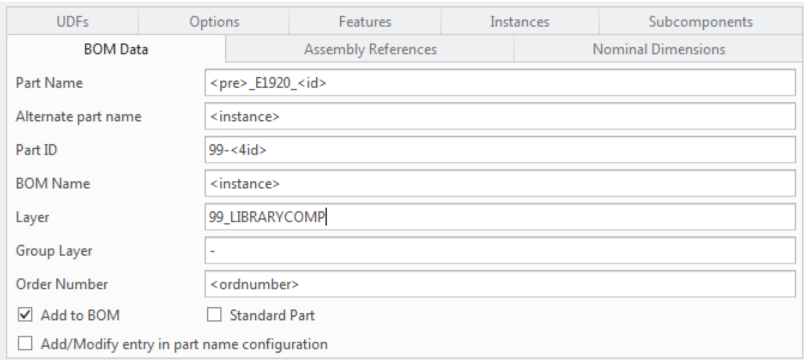
3. .Adjust the model orientation and click 

NOTE:
It is recommended to display the assembly reference features so they can be identified in the image. You may edit the picture with a regular image editor afterwards.



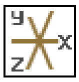
4. Define the **BOM data** for this component.

This is the data that can be found for regular EMX components in the **EMX Options ▸ Part Names** sheet.

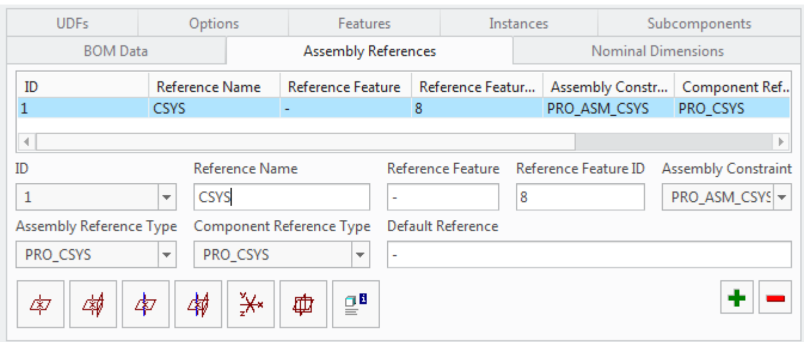


5. Enter the **Part name** `<pre>_E1920_<id>`.
6. Enter the **Alternate part name** `<instance>`.
7. Keep **Part ID** as `99<2id>`.
8. Keep **BOM Name** as `<instance>`.
9. As target **Layer** enter `99_LIBRARYCOMP`. The component will be place on this layer after the assembling process.
10. Enter **Group Layer** `-`.
11. Keep **Order Number** as `<ordnumber>`.
12. Enter **Customer number** `-`.
13. Enable **Add in BOM** so this component will be displayed in the EMX bill-of-materials later.

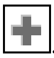
30.9.3 Define the Assembly References

1. Click the tab **Assembly References**.
2. Click  and select the coordinate system `PLACEMENT_CSYS` from the model `E1920.PRT`.

   A new line appears in the assembly constraints.
3. Select this line (with ID1) and edit the entry for **Reference Name** in the edit area under the list to `CSYS`.



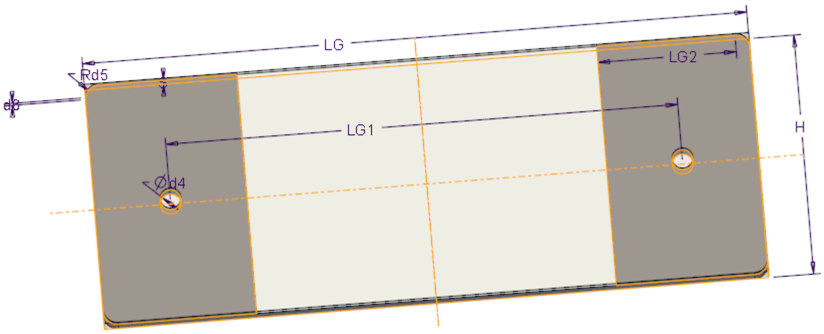
30.9.4 Define the Nominal Dimensions

1. Click the tab **Nominal Dimensions**.
2. Click .

   All dimensions are displayed in the 3D model.
3. Hold down the CTRL-Key and select the dimensions: `LG`, `LG1`, `LG2`, `H` and `S`.Click MMB to finish the selection.

   NOTE:
   The order of selection defines the order in the dialog entries.

   Five new lines appear in the **Nominal Dimension** list.



4. Select a line to edit the entries in the edit area below the list.
5. Select the `LG` line and overwrite the **Dimension Name** (symbol **LG**) with **LENGTH**
6. Select the `H` line and overwrite the **Dimension Name** (symbol **H**) with **HEIGHT**.
7. Set the **Dimension Type** of `LG1`, `LG2` and `S` to PULLDOWN_EDIT.

Dimension Name	Dimension Sym...	Dimension Type
H	H	PULLDOWN
LG	LG	PULLDOWN
LG1	LG1	PULLDOWN_EDIT
S	S	PULLDOWN_EDIT
LG2	LG2	PULLDOWN_EDIT

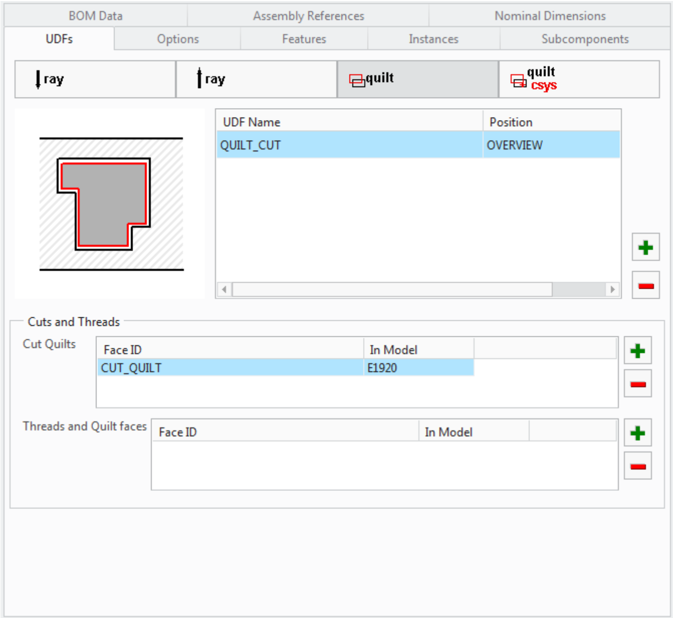
30.9.5 Define cutout UDFs

1. Click the tab **UDFs** .

For library components the UDF QUILT_CUT is used by default.

2. Click the beside the Cut quilts list and select the Quilt feature `CUT_QUILT` in the `E1920.PRT`.

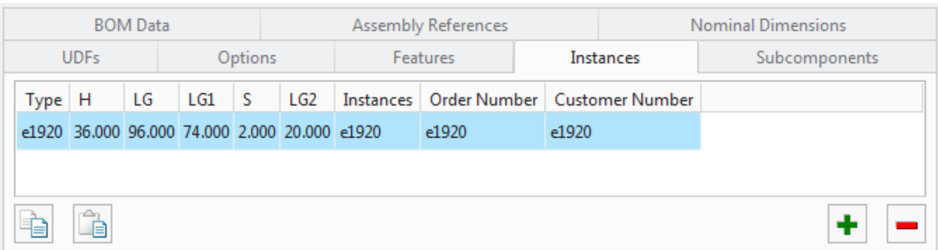
NOTE:
Valid cut quilts will be automatically detected and added to the cut quilts list.



No **Predefined threads** should be defined for this example.

30.9.6 Define instances with Excel

1. Click the tab **Instances**.



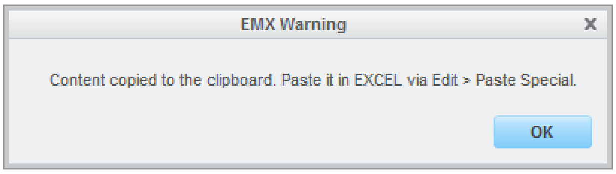
In the table all instances that are defined for the library component can be seen.

Each **Nominal Dimension** is represented as a column.

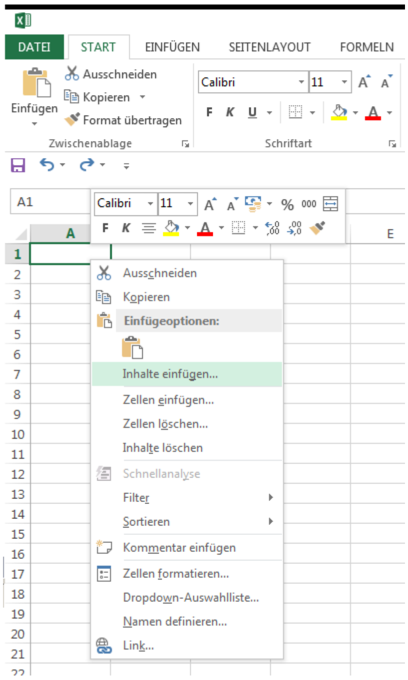
By default only one instance with the current model values is added to the table.

Excel can be used for additional instances.

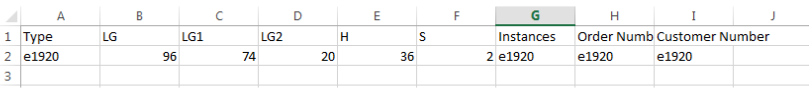
2. Click **Copy**  to copy the current content of the table.



3. Open an MS Excel instance and paste the content to the Excel-Sheet.



All rows will be displayed in the Excel-Sheet.



	A	B	C	D	E	F	G	H	I	J
1	Type	LG	LG1	LG2	H	S	Instances	Order Numb	Customer Number	
2	e1920	96	74	20	36	2	e1920	e1920	e1920	
3										

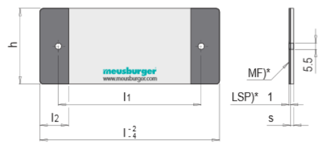
4. Enter instances that can be found in the Meusburger Digital catalog.

meusburger

E 1920

Auswerferabdeckung transparent, magnetisch

Transparent and magnetic ejector set cover

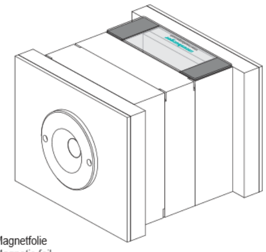


Mat.: PC transparent

l_2	l_1	l	h	s	Nr. /No.
20	74	96	36	2	E 1920/ 96 36/2
			46		E 1920/ 96 46/2
			56		E 1920/ 96 56/2
			66		E 1920/ 96 66/2
20	104	126	36		E 1920/126 36/2
			46		E 1920/126 46/2
			56		E 1920/126 56/2
			66		E 1920/126 66/2
			76		E 1920/126 76/2
			86		E 1920/126 86/2
			96		E 1920/126 96/2
25	124	156	46		E 1920/156 46/2
			56		E 1920/156 56/2
			66		E 1920/156 66/2
			76		E 1920/156 76/2
			86		E 1920/156 86/2
			96		E 1920/156 96/2
30	156	196	46		E 1920/196 46/2
			56		E 1920/196 56/2
			66		E 1920/196 66/2
			76		E 1920/196 76/2
			86		E 1920/196 86/2
			96		E 1920/196 96/2
			116		E 1920/196 116/2
30	176	216	46		E 1920/216 46/2
			56		E 1920/216 56/2
			66		E 1920/216 66/2
			76		E 1920/216 76/2
			86		E 1920/216 86/2
			96		E 1920/216 96/2
30	206	246	46		E 1920/246 46/2
			56		E 1920/246 56/2
			66		E 1920/246 66/2
			76		E 1920/246 76/2
			86		E 1920/246 86/2
			96		E 1920/246 96/2
			116		E 1920/246 116/2
			136		E 1920/246 136/2
40	248	296	56		E 1920/296 56/2
			66		E 1920/296 66/2
			76		E 1920/296 76/2
			86		E 1920/296 86/2
			96		E 1920/296 96/2
			116		E 1920/296 116/2
			136		E 1920/296 136/2
40	268	316	56	3	E 1920/316 56/3
			66		E 1920/316 66/3
			76		E 1920/316 76/3
			86		E 1920/316 86/3
			96		E 1920/316 96/3
			116		E 1920/316 116/3
			136		E 1920/316 136/3
			156		E 1920/316 156/3
40	298	346	56		E 1920/346 56/3
			66		E 1920/346 66/3
			76		E 1920/346 76/3
			86		E 1920/346 86/3
			96		E 1920/346 96/3
			116		E 1920/346 116/3
			136		E 1920/346 136/3
			156		E 1920/346 156/3

t max. = 120°C

l_2	l_1	l	h	s	Nr. /No.
50	324	396	66	3	E 1920/396 66/3
			76		E 1920/396 76/3
			86		E 1920/396 86/3
			96		E 1920/396 96/3
			116		E 1920/396 116/3
			136		E 1920/396 136/3
			156		E 1920/396 156/3
50	374	446	76		E 1920/446 76/3
			86		E 1920/446 86/3
			96		E 1920/446 96/3
			116		E 1920/446 116/3
			136		E 1920/446 136/3
			156		E 1920/446 156/3
50	424	496	76		E 1920/496 76/3
			86		E 1920/496 86/3
			96		E 1920/496 96/3
			116		E 1920/496 116/3
			136		E 1920/496 136/3
			156		E 1920/496 156/3
50	474	546	76		E 1920/546 76/3
			86		E 1920/546 86/3
			96		E 1920/546 96/3
			116		E 1920/546 116/3
			136		E 1920/546 136/3
			156		E 1920/546 156/3
			176		E 1920/546 176/3
50	508	596	86		E 1920/596 86/3
			96		E 1920/596 96/3
			116		E 1920/596 116/3
			136		E 1920/596 136/3
			156		E 1920/596 156/3
			176		E 1920/596 176/3



MF)* Magnetfolie
Magnetic foil

LSP)* Luftspalt - verhindert Hitzestau
Air gap - avoids from heat accumulation

Not intended for use in the USA!

	A	B	C	D	E	F	G	H	I
1	Type	LG	H	LG1	LG2	S	Instances	Order Number	Customer Number
2	e1920	96	36	74	20	2	e1920_96_36_2	e1920/96/36/2	
3	e1920	96	46	74	20	2	e1920_96_46_2	e1920/96/46/2	
4	e1920	96	56	74	20	2	e1920_96_56_2	e1920/96/56/2	
5	e1920	96	66	74	20	2	e1920_96_66_2	e1920/96/66/2	
6	e1920	126	36	104	20	2	e1920_126_36_2	e1920/126/36/2	
7	e1920	126	46	104	20	2	e1920_126_46_2	e1920/126/46/2	
8	e1920	126	56	104	20	2	e1920_126_56_2	e1920/126/56/2	
9	e1920	126	66	104	20	2	e1920_126_66_2	e1920/126/66/2	
10	e1920	126	76	104	20	2	e1920_126_76_2	e1920/126/76/2	
11	e1920	126	86	104	20	2	e1920_126_86_2	e1920/126/86/2	
12	e1920	126	96	104	20	2	e1920_126_96_2	e1920/126/96/2	
13	e1920	156	46	124	25	2	e1920_156_46_2	e1920/156/46/2	
14	e1920	156	56	124	25	2	e1920_156_56_2	e1920/156/56/2	
15	e1920	156	66	124	25	2	e1920_156_66_2	e1920/156/66/2	
16	e1920	156	76	124	25	2	e1920_156_76_2	e1920/156/76/2	
17	e1920	156	86	124	25	2	e1920_156_86_2	e1920/156/86/2	
18	e1920	156	96	124	25	2	e1920_156_96_2	e1920/166/96/2	
19	e1920	196	46	156	30	2	e1920_196_46_2	e1920/196/46/2	
20	e1920	196	56	156	30	2	e1920_196_56_2	e1920/196/56/2	
21	e1920	196	66	156	30	2	e1920_196_66_2	e1920/196/66/2	
22	e1920	196	76	156	30	2	e1920_196_76_2	e1920/196/76/2	
23	e1920	196	86	156	30	2	e1920_196_86_2	e1920/196/86/2	
24	e1920	196	96	156	30	2	e1920_196_96_2	e1920/196/96/2	
25	e1920	196	116	156	30	2	e1920_196_116_2	e1920/196/116/2	

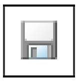
NOTE:
The first column always contains the type of the component in the case `E1920`.

Make sure that the column for **Instances** and **Order Number** is correct and has a unique name. Otherwise EMX can not added the correct ordering number to the BOM. The entries in Instances should be proper Creo model names, so they must not contain "/" or ",". This names will be used as partname if the wildcard `<instance>` is used.

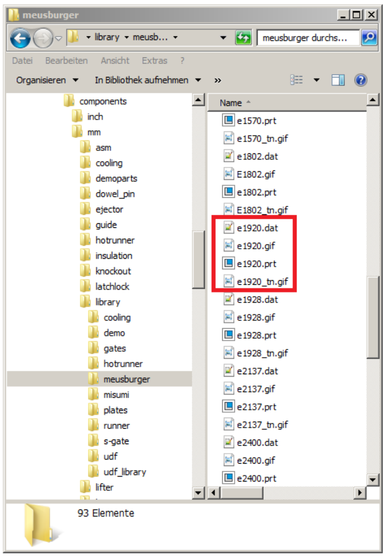
5. Copy the final instance sheet to the EMX instances list with **Paste** 

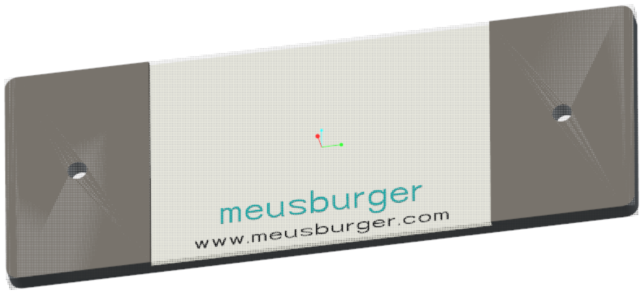
NOTE:
In MS Excel select the exact area of cells you want to paste including the header row.

30.9.7 Save the Library Component

1. Click 

EMX will create a `e1920.dat` file and a `e1920.gif`-file in the directory where the library part is located. The model will also be saved again as `e1920.prt.1`. Delete the original file `e1920.prt` and rename the new file without an index. This step is not required, but it helps you to keep the library folders clean.



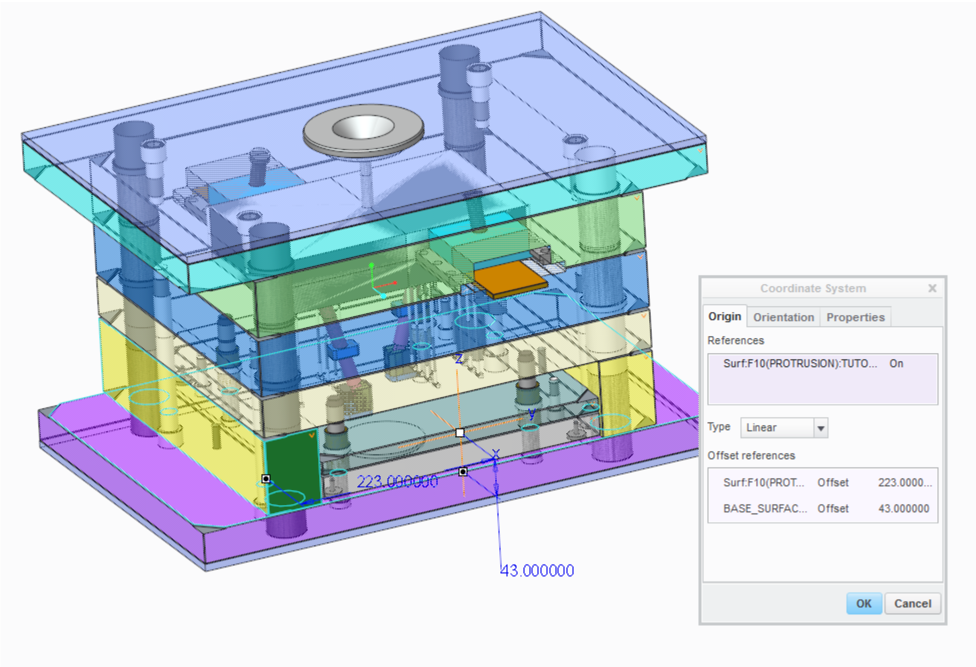


NOTE:
You don't have to create the tiny image `e1920_tn.gif` manually. This image is used in the **Library Component** dialog box for preview. If the image does not exist, EMX will create it use the setting of `LIBRARY_THUMB_SIZE` to determine the size.

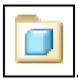
The component is now fully defined.

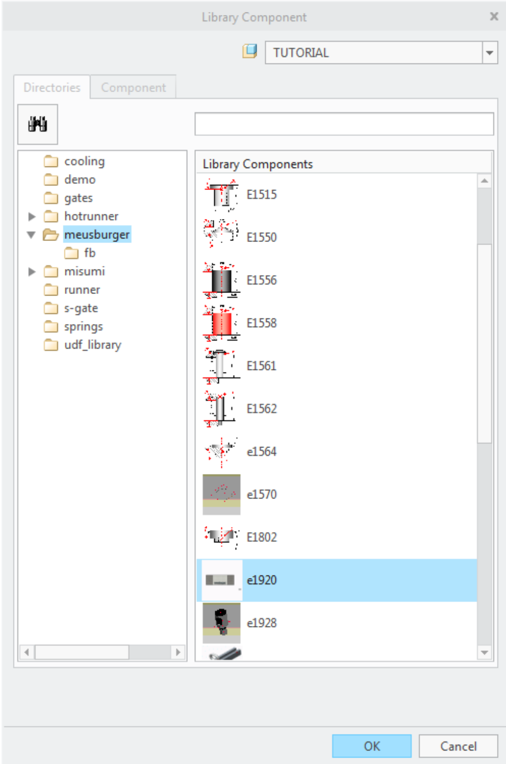
30.9.8 Assemble the component

1. Open the mold base `TUTORIAL.ASM`
2. Create a coordinate system as assembly reference for the plate as shown in the picture below.



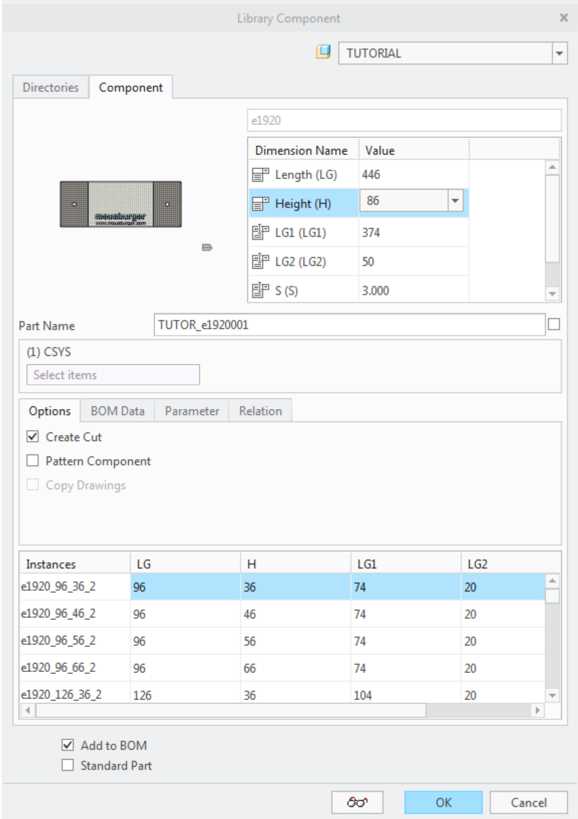
3. Make sure that the orientation of the coordinate system is correct. **X-Vector** is normal to surface and **Z-Vector** points into opening direction of the mold base.

4. Select **EMX Components ► Library ► Assemble** .

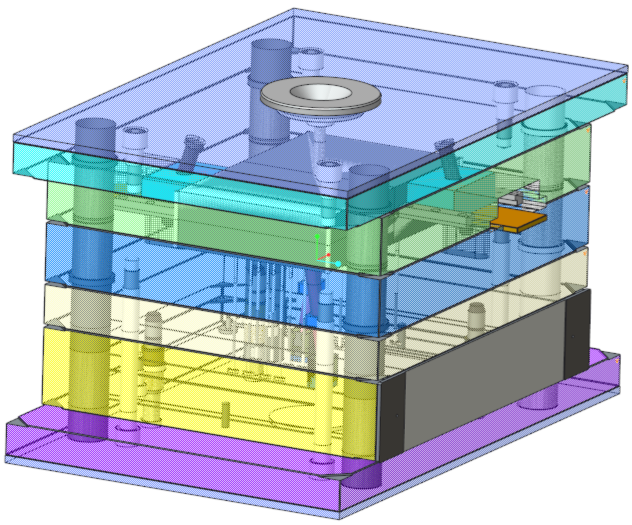


5. Select the directory **meusburger** from the directory tree.
6. Select part **e1920** with Double-Click.
7. Set the **Length (LG)** value to 446.
8. Set the **Height (H)** value to 86.

9. As assembly reference **(1) CSYS** select the recently created coordinate system.

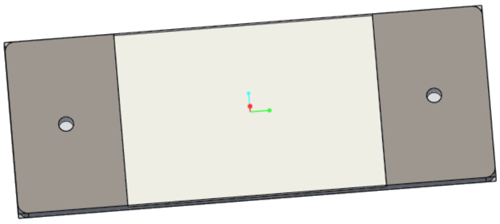


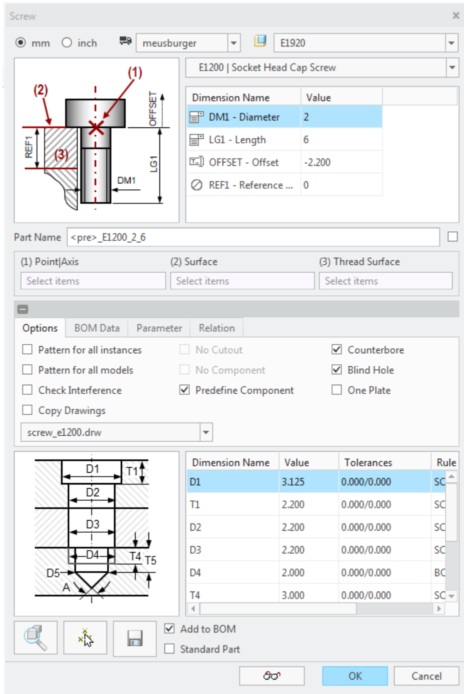
10. Leave the **Library Component** dialog with **OK** .



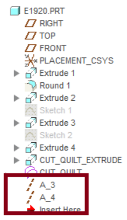
30.10 To Add Predefined Screws to Components

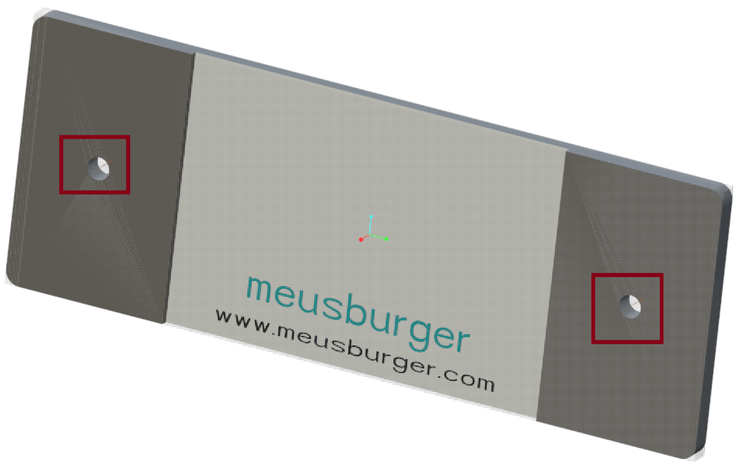
1. Erase all models from the Creo session.
2. Open part `E1920.PRT` from directory `library/tutorial`.
3. Click **EMX Part Mode ▶ Components ▶ Screw** .



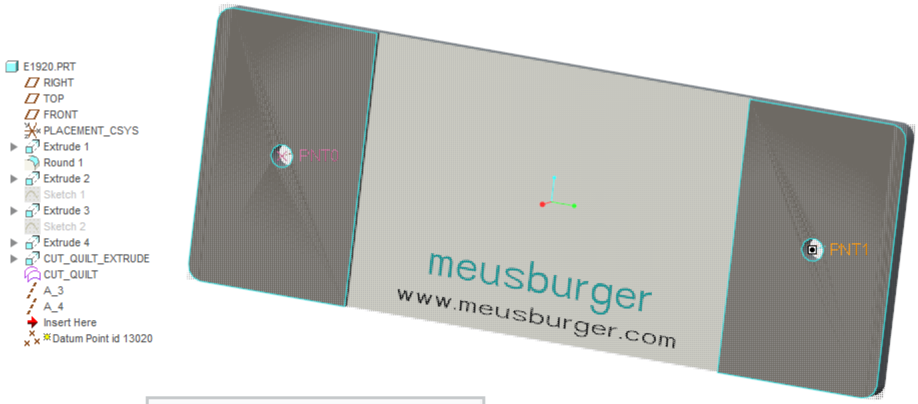


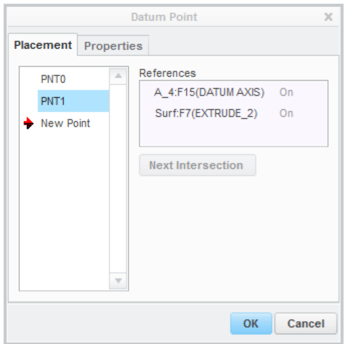
4. Create a **Datum Axis** at each of the existing bores.





5. Create a **Datum Points** on each of the recently created axes and the upper surface of the plat as seen in the picture below.



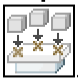


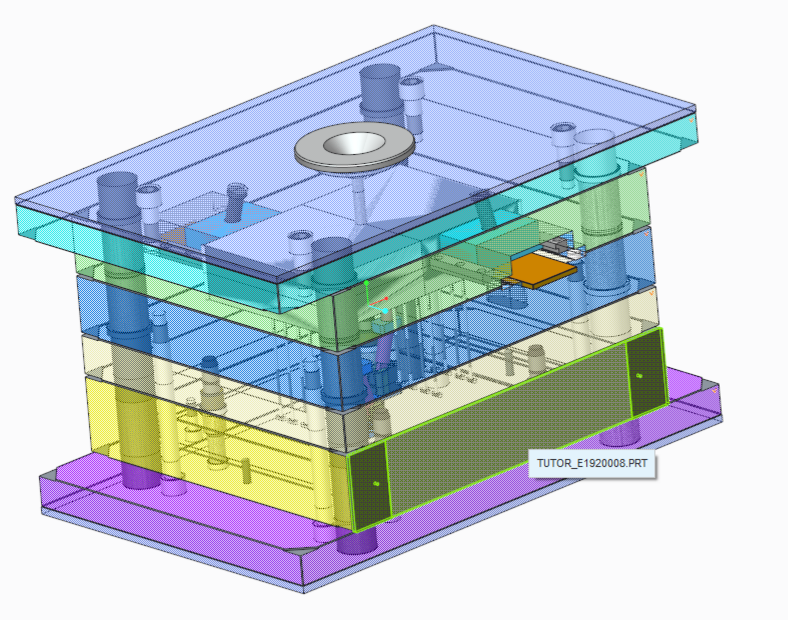
6. Within the **Screw** dialog box select the desired screw supplier **meusburger** and screw type **E1200 | Socket head cap screw**
7. Define the references.
   a. Click **(1) Point|Axis** and select the recently created datum points.
   b. **(2) Surface** - Select the top face of the `E1920.PRT`.
   c. **(3) Thread surface** - Select the bottom face of the `E1920.PRT`.
8. Double click the **Diameter Value** and select 4.
9. Double click the **Length Value** and select 10.
10. Make sure that **Counterbore** is disabled.

    When working in Part Mode, the **Predefined Component** flag is set by default.
11. Leave the **Screw** dialog box with **OK** and find the predefined component UDFs in the model tree.
12. Save the modified library part with **File > Save**.

30.10.1 Assemble the Modified Library Component

1. Open the mold base `TUTORIAL.ASM`
2. Assemble the `E1920.PRT` as described in the chapter Add a Part to Library on page 388.

3. Select **EMX Components ► Component Handling ► Assemble predefined components** .
4. Select the model TUTOR_E1920008 in the graphics window.



As the result the predefined screws are assembled to the mold base.

Background Information

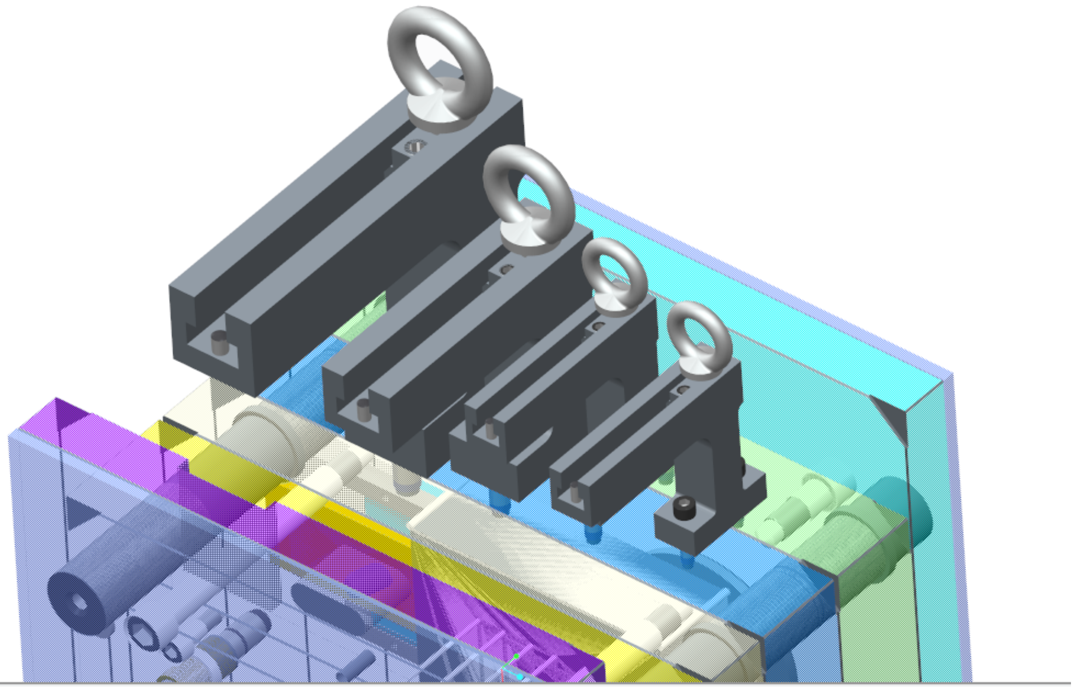
Set the **EMX Option** AUTO_ASSEMBLE_SCREWPINS to YES to assemble predefined components directly after assembling the owner component automatically.

This way of predefining components can be used for any EMX component, even sliders or lifters. But in most cases it is dowel pins and screws that are predefined.
If you set the **EMX Option** USE_41_SCREWPIN_PREDEFINE to YES, EMX will not add REF-GROUPS on the selected references but save all component information directly to the selected placement points..

30.11 To Add an Assembly to the Library

Download 00_library_asm to start with this chapter.

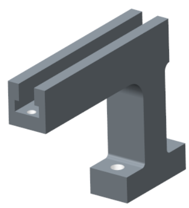
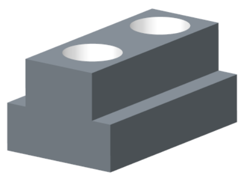
Adding assembly components to library is more complex than just adding a single part. In this chapter we will add the adjustable transportation bar `E1930` from the supplier Meusburger which can be seen in the picture below.

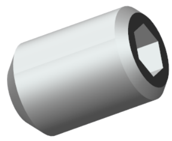
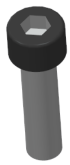
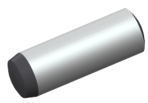


30.11.1 The Assembly Structure

Before adding the assembly to the EMX Library all containing single components need to be implemented.

The `E1930` assembly contains the following parts:

`E1930_1`: The transportation bar	
`E1903_2`: The t-nut	

E1270: A hoist ring	
E1200: 2 x socket head cap screws	
E1230: A set screw	
E1300: 2 dowel pins	

The components E1270, E1200, E1230 and E1300 are already available as regular EMX Components.

The bar E1930_1and the T-Nut E1930_2 have to be created.

Go to the Meusburger catalog and extract the data sheets for the components E1930_1 and E1930_2.

Use the regular Creo functions to create the PRT files.

30.11.2 Design the component E1930_1

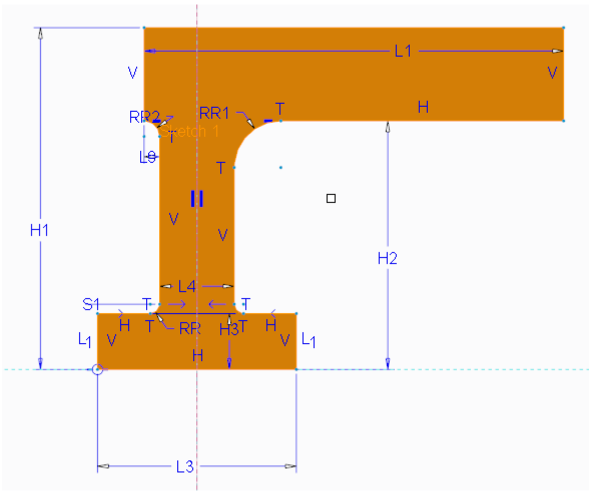
All model described in this chapter can be found in folder <install_tutorial>/models/library_asm. E1930_1 contains a couple of features:

- PLACEMENT_CSYS : Use this csys as assembly reference.
- BODY and SHAPE: This features describe the geometry.
- POINTS_FOR_E1300: points where the dowel pins E1300 will be placed in the assembly.
- POINTS_FOR_E1200: points where the screws E1200 will be placed in the assembly.
- CUT_QUILT: A quilt surface which surrounds the whole part this will be used as cut quilt later.

NOTE:
The UDF Group features are later created during the design of the assembly.

30.11.3 Add the component E1930_1 to the EMX Library

1. Erase all models from the Creo session.
2. Add the previously created part file `E1903_1.PRT` to the library folder <library_emx90>/tutorial and open the part.
3. Add the component to the EMX library just as you have seen it the chapter To Add a Part to the Library on page 388.
4. Use the `PLACEMENT_CSYS` as the assembly **Reference**.
5. Make sure all dimensions which need to be varied are added as **Nominal Dimensions**



6. Set up the **Instances** correctly.

EMX_INSTANCES

TYPE	L2	L1	S1	H1	H3	L3	H2	L5	L4	L6	T7	T8	B7	B8	R1	R2	R	L9	INSTANCE	ORD_NUMBER
e1930_1	44	135	29	110	18	64	80	20	24	145	20	8	12	19	15	5	3	5	e1930_1_44	e1930/1/44
e1930_1	64	135	29	127	25	84	97	30	30	155	20	8	12	19	15	2	8	2	e1930_1_64	e1930/1/64
e1930_1	82	181	49	147	27	106	107	35	40	204	30	12	18	30	20	4	8	4	e1930_1_82	e1930/1/82
e1930_1	100	242	59	209	36	134	155	40	50	268	38	16	22	37	20	9	8	9	e1930_1_100	e1930/1/100

EMX_INSTANCES_END

7. Use the `CUT_QUILT` feature as **Cut Quilt**.

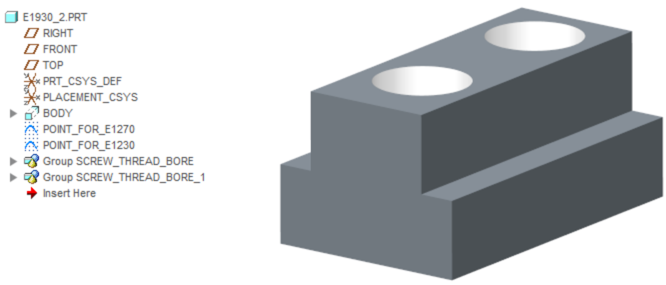
   As a result you will find the `E1930_1.dat` file and the picture `E1930_1.gif` in <library_emx90>/tutorial folder.

30.11.4 Design the component E1930_2

These are the important features:

- PLACEMENT_CSYS : Use this csys as assembly reference.
- BODY : This features describes the geometry.
- POINTS_FOR_E1270: points where the hoist ring E1270 will be placed in the assembly.
- POINTS_FOR_E1230: points where the set screws E1230 will be placed in the assembly.

NOTE:
The UDF Group features are later created during the design of the assembly.

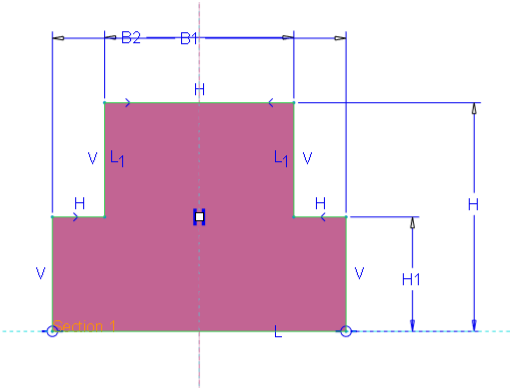


30.11.5 Add the component E1930_2 to the EMX Library

1. Erase all models from the Creo session.
2. Add the previously created part file `E1903_2.PRT` in the library folder <library_emx90>/tutorial and open the part.
3. Add the component to the EMX library just as you have seen it the chapter
   To Add a Part to the Library on page 388.

   Use the `PLACEMENT_CSYS` as the assembly **Reference**.

4. Make sure all dimension which need to be varied are added as **Nominal Dimensions**



5. Set up the **Instances** correctly.

```
EMX_INSTANCES
TYPE      L      B1     B2     H1     H      L1     L2       INSTANCE        ORD_NUMBER     USER_NUMBER     PRICE
e1930_2  30      18     11.6   7      14     8      24.000   E1930_2_10_10   E1930_2/10/10  -
e1930_2  42      28     17.6   10     20     13     35.000   E1930_2_16_12   E1930_2/16/12  -
e1930_2  51      35     21.6   14     28     17.5   42.000   E1930_2_20_16   E1930_2/20/16  -
EMX_INSTANCES_END
```

As a result you will find the `E1930_2.dat` file and the picture `E1930_2.gif` in `<library_emx90>/tutorial` folder.

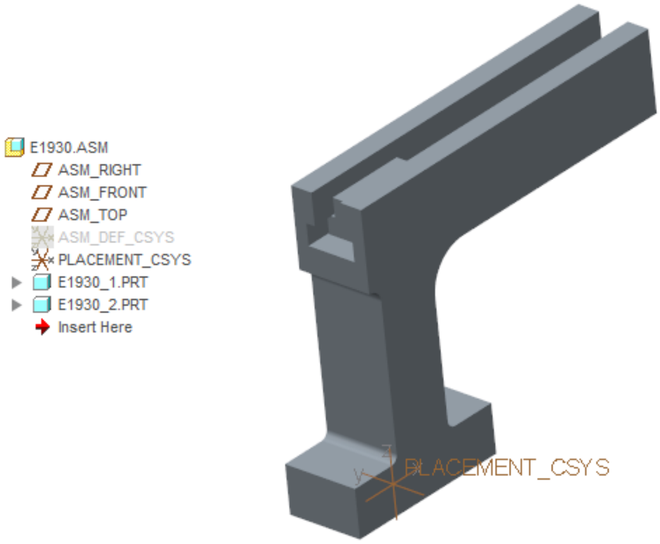
30.11.6 Copy Screws and Dowel Pins to Library Folder

To design the assembly E1930.ASM copies of the template parts of dowel pin E1300, set screw E1230, head cap screw E1200 and hoist ring E1270 are required in the library folder `<library_emx90>/tutorial`.

1. Copy `screw_e1200.PRT` from “`<install_emx90>/components/mm/screw`” to “`<library_emx90>/tutorial`”
2. Copy `screw_e1230.PRT` from “`<install_emx90>/components/mm/screw`” to “`<library_emx90>/tutorial`”
3. Copy `screw_e1270.PRT` from “`<install_emx90>/components/mm/screw`” to “`<library_emx90>/tutorial`”
4. Copy `dwlpin_e1300.PRT` from “`<install_emx90>/components/mm/dowel_pin`” to“`<library_emx90>/tutorial`”

30.11.7 Design the Assembly E1930

1. Erase all models from the Creo session.
2. Create a new assembly and assemble the previously created components as seen in the picture below.



3. Open **EMX Assembly ► Project ► Modify** .

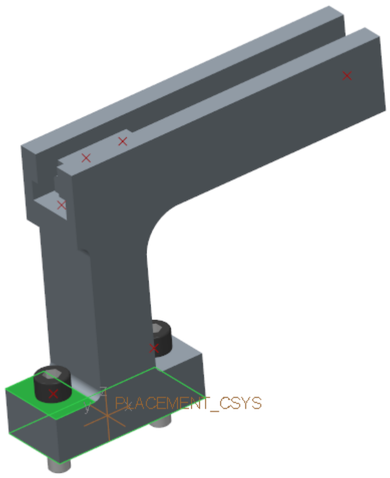
   The **Project** dialog box appears.
4. Leave the dialog box again with **OK**.

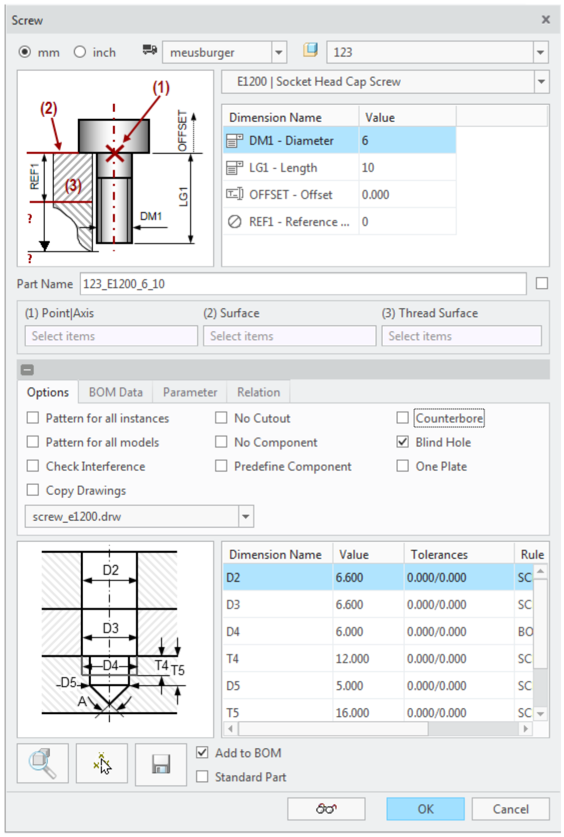
   The Assembly is now considered to be an EMX project.

In the next steps the screws, pins and the hoist ring will be assembled.

1. Click **EMX Components ► Components ► Screw** .
2. Assemble the `E1200` component as seen in the picture below.

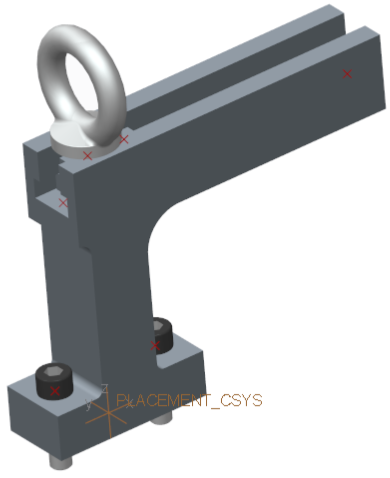
3. Set **Part Name** to `screw_E1200`.This is essential to allow EMX to find the correct template model.

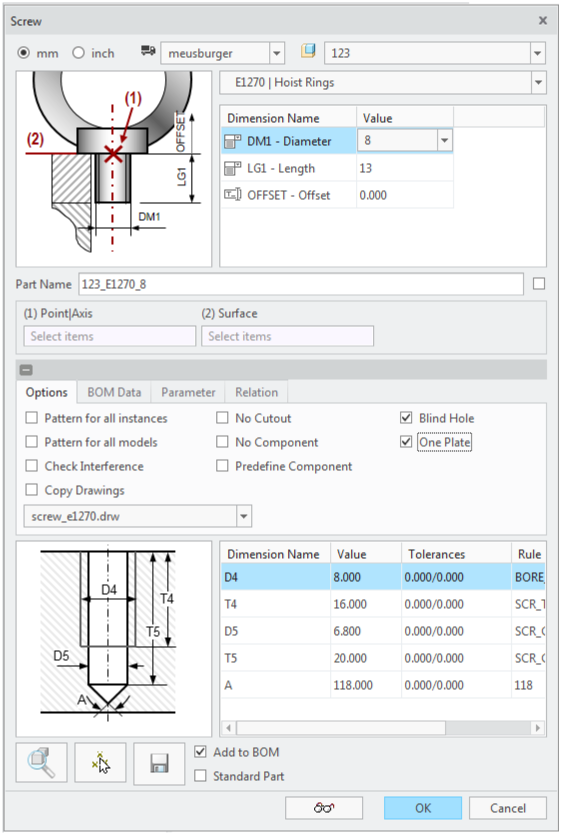




4. Repeat the same steps for the hoist ring `E1270`.

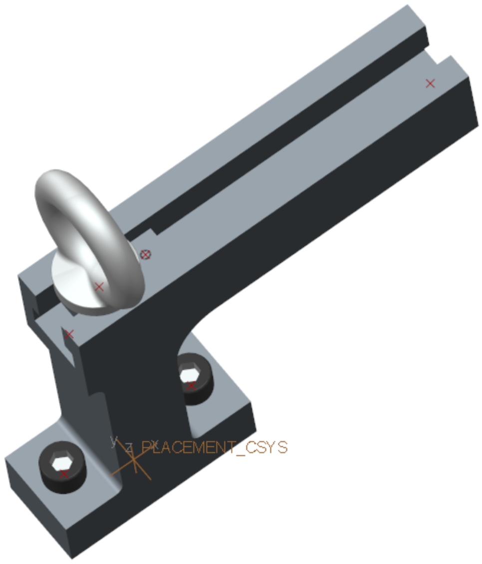
5. Set **Part Name** to `screw_E1270`.

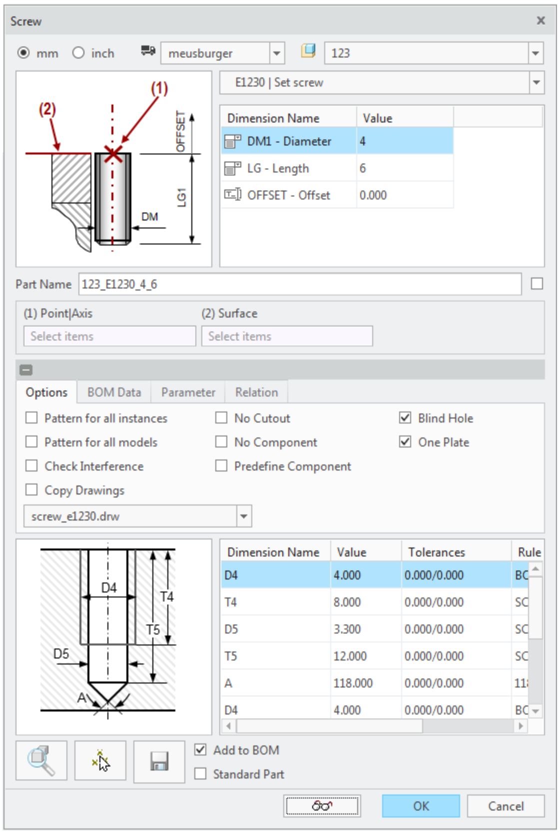




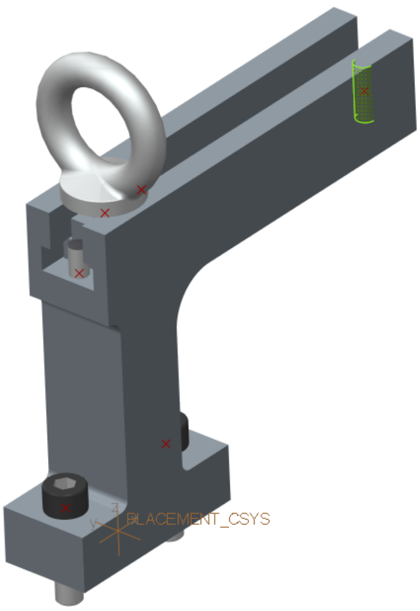
6. Repeat the same steps for the set screw `E1230`.

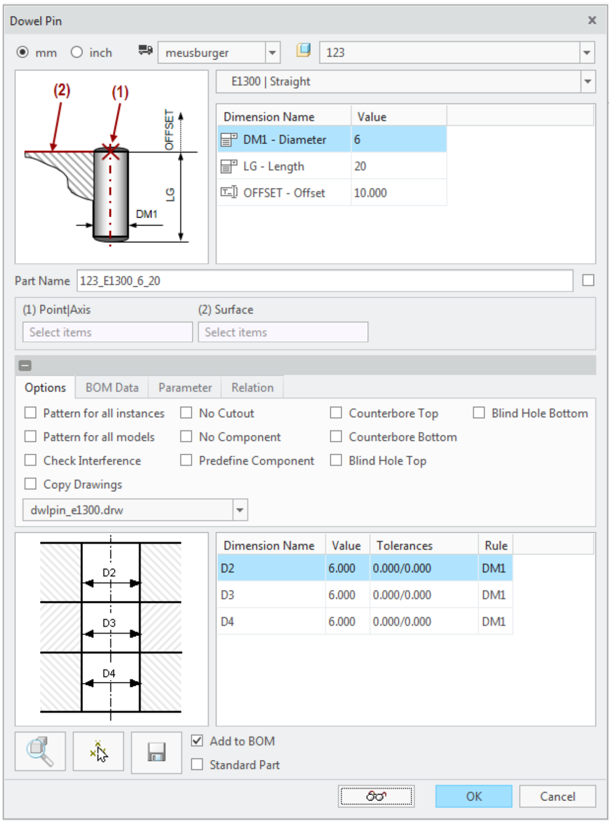
7. Set **Part Name** to `screw_E1230`.



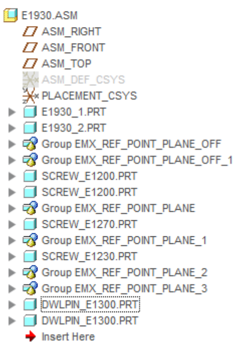


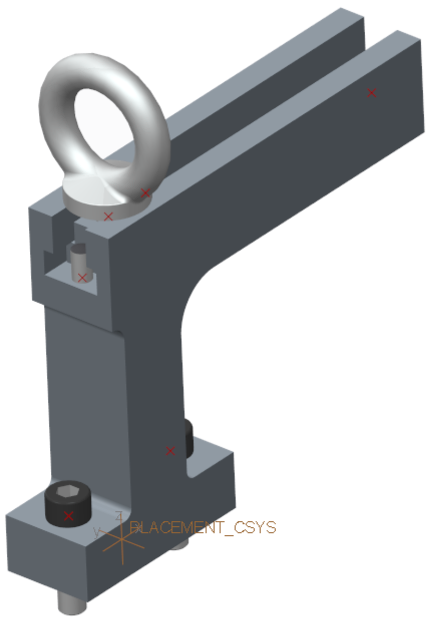
8. Repeat the same steps for the dowel pins E1300.
9. Set **Part Name** to `dwlpin_E1300`.



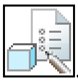


The Assembly is completed.





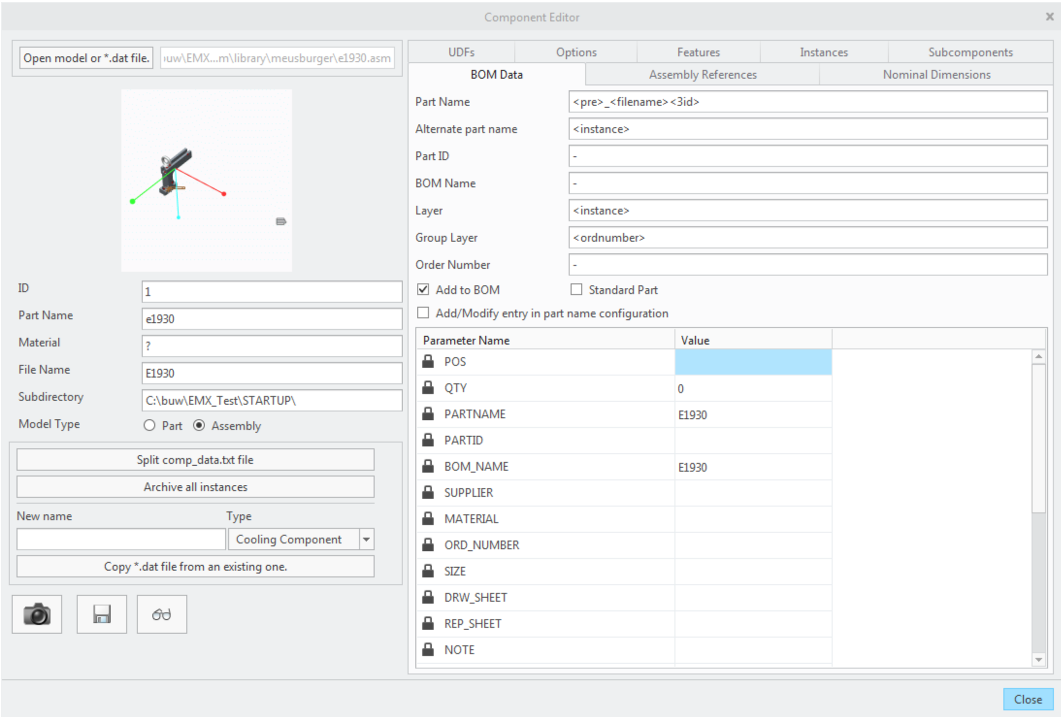
30.11.8 Add the assembly E1930 to the EMX Library

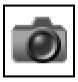
1. Start **EMX Assembly ► Administrator Tools ► Component Editor** .

   The **Component Editor** dialog box opens.

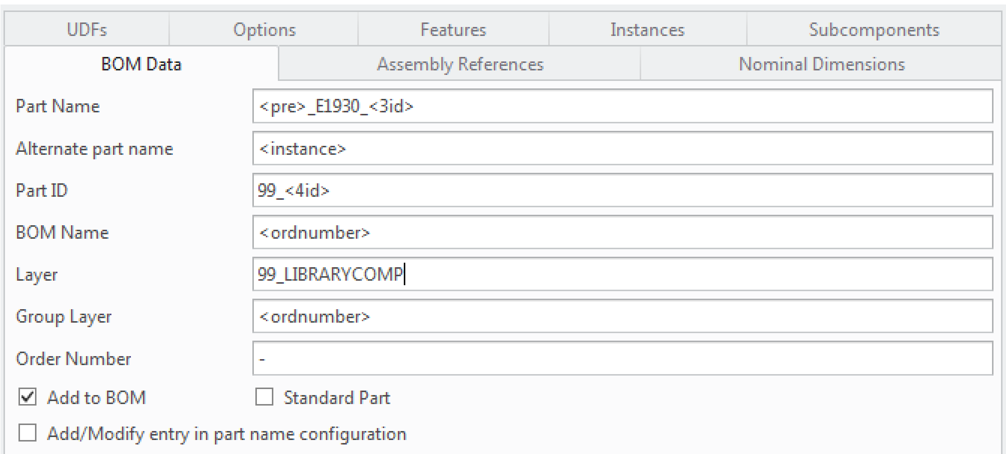
2. .Click **Open model or *.dat file**

   The **File Open** dialog box appears. Select `e1920.prt` from current working directory. The dialog is filled with default library component information.



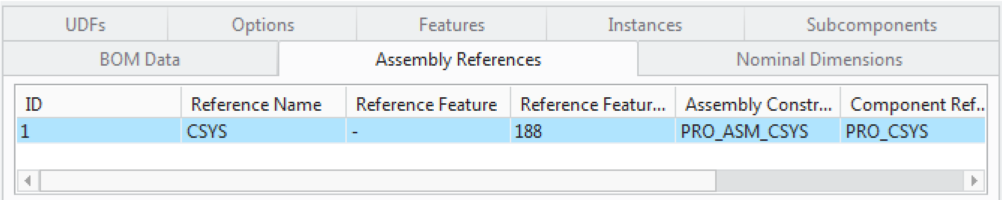
3. Reorient the model and click  to update the image for this library component. This image will be displayed in the **Library Component** dialog box later.

4. Define the **BOM data** for this component.

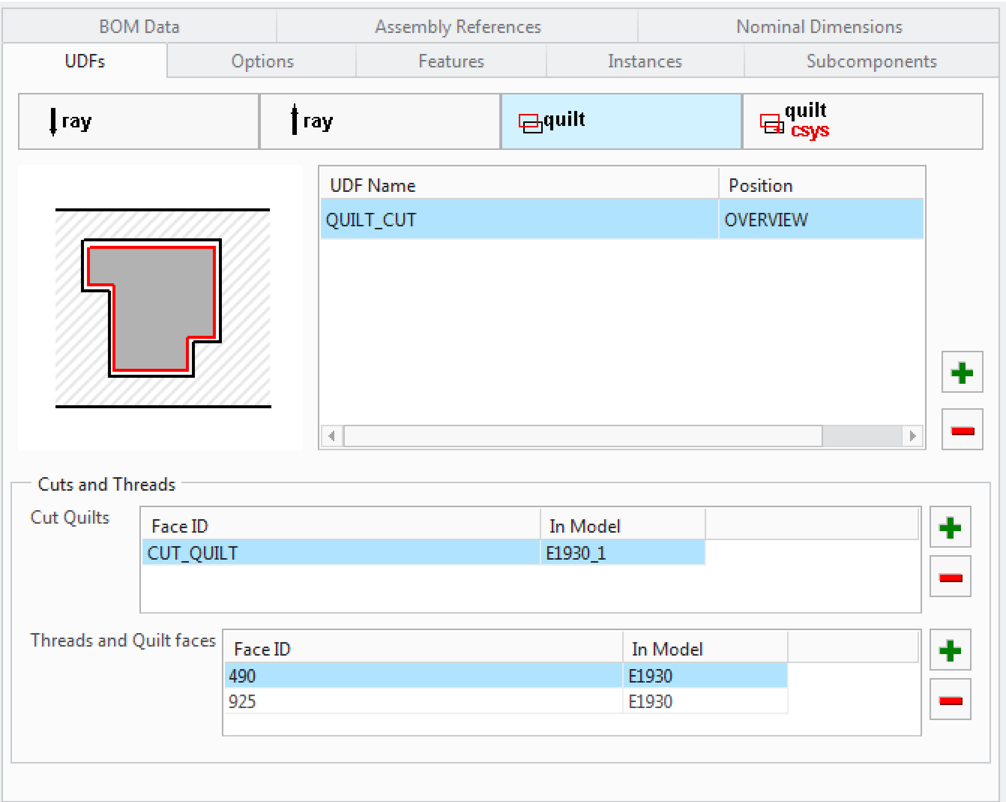


5. Got to the tab **Assembly references**.

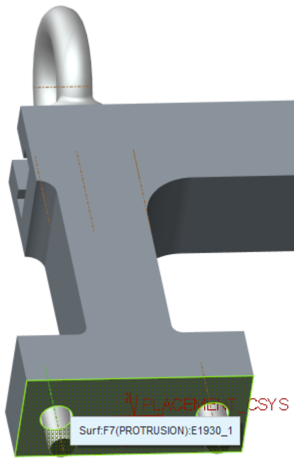
6. Set up `PLACEMENT_CSYS` as assembly reference



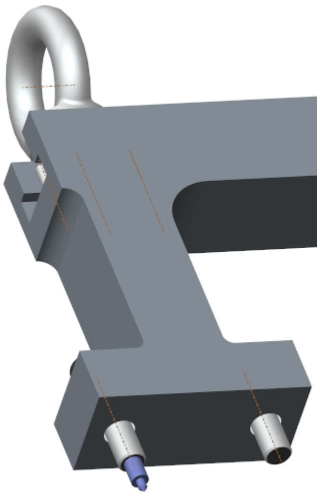
7. Got to the tab **UDFs**.
8. Create **Threads and Quilt Faces** in the bottom area of the **Cuts and Threads** tab.



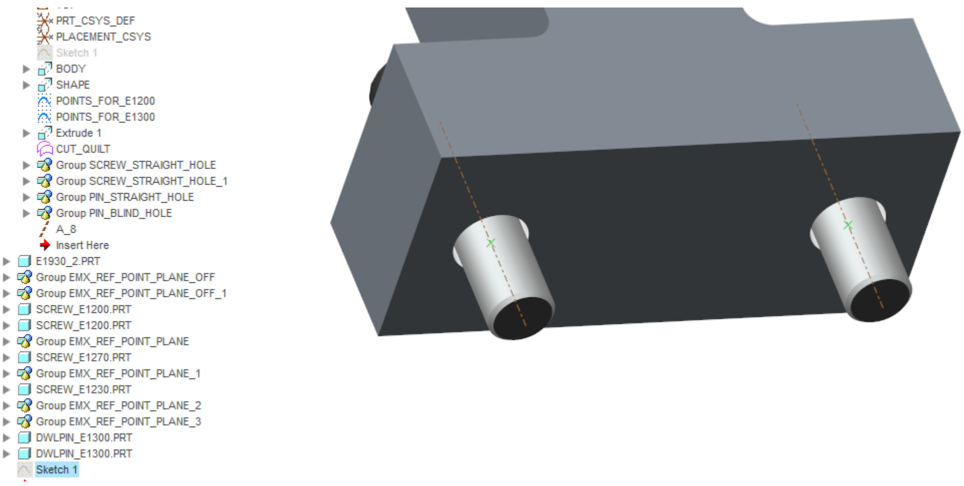
9. Add a new **Threads and Quilt Face** with . You are prompted to select an **Axis** and **Surface**.



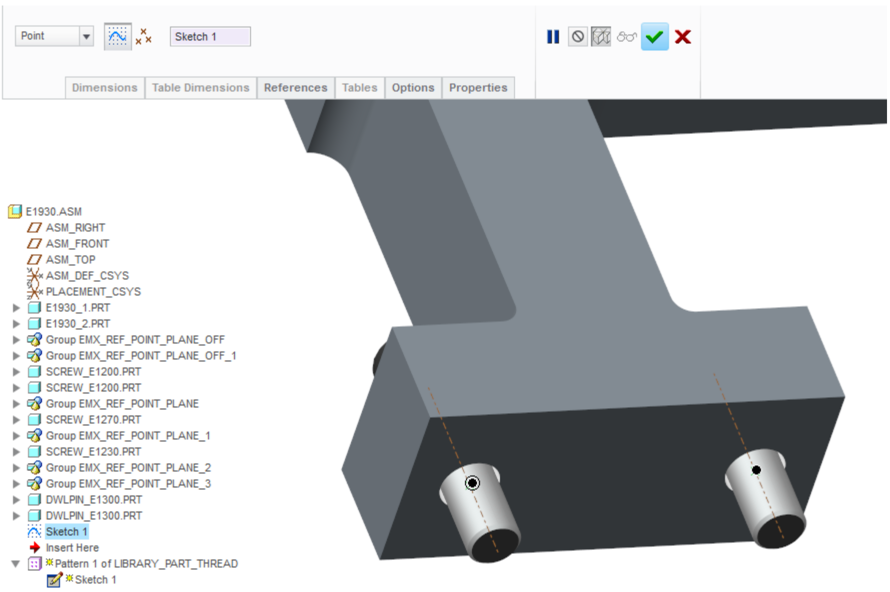
EMX will add a **Predefined Thread** UDF.

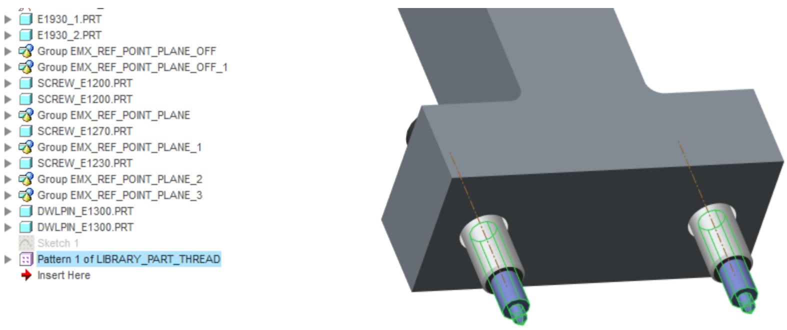


10. Pattern this UDF by creating a **Sketch** which contains 2 datum points directly in the center of the two bores.



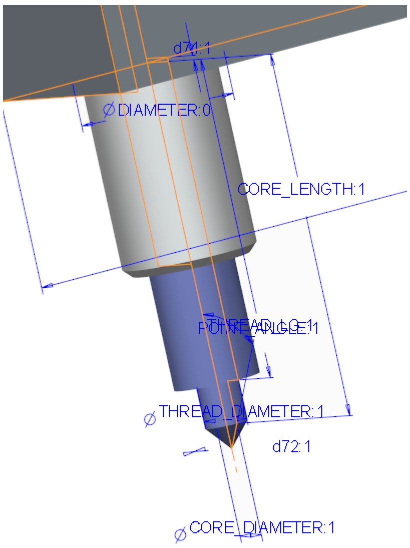
11. Create a pattern of the **Predefined Thread** UDF.



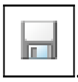


NOTE:
For each instance of the assembly change the dimensions of the **Predefined Threads**.

12. Go to the **Nominal Dimensions**.
13. Add the dimensions:



THREAD_DIAME...	THREAD_DIAME...	PULLDOWN
CORE_DIAMETER	CORE_DIAMETER	PULLDOWN
THREAD_LG	THREAD_LG	PULLDOWN
CORE_LENGTH	CORE_LENGTH	PULLDOWN
POINT_ANGLE	POINT_ANGLE	PULLDOWN

14. Save the assembly with .

    As a result you will find the `E1930.dat` file and the picture `E1930.gif` in `<library_emx90>/tutorial` folder.

30.11.9 Final Changes to the E1930.dat File

The `E1930.dat` file contains all information to control the component. This `dat-file` can be change manually as well.

NOTE:
Be aware of what you are doing. Wrong entries in the dat file can cause trouble in EMX.

1. Open the file within a Text Editor.
2. Go to the Section `EMX_NOM_VALUES`.

   This section contains all nominal dimension shown in the **Library** Component dialog box.

   The only "nominal dimension" which will be set up is the component `E1930_1`. This is the leading component and can later be selected as **PULLDOWN** in the **Library** Component dialog box.
3. Comment out the dimension lines of the **Predefined Thread**. We do not want to change them in the dialog box, this dimensions will be set thru the instance lines.

```
!listing of nominal dimensions
!NAME   DISPLAY_NAME     DIM_NAME         TYPE     RELATION
EMX_NOM_VALUES
e1930   E1930_1 E1930_1 PULLDOWN
!e1930  THREAD_DIAMETER DIAMETER          PULLDOWN_EDIT
!e1930  CORE_LENGTH     CORE_LENGTH       PULLDOWN_EDIT
!e1930  THREAD_LG       THREAD_LG         PULLDOWN_EDIT
!e1930  POINT_ANGLE     POINT_ANGLE       PULLDOWN_EDIT
!e1930  CORE_DIAMETER   CORE_DIAMETER     PULLDOWN_EDIT
EMX_NOM_VALUES_END
```

4. Go ahead to the section `EMX_COMPONENTS`. In this section you let EMX know which component is part of the assembly and where EMX can find the component information.

   The components `E1930_1` and `E1930_2` can be found in the **SUBFOLDER** `library`. The **SUPPLIER** is `meusburger`. This is exactly the folder structure that is set up.

   All other components can be found in their regular components folders, e.g. `E1300` in **SUBFOLDER** `dowel_pin` and **SUPPLIER** `meusburger`.

```
!listing of assembly-subcomponents
!NAME  DISPLAY_NAME  UNIT  SUBFOLDER  SUPPLIER    FILE          TYPE      SIDE
EMX_COMPONENTS
e1930  E1930_1       mm    library    meusburger  E1930_1       E1930_1   ES
e1930  E1930_2       mm    library    meusburger  E1930_2       E1930_2   ES
e1930  E1300         mm    dowel_pin  meusburger  dwlpin_E1300  E1300     ES
e1930  E1270         mm    screw      meusburger  screw_E1270   E1270     ES
e1930  E1230         mm    screw      meusburger  screw_E1230   E1230     ES
e1930  E1200         mm    screw      meusburger  screw_E1200   E1200     ES
EMX_COMPONENTS_END
```

5. Finally, go to the section `EMX_INSTANCES`. The instances are set up similar to a part library component with one exception: For each sub component a valid component instance needs to be provided.

   Additionally, the dimension for the **Predefined Thread** features need to be listed here:

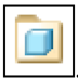
```
EMX_INSTANCES
TYPE   E1930_1      E1930_2        E1270     E1200        E1300        E1230        THREAD_DIAMETER
e1930  E1930_1_44   E1930_2_10_10  E1270_10  E1200_8_30   E1300_6_20   E1230_10_16  8.000
e1930  E1930_1_64   E1930_2_10_10  E1270_10  E1200_10_40  E1300_6_20   E1230_10_16  10.000
e1930  E1930_1_82   E1930_2_16_12  E1270_16  E1200_12_45  E1300_8_20   E1230_12_16  12.000
e1930  E1930_1_100  E1930_2_20_16  E1270_20  E1200_16_60  E1300_10_20  E1230_16_30  16.000
EMX_INSTANCES_END
```

```
CORE_LENGTH  THREAD_LG  POINT_ANGLE  CORE_DIAMETER  INSTANCE         ORD_NUMBER       USER_NUMBER  PRICE
20           16         61           6.65           E1930_44_80      E1930/44/80      -     -
24           20         61           8.38           E1930_64_97      E1930/64/97      -     -
27           23         61           10.11          E1930_82_107     E1930/82/107     -     -
33           29         61           13.84          E1930_100_155    E1930/100/155    -     -
```

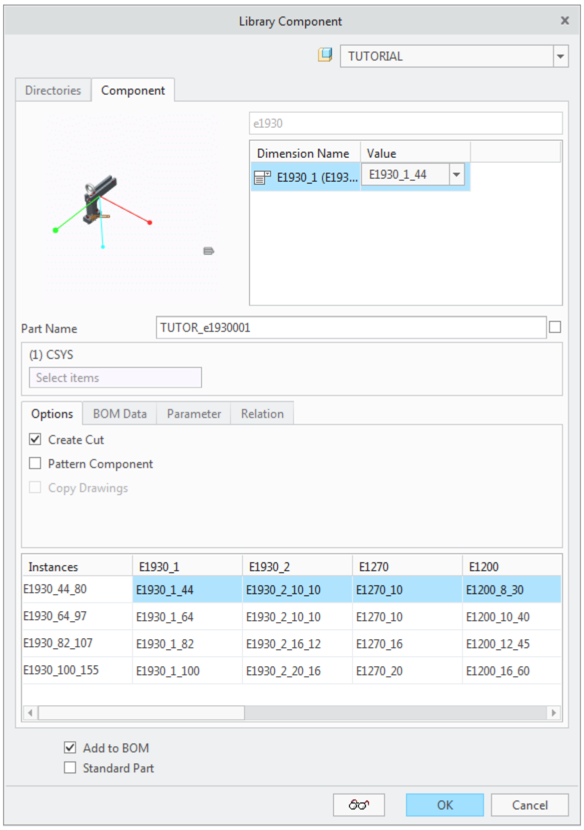
NOTE:
Write the instances one after another. In the tutorial they are simply displayed in two separated picture because of space restrictions.

30.11.10 Assemble the New Library Assembly E1930

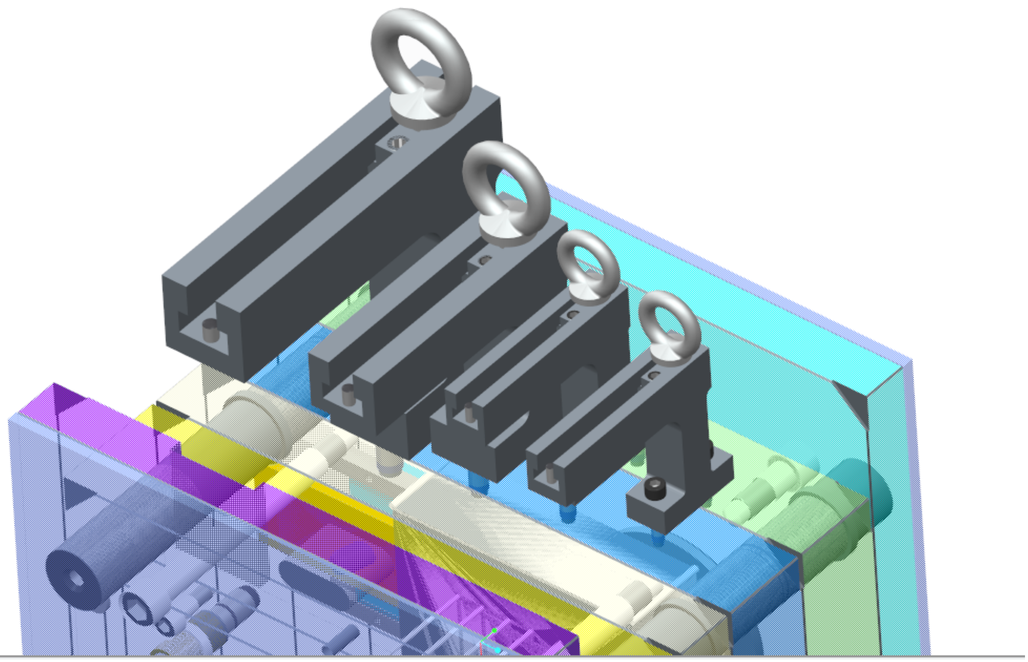
If everything is set up properly you can now assemble the new **Library Component**. Use the already provided `E1930` from the `library/meusburger` folder as a comparison reference.

1. Open the mold base `TUTORIAL.ASM`
2. Add a coordinate system to the mold base with the appropriate orientation.
3. Select **EMX Components ► Library ► Assemble** .
4. Select the directory **meusburger** from the directory tree list.

5. Double-click **e1930** .



6. Select on of the 4 available instances and leave the dialog with **OK**.As a result you will find the **Assembly Component** add to the mold base.



Background Information

When working with complex library components they might have several independent cut quilts and generate a huge amount of individual cuts in plates or other models. During the redefine of such library components you might face problems with the cut generation. If you set the **EMX Option** `UPDATE_CUTS_IN_REDEFINE` to `YES`, you force EMX to simply remove previous cuts and generate new ones.

30.12 About Migrating comp_data.txt and udf_data.txt Files to *.dat Files

1. Click **EMX** ►  **Component Editor** or click **EMX Assembly** ► **Administrator Tools** ► **Component Editor**. The **Component Editor** dialog box opens.
2. Select **Split comp_data.txt file**. The **Open comp_data.txt-file to split its content into separate *.dat-files** dialog box opens.
3. Select an EMX 9 formatted `comp_data.txt` file.

   `%EMX_INSTALL%/components/mm/cooling/din/comp_data.txt`

   The splitting process is triggered and the progress is shown in the status bar. As a result the new `*.dat` files will be written to the component data folder.
4. Erase all content from the old `comp_data.txt` files except the section EMX_TYPES
5. Proceed with all `comp_data.txt` files of the different suppliers for one component type, such as all suppliers of type cooling.

   `%EMX_INSTALL%/components/mm/cooling/d-m-e/comp_data.txt`

   `%EMX_INSTALL%/components/mm/cooling/dms/comp_data.txt`

   `%EMX_INSTALL%/components/mm/cooling/hasco/comp_data.txt`
6. To finish the migration process delete `udf_data.txt` file in the folder

   `%EMX_INSTALL%/components/mm/cooling/udf/udf_data.txt`

30.13 About Creating Copies of Existing *.dat Files

1. Click **EMX** ► **Administrator Tools** ►  **Component Editor** to open the **Component Editor** dialog box.
2. On the bottom right of the dialog box, in the **New name** box, enter a new name for the new component `*.dat` file,
3. In the box to the left of **New name**, select a component type.
4. Click **Copy *.dat file from an existing one**. A generic **Component** dialog box opens. The content is populated from the archive folder. This folder contains a list of `*.dat` file templates that you can use for copying `*.dat` files.
5. Select the `*.dat` file to use for copying and click **OK**. A new `*.dat` file and the associated image and detail images are copied into the current working directory

6. In the **Component Editor** dialog box, you can modify the new `*.dat` file.
7. Click **Close** to close the dialog box.

31

Administrator Tools - Admin Tutorial

31.1 About Administrator Tools

Click **EMX** or **EMX Assembly** and then click **Administrator Tools** to access the tools described below:

- —Opens the **Mold Base Editor** dialog box.
- —Opens the **Component Editor** dialog box.
- **Add Drawing Formats**—Adds model parameters and drawing formats.
- **Verify unused models**—Verifies whether models in the current working directory are in use.
- **Replace Parameters**—Replaces an existing set of parameters with new parameters defined in the file <installdir>/configuration/param_replace.cfg.

31.2 About Verifying Unused Files

Click **EMX Assembly ► Administrator Tools ► Verify unused models** to identify and save unused assembly *.prt and *.asm files to the emx_unused_mdls.inf file in the current working directory. The emx_unused_mdls.inf file appears when the verification process is complete.

31.3 About Adding Drawing Formats

You can customize plate drawings using the following steps:

1. Click **EMX Assembly ► Administrator Tools ► Add Drawing Formats** and choose the format to use with the current drawing.
2. Click **OK** in the **Update all drawings in this directory?** dialog box to apply the format to all drawings in the current directory or click **Cancel** to add the format to only one drawing.

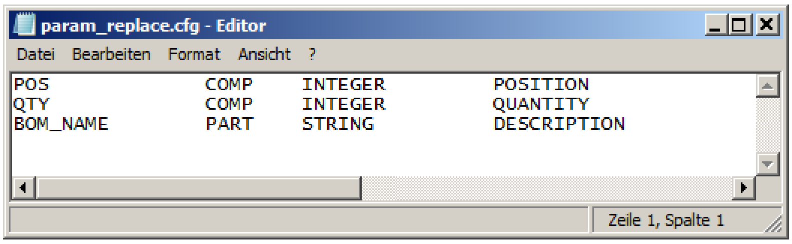
31.4 About Replacing Parameters

1. Open an assembly to modify.
2. Enter new parameter names in the file <installdir>/configuration/param_replace.cfg.
3. Click **EMX Assembly ► Administrator Tools ► Replace Parameters**.
4. The contents of the old parameters are copied to the new parameters and are removed from all models of the current assembly.

31.5 To Replace Parameters in existing Mold Base Assembly

EMX can change the parameter set for all models in the mold base assembly. Therefore create a file `configuration/param_replace.cfg`.

This file has four columns: Original Parameter name, Owner (COMP or PART), Type (DOUBLE, INTEGER, STRING), New Parameter name.




```
POS        COMP   INTEGER    POSITION
QTY        COMP   INTEGER    QUANTITY
BOM_NAME   PART   STRING     DESCRIPTION
```

1. Start the function **Administrator Tools ► Replace Parameters**.

   EMX reads the parameter values from all models, creates the new parameters, copies the values and deletes the original parameters.

Notes:

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The Mold Base Editor - Admin Tutorial

32.1 About the Mold Base Editor

Click **EMX Assembly ► Administrator Tools ► Mold Base Editor** to open the **Mold Base Editor** dialog box. Use this dialog box to modify the mold base supplier database.

32.2 About Setting Overall Dimensions in Mold Base Editor

1. Click **EMX Assembly ► Administrator Tools ► Mold Base Editor**. The **Mold Base Editor** dialog box opens.
2. Select the **Overall Dimensions** tab.
3. Select the unit of measurement as **mm** or **inch**. When the unit is selected, the list of available suppliers is updated.
4. Select a supplier from the list of existing suppliers.
5. Select a mold base to edit from the list of existing mold base sizes. New sizes are automatically added to the list.
6. Select the default supplier for **Supplier Plates**, **Supplier Screws**, **Supplier Equipment**, **Supplier Guides**, **Supplier Stop System**, and **Supplier Return Pin**.
7. Select **Rectangular** or **Round** mold base type.
8. Enter a new **Mold Base Size**.
9. Enter a value for the **Rail Width** to create a new rail size.
10. Select a dimension name in the **Feature Dimensions** list. The name and value appear for the selected dimension.
11. Click or to add or remove the selected dimension from the list.
12. Click to save the current settings or click **Close** to close the dialog box.

32.3 About Setting Pattern Dimensions in Mold Base Editor

1. Click **EMX Assembly ► Administrator Tools ► Mold Base Editor**. The **Mold Base Editor** dialog box opens.
2. Click **Pattern Dimensions**.
3. Select the active pattern from the **Pattern Name** list.
4. Preview the active pattern.
5. View or edit the **Pattern Diameter**.
6. Set the pattern **Quantity**.
7. Set the **Pattern Offset** value. This is the x-offset of the first instance for a metric database and the second instance for an imperial database.
8. In the **Pattern Width** box, type a value for the x-direction.

9. In the **Pattern Length** box, type a value for the y-direction.
10. The new instance appears on the pattern list. A visible instance is indicated by . Click the first column to toggle visibility on and off.

32.4 About Setting Plate Types in Mold Base Editor

1. Click **EMX Assembly ▸ Administrator Tools ▸ Mold Base Editor**. The **Mold Base Editor** dialog box opens.
2. Select the **Plate Type** tab. The following plate types are available:
    - plate_1—Clamping plate.
    - plate_2—Ejector bottom plate.
    - plate_3—Intermediate/Support plate.
    - plate_4—Rails (one model).
    - plate_4_left—Rails (two models).
    - plate_5—Cavity plate.
    - plate_6—Ejector base plate.
    - plate_7—Ejector retainer plate.
    - plate_8—stripper plate.
3. Click a line in the list to activate a plate instance.
4. Edit values of the activated plate instance.
5. Click to add a new plate instance to the list, or click to remove the active plate instance from list.

32.5 About Setting Plate Features in Mold Base Editor

1. Click **EMX Assembly ▸ Administrator Tools ▸ Mold Base Editor**. The **Mold Base Editor** dialog box opens.
2. Select the **Plate Features** tab.
3. To activate a feature, select a row in the list of plate features.
4. Edit the plate name and display name if required. You can also select a feature name from the list.
5. Edit the feature dimensions.
6. Click to add a new plate feature to the list, or click to remove the active plate feature from the list.

Notes:

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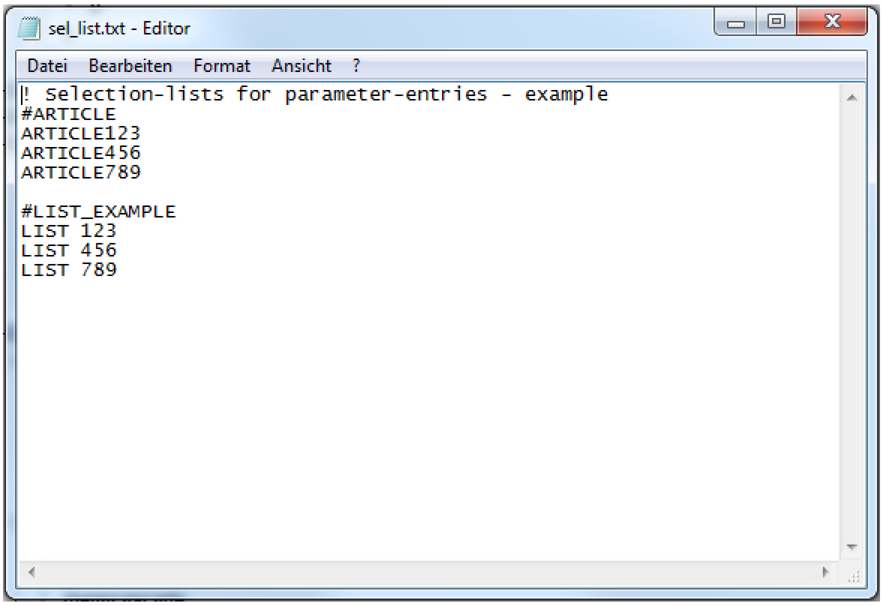
Additional Configuration Capabilites - Admin Tutorial

Create a Quick Selection pull down menu with sel_list.txt
Working with Standard Parts from Windchill
To Use Relation Files (*rel) to Add Relations to Components

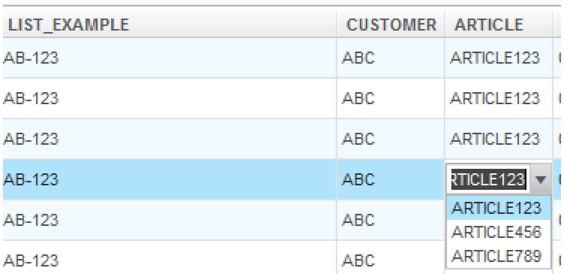
33.1 Create a Quick Selection pull down menu with sel_list.txt

For some parameter it is required to only allow a list of valid parameter values. This can be achieved with the sel_list.txt files in the configuration folder.

1. To define a list of values (like for ARTICLE or LIST_EXAMPLE), edit the file `/configuration/sel_list.txt` as follows:



2. Identify the Parameter with a "#"-symbol. All following lines are the values that will be displayed. You still can manually enter any value, even a drop-down list exists.



33.2 Working with Standard Parts from Windchill

To assemble standard parts from Windchill several configuration tasks need to be done.

33.2.1 Provide a Custom Configuration using the EMX _USER_CONFIG_PATH variable

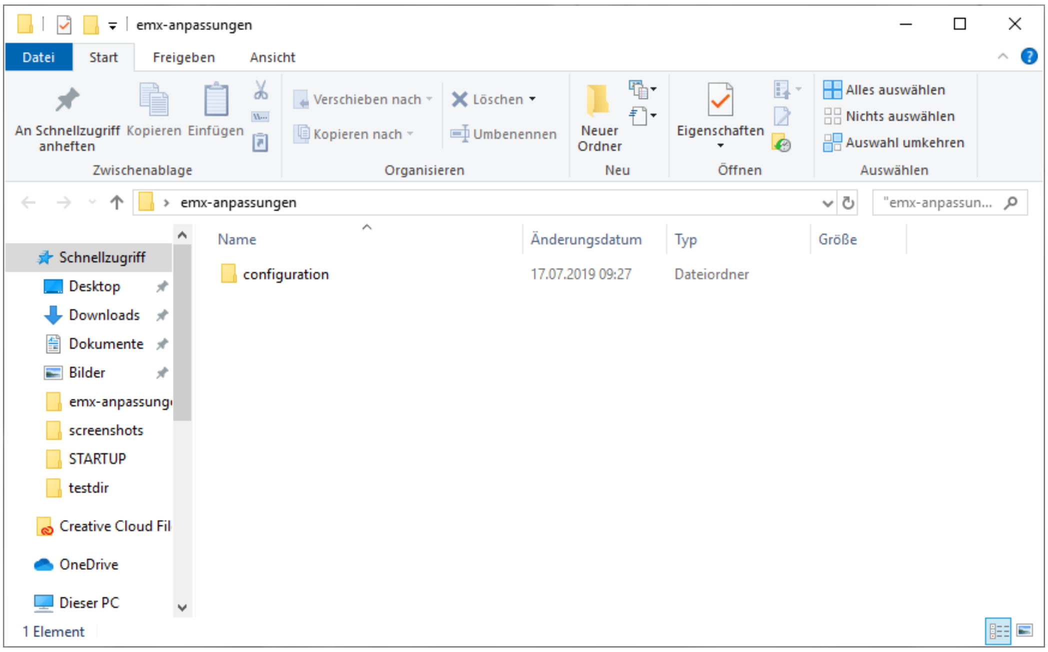
In the beginning it is necessary to provide a custom configuration.

The goal is that all users within a company are working with the same configuration.

To fulfill this goal several tasks need to be done:

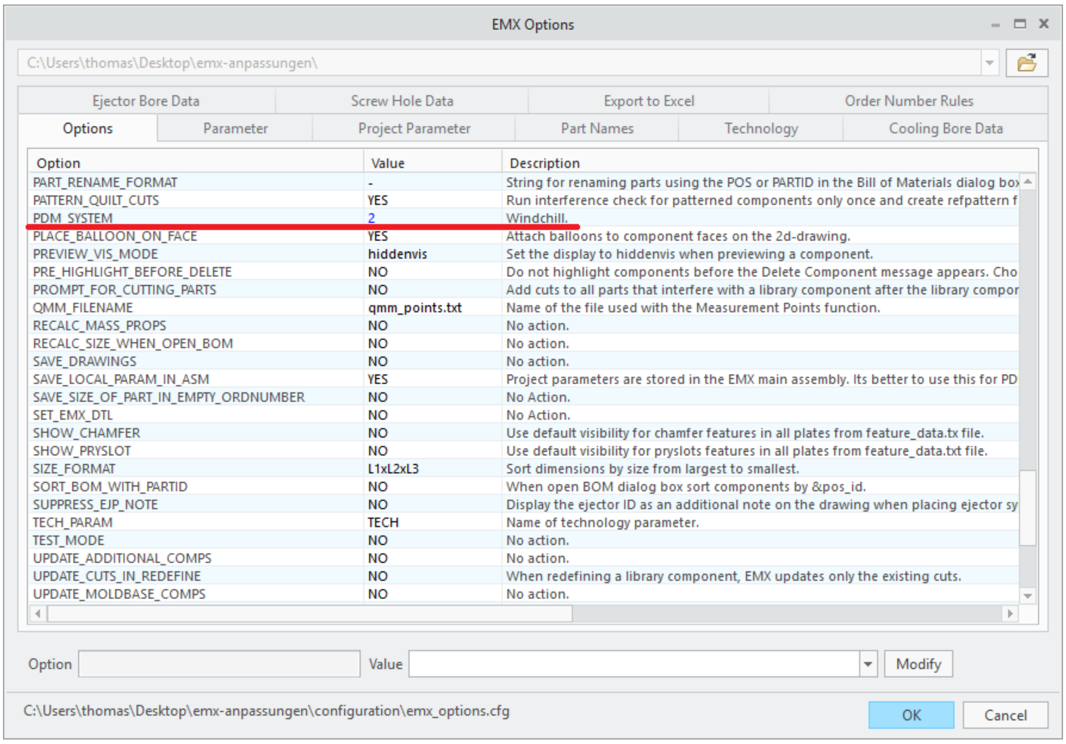
1. The standard configuration folder from the EMX installation `"emx-installation-path\configuration"` needs to be copied to a server location which is visible for all users, i.e. `"<server-path>\emx-adaptions\configuration"`.
2. This folder should have writing restrictions for all users and should only be modified by the administrator.
3. Finally, it is required to set the Windows variable `EMX_USER_CONFIG_PATH` to the value `<server-path>\emx-adaptions` for all users.

   EMX will then find the custom configuration at this location.



33.2.2 Set EMX Option PDM_SYSTEM

1. In the **EMX Options**  the configuration option `PDM_SYSTEM` needs to be set to `2 – Windchill`.

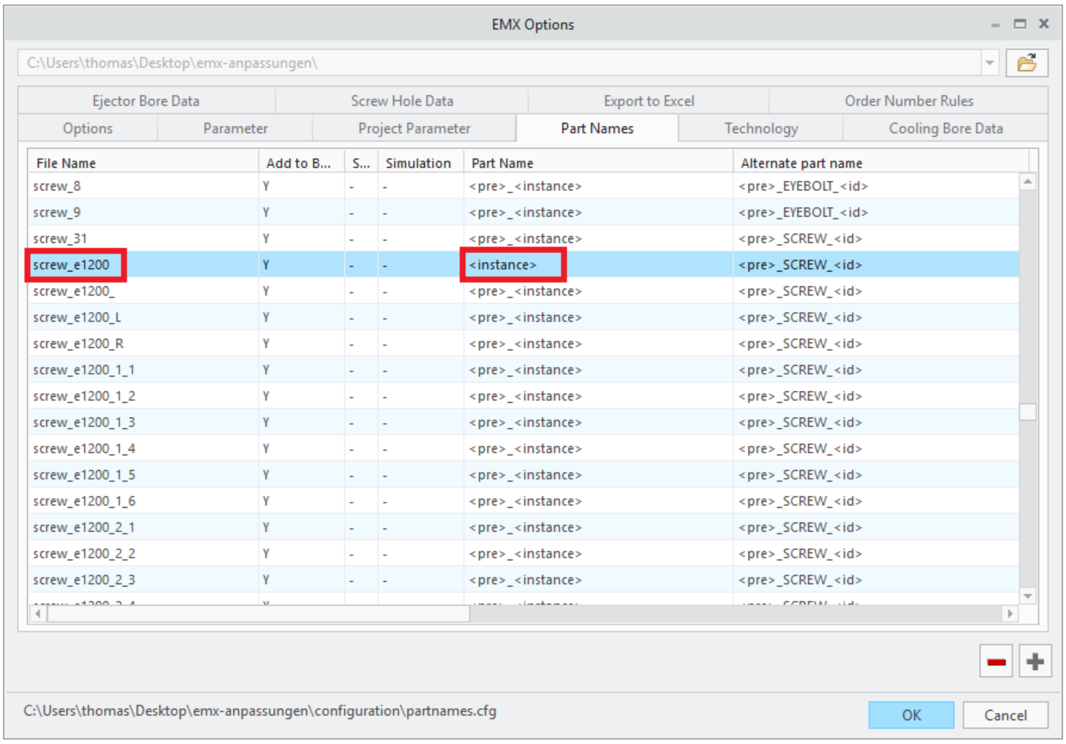


33.2.3 Configure Part Names in EMX Options

In the **EMX Options**  ► **Part Names** the part name rules have to be adapted in a way that the resulting part name is project independent.

ID (<id>) or project prefix (<pre>) have to be removed.

1. For the screw `E1200` with the according template file `screw_E1200.prt` the part name rule is set to `<instance>`.



As a result the name will be, i.e. `E1200_8_60`. These instance names come from the dimension tables in the *.dat file.

33.2.4 Reduce Variety of Selectable Components in the Screw Dialog Box

When working with standard parts from Windchill it make sense to only provide the component types which are actually administered in Windchill.

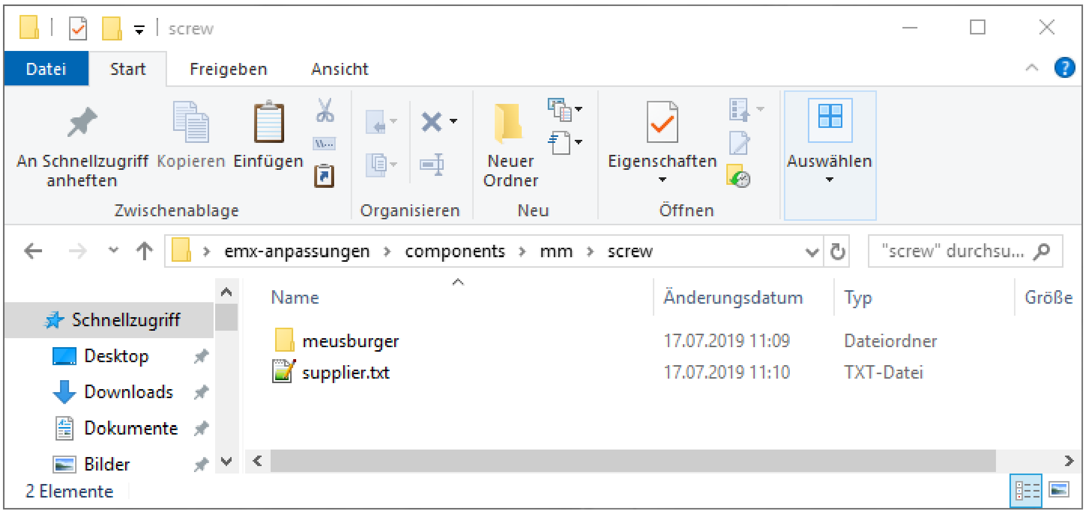
To achieve these goal the available suppliers and screw types need to be reduced.

The supplier list can be restricted by providing a adapted `supplier.txt` file in the correct location. This file needs to be placed to the following path in the custom configuration.

`EMX_USER_CONFIG_PATH\components\mm\screw\supplier.txt`

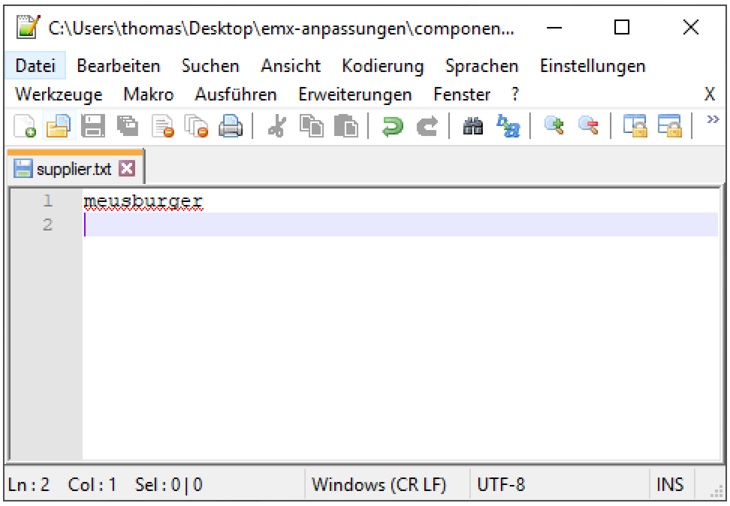
As template, the equivalent file from the EMX standard installation can be copied.

`<emx-installation-path>\components\mm\screw\supplier.txt`



In general can be said: If a file is found in the custom configuration, it has a higher priority compared to the equivalent file from the standard installation and will be used instead.

1. In the given example the content of the `supplier.txt` is therefore reduce to `meusburger` only.

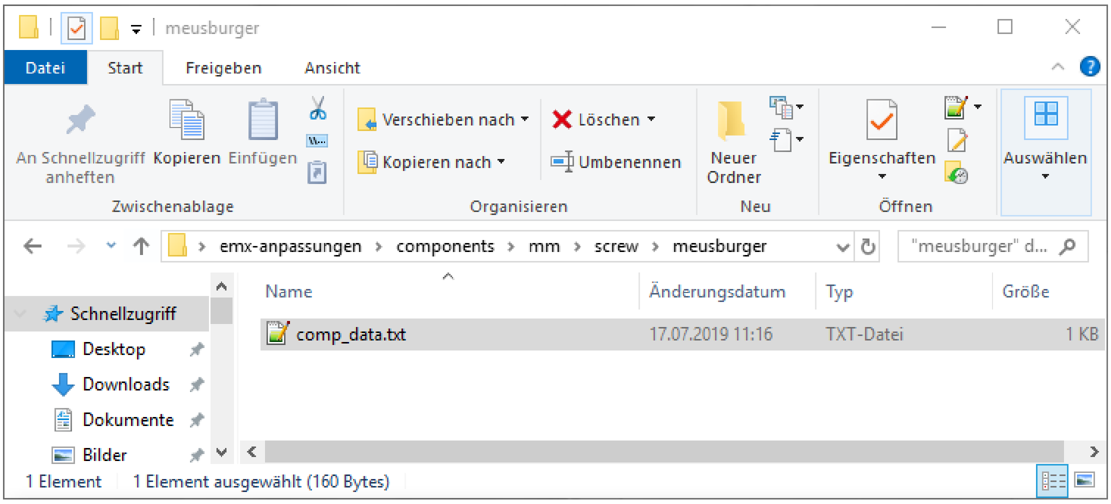


The restrictions for the screw types follow a similar pattern. An adapted `comp_data.txt` needs to be created. The location is however inside a supplier specific folder.

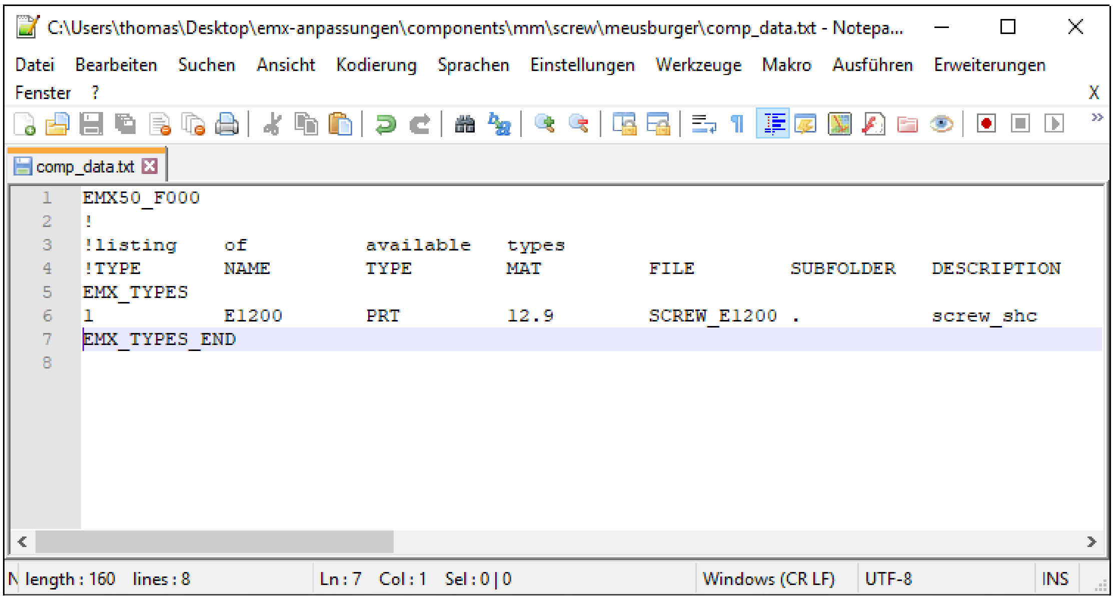
`EMX_USER_CONFIG_PATH\components\mm\screw\meusburger\comp_data.txt`

As template, the equivalent file from the EMX standard installation can be copied.

`<emx-installation-path>\components\mm\screw\meusburger\comp_data.txt`



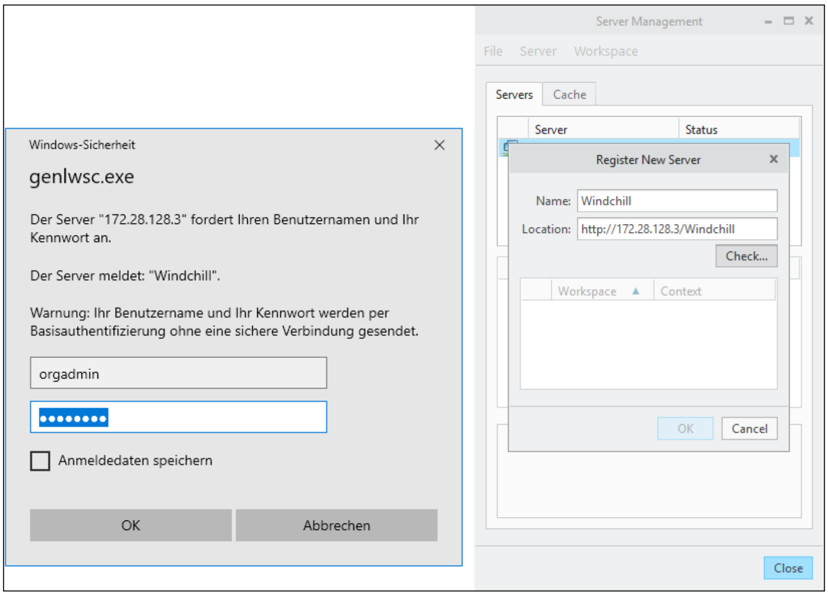
In this `comp_data.txt` all lines except the relevant `E1200` line have to be deleted.



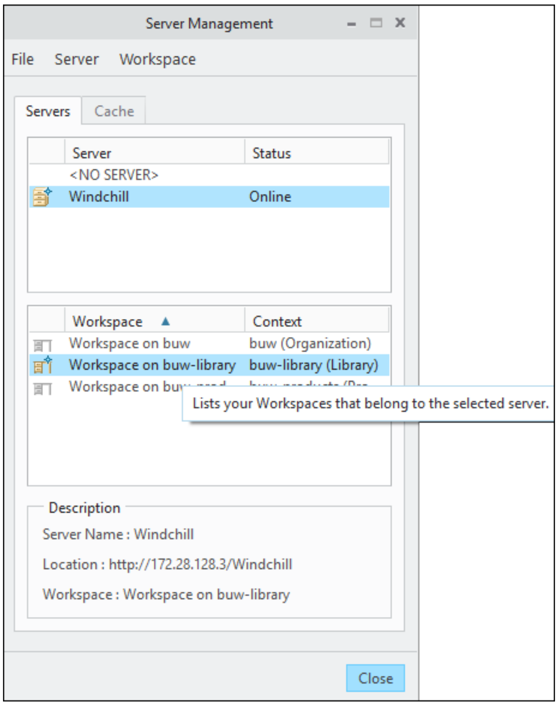
In the screw dialog only `E1200` screws from Meusburger can be assembled.

33.2.5 Create All Screw Instances

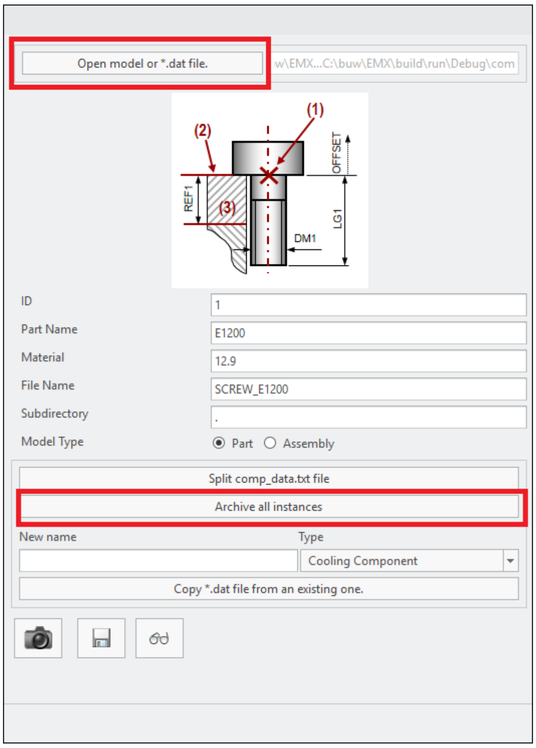
Before creating the all instances of the `E1200` screw one should connect to the Windchill-Server and login as and administrator.



The Workspace for the Windchill library should also be activated.



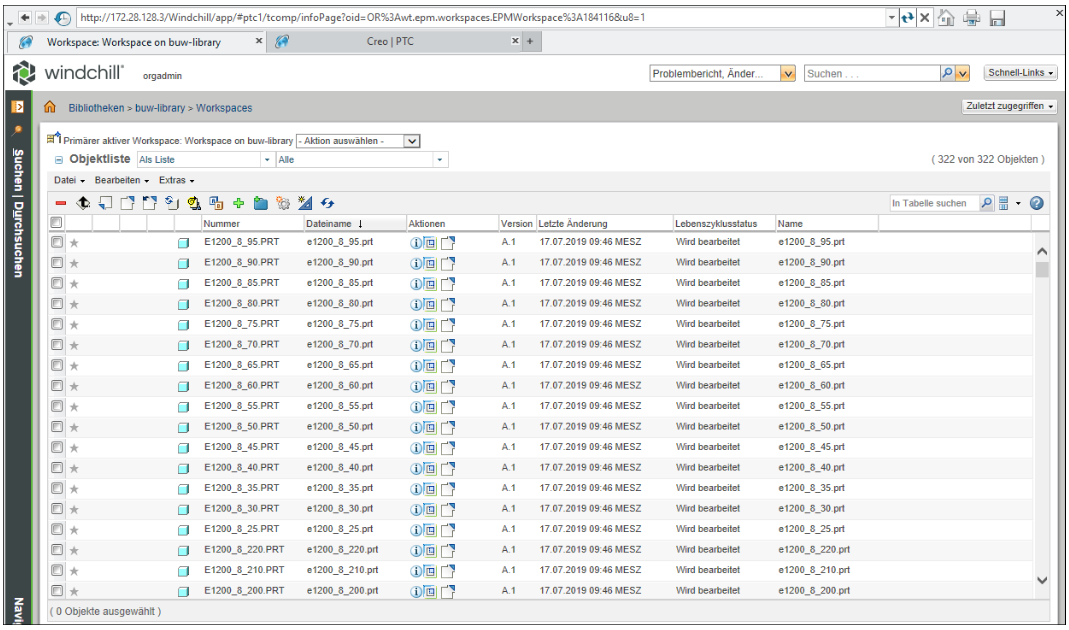
Using the **Component Editor**  all instances of a component can be created.



Therefore theses steps a required:

1. Open the **Component Editor** 
2. Open the file `E1200.dat` from the standard installation `<emx-installation-path>\components\mm\screw\meusburger\E1200.dat`using **Open model or *.dat file**
3. Click the button **Archive all Instances**.

   All Instances are saved to the current workspace.



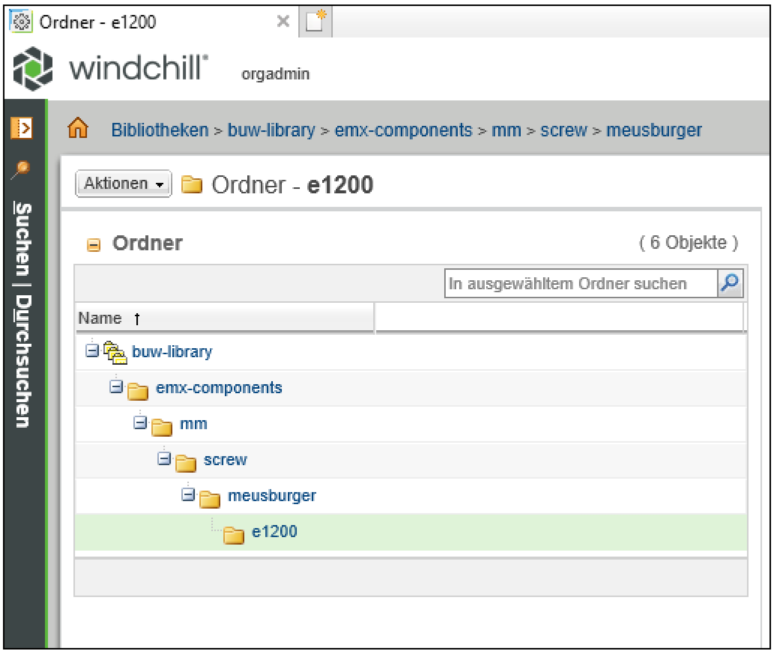
33.2.6 Check in all Instances to the Windchill Library

To provide the created instances to all users it is now necessary to store the models in a structured way in the Windchill library.

It is recommended to use a folder structure which is similar to the folder structure in the EMX standard installation.

For the example screw Meusburger E1200, the following structure is used.

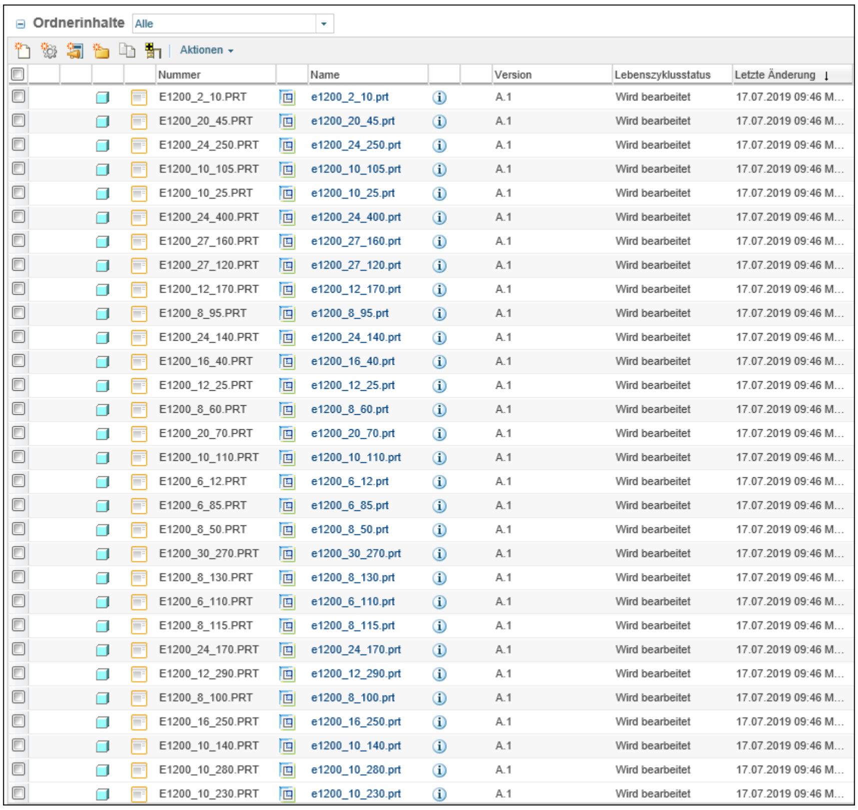
`buw-library` > `emx-components` > `mm` > `screw` > `meusburger` > `e1200`



Using this structure has several advantages. The stored EMX standard parts library can be extended easily step-by-step in the future. Furthermore, it is quite simple to find the models even after a longer period of time.

Using the Windchill Check-In mechanism the files can be stored in the e1200 folder.

After this step EMX is fully configured and screws can now be pulled from Windchill.



Ordnerinhalte Alle

Aktionen

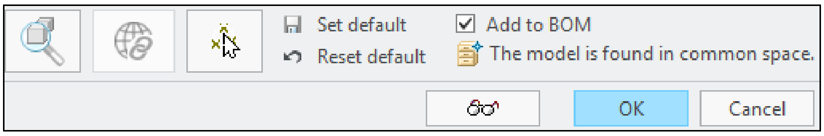
Nummer	Name	Version	Lebenszyklusstatus	Letzte Änderung
E1200_2_10.PRT	e1200_2_10.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_20_45.PRT	e1200_20_45.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_24_250.PRT	e1200_24_250.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_10_105.PRT	e1200_10_105.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_10_25.PRT	e1200_10_25.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_24_400.PRT	e1200_24_400.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_27_160.PRT	e1200_27_160.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_27_120.PRT	e1200_27_120.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_12_170.PRT	e1200_12_170.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_8_95.PRT	e1200_8_95.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_24_140.PRT	e1200_24_140.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_16_40.PRT	e1200_16_40.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_12_25.PRT	e1200_12_25.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_8_60.PRT	e1200_8_60.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_20_70.PRT	e1200_20_70.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_10_110.PRT	e1200_10_110.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_6_12.PRT	e1200_6_12.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_6_85.PRT	e1200_6_85.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_8_50.PRT	e1200_8_50.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_30_270.PRT	e1200_30_270.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_8_130.PRT	e1200_8_130.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_6_110.PRT	e1200_6_110.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_8_115.PRT	e1200_8_115.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_24_170.PRT	e1200_24_170.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_12_290.PRT	e1200_12_290.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_8_100.PRT	e1200_8_100.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_16_250.PRT	e1200_16_250.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_10_140.PRT	e1200_10_140.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_10_280.PRT	e1200_10_280.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...
E1200_10_230.PRT	e1200_10_230.prt	A.1	Wird bearbeitet	17.07.2019 09:46 M...

33.2.7 Assembling EMX Components from Windchill (User)

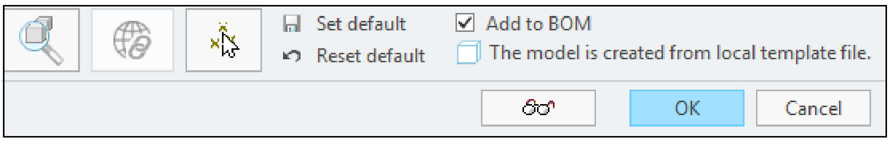
From a user perspective not much changes during the assembly procedure of components.

However, within the screw dialog box the user is now informed about the source of the model.

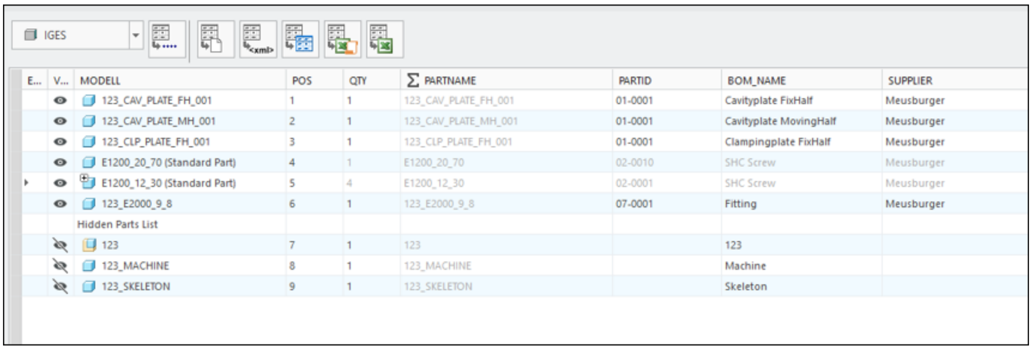
In case a model is found in the common space by the part names rule, an additional icon with a short message **The model is found in common space** is displayed in the bottom area of the dialog.



In case no model was found in common space and the standard assembly procedure is used, an alternative message **The model is created from local template file** appears.



Additionally, the standard part status is also visualized in the Bill of Materials. Parameters can not be edited and are therefore disabled.



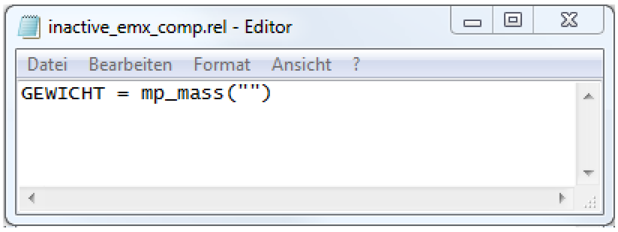
E...	V...	MODELL	POS	QTY	PARTNAME	PARTID	BOM_NAME	SUPPLIER
		123_CAV_PLATE_FH_001	1	1	123_CAV_PLATE_FH_001	01-0001	Cavityplate FixHalf	Meusburger
		123_CAV_PLATE_MH_001	2	1	123_CAV_PLATE_MH_001	01-0001	Cavityplate MovingHalf	Meusburger
		123_CLP_PLATE_FH_001	3	1	123_CLP_PLATE_FH_001	01-0001	Clampingplate FixHalf	Meusburger
		E1200_20_70 (Standard Part)	4	1	E1200_20_70	02-0010	SHC Screw	Meusburger
		E1200_12_30 (Standard Part)	5	4	E1200_12_30	02-0001	SHC Screw	Meusburger
		123_E2000_9_8	6	1	123_E2000_9_8	07-0001	Fitting	Meusburger
		Hidden Parts List						
		123	7	1	123		123	
		123_MACHINE	8	1	123_MACHINE		Machine	
		123_SKELETON	9	1	123_SKELETON		Skeleton	

33.3 To Use Relation Files (*rel) to Add Relations to Components

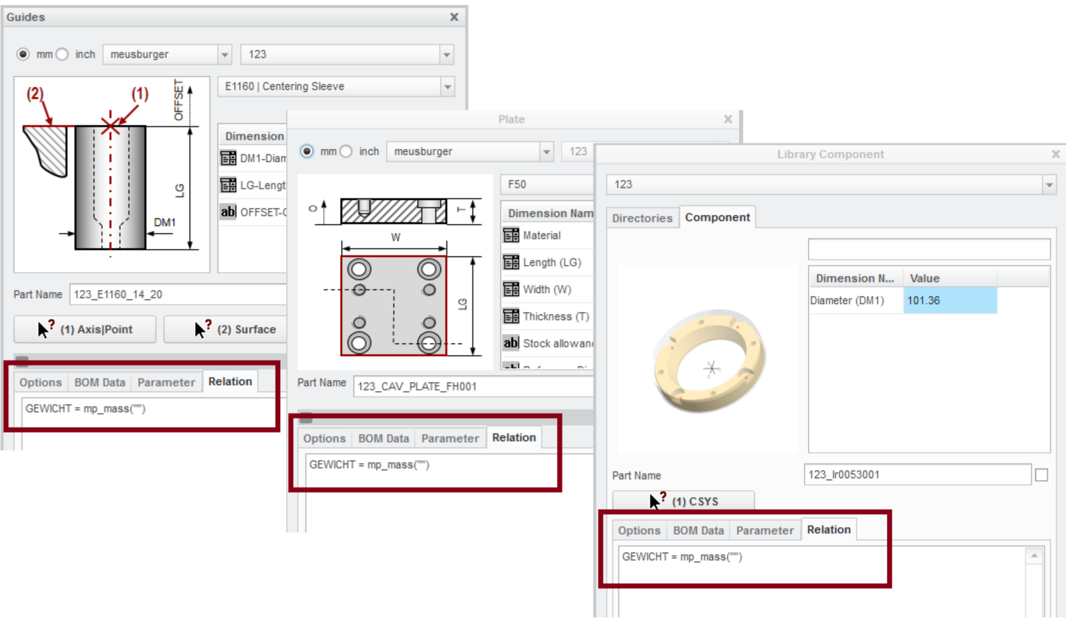
33.3.1 Using the general relation file “emx_comp.rel”

The file `emx_comp.rel` can be used to define a relation for all EMX components (cooling, ejector pins etc.). It will also be applied for components from the EMX library.

NOTE:
In the default installation you will find a file called `inactive_emx_comp.rel`. This is a template file that can beused as a reference. As soon as the file is renamed to `emx_comp.rel` the content will be written into relation area of the Creo part.

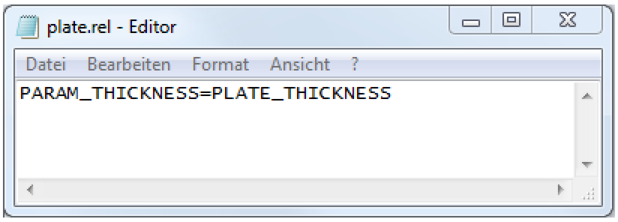


When adding an EMX component you will find the content in the **Plate**, **Component** or **Library Dialog** on the **Relation** tab:



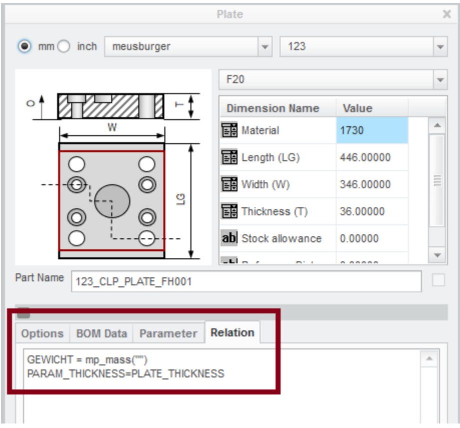
33.3.2 Using plate relation files

In case a relations needs to be added to all plates of one specific supplier the `plate.rel` files can be used.



The `plate.rel` file needs to be added to the location `<emx_install>\mm\components\plates\<supplier>\`

This relation will be written into the **Relation** tab of the **Plate Dialog**



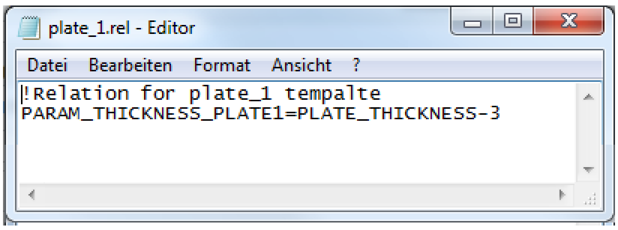
NOTE:
The relation from `emx_comp.rel` will not be overwritten, instead it will be added underneath the existing relations.

33.3.3 Using plate type specific relation files

In case a plate type specific relation needs to be added the `plate_#.rel` file can be used.

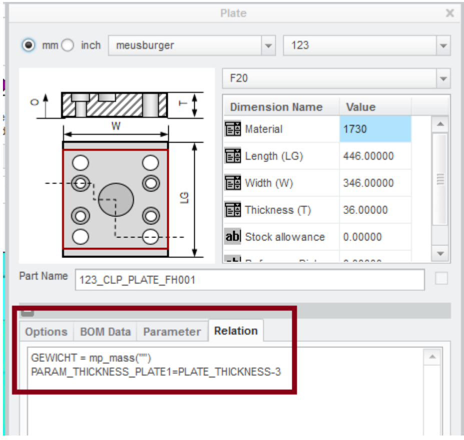
In this example a relation file for the plate template `plate_1.prt` is used.

In case the plate type specific file `plate_1.rel` exists the content of `plate.rel` is ignored and the content of the type specific relation is used.



The `plate.rel` file needs to be added to the location `<emx_install>\mm\components\plates\<supplier>\`

This relation will be written into the **Relation** tab of the **Plate Dialog**



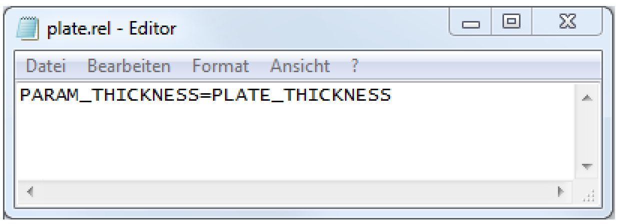
NOTE:
In EMX eight default plate template types exist. `plate_1.prt` is used as default for clamping plates. Other template types are as follow:

- `plate_1`: Clamping Plate
- `plate_2`: Core Pin Retainer Plate
- `plate_3`: Support Plate
- `plate_4`: Rail Plate
- `plate_5`: Cavity Plate
- `plate_6`: Ejector Retainer Plate
- `plate_7`: Ejector Base Plate
- `plate_8`: Stripper Plate

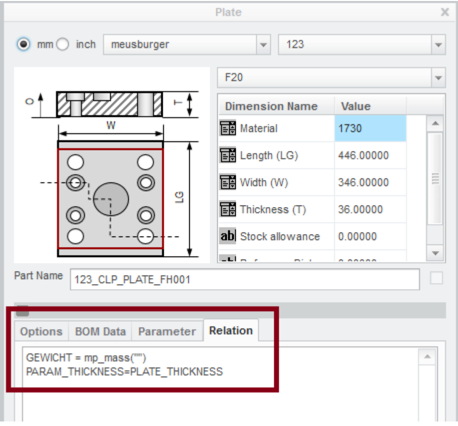
33.3.4 Using component relation files

Similar to plates, relations can be added to other components with the component relation files. In this example the screw specific file is used. This can be done for other component types as well.

In case the file `screw.rel` is located in the location <emx_intall>\components\mm\screw\



This content will be display in the **Relation** tab of the **Component Dialog**



This relation will be written into the **Relation** tab of the **Plate Dialog**

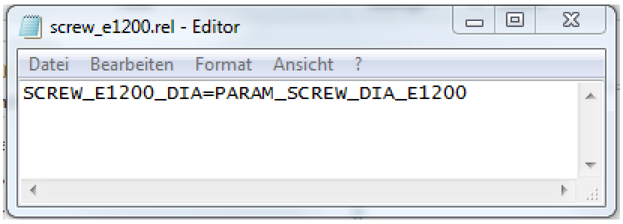
NOTE:
The relation from `emx_comp.rel` will not be overwritten, instead it will be added underneath the existing relations.

33.3.5 Using specific type component relation files

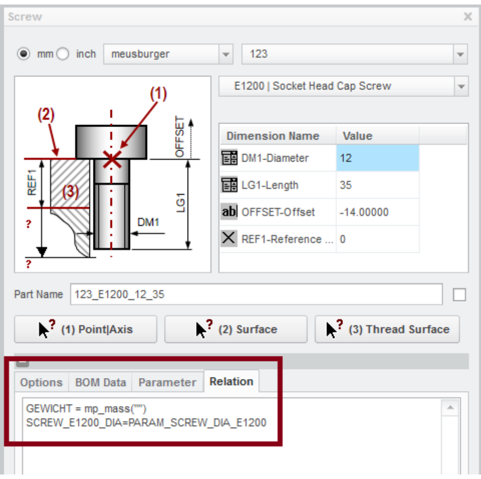
In case the relation should only be applied for one specific template of one supplier you can use type specific relation files.

In this example a screw with the template name `screw_e1200.prt` is used

The `screw_e1200.rel` file needs to be added to the location <emx_install>\mm\components\screw\<supplier>\



This content will be display in the **Relation** tab of the **Screw Dialog**



33.3.6 Adding relations to the Post-Regeneration-Area

In case an relation should be added to the post regeneration are of the part. a relltion file with the post fix `_post_regen` needs to be added in the same way as described above.

`emx_comp_post_regen.rel`, `plate_post_regen.rel` etc,

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