**CATIA - Virtual Design Training**
Foils

# Mold Tooling Design

**Version 5 Release 21**
**January 2012**

**EDU_CAT_EN_MTD_FF_V5R21**

# Course Presentation

## Objectives of the course

**Upon completion of this course you will be able to:**

**- Create a mold base using guided and fixed components**

**- Build a Plastic Injection Mold assembly from scratch**

## Targeted audience

**Mold Tooling Designers**

## Prerequisites

**Students attending this course should have knowledge of CATIA V5 fundamentals and Tooling Design fundamentals**

*8 hours*

# Table of Contents (1/4)

# Table of Contents (2/4)

# Table of Contents (3/4)

*Student Notes:*

# Table of Contents (4/4)

# Introduction to Mold Tooling Design

*You will discover the CATIA V5 Mold Tooling Design user interface and review the general process to create a Mold.*
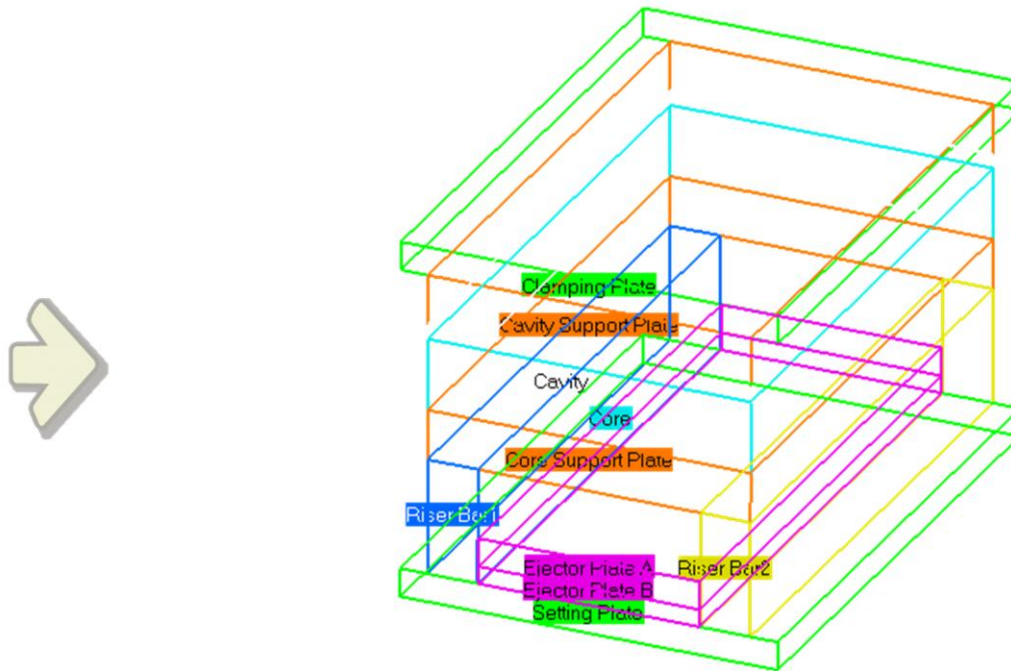
# Accessing the Workbench

**The Mold Tooling Design Workbench is a member of Mechanical Design applications (P2) family:**
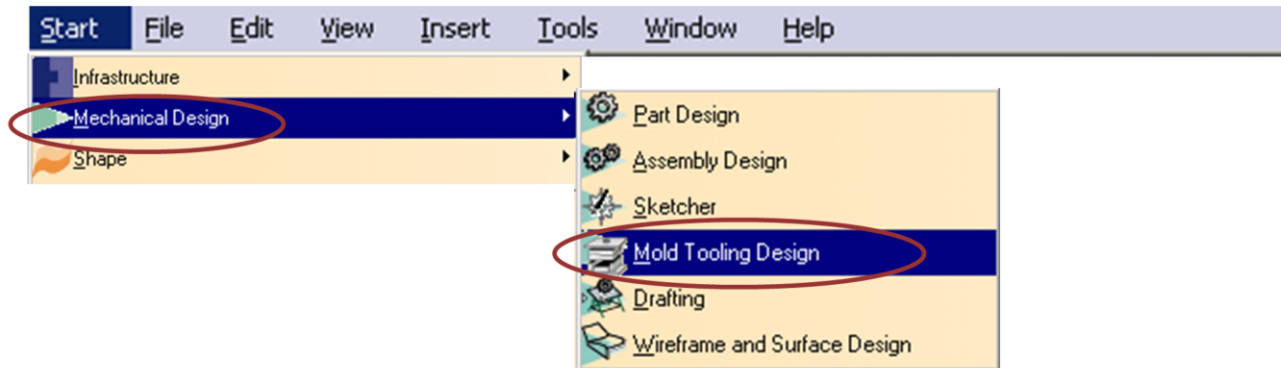
# User Interface Presentation

**Once inside the Mold Tooling Design Workbench, you have access to the:**

A. **General Assembly-level tools (MTD Tools)**

B. **Standard tools**

C. **Dedicated Mold Tooling Design tools**

# Mold Tooling Design Functions (1/2)

**New Mold Base**

**Add Mold Plate**

**Mold Base Elements**

**Add Slider**

**Add Retainers**

**Add Insert**

**Guiding Components**

**Add Leader Pin**

**Add Bushing**

**Add Sleeve**

**Add Locating Ring**

**Add Dowel Pin**

**Locating Components**

**Fixing Components**

**Cap Screw**

**Countersunk Screw**

**Locking Screw**

# Mold Tooling Design Functions (2/2)

Ejection Components → Ejector Pin

Ejector

Flat Ejector

Ejector Sleeve

Core Pin

Stop Pin

Angle Pin

Knockout Pin

Injection Components → Sprue Bushing

Sprue Puller

Support Pillar

O-Ring

Plug

Baffle

Injection Feature: Gate

Injection Feature: Runner

Coolant Channel

Miscellaneous Components → User Component

Eye Bolt

Spring

# Mold Tooling Design Catalogs

## For Mold Bases and components, catalogs of major market providers are available:

**Creation of a
new Mold Base**
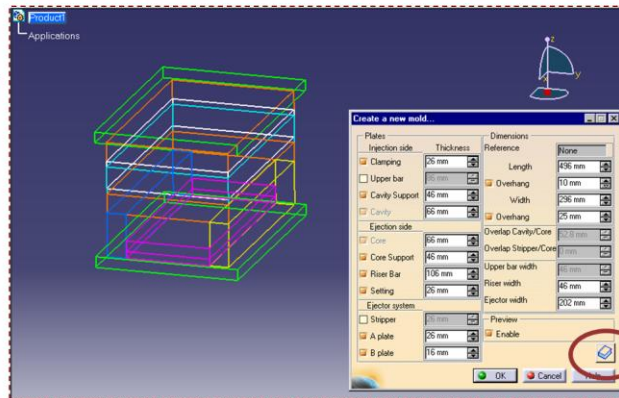
**Creation of a new
component**

**(e.g. Leader Pin)**
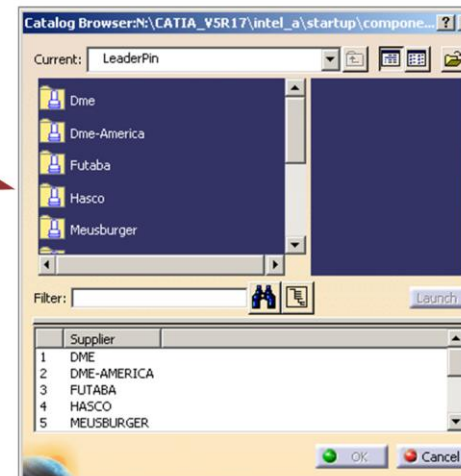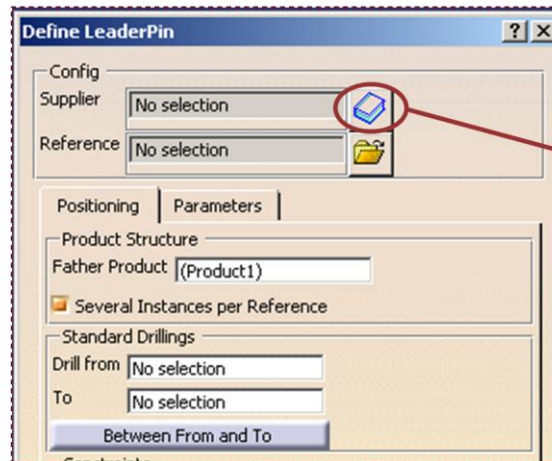
# Specific Tooling Design Settings (1/7)

1. **Select Tools > Options**
2. **Select the item dedicated to Mold Tooling Design in Mechanical Design.**

- **In the Catalogs tab, you can update the location of the Tooling catalogs (field In the Directory field for file-based mode or dedicated definition for catalogs stored in SMARTEAM).**

**Student Notes:**
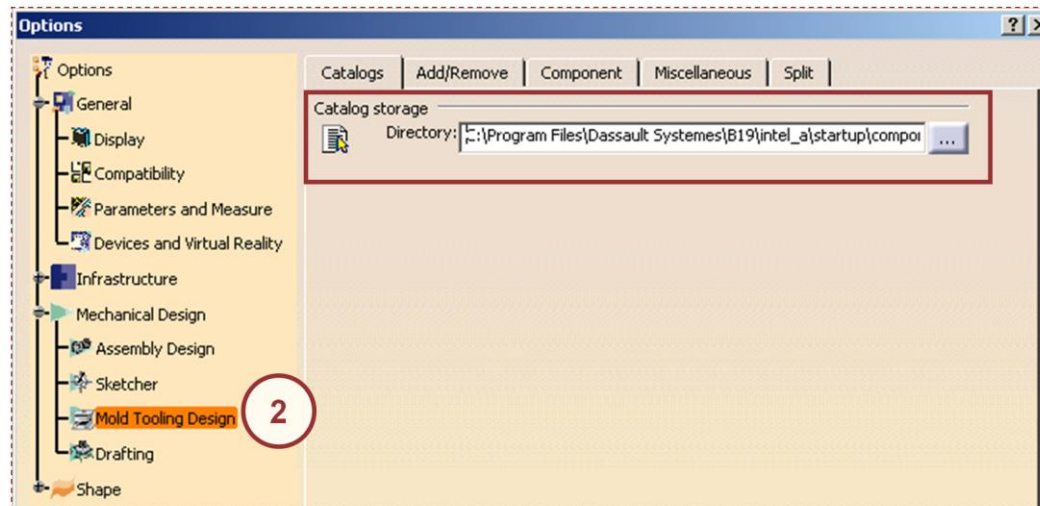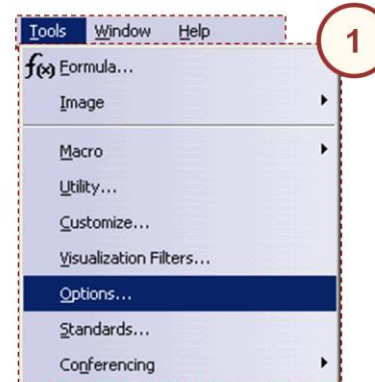
# Specific Tooling Design Settings (2/7)

- In the component tab, you can define for each type of component:
  a. How it should appear in drawing sections (cut/uncut).
  b. If predefined selection filters should be applied at creation time.
  c. If it is allowed to create several instances per reference.
  d. On which plate it should be positioned by default (only when using MTD Mold Bases).
  e. On which side of this plate (top or bottom) it should be positioned by default.

14
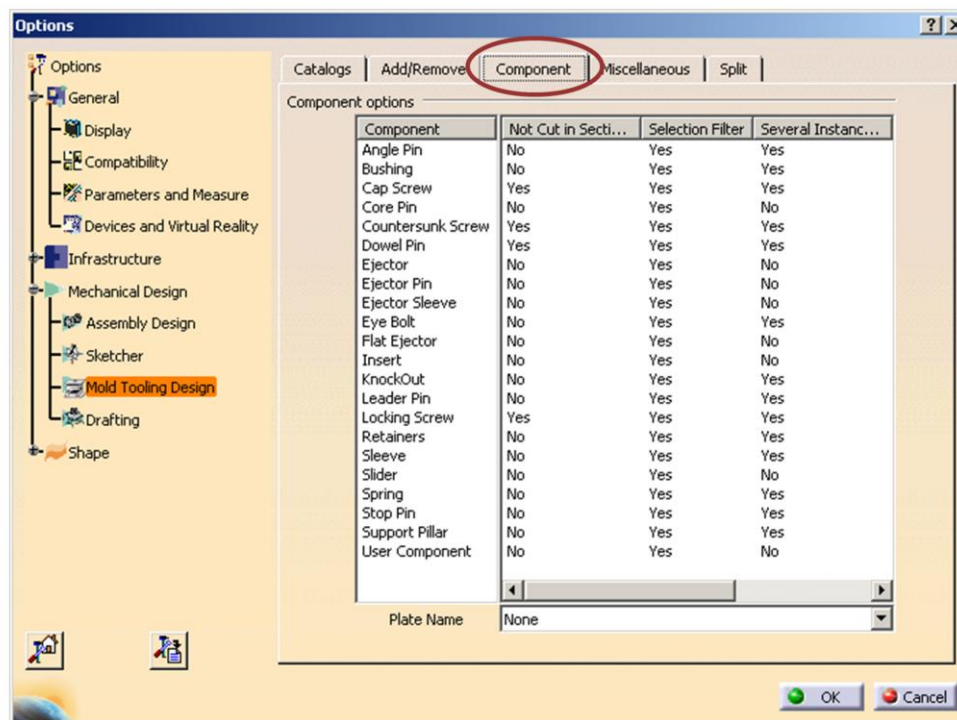
# Specific Tooling Design Settings (3/7)

- In the Add/Remove tab, you can define information relevant for TG1 command Distributed Drill Component:
    a. Extend or restrict the list of Add/Remove bodies. By default, this list contains DrillHole, TapHole, Pad, Pocket
    b. Extend or restrict the list of Receiving bodies
    c. Extend or restrict the list of Associations between elements of the two previous lists (use buttons Associate and Dissociate)
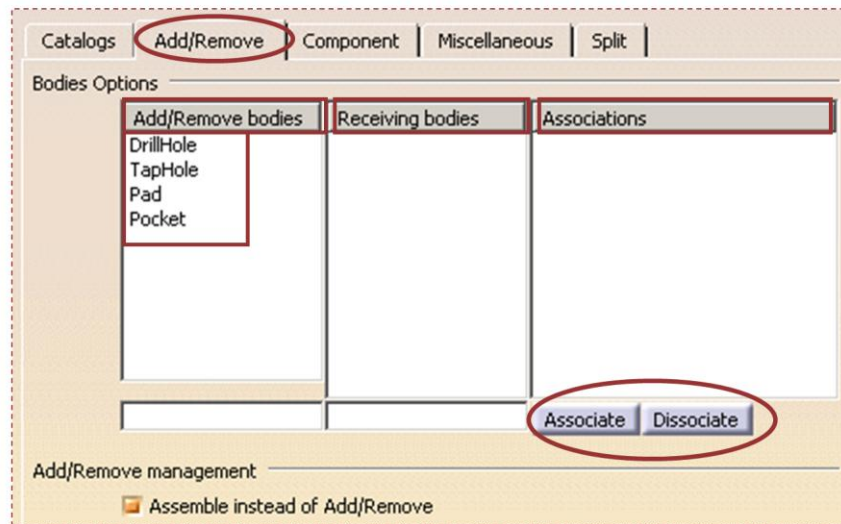


Note that only solid Body elements (standard or hybrid) are usable (and not Geometrical Sets or Ordered Geometrical Sets).

# Specific Tooling Design Settings (4/7)

- See an example of additional bodies and association:

  1. Type name in Add/Remove Bodies.
  2. Type name in Receiving Bodies.
  3. Select successively in the two lists the names to be associated.
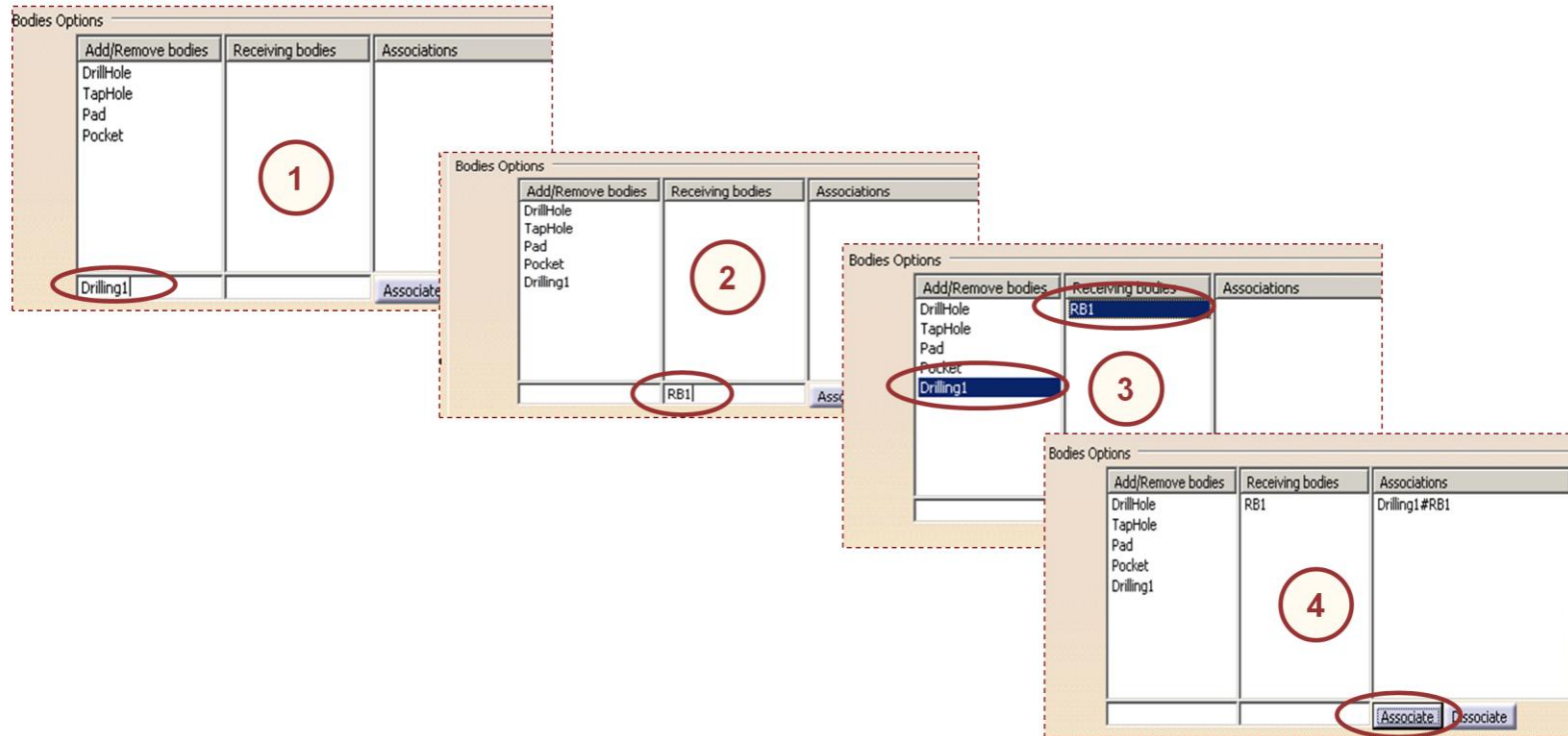  4. Click the Associate button.

# Specific Tooling Design Settings (5/7)

- **Naming rules for Add/Remove bodies:**

- **Each name in this list is used by TG1 as a beginning string, not as an exact string. Therefore it is not allowed to define a name which is "included" in another:**
  - ◆ **For example, you can define Drilling1 and Drilling2**
  - ◆ **But you cannot define Drilling1 and Drilling11**

- **It is not allowed to use the characters "_" (underscore) and "-", except in the following case.**

- **"_PS0" to "_PS9" and "-PS0" to "-PS9" at the end of the name are considered by the function Distributed Drill Component as pre-selection codes to minimize the number of user interactions.**

- **"__" (double underscore) at the beginning of the Body name is considered by the function Distributed Drill Component the same way as other drilling functions do (Drill From/To and Drill Component) when the Component is split, the Split operation is not performed on this Body.**
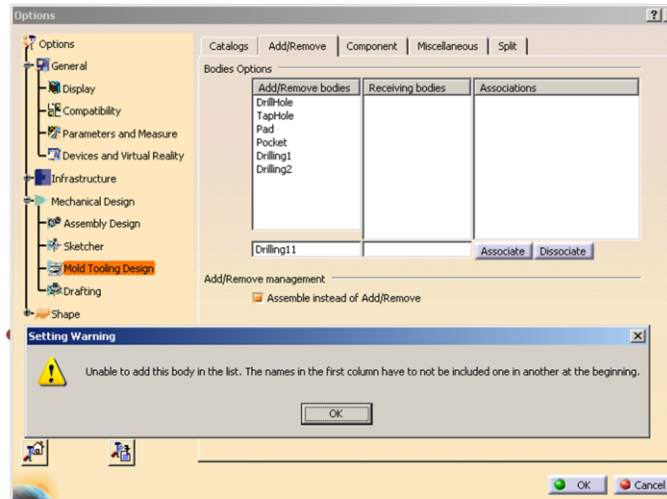
# Specific Tooling Design Settings (6/7)

- **Editing Body names:**
  - To edit the name of a Body (or to erase it), click it twice in the list and modify it as required.

- **About Receiving bodies:**
  - There is no naming rule.
  - No check is performed when defining the name (duplicates).
  - A Receiving body can be located in the Part at the same level as the PartBody, or as a sub-body through an Assemble operation.

- **About Associations:**
  - If the name of a Body is changed, you must manually recreate the corresponding Association (s).
  - An Add/Remove body can belong to only one Association
  - A Receiving body can belong to several Associations.
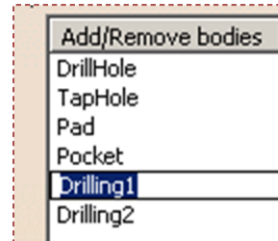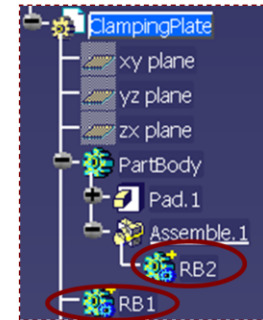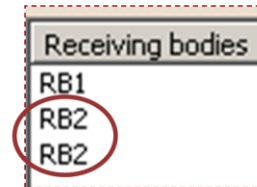
# Specific Tooling Design Settings (7/7)

**In the Add/Remove tab, you can define the way add or remove of material should be performed:**

- **If option Assemble instead of Add/Remove is deactivated (default value), Add or Remove Boolean operations will be created when a Tooling Component is instantiated**

- **If the option Assemble instead of Add/Remove is active (recommended), an Assemble Boolean operation will be created when a Tooling Component is instantiated. The polarity of the Component Body will therefore determine if material is added or removed.**
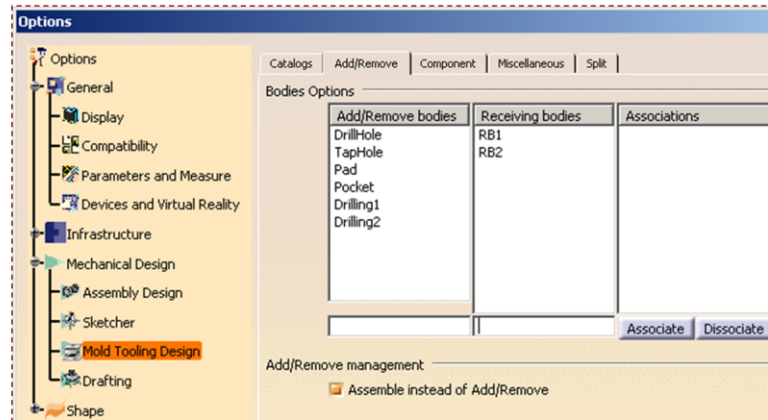


**The Distributed Drilling Component function works only if the Add/Remove option is active instead of the Assemble one.**

# Recommended General Settings (1/2)

- **In General / Parameters and Measure / Knowledge, activate:**
  - ◆ **Parameter Tree View: With value**
  - ◆ **Parameter Tree View: With formula**
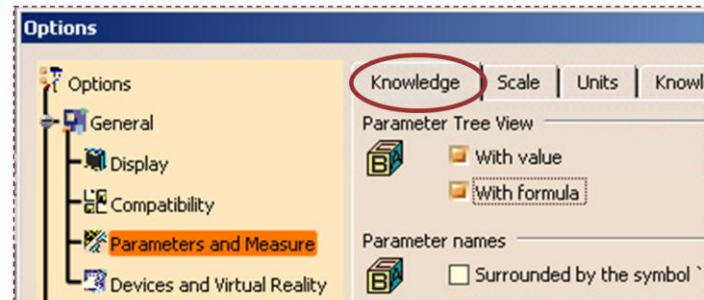
- **In Infrastructure / Product Structure / Tree Customization, activate:**
  - ◆ **Specification Tree Node Name: Parameters**
  - ◆ **Specification Tree Node Name: Relations**
  - ◆ **Specification Tree Node Name: Constraints**
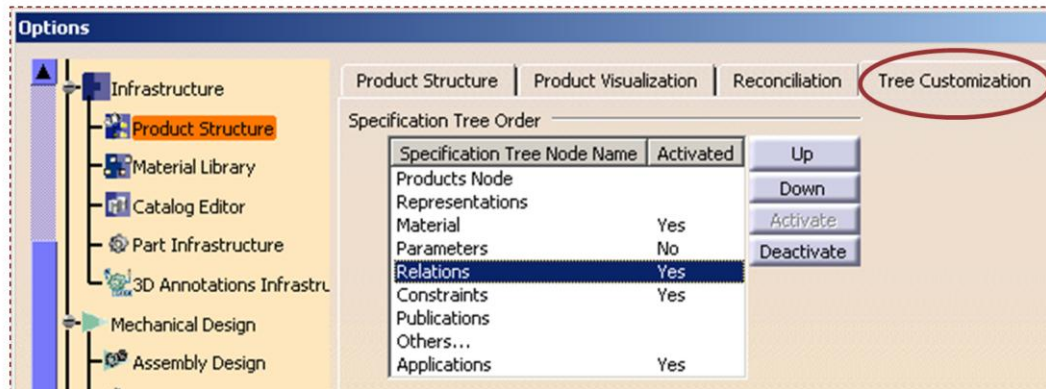
# Recommended General Settings (2/2)

- **In Infrastructure / Part Infrastructure / Display, activate:**
  - ◆ **Display in Specification tree: Constraints**
  - ◆ **Display in Specification tree: Parameters**
  - ◆ **Display in Specification tree: Relations**

- **In Mechanical Design / Assembly Design / General, activate:**
  - ◆ **Update: Automatic**
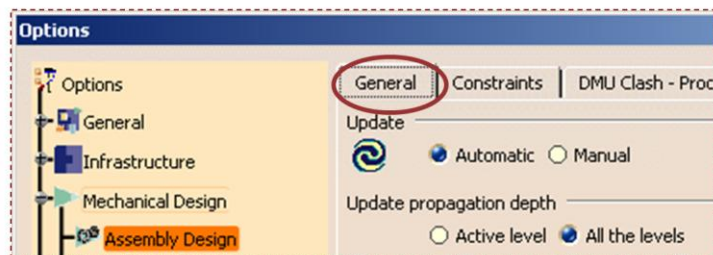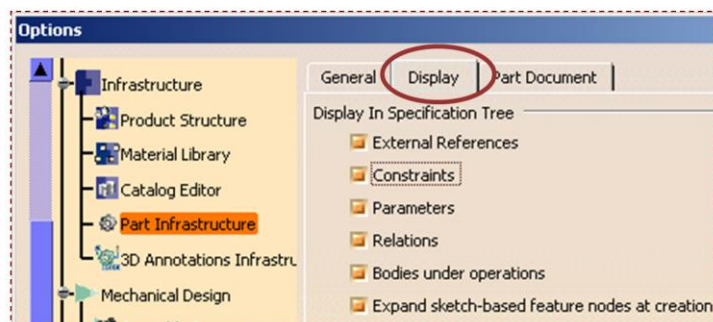
# Mold Creation: General Process



**Part to be molded**

**Define a Mold Base**

**1**

**2**

**Adjust position of the Molded Part**

**3**

**Core and Cavity separation**

**4**

**Add components**

**5**

**Add injection runners and coolant channels**

**End of Mold creation**

**6**
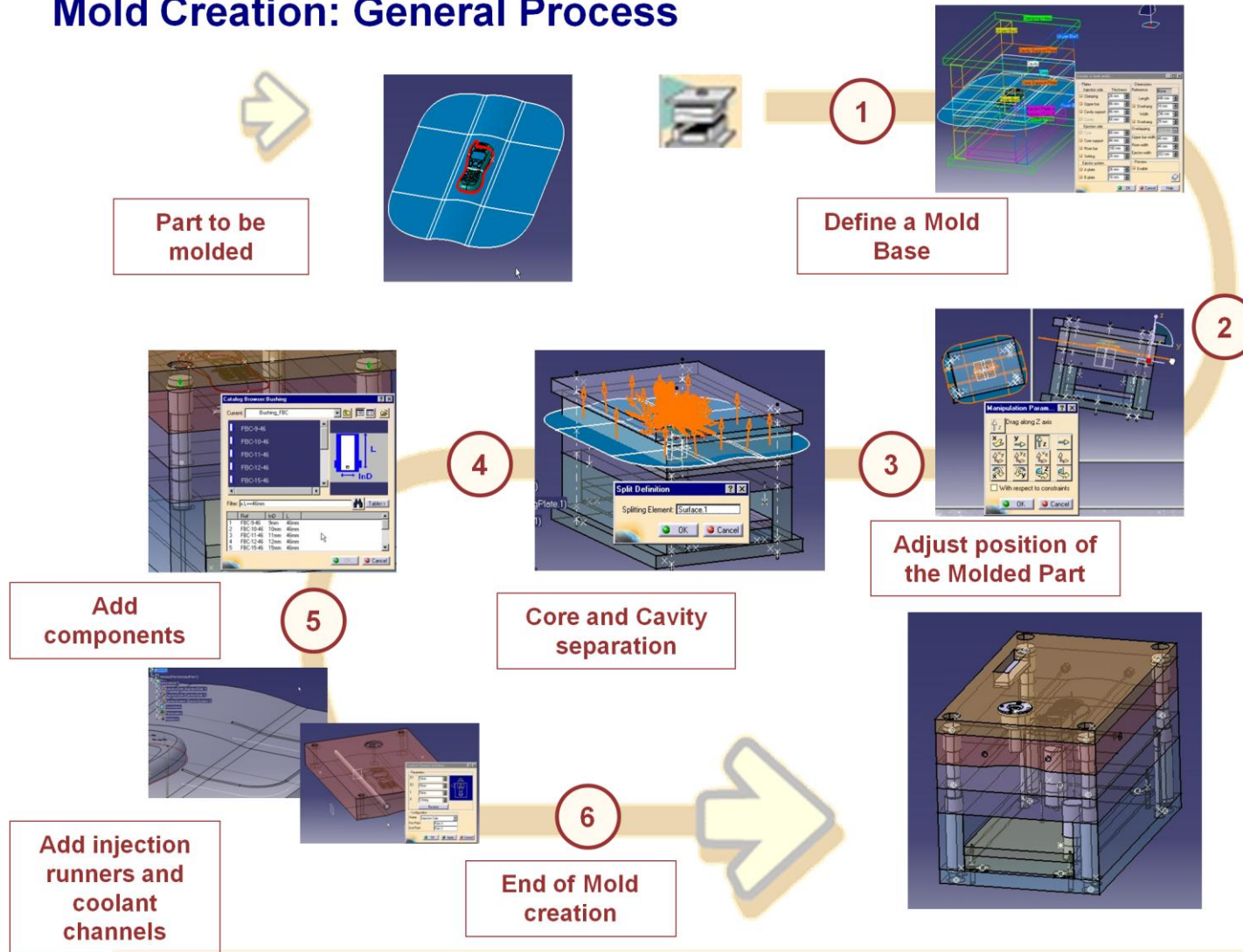
# Creating a New Mold

*You will see how to create a new Mold Product from scratch, insert the Molded Part and create the corresponding Mold Base.*
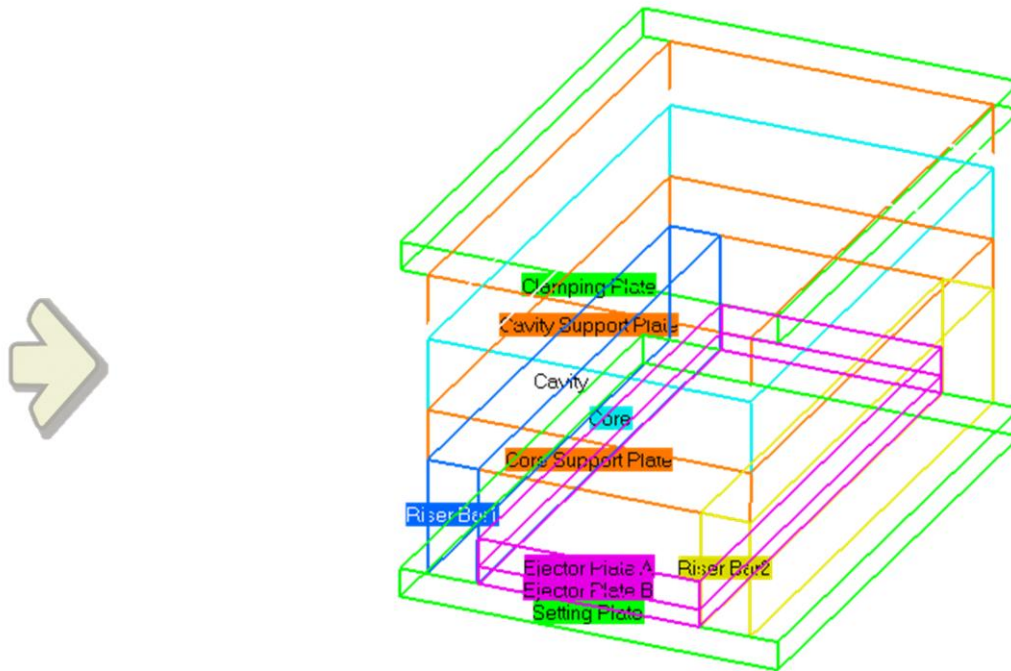
# Creating a New Mold Product

**A Mold Product is a standard CATIA V5 Product where specific Mold elements will be inserted later:**

- **You can create a new CATIA V5 product by:**

    - ◈ **Starting workbench Mold Tooling Design:**
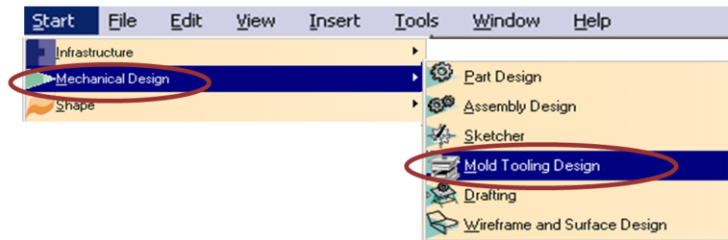


    - ◈ **Or explicitly creating a new Product:**
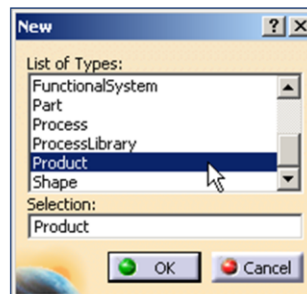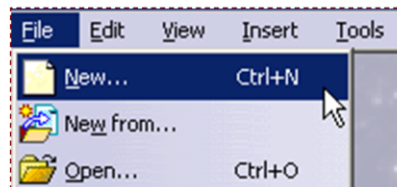


**A new Product is created**

# Inserting the Molded Part in the Mold Product

- The Molded Part can be inserted in the Mold Assembly by inserting the already existing CATPart:
    - From the contextual menu of the top Product: Components / Existing Component, then selecting the name of the CATPart file
    - Or by using standard menu Insert / Existing Component, then selecting the name of the CATPart file

**The Molded Part is inserted in the Product**
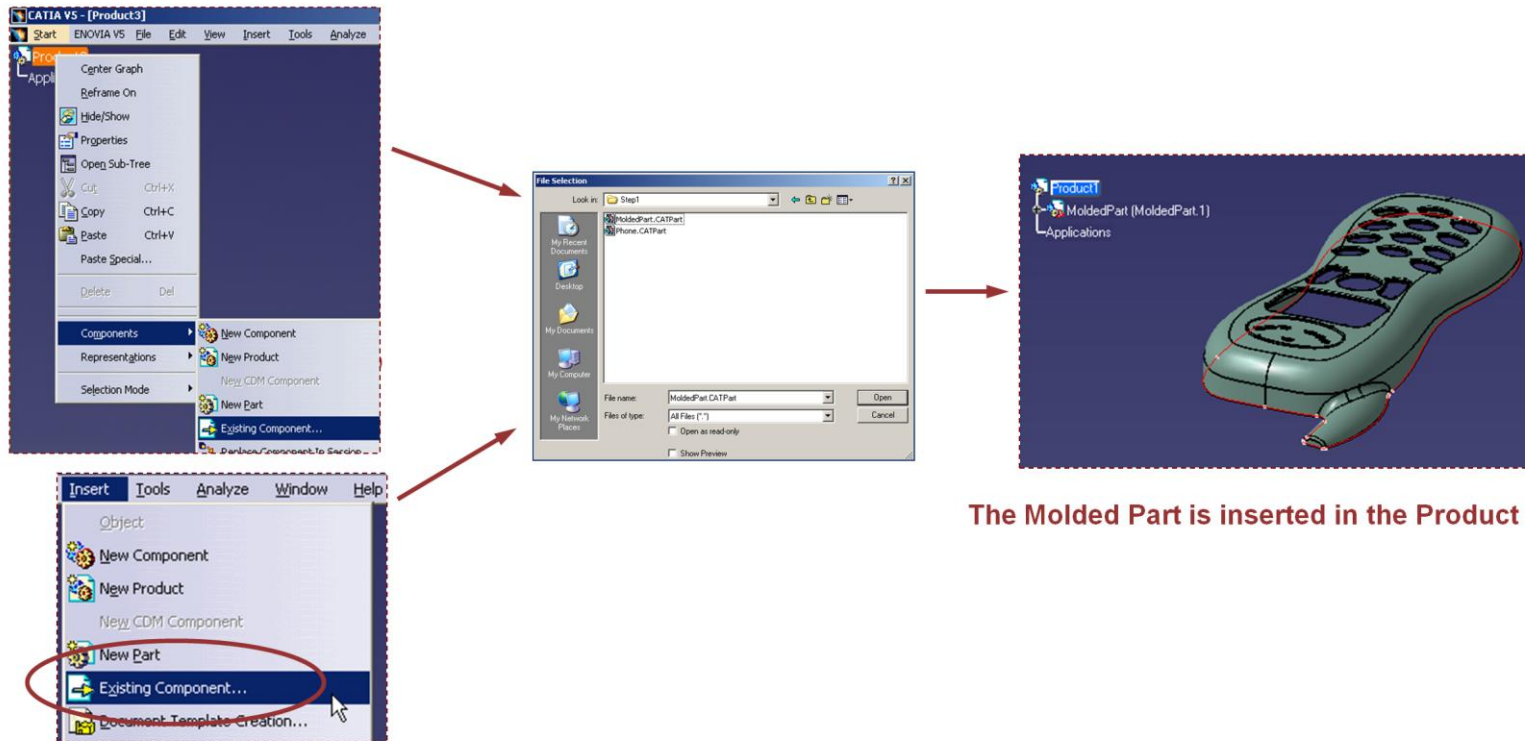
# Structure of the Molded Part: Requirements (1/3)

- **The Part Number (see tab Product in contextual menu Properties) must be MoldedPart**

- **The Instance name and the name of the CATPart file can be modified as required**
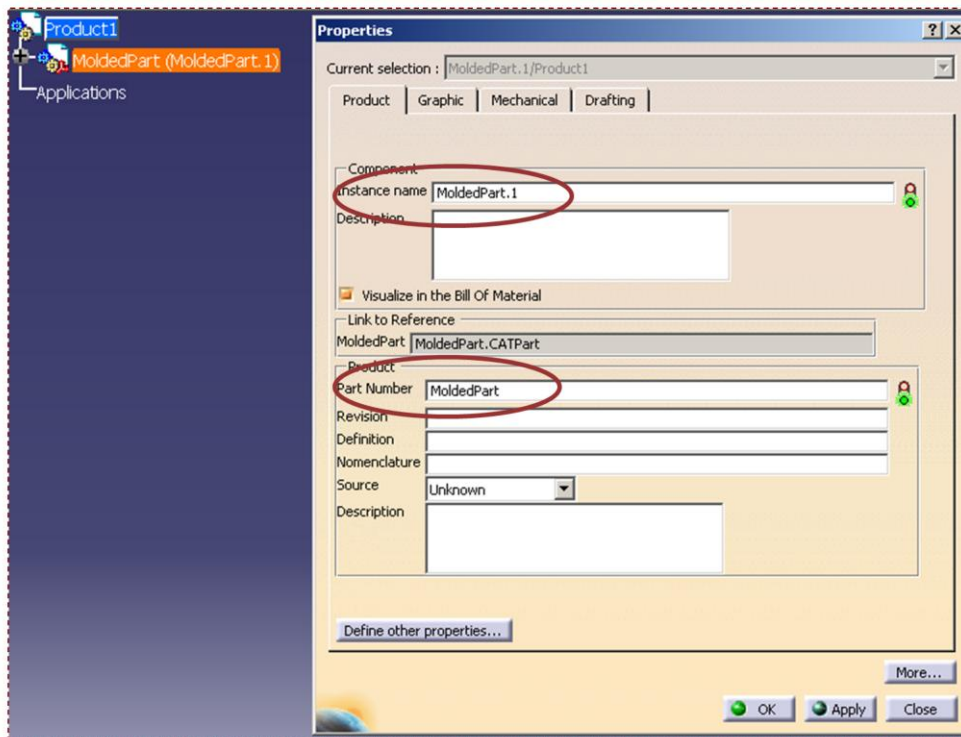
# Structure of the Molded Part: Requirements (2/3)

- The Molded Part must contain the part itself and also all the surfaces required for the core/cavity separation.
- It is generally created as the result of the preparation work done in the Core & Cavity Design or Generative Shape Design workbenches.

- It is recommended that the user defines the following elements in the Molded Part, in Geometrical set called PartingBody:
  - PartingSurface: the external parting surface created from the parting line of the Molded Part,
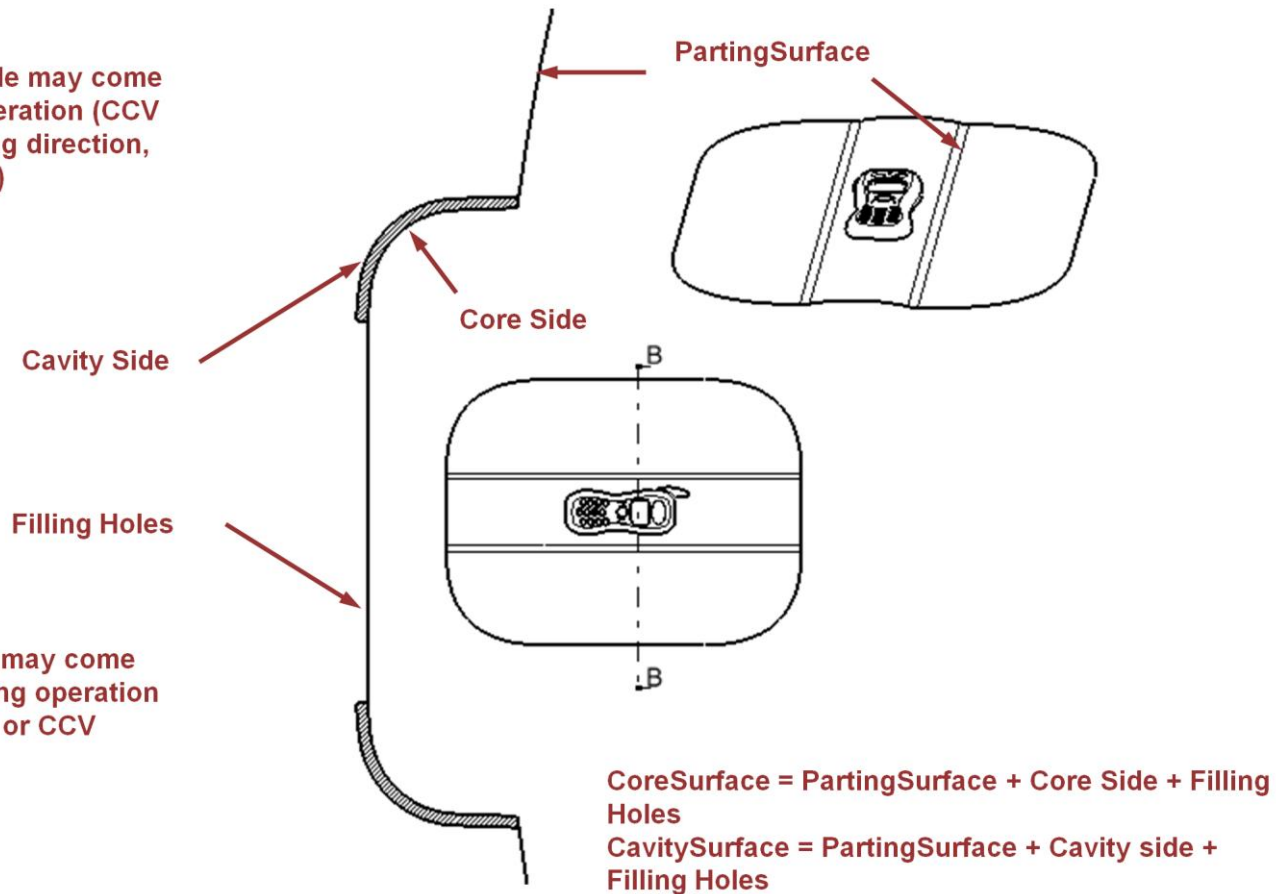  - CoreSurface: Join of the core side of the part (including fills of functional holes) and of PartingSurface
  - CavitySurface: Join of the cavity side of the part (including fills of functional holes) and of PartingSurface
- The elements bearing these names will be used by the application as (editable) default elements for core/cavity split operations and for creating gates and runners.
- If they are not present, you will have to explicitly select other elements in the Molded Part when performing core/cavity split operations.

# Structure of the Molded Part: Requirements (3/3)

**The Core Side may come from any operation (CCV Define Pulling direction, GSD Extract)**

**PartingSurface**

**Cavity Side**

**Core Side**

**Filling Holes**

**Filling Holes may come from any filling operation done in GSD or CCV**

**CoreSurface = PartingSurface + Core Side + Filling Holes**

**CavitySurface = PartingSurface + Cavity side + Filling Holes**

# Creating the Mold Base (1/3)

- You can define a new Mold Base by:
  - ◆ Defining a complete, fully dimensioned set of plates
  - ◆ Selecting an existing Mold Base in a catalog
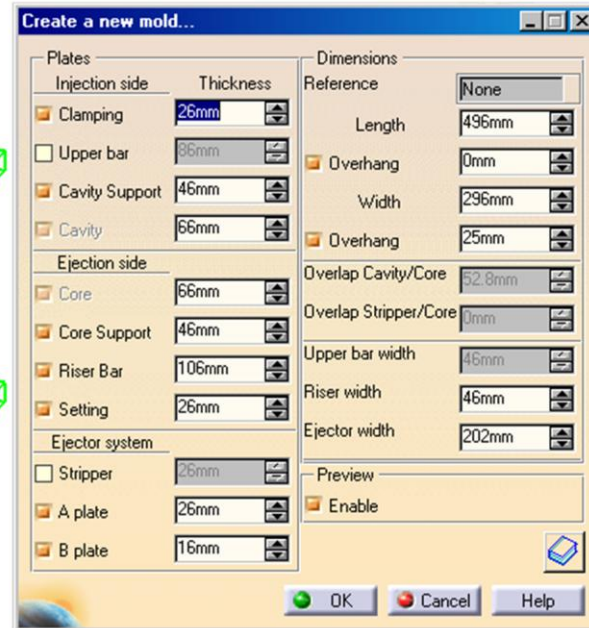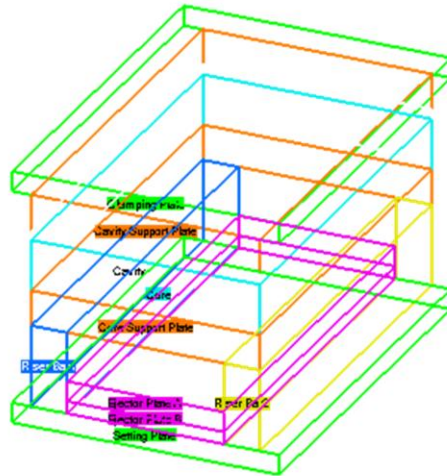
# Creating the Mold Base (2/3)

- You can create a new Mold Base by defining the plates' number, type and dimensions:
  - The default configuration includes most of the predefined plate types with their default dimensions.
  - Plates may be deactivated (with the exception of the Core and Cavity plates which are mandatory).
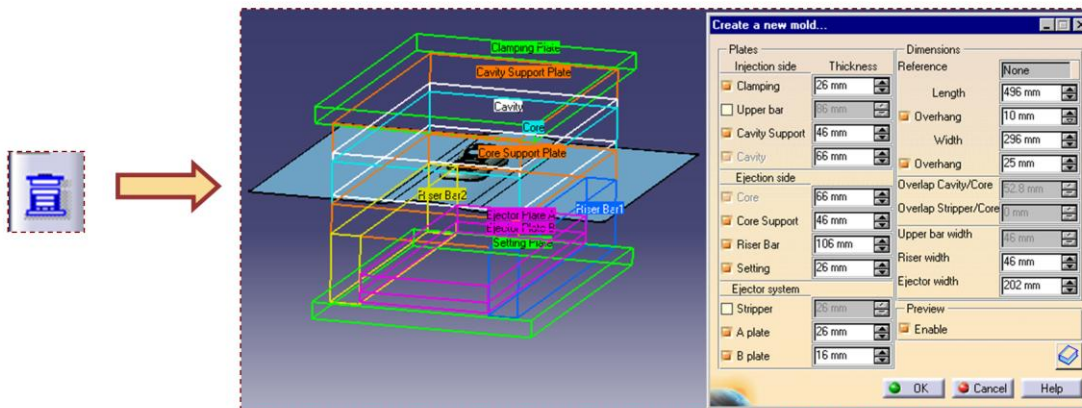    - Plates that cannot be deactivated are grayed out in the dialog box
    - The preview of the Mold Base is automatically updated when plates are deactivated
  - Plate dimensions may be modified in accordance with design needs.

The Create a new Mold function allows you to create a new Mold Base



The Mold Base dialog Box lets you control the definition of the plates that will be included in the Mold Base

# Creating the Mold Base (3/3)

- **Structure of the resulting Mold Product**
  - **Once the Mold Base is created, the structure of the Mold Product consists of:**
    - **A Component corresponding to the Molded Part.**
    - **A Product corresponding to the Mold tool assembly which contains the Mold Base plates. Other components will be later inserted in this assembly. This Product has 3 sub-assemblies: InjectionSide, EjectionSide and EjectorSystem.**
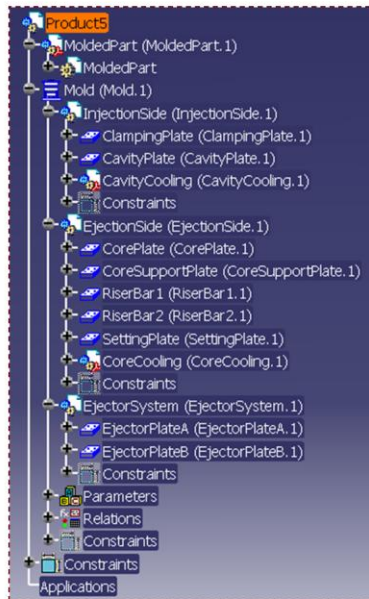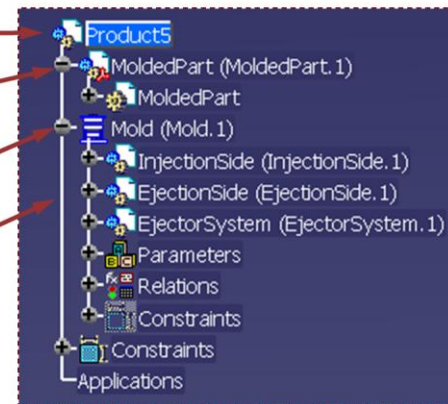


**Top Product**

**Component Molded Part**

**Mold Base product**

**Sub-products**

# Creating the Mold Base Using Catalogs

- **You can also select an existing Mold Base variant in a catalog**



**Catalog Browser**
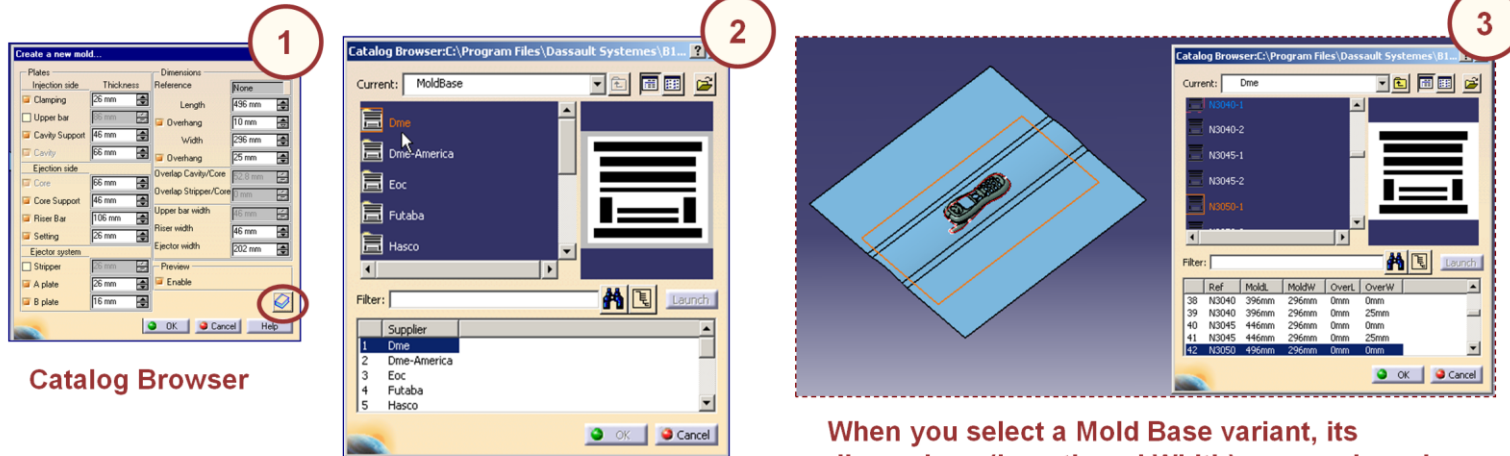
**You have access to the catalogs of many major market providers**

**When you select a Mold Base variant, its dimensions (Length and Width) are previewed**

**The new Mold Base is created as a new Product inside the top Product**
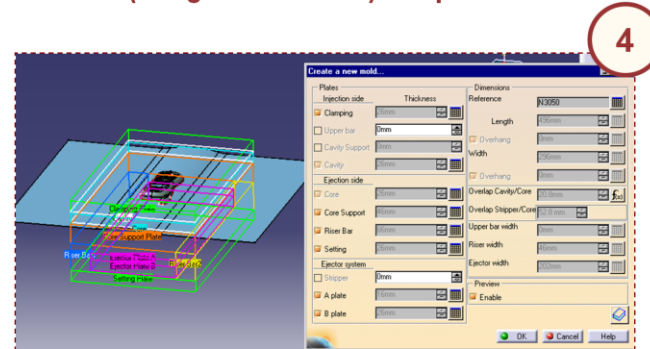
**Once you have selected your Mold Base, a full preview is displayed**

# Defining a Stripper Plate

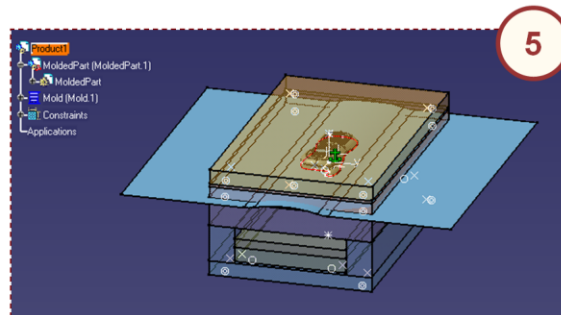🔹 **Using a Stripper Plate**

**When the Molded Part contains deep hollows, you may decide not to use Ejectors, but instead a Stripper Plate to avoid creating marks on the Part.**

🔹 **Creating a Stripper Plate:**

◆ **You can select it when creating the Mold Base, or add it later using Add Mold Plate function**

◆ **The Stripper Plate is located on top of the Core Plate, in the Ejection System sub-product. Its overlap with the Core Plate can be adjusted.**

◆ **Its shape to fit the Molded Part can be adjusted through Split Component operation, similar to what is done on Core or Cavity Plates.**
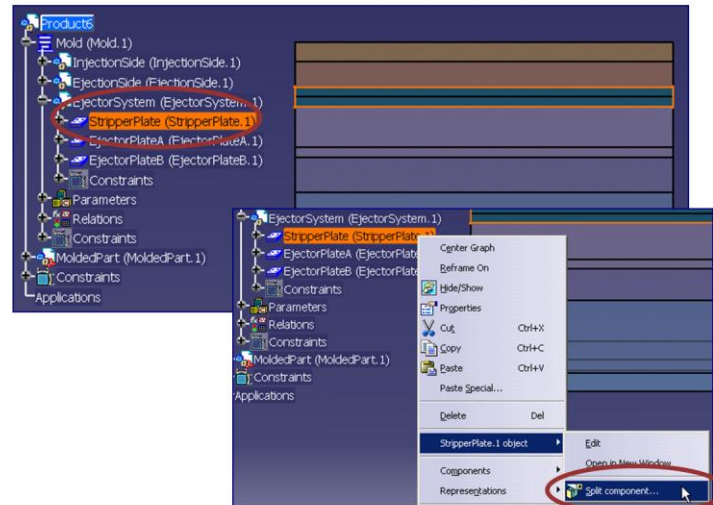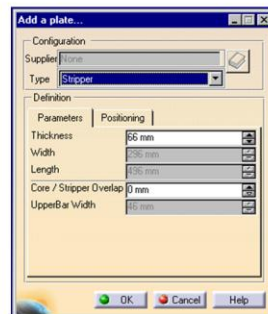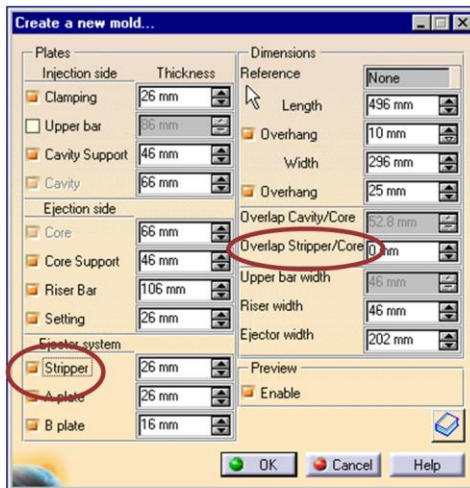
# Creating Mold Base Before the Molded Part

- **Creation of the Molded Part:**
  - ◆ **If the Mold Base is created prior to insertion of corresponding Molded Part, a new empty Component named MoldedPart will be automatically created by Mold Tooling Design application**
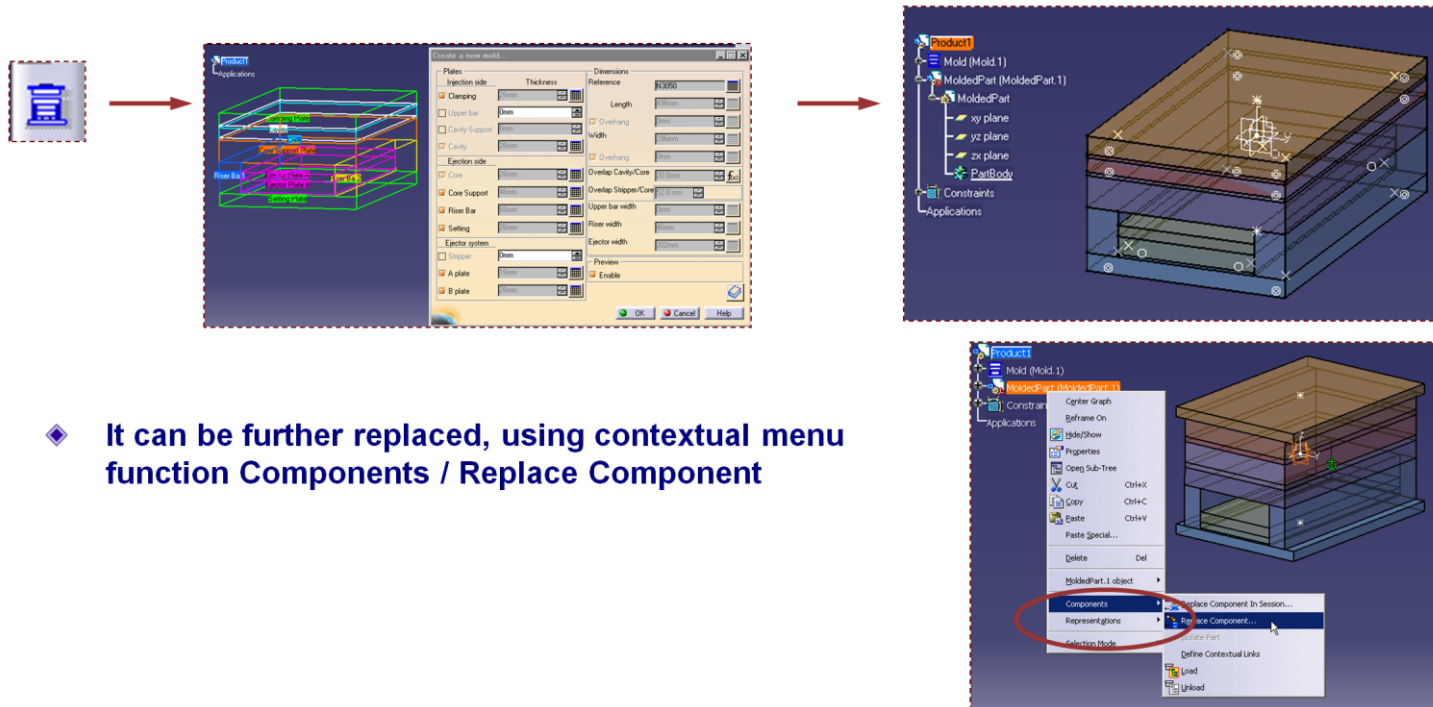  - ◆ **This component can then be interactively completed, using standard Part Design, Core&Cavity Design, Generative Shape Design and Assembly Design functions.**



  - ◆ **It can be further replaced, using contextual menu function Components / Replace Component**

# Positioning the Molded Part: Automatic Positioning

- When the Mold Base Product is created, it is automatically positioned with regard to the Axis System defined as active in the Molded Part:
  - First, the reference Axis System of the Mold Base (located at the bottom of the Core Plate) is snapped onto the active Axis System of the Molded Part.
  - Then the Mold Base position is adjusted along Z in order to position the origin of the Molded Part at medium distance between the top of the Cavity Plate and the bottom of the Core Plate

Core Plate and Cavity Plate are in overlap

Extreme sides of the Cavity Plate (top) and Core Plate (bottom)
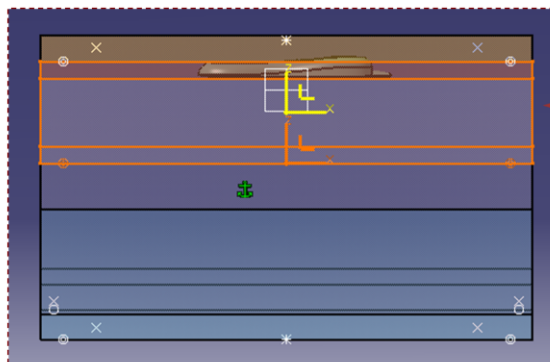
If there is no active Axis System in the Molded Part, the system uses the default origin and standard planes of the Part.

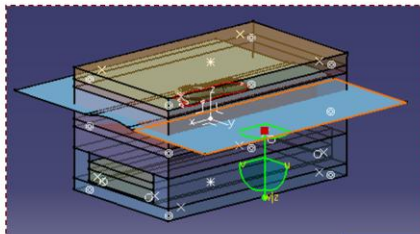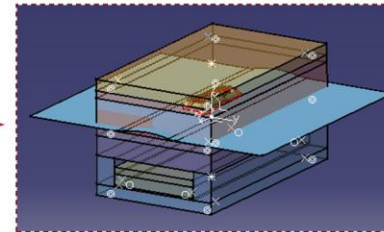Active Axis System of the Molded Part (yellow)
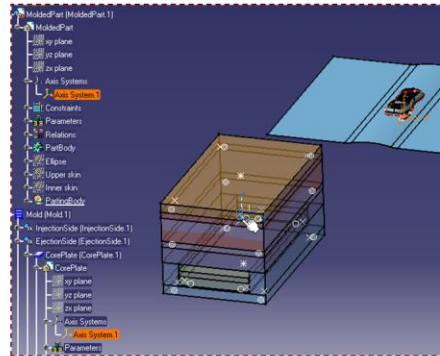
Axis System of the Mold Base (pink)
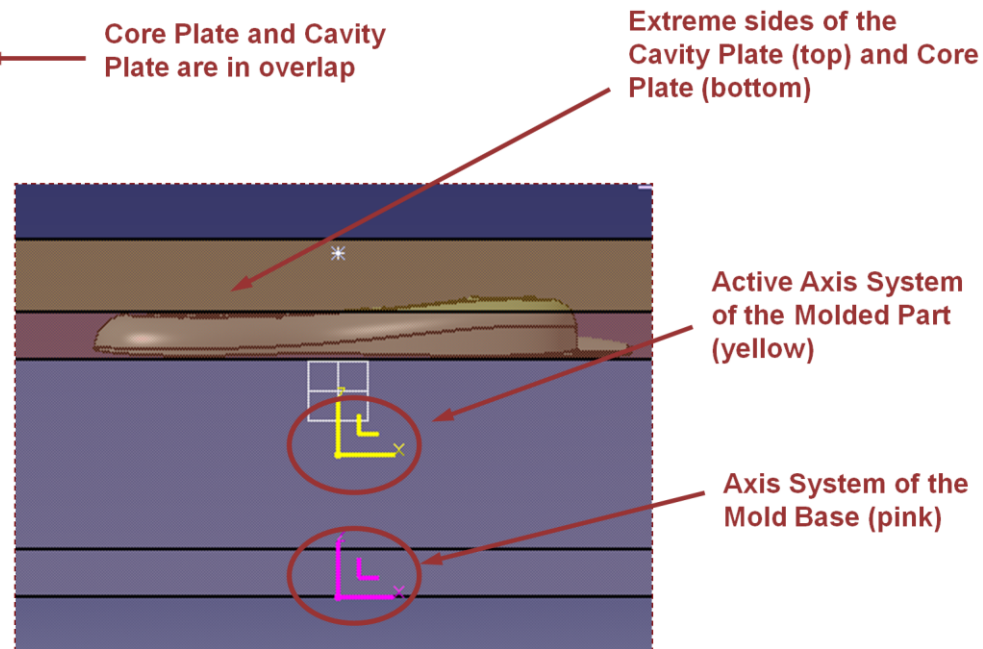
# Positioning the Molded Part: Additional Capabilities



You can use Snap function to apply coincidence between axis systems of the Molded Part and Mold Base.
You can also use main planes for snapping purposes.

You can also use the Compass to dynamically move the Molded Part in order to ensure that the Parting Surface will fully split Core and Cavity plates, or to define precise location values.
You can for example snap the Compass to the local axis system of the Molded Part or to the Parting Surface, in order to drag it as required.
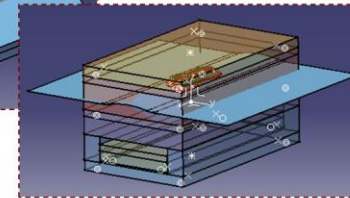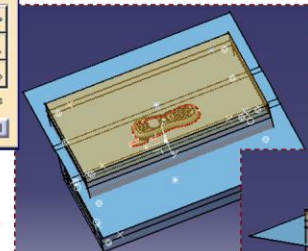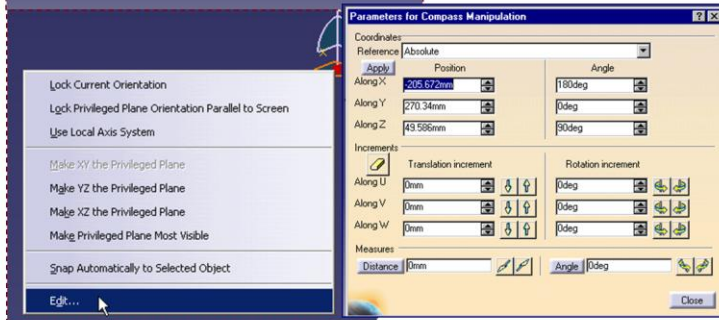
You can use Manipulation function to position the Part

# Editing the Mold Base

*You will see how to edit the definition of a Mold Base.*
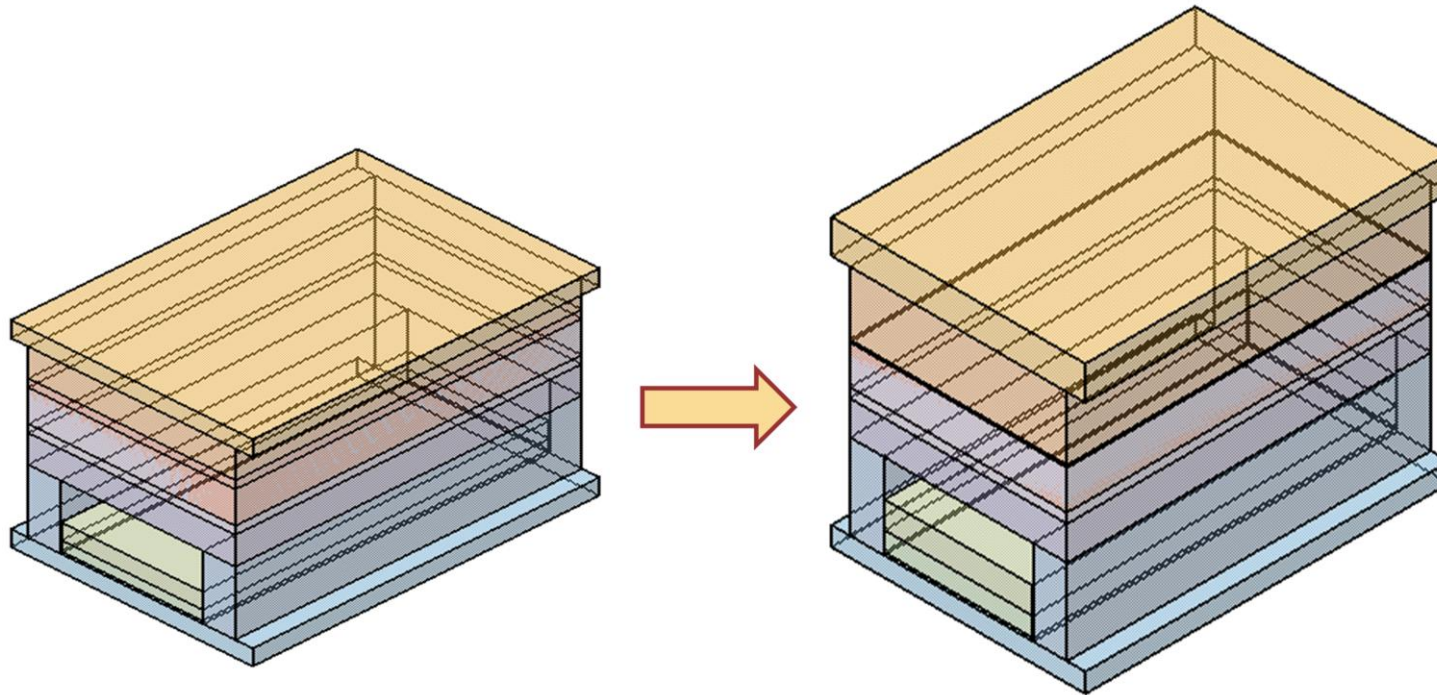
# Editing the Definition of the Existing Mold Base (1/2)

- Modify the definition of the current Mold Base using the contextual menu function Edit Mold.
  - ◆ Select another Mold Base reference in the Catalog.
  - ◆ The Mold Base preview will be automatically updated.
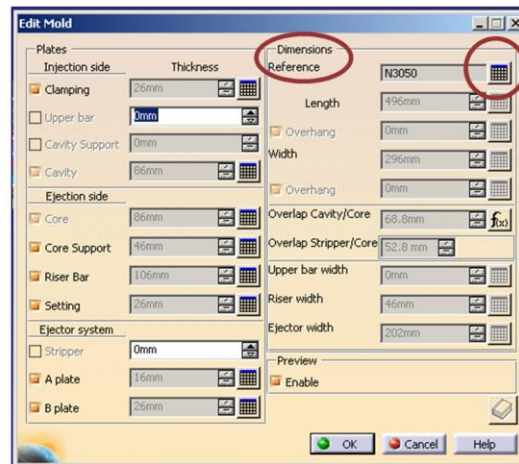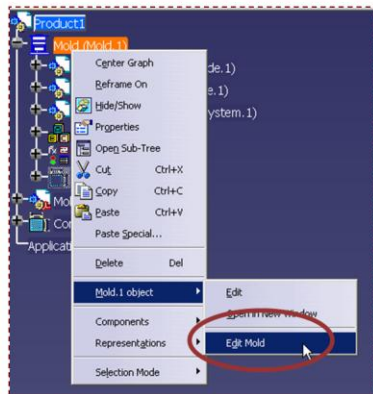
You can browse the Design Table containing the list of Mold Base variants and select a new one

# Editing the Definition of the Existing Mold Base (2/2)

- **Modify the definition of the current Mold Base using the contextual menu function Edit Mold.**
    - ◆ **Update the dimension of plates**
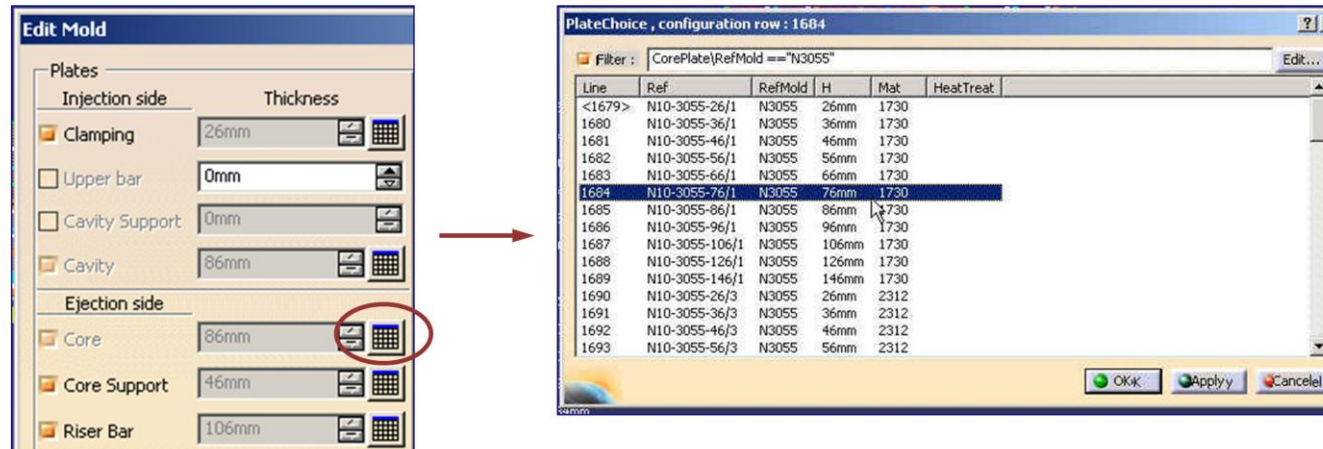


    - ◆ **Remove optional plates by deselecting them in the dialog box**
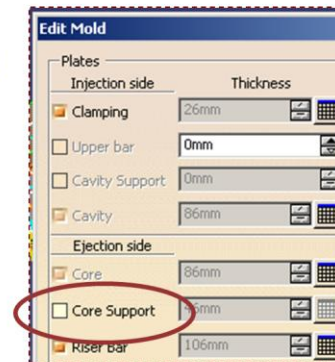
# Adding a Plate

- It is possible to add a plate inside an existing Mold Base. The new plate must be of one of the available predefined types allowed for the current Mold Base.

-  Current limitation: it is not possible to add other types of plates (user-defined types of plates).



**Only types allowed for the current Mold Base are displayed**
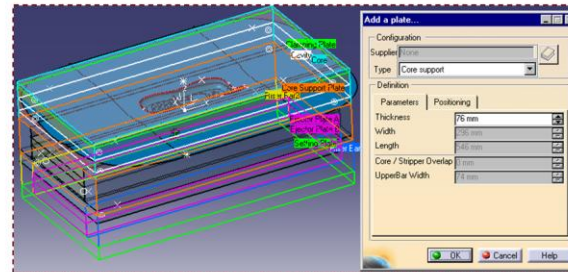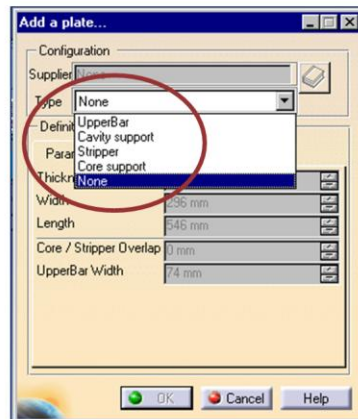
**The Mold Base preview is updated with the selected plate**

**Note that it is not possible to modify parameters which are related to the overall size of the Mold Base**
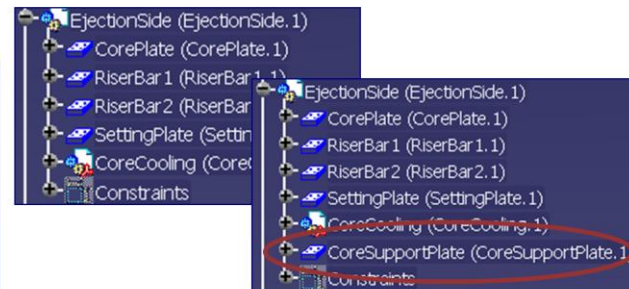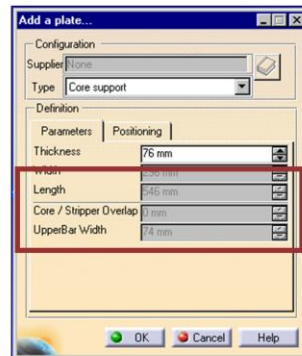
**Once created, the new plate is inserted in the tree structure**

# Splitting Core and Cavity Plates

- **Core and Cavity Plates can be split to fit the shape of the Molded Part.**
- **This operation means that the Core and Cavity Plates will be driven by the Molded Part. They will depend upon the design context. This subject will be addressed later.**
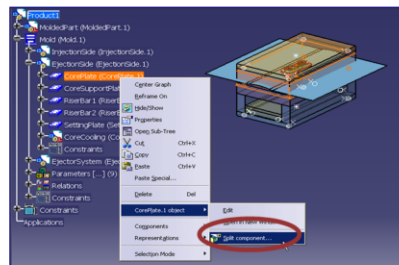
**For both Core and Cavity plates, there is a dedicated function Split Component in their contextual menu.**
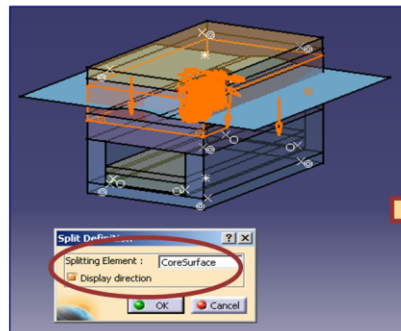
**The application will use by default as splitting surface the one named CoreSurface (resp. CavitySurface), if present in the Molded Part. You can select another surface if you want.**
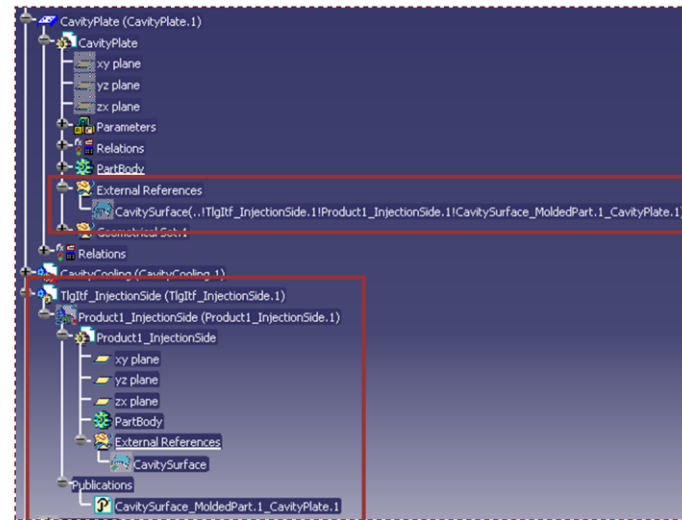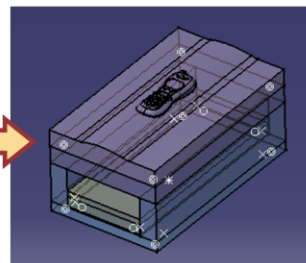
**Product Interface 'TlgItf_InjectionSide' is comprised of Part Interface 'Product1_InjectionSide' which contain a Published Copy of 'CavitySurface' from the MoldedPart**

# Working with Mold Components

*You will learn how to manage Mold Tooling Components.*

# Generic (TG1) / Mold (MTD)

- The Tooling Design (TG1) and Mold Tooling Design (MTD) workbenches share the same Tooling Component concept.

- Component information and behavior described in the TG1 training course is also applicable to MTD.

- The MTD workbench contains:

  - Standard catalog components available in TG1 such as:
    - Leader Pin, Bushing, Sleeve, Ejector Pin, Ejector, Flat Ejector, Ejector Sleeve, Core Pin, Stop Pin, Angle Pin, Knock-out, Support Pillar, Eye Bolt
  - Additional catalog components compared to TG1 such as:
    - Locating Ring, Sprue Bushing, Sprue Puller, O-Ring, Plug, Baffle
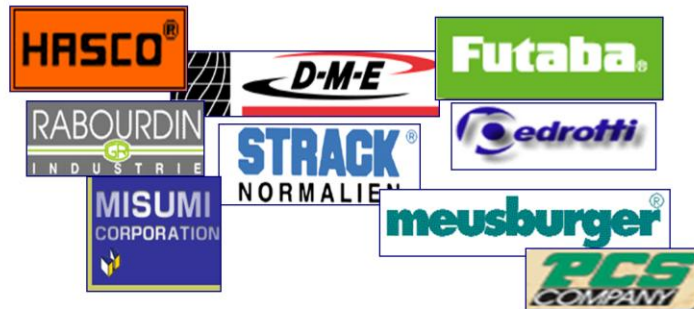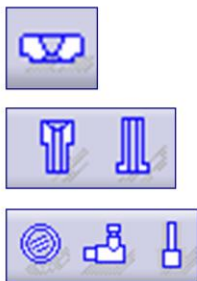
# Creating Components

*You will learn how to select and instantiate any type of component in a Mold or Die.*
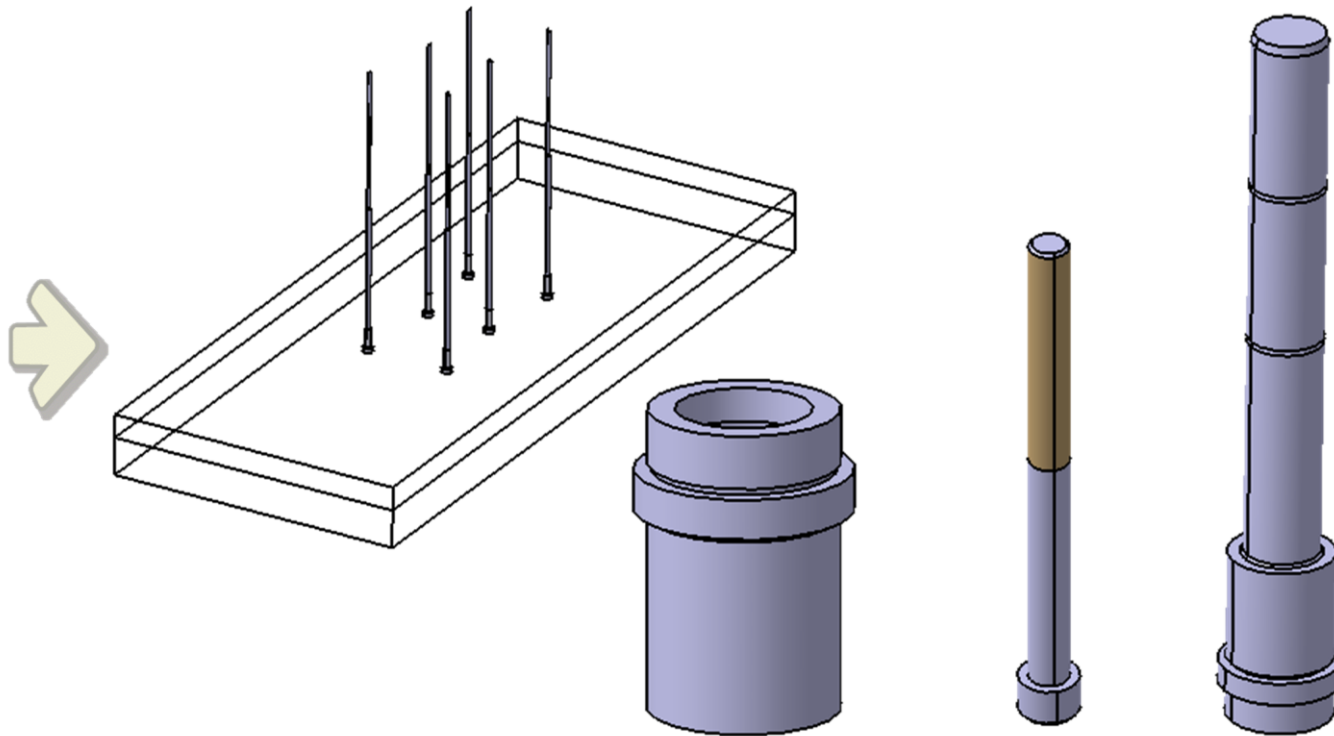
# What is a Tooling Component? (1/2)

- A Tooling Design component can be any kind of element. Components can be simple, such as a screw, wear plate or slider, or complex. A simple component may be defined as a Part, but a complex component may be defined as a Product.

- A list of predefined component types is available in the TG1 application. Components such as Leader Pins or Cap Screws can be found in the catalogs of leading market providers such as Hasco, DME, etc.

- It is possible to create your own predefined components (e.g. a custom Leader Pin) to extend or replace the libraries included in the application.

- It is possible to create your own component types (locks, stamps, washers, sliders, etc.), to create generic User Components.

# What is a Tooling Component? (2/2)

- **A Tooling Component is usually a CATPart file. This can contain specific features:**

  - **Drill holes.**

  - **A reference point used for positioning when instantiating the Component.**

  - **Dedicated properties and parameters to identify the component.**

- **The Component can have variants (defined in a Design Table) to allow for different sizes.**

- **The structure of a standard manufacturers' catalogs Component or a user-defined Component is identical.**

- **If the Tooling Component is an assembly (CATProduct), each of its component parts respect the dedicated structure.**

- **When a Component is instantiated it can be selected from a catalog or a file document.**

## Tooling Component Structure Basics: Associated Drillings (1/2)

- The shape of the object itself must be defined in PartBody. It may consist of any sequence of features like Pads, Shafts, etc.

- The negative volumes associated to the shape must be created in a Body named Drill Hole. They can be defined using any type of negative-polarity feature (e.g. Hole, Pocket ).

- When the Component is inserted in the Mold, the DrillHole is automatically subtracted (creation of Remove or Assemble features) from all affected elements.
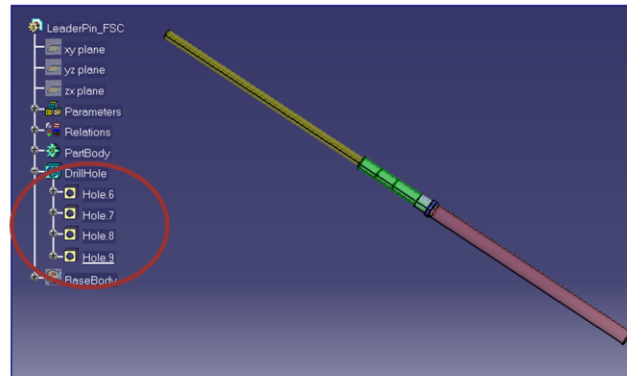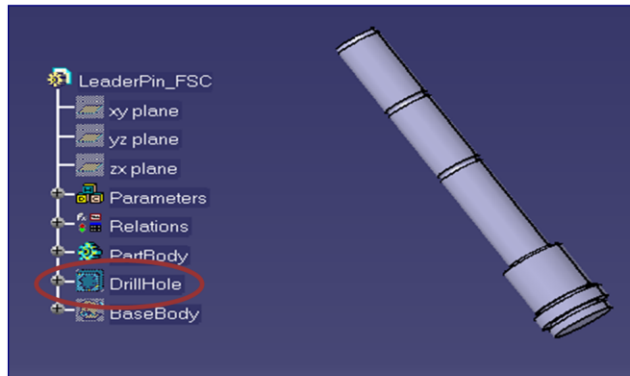
# Tooling Component Structure Basics: Associated Drillings (2/2)
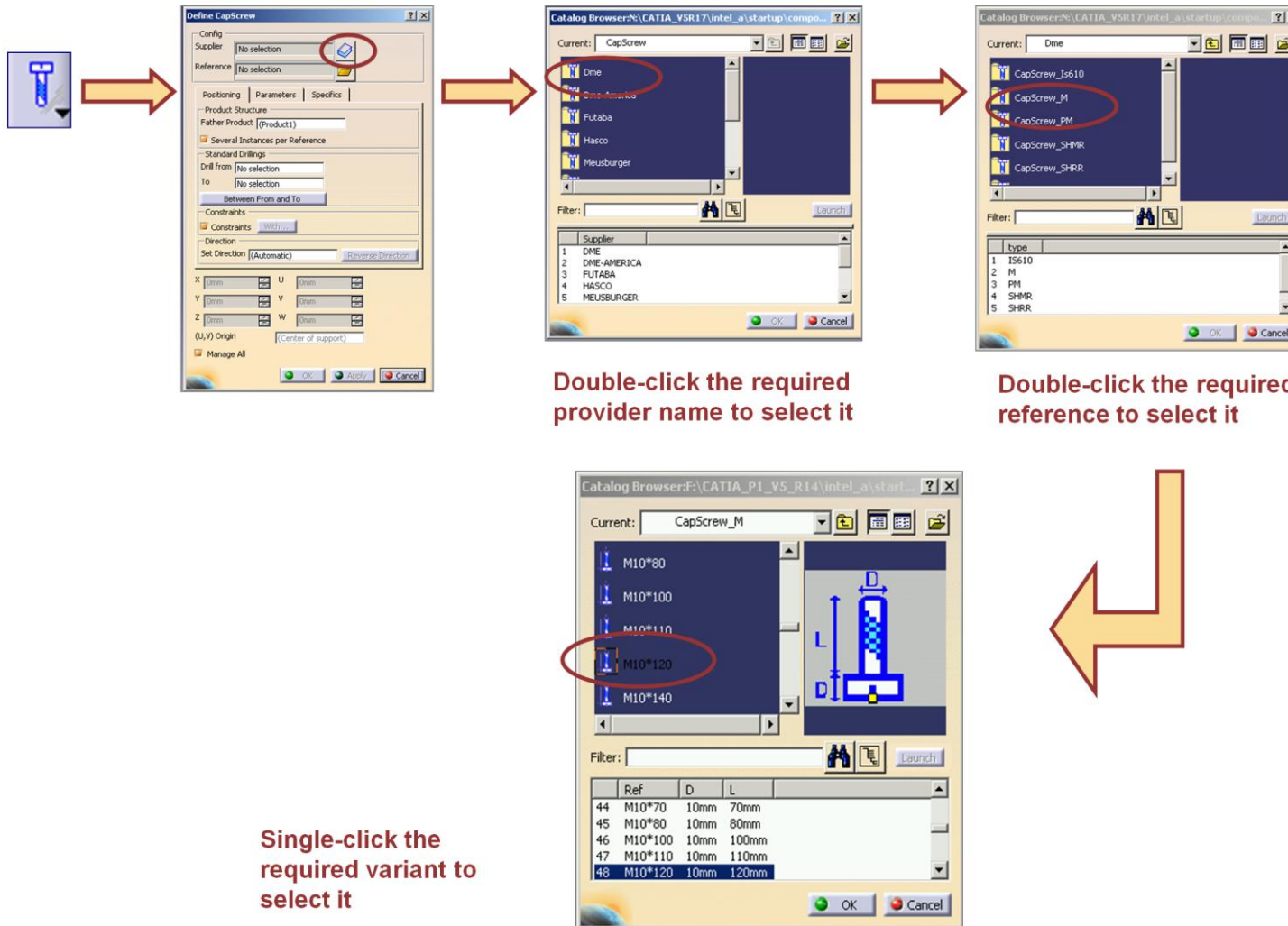
- **Another Body, named TapHole, can also be present to define additional negative volumes corresponding to threaded holes (typically in screws). This Body is optional.**

- **Like body DrillHole, Body TapHole will be used to perform Remove or Assemble operations.**

- **When the Component is inserted in the Mold, the TapHole is automatically subtracted (creation of Remove or Assemble feature) from the appropriate element, while the DrillHole is applied to all other affected elements.**
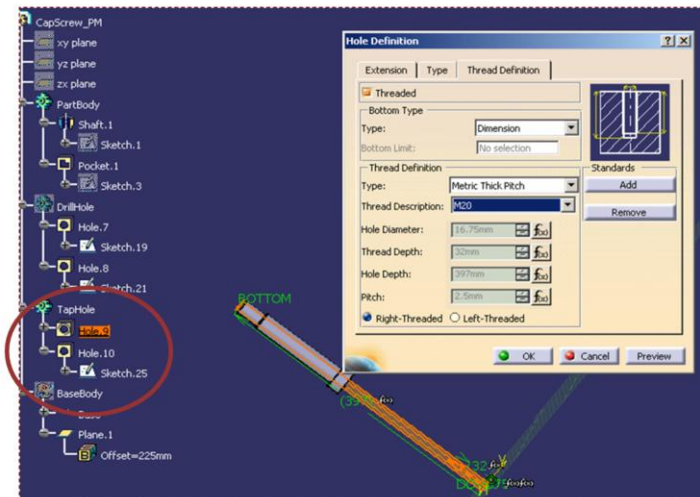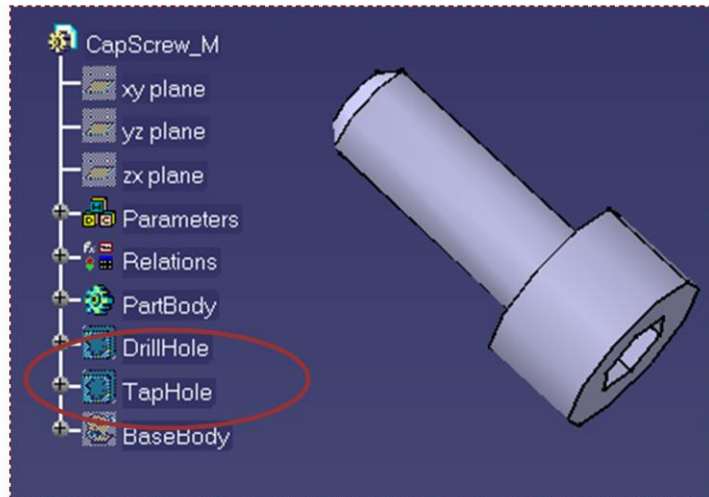
# Selecting a Component (1/2)

- The procedure to select, then position a Component is similar for all available types of components (Leader Pin, Bushing, Cap Screw)

- Select the required Component variant:

  - Click the icon corresponding to the required type of component

  - The definition dialog box of the component is displayed

  - Pick in the 3D viewer the required location for the component

  - Select the required catalog provider (if catalog is available: generic components only): DME, Hasco, Rabourdin, etc.

  - Select the required reference for this component (only inside a catalog)

  - Select the required variant for this component.

- Note 1: it is also possible to select first the component variant and then pick the required location in the 3D viewer.

- Note 2: if Selection / Filter is active in Tools / Options / Mechanical Design / Mold Tooling Design (default setting), a knowledge rule is applied to filter only the relevant dimensions. This knowledge filter is specific to each type of component (please refer to the on-line documentation for the exhaustive description of these rules). You can erase the proposed filter if you want to get the whole list.

# Selecting a Component (2/2)



**Double-click the required provider name to select it**

**Double-click the required reference to select it**

**Single-click the required variant to select it**

# Instantiating a Component

*You will learn how to select and instantiate any type of component in a Mold or Die.*
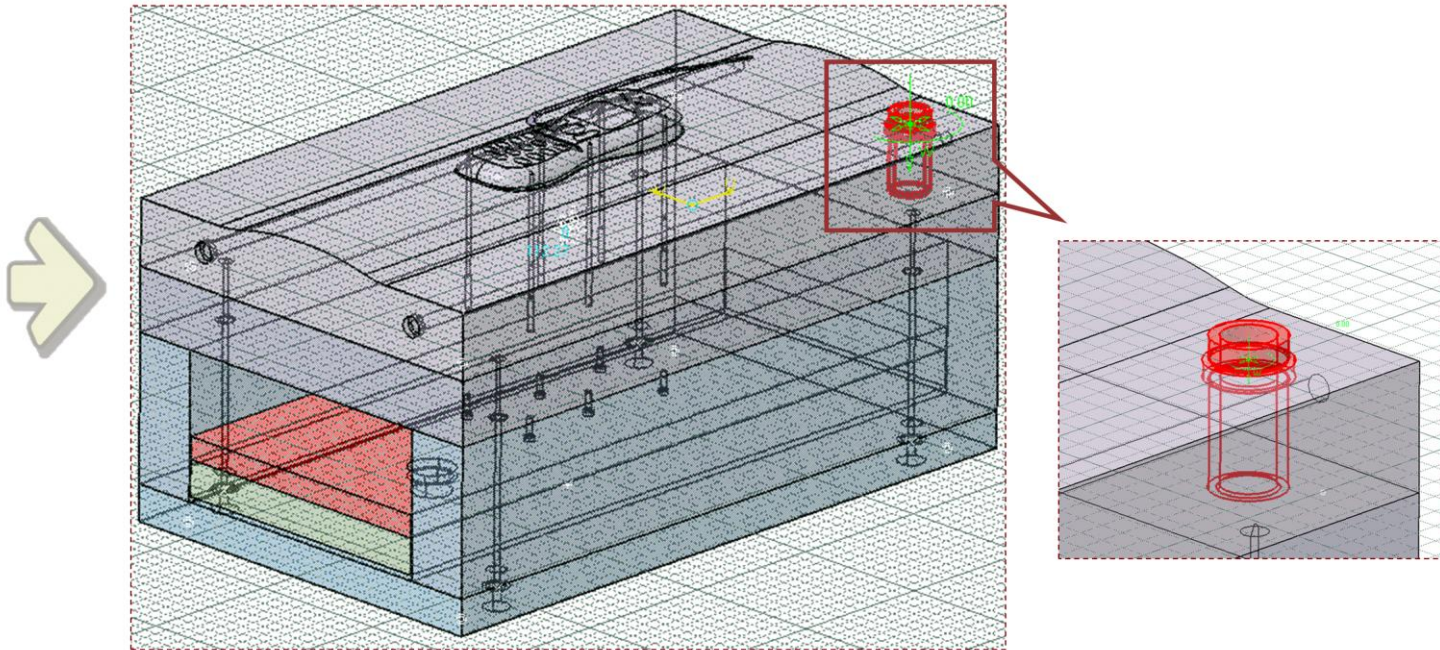
# Positioning the Component (1/4)

- 🔹 **To define the component's position, you can pick:**
  - ◆ **Either a point (3D point or vertex or sketch point).**
  - ◆ **A preview of the component is then immediately displayed.**
  - ◆ **Or a face (planar face or plane). The 3D display is automatically adjusted according to the value of Sketcher Settings Options Position sketch plane parallel to screen and Grid display. Sketcher option Snap to point if also taken into account. Pick then a position on the selected face to get a preview of the component.**

- 🔹 **You can adjust the position:**
  - ◆ **Either by changing the numerical values in the dialog box. These values can be defined as absolute: X,Y, Z, i.e. considering the axis system of the Mold or Die, or as local: U,V,W, i.e. relatively to (U,V) Origin.**
  - ◆ **Or you can also work in the 3D viewer and use the green arrows displayed in the viewer (for translation) or the green half-circle (for rotation).**
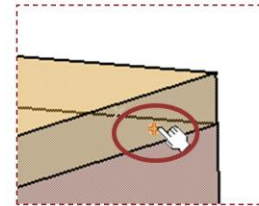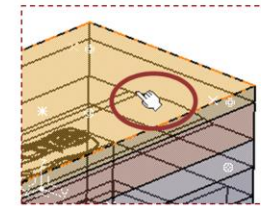
**3D point**          **Planar face**

**Sketch point**
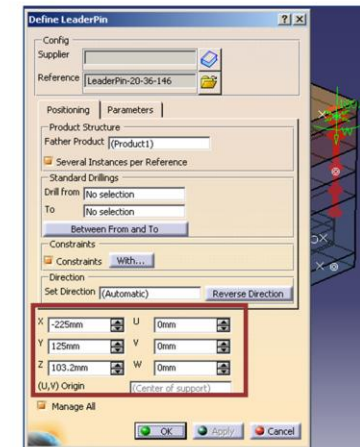
**Numerical values and arrows positions are updated simultaneously**
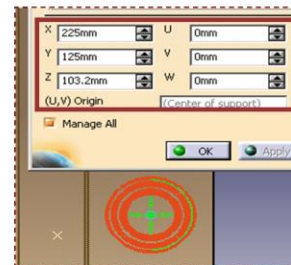
# Positioning the Component (2/4)

- Field Set Direction allows to define explicitly the orientation of the component. You can pick an axis, an edge, a 3D line, a planar face, a plane.
  - ◆ Without explicit selection, its value is set to Automatic: it corresponds to the normal to the planar face (if you are positioning the component on a planar face) or to Z axis of the assembly (if you are positioning the component on a 3D point in space).
- Button Reverse Direction allows to invert orientation if needed
  - ◆ When inserting a component on a planar face or on sketch points, the default positioning is: its Z+ axis oriented inwards into the material.
- To erase a previewed component, just click the vertical green arrow in the 3D viewer.

**Click green arrow to erase preview**

**Orientation into material**
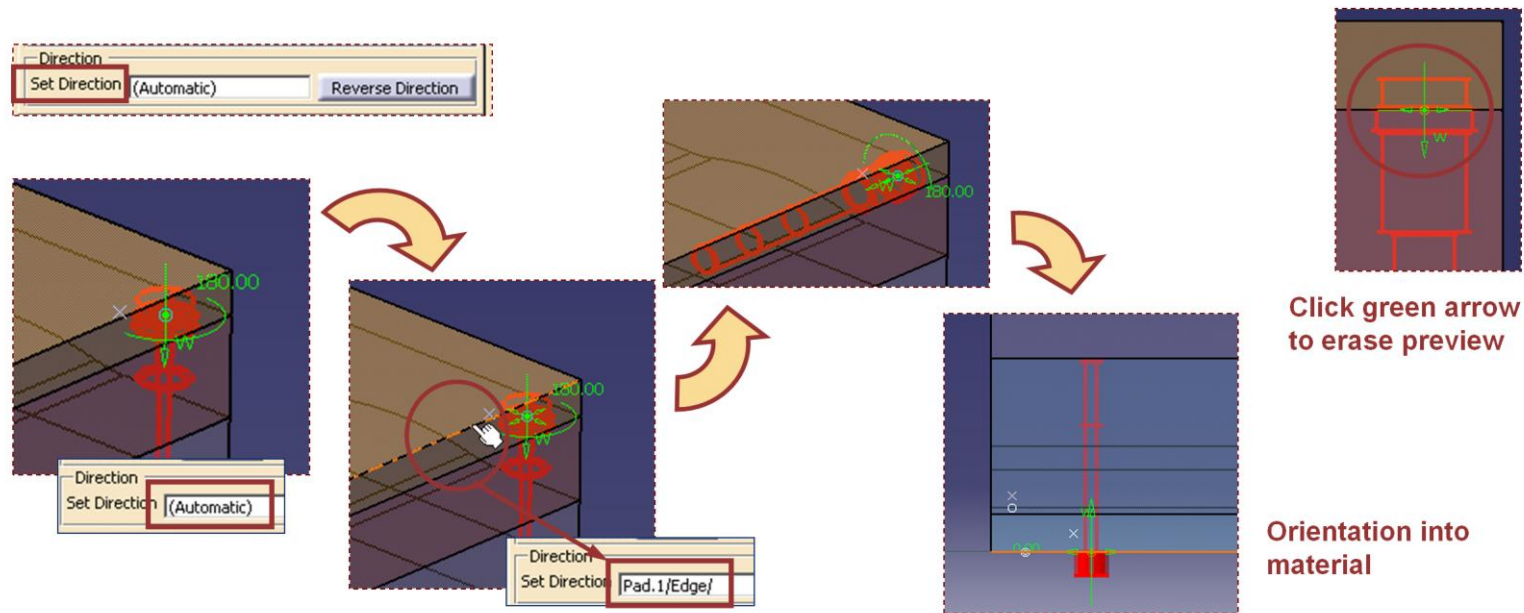
# Positioning the Component (3/4)

- **Button Constraints (active by default) enables creation of one or two Offset constraints :**
  - ◆ **If the component is positioned on a 3D Point, one Offset constraint is created (type = Point/Point).**
  - ◆ **If the component is positioned on a planar face, one Offset constraint is created (type = Plane/Point).**
  - ◆ **If the component is positioned on a sketch point, two Offset constraints are created (types = Point/Point and Plane/Point).**
- **To view and edit if required the geometrical element (point, face, plane) to which the component is constrained, click button With, select the element in the window to highlight it and select the new geometrical element in the 3D viewer or in the specification tree.**
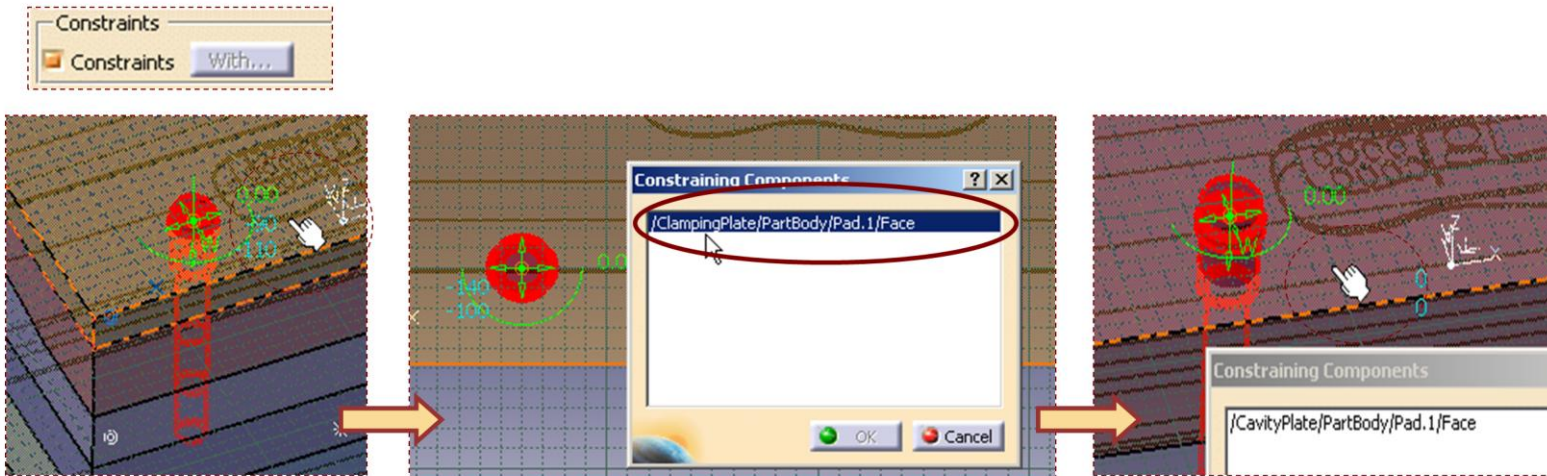
# Positioning the Component (4/4)
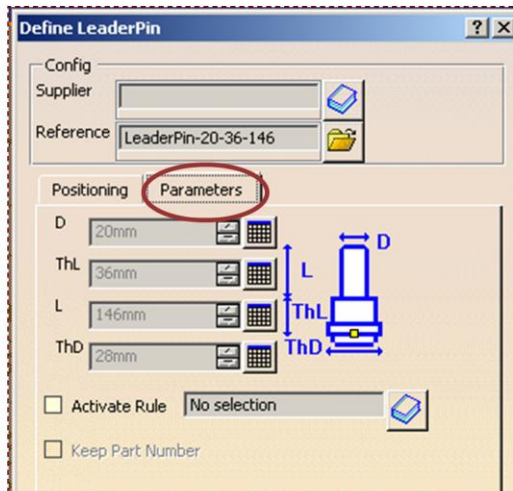
**The list of variants (Design Table) for the current type of components is accessible in tab Parameters.**
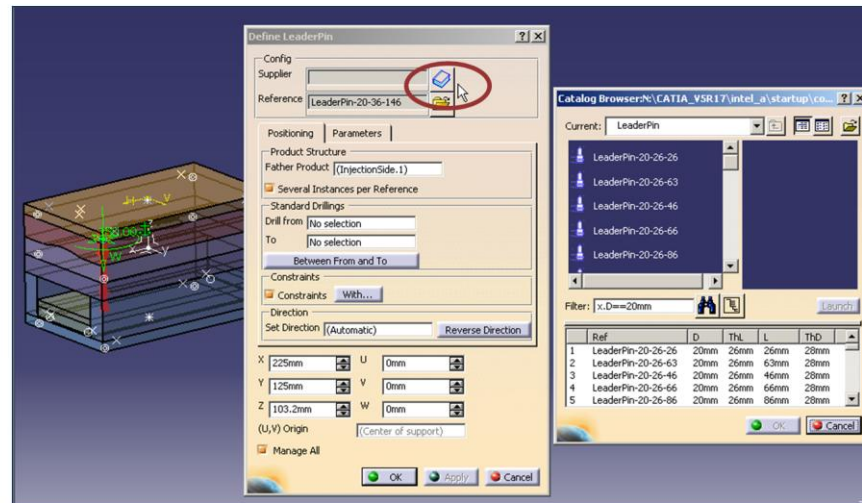
**Note that the 3D preview, the selection panel and the definition panel can be displayed simultaneously (click catalog icon in the component's definition dialog box).**
**If the preview shows that the selected variant (or provider, or subtype) is inadequate, just pick another one in the list and the preview will be updated automatically.**

# Drilling Holes (1/2)

- **Fields Drill From and To allow to define the first and last elements to be drilled by the component when it is inserted in the Mold or Die Structure. These elements are automatically drilled when validating creation.**
- **Button Between From and To gives access to another dialog box which enables you to explicitly define other affected elements between element From and element To.**
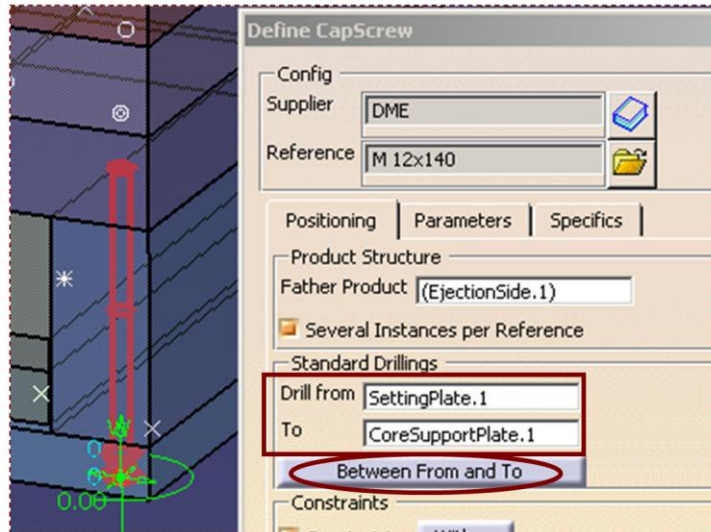
The elements Drill From and To can be selected in the viewer or in the tree
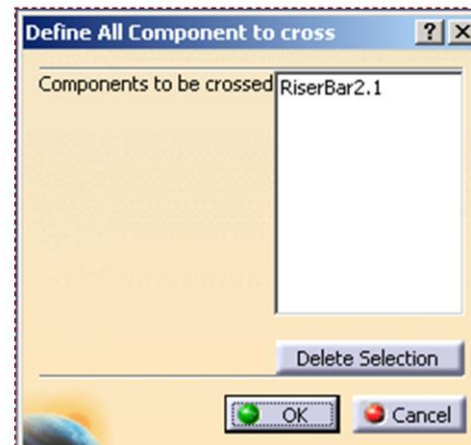
The list of affected elements Between From and To can be adjusted (add by selecting in the viewer or in the tree, remove in the dialog box list)

# Drilling Holes (2/2)

- **Drillings in the affected elements are created as Boolean features (Remove or Assemble, depending on Settings option) in the PartBody of these elements.**

- **These Boolean features are formed due to associated Technological Results, the technological information of the initial Hole features defined in the component (size, thread ).**
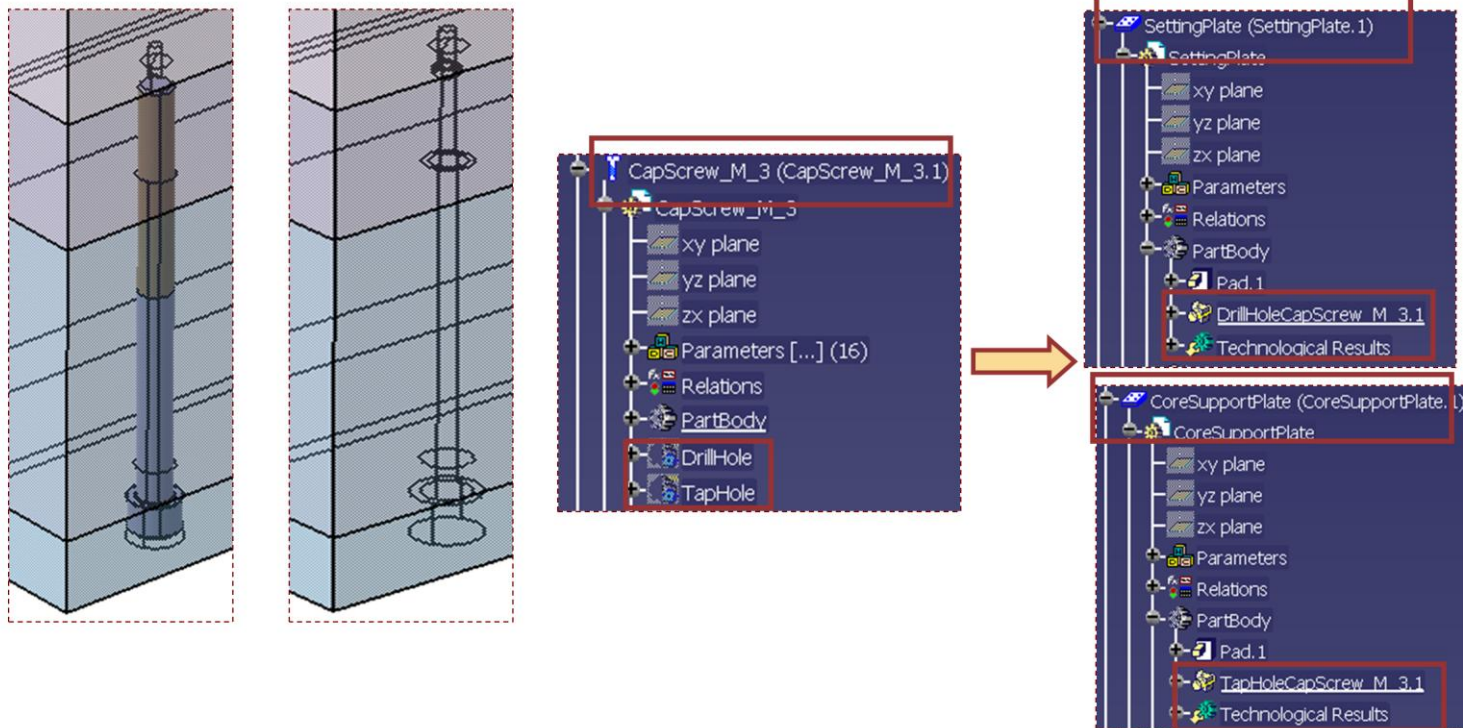
# Creating Product Structure (1/4)

- Field **Father Product** defines the position of the element in the specification tree.
  - ◆ It is set by default to the current active product in the assembly (blue highlight).
- If field **Drill From** is defined with an element, **Father Product** is automatically set by default to the Father Product of that element, unless the current active product is a child of this Father Product.
- If no **Drill From** is defined and if the corresponding product exists, the Father Product is defined by the following rules (relevant if you use an MTD Mold Base):
  - ◆ If the component is an Angle Pin or an Eye Bolt, the Father Product is **InjectionSide**
  - ◆ If the component is a Core Pin, an Ejector, an Ejector Sleeve, an Ejector Pin, a Flat Ejector, a Knock Out, or a Support Pillar, the Father Product is **EjectorSystem**
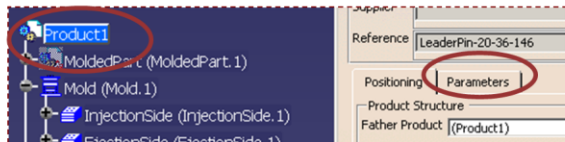  - ◆ If the component is a Slider, the Father Product is **Mold**.
- You can explicitly select an element in the specification tree as Father Product to override the default or automatic values.

**Initial status**

**After selecting ClampingPlate (father = InjectionSide) as positioning face**
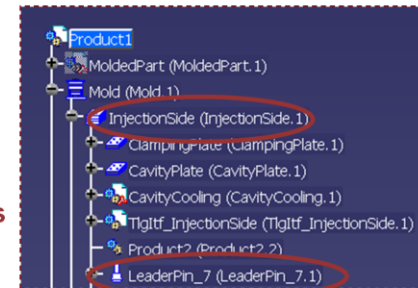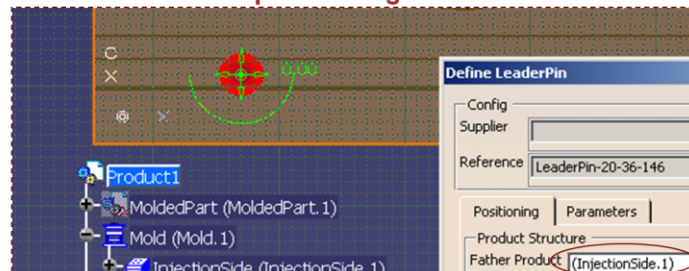
**Result = Component is located in sub-assembly InjectionSide**

# Creating Product Structure (2/4)

- **It is possible to define sub-structures inside original sub-products to better organize the location of the Mold or Die components in the tree structure.**
- **Sub-structures can be defined at any depth inside the structure tree.**
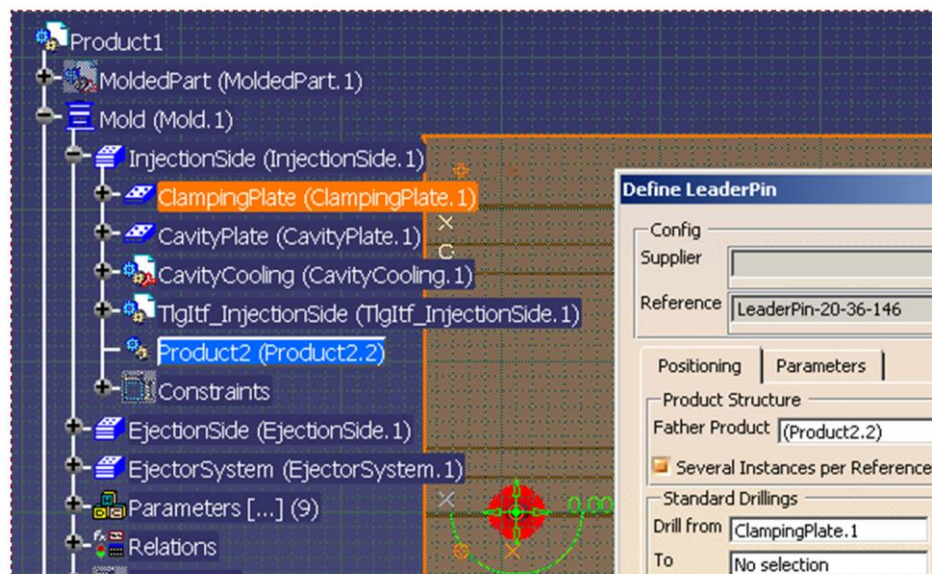
# Creating Product Structure (3/4)

- **Note when using sub-structures: Inserting a component automatically defines Offset constraints (if option Constraints is active in the Component instantiation panel) between the origin of the component and the element onto which it is inserted.**

- **Since Assembly constraints can be created only between children of a given structure (general CATIA V5 Assemble Design behaviour), it is therefore recommended to define a positioning part in the sub-structure where the location elements for the components (e.g. a sketch of points on top of a Mold plate) will be located.**



**Sub-structure to put screws**

**Positioning part containing sketch of points placed on top of ClampingPlate**
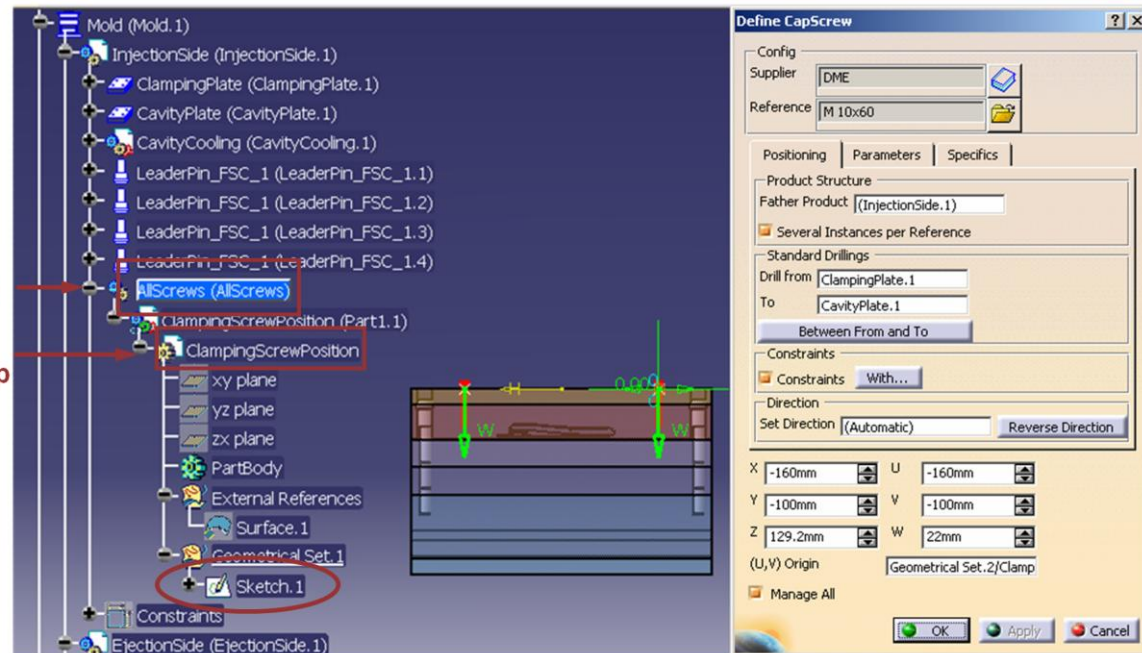
# Creating Product Structure (4/4)

- **Activating Settings option Keep link with selected object is useful to maintain associativity for example between the positioning points of a Sketch and the face of a Plate).**

61
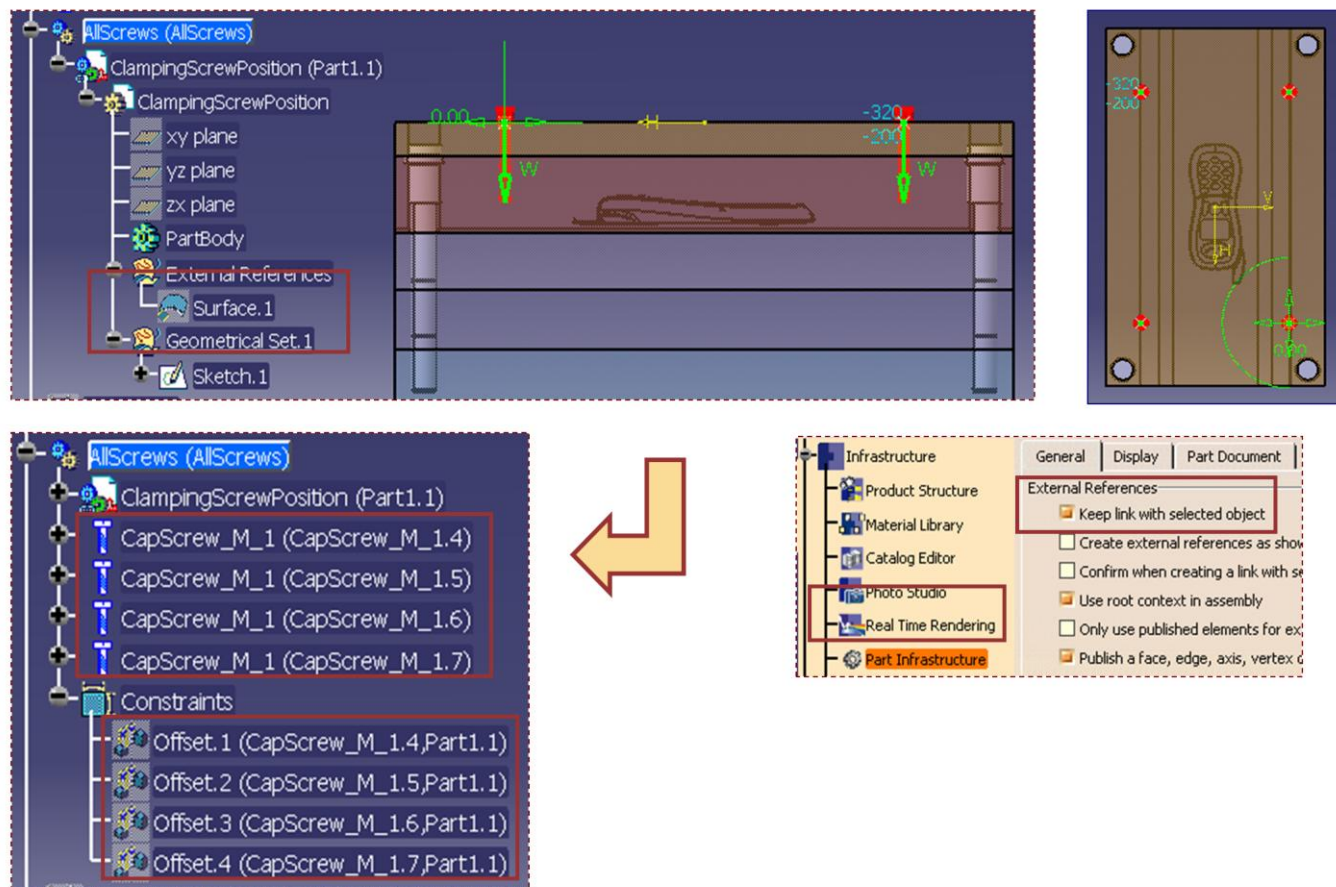
# Multi-instantiation of Components (1/3)

- Note that that you can create in the Tool Structure some predefined sets of 3D points, automatically recognized (naming convention based on the type of the Component) by the application when inserting a given type of component to perform automatic multi-instantiation.

- This automatic multi-instantiation is done only if option Manage All is active. You can erase one instance just by picking its W axis in the preview.

- For example, when picking one of the points dedicated to the positioning of Leader Pins, the corresponding additional instances of the Leader Pin are automatically positioned on the other predefined points.
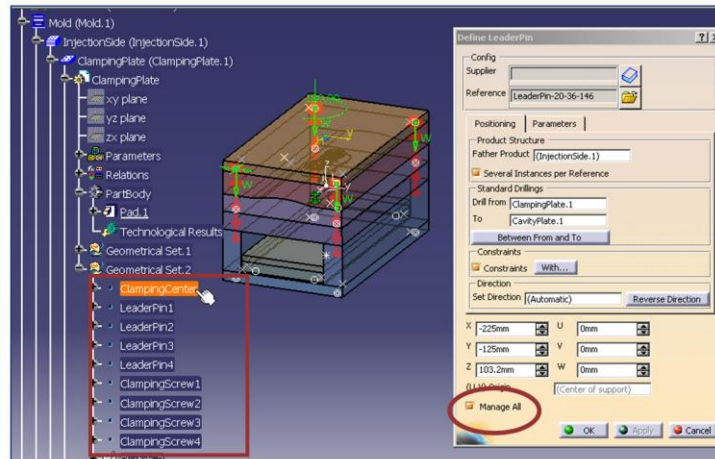


- Manual multi-instantiation (i.e. there are no predefined points): it is possible to multi-instantiate any given component in a single operation. You just have to pick several positioning points instead of one.

# Multi-instantiation of Components (2/3)

- **Option Manage All allows to control the number of current active instances:**
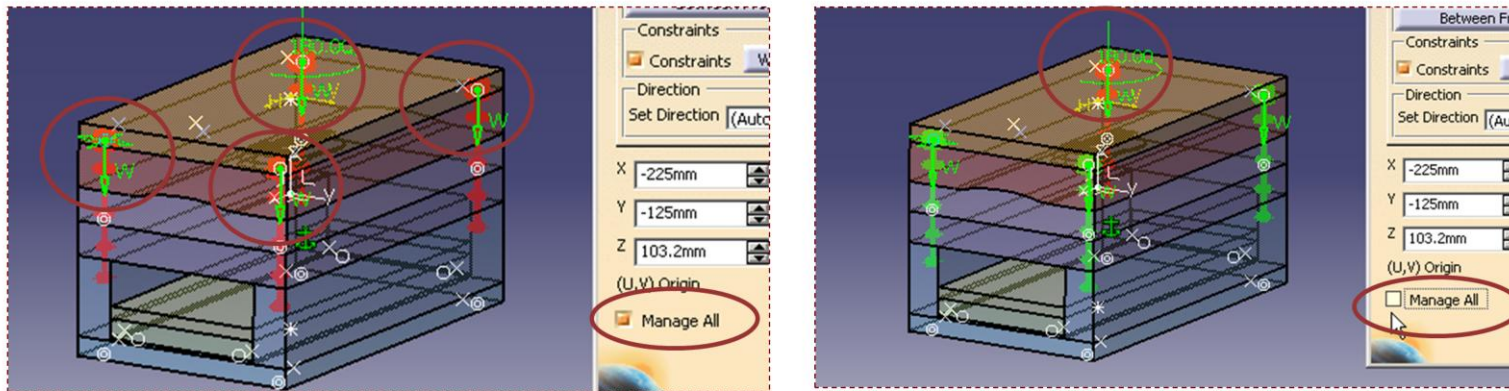


**Current instance(s) is (are) highlighted in red**

- **When option Manage All is active, positioning operations will be applied to all instances simultaneously. This allows for example to move all instances along Z in a single operation.**

- **If option Several Instances per Reference is active, only one CATPart (the Reference) is created at the end of the operation. This button in the Create dialog box allows to override, for the current operation, the value defined in Tools/Options.**
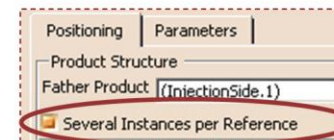
# Multi-instantiation of Components (3/3)

- A convenient method to position several identical Components in a single operation is to pick an existing Sketch of points: picking just one of the points is enough for TG1 application to automatically multi-select all the other points inside the Sketch (if option Manage All is active).

- For example, a group of Ejector Pins can be created using a Sketch of Points located on top of the Ejector Plate.
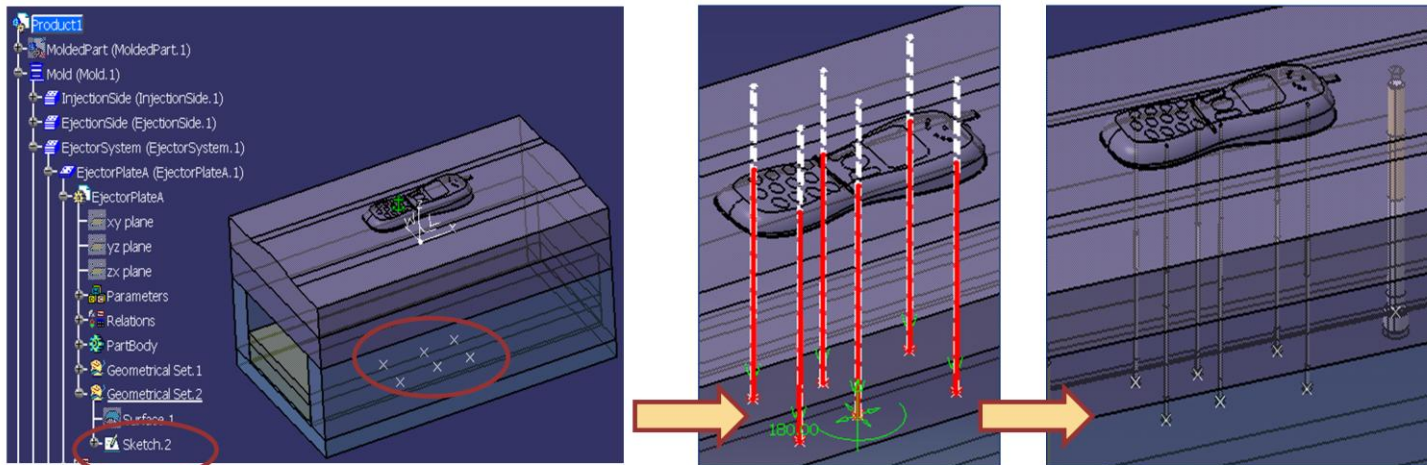
# More about Positioning (1/3)

- When working in Planar mode (component positioned on a planar face or a plane), the following pick possibilities are available to place the component:
  - A 2D point
  - A 2D point on the face
  - A 2D point outside the face (it will automatically be projected onto the face)
  - A Sketch (made up only of points) on the face
  - A Sketch (made up only of points) outside the face (it will automatically be projected onto the face)
  - A 3D point on the face
  - A 3D point outside the face (it will automatically be projected onto the face)
  - An axis, an edge, or a line that intersects with the face (the intersection point will automatically be computed)
  - The axis of an existing component (this axis will be displayed when moving the mouse over the component). Note that this is just a positioning capability and does not implicitly create a concentricity constraint between the existing component and the new one.

LeaderPin axis, visible when placing the mouse over the component.
It can be used to position component Bushing.
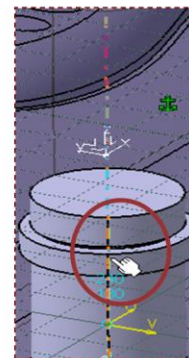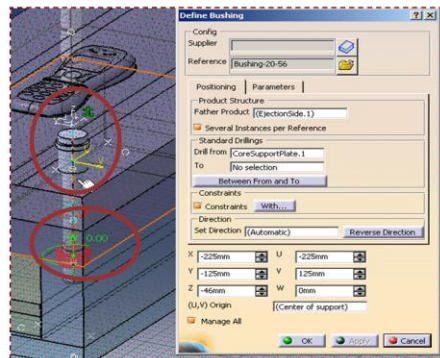
# More about Positioning (2/3)

- When working in Planar mode (component positioned on a planar face or a plane), the position of the anchor point is defined by (U,V) Origin:
  - (U,V) Origin is by default the center of the positioning plane,
  - U and V directions are the main directions in the positioning plane, W is the alignment direction and its origin is equal to its intersection with the positioning plane.

- To modify (U,V) Origin:
  - Pick a line, an axis or an edge: intersection with positioning plane will define (U,V) Origin.
  - Pick any point, even outside the positioning plane: normal projection on the positioning plane will define (U,V) Origin.

Point (U,V) Origin is displayed with a blue marker

The (U,V) position of the pointer is displayed and is updated as you move the mouse.
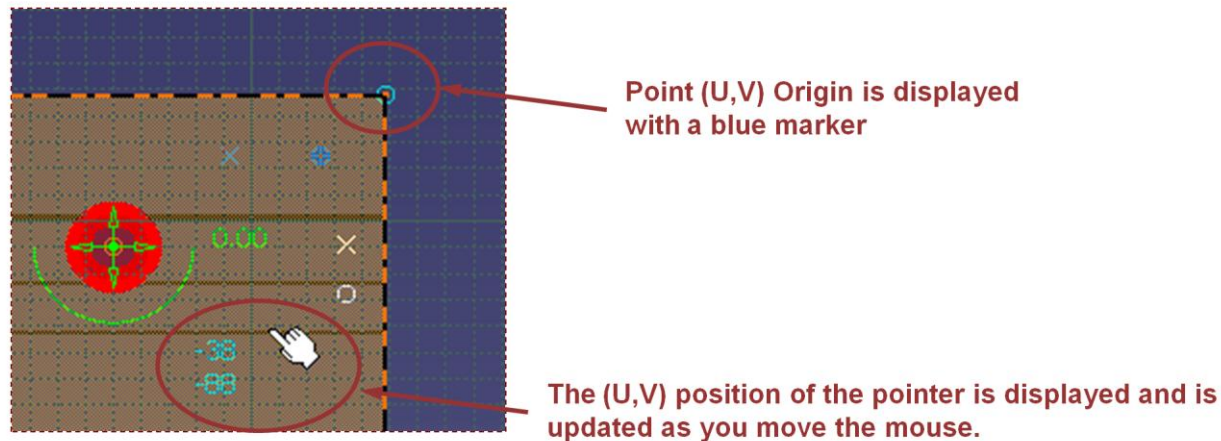
# More about Positioning (3/3)

- When positioning a component, you can apply translation movements, but you can also define a rotation around W by using the green circle arc displayed in the preview:
  - Either by dragging this arc: it will rotate the component accordingly,
  - Or by selecting MB3 while this arc is selected (highlighted in orange): contextual menu Edit angle will be displayed, allowing to rotate the component by increment steps of the displayed value.
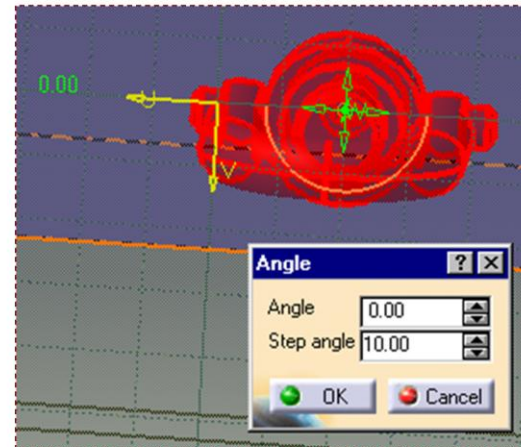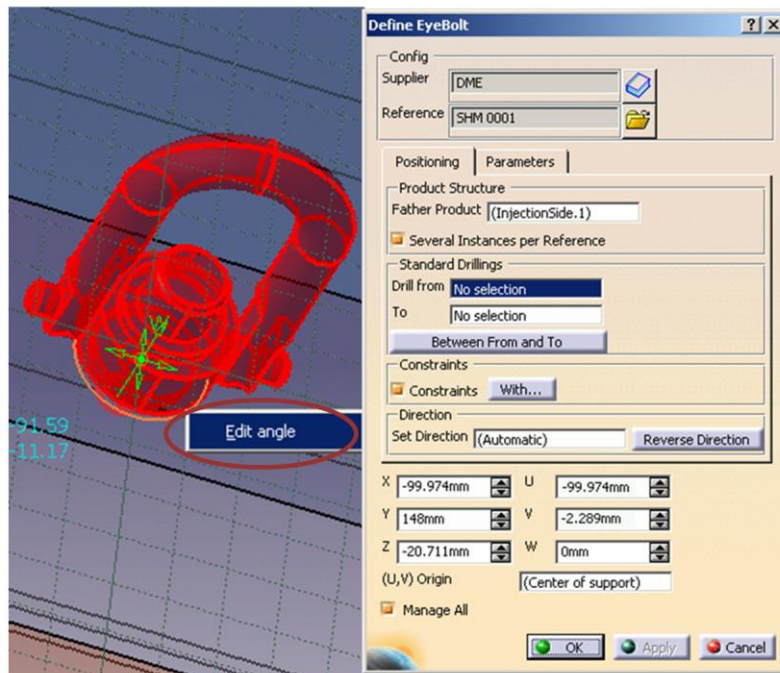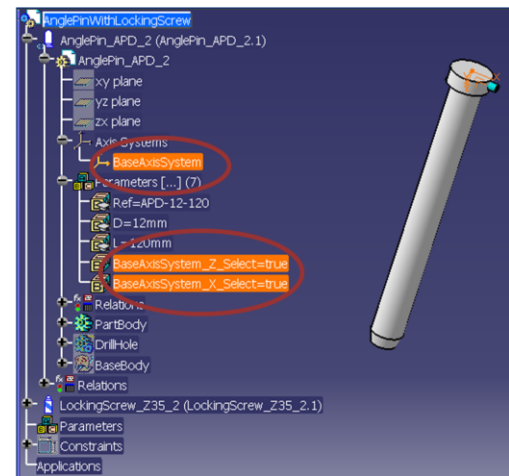
# Guided Mode Positioning (1/3)

- **This positioning mode enables to instantiate a Component in a more controlled way compared to the standard positioning capabilities.**

- **Based on additional information defined inside the Component, the user will be guided in the selection of positioning axes when instantiating the Component.**

- **Additional information to be defined in the Component (in the first Part, if the Component is an assembly):**

    - **Axis System, whose name must be BaseAxisSystem**

    - **Up to three Boolean Parameters, whose names must be BaseAxisSystem_X_Select, BaseAxisSystem_Y_Select, BaseBaseAxisSystem_Z_Select. Depending on the value (true/false) and order of these Parameters, the TG1 application will prompt the user for the corresponding information at instantiation.**

**Only two user-selected axes are necessary to define the position: the third axis is computed for the first two. Therefore, if there is a third Boolean Parameter set to true, the system will ignore it.**
**If zero or one only axes are defined, the system will implicitly use information from the general axis system.**

**In this example, the user will be first prompted for axis Z, then for axis X. Axis Y will automatically be computed.**

# Guided Mode Positioning (2/3)

- When instantiating the Component, the user is guided by a symbol displayed next to the cursor in 3D space:
  - First a small "x" symbol to pick the locating point of the Component: you can pick a 3D Point or a Sketch point or a vertex.
  - Then successively "X", "Y", or "Z" symbol with axis to pick the corresponding axes: you can pick an edge, or line, or axis.
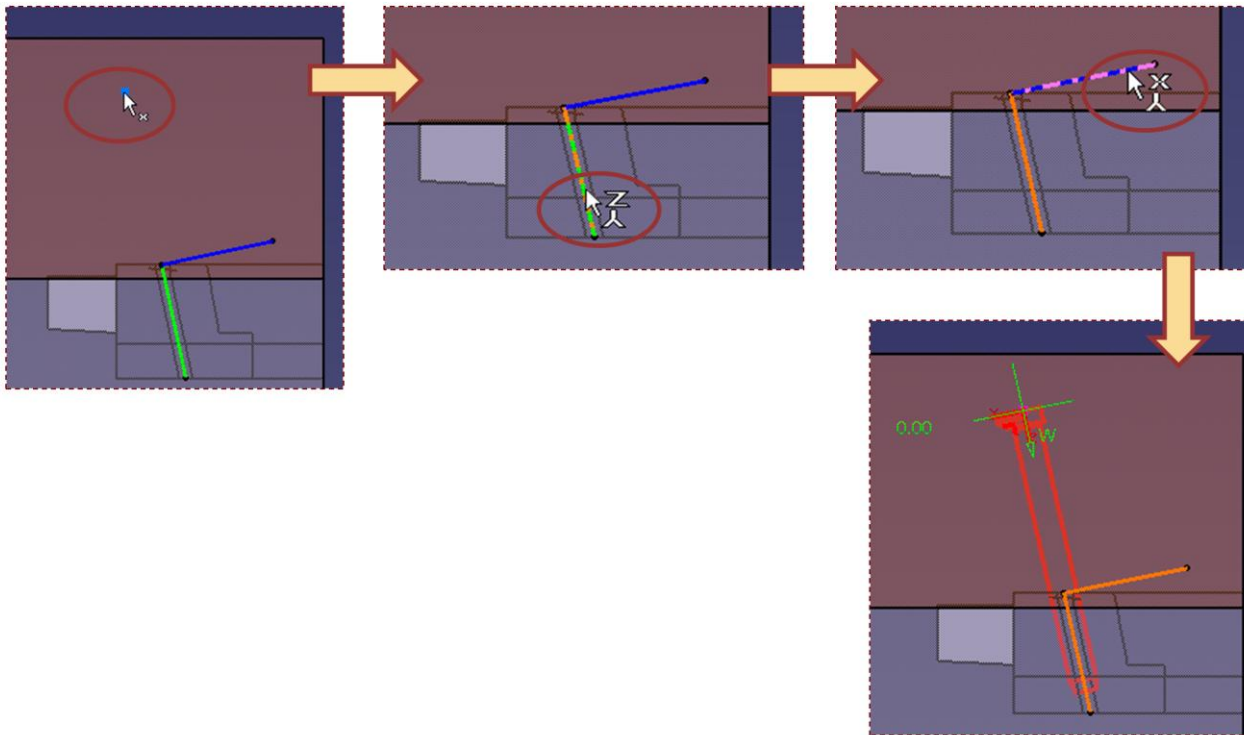
# Guided Mode Positioning (3/3)

- When the Component is instantiated, the BaseAxisSystem of the Component is displayed in the 3D viewer.
- The following constraints are created in the assembly:
  - A Coincidence constraint between the Component and the Point where it has been placed. If the user has applied a displacement (using for example the green arrows in the preview), and Offset constraint is created instead of a Coincidence.
  - One or several Angle constraints between the Component and the positioning axes.
- If the positioning elements are changed, the Components moves accordingly, using the constraints.
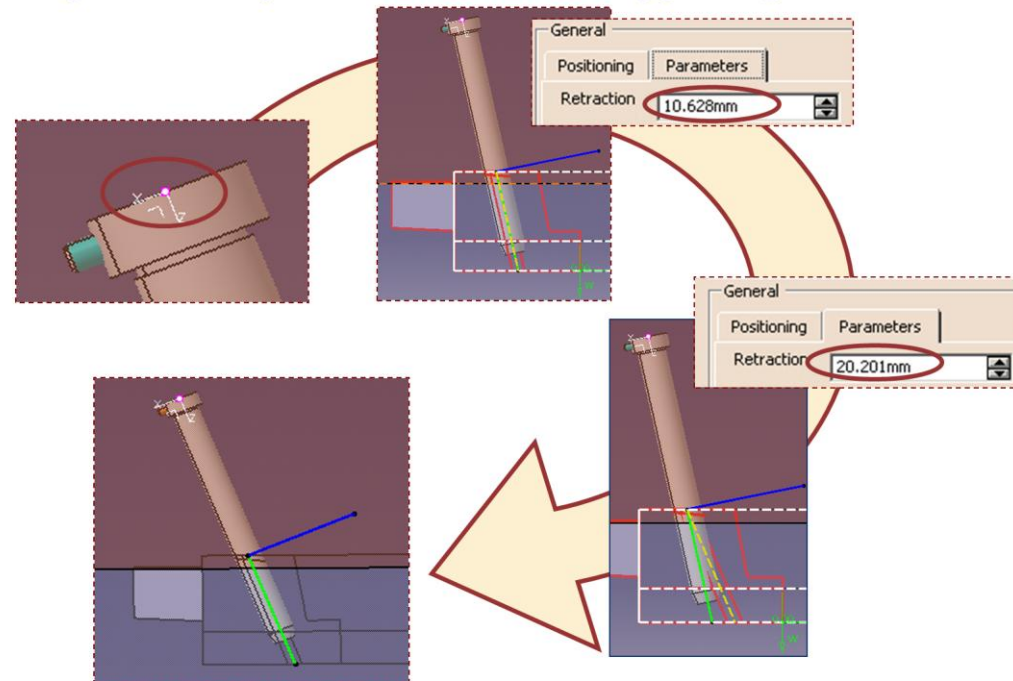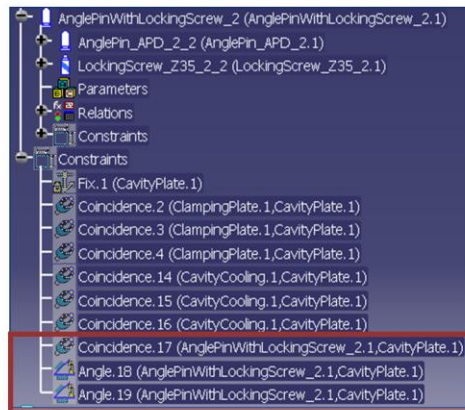
# Context Management (1/3)

- When creating Tooling Components, TG1-managed operations for add/remove of material (Drill From/To, Drill Component, Distributed Drill Component) and splitting (Split Plate, Split Component) automatically imply Copy/Paste As Result With Link of the corresponding elements (drilling Body, splitting Surface).

- As a result, the impacted Part then becomes contextual to the one containing the copied element: for example, the ClampingPlate of a Mold has a contextual link with the LeaderPin which drills it.

- If the impacted Part belongs to a sub-assembly different from the one of the impacting Component, it means that both sub-assemblies are simultaneously needed to continue working.

- This makes concurrent engineering difficult: for example if one user has to work only on mold sub-assembly EjectionSide, impacted by Ejectors located in sub-assembly EjectorSystem managed in parallel by another user.

- To facilitate concurrent engineering, a specific mechanism is automatically applied by application TG1: a dedicated Part (call Part Interface) is created in the impacted sub-assembly to gather Published copies of impacting elements. These copies are used in the sub-assembly to perform TG1-managed drilling or splitting operations. Therefore contexts due to TG1 Components are "confined" there and the existing contextual links of the impacted element are not modified.
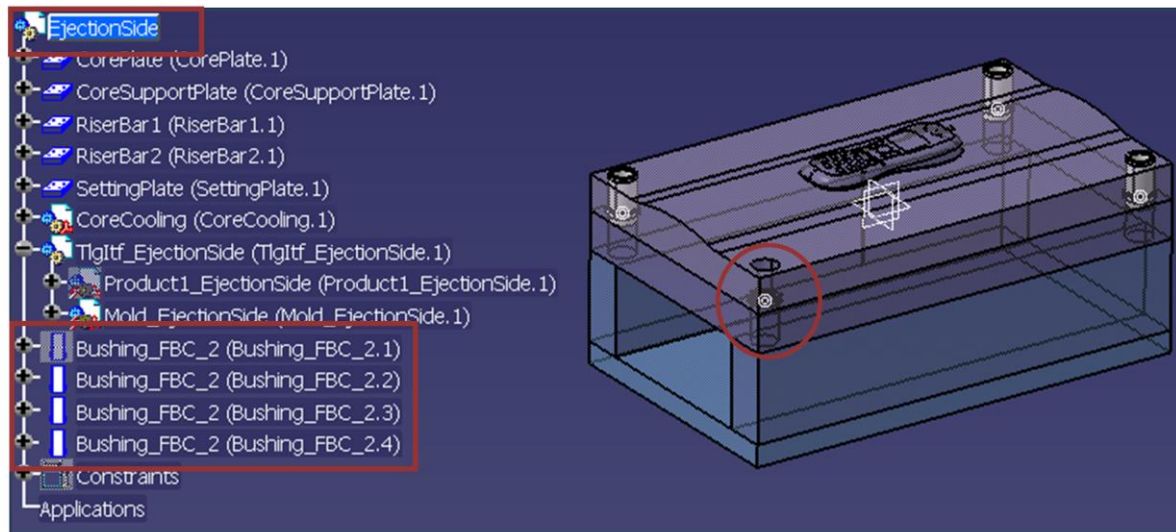
# Context Management (2/3)

**EjectorPin Component created in sub-assembly EjectorSystem impacts CorePlate and CoreSupportPlate located in sub-assembly EjectionSide**
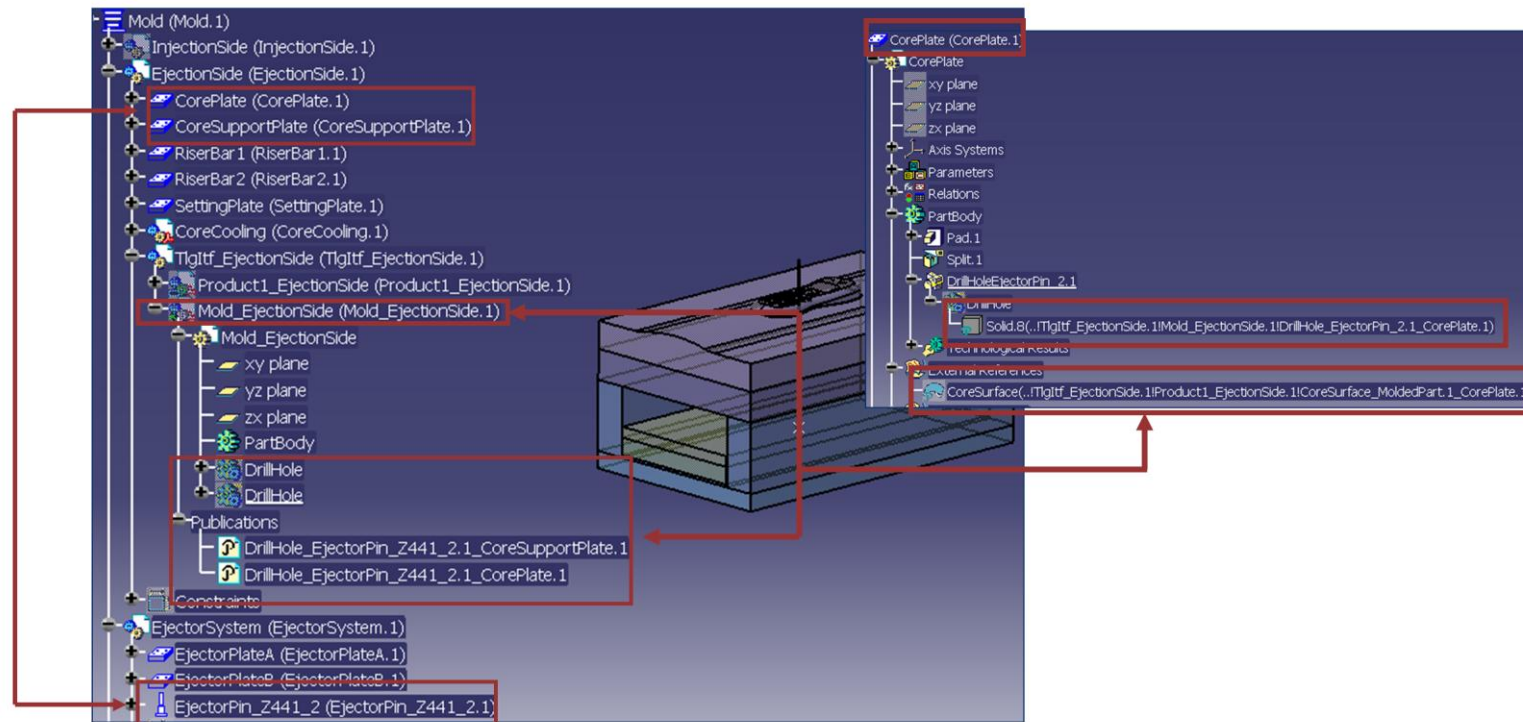


**Product Interface TlgItf_EjectionSide is consistd of:**
- **Part Interface Mold_EjectionSide containing two Published Copies of the EjectorPin Drill Hole: one for CorePlate and one for CoreSupportPlate**
- **Part Interface Product1_InjectionSide, resulting from the same mechanism applied when splitting the CorePlate with the MoldedPart CoreSurface**

# Context Management (3/3)



**Using the Part Interface mechanism, it is then possible to open individually Product EjectionSide and add for example Bushing Components**

- **NOTE 1: Part Interface context management does not take into account contextual links created by other means than TG1 functions mentioned above.**

- **NOTE 2: The Part Interface is not intended for access by the end-user. Therefore there is only a limited set of possibilities like show/noshow, expanding/collapsing the tree, etc. But for example it is not possible to manually create a feature inside.**
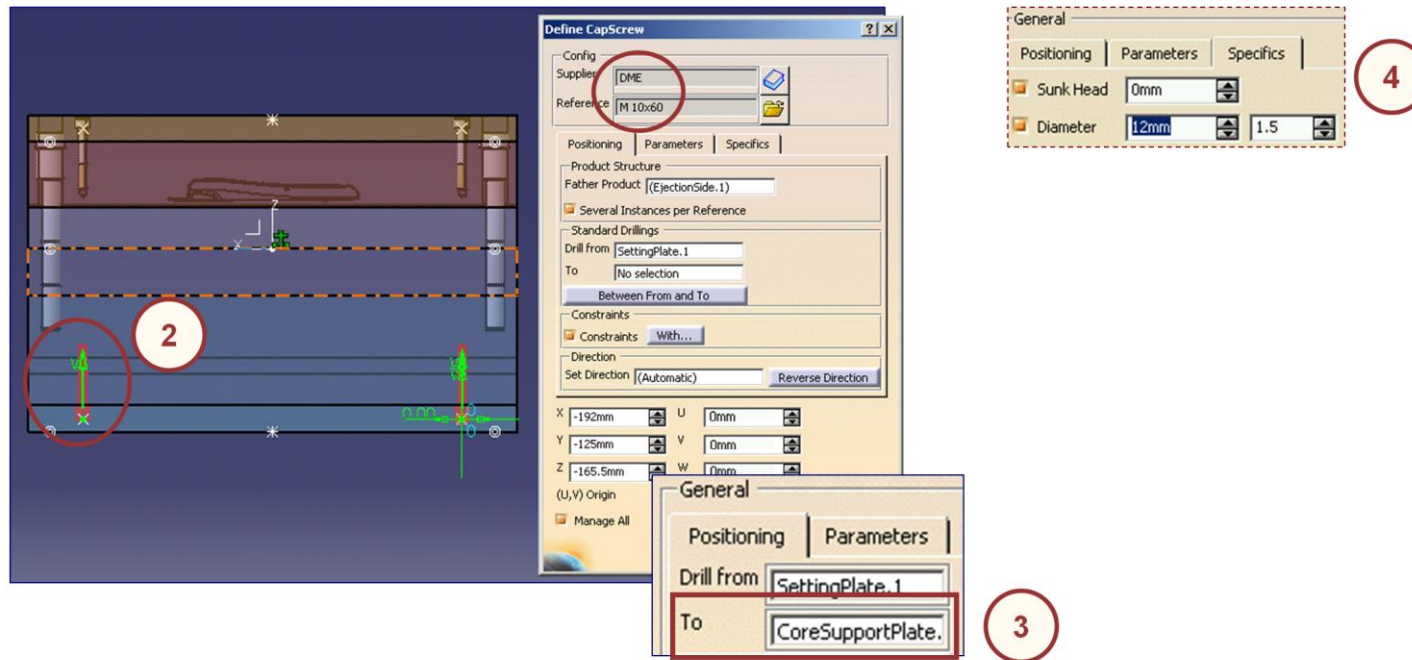
# Assisted Selection of Screw Length (1/5)

- For screw components (Cap Screw, Countersunk Screw, Locking Screw), additional assistance is provided to help select the most appropriate length variant in the catalog.

- To benefit from this capability, you must first define element To and then select the required sub-type of Screw (e.g. DME type M).

1. **Click required Screw icon.**
   When the Screw dialog box is displayed, the variant of Screw used in the previous operation is pre-selected (usual behavior for any type of component).
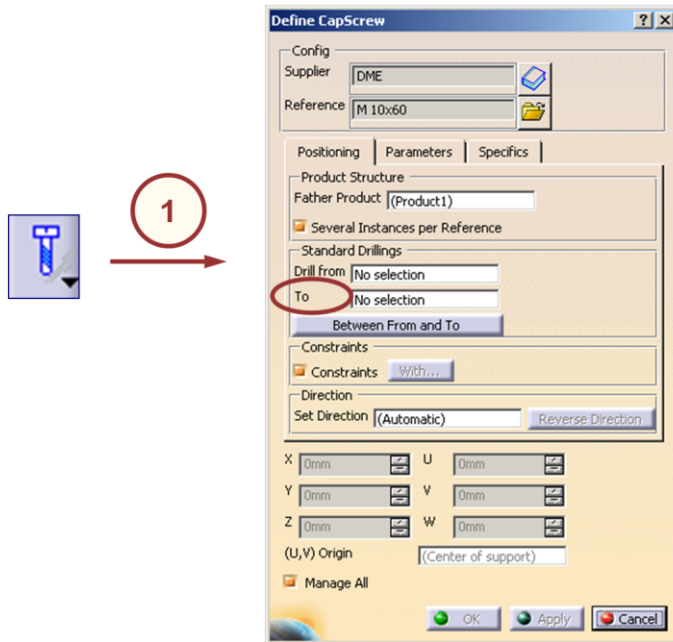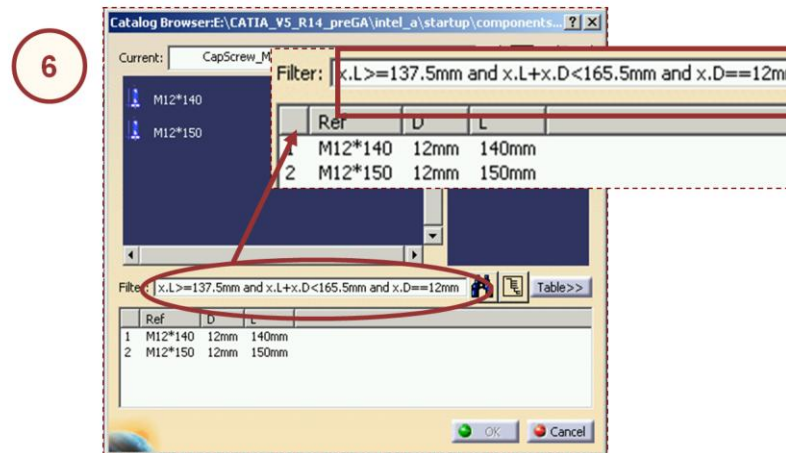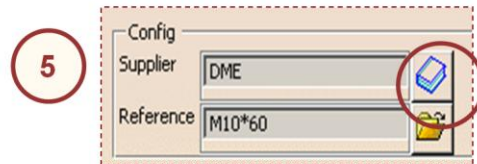
# Assisted Selection of Screw Length (2/5)

2. **Select the required location(s) for the Screw(s) to be inserted. The preview displayed still corresponds to the size variant selected in the last operation.**
3. **Select Drill element To in order to indicate what is the extremity element to fix. For example: CoreSupportPlate. The element Drill From can be defined now or later.**
4. **In tab Specifics:**
   a. **Activate Diameter and set required value.**
   b. **Modify if required the minimum value for Thread Engagement ratio (default = 1.5 * Diameter).**
   c. **Activate and define if required a depth value for the screw head.**

# Assisted Selection of Screw Length (3/5)

5.  **Click Catalog icon.**
6.  **The list displayed is restricted according to a filter. This filter is automatically computed to select only the screw variants whose size can fit the requirements previously defined: positioning point, target drilled element, diameter, thread engagement ratio, Sunk Head.**
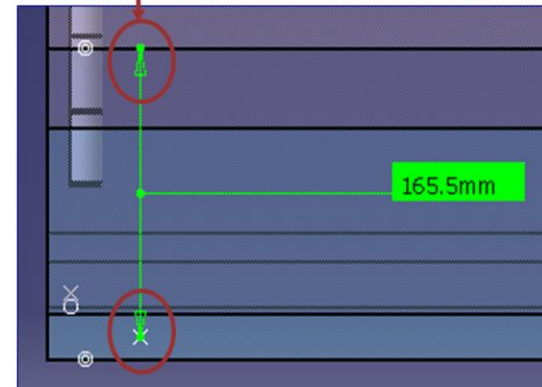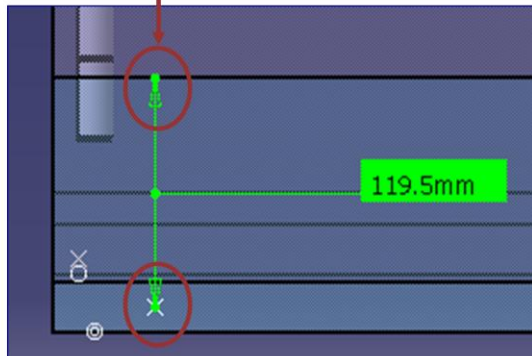
# Assisted Selection of Screw Length (4/5)

- The minimum and maximum values for the screw length (parameter L) computed by the application correspond, respectively, to the distance between the active preview location and the first face and last face of the extremity element (To).

- Note that only the rough form of the element is taken into account for this computation: if for example the element is the CorePlate split to the shape of the Molded Part, the distances considered for computation are those of the original Pad.

Filter: x.L>=137.5mm and x.L+x.D<165.5mm and x.D==12mm

| | Ref | D | L | |
|---|---------|------|-------|---|
| 1 | M12*140 | 12mm | 140mm | |
| 2 | M12*150 | 12mm | 150mm | |

119.5mm

165.5mm

# Assisted Selection of Screw Length (5/5)
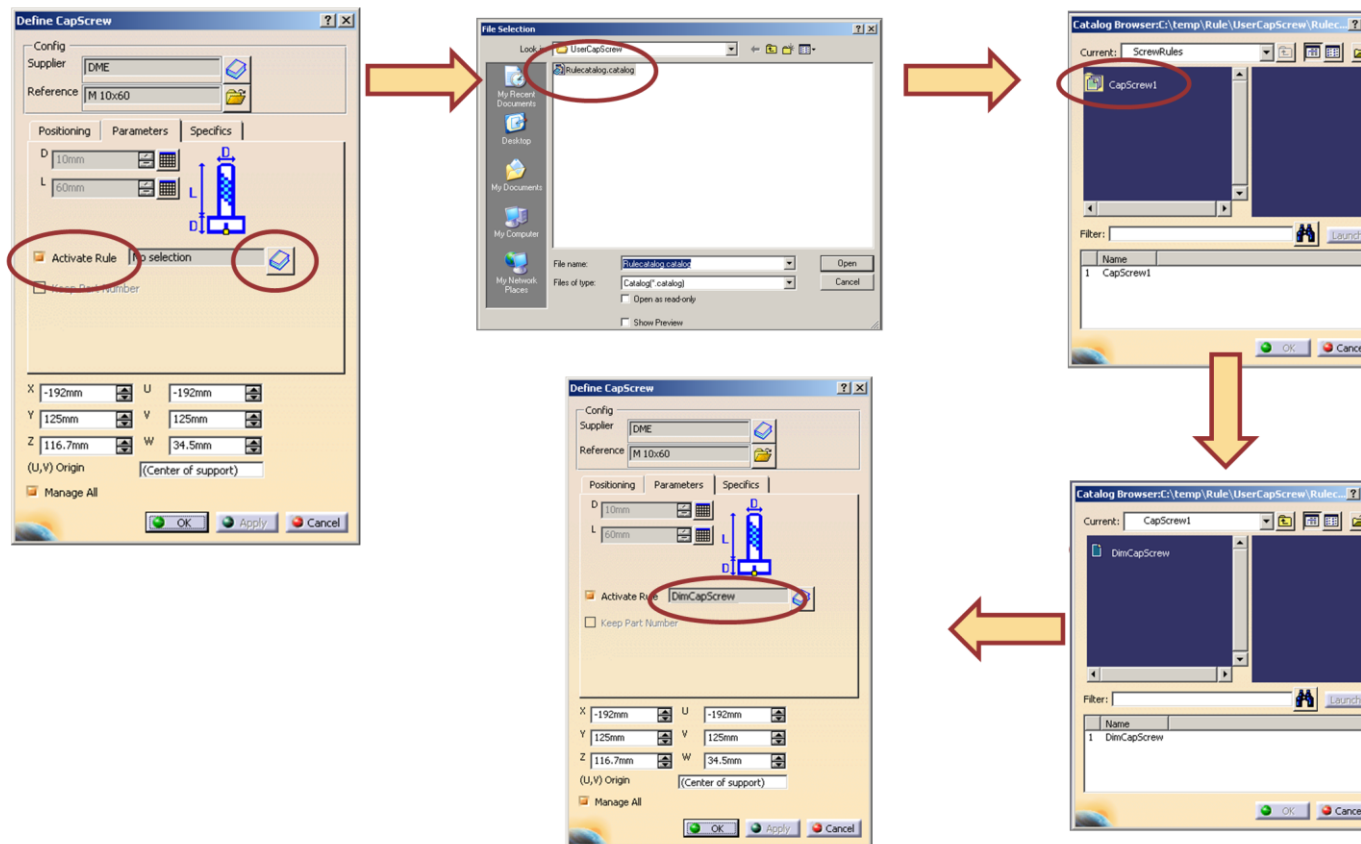
- You can select a variant in the restricted list, depending on the diameter you want to use. The preview is updated accordingly.
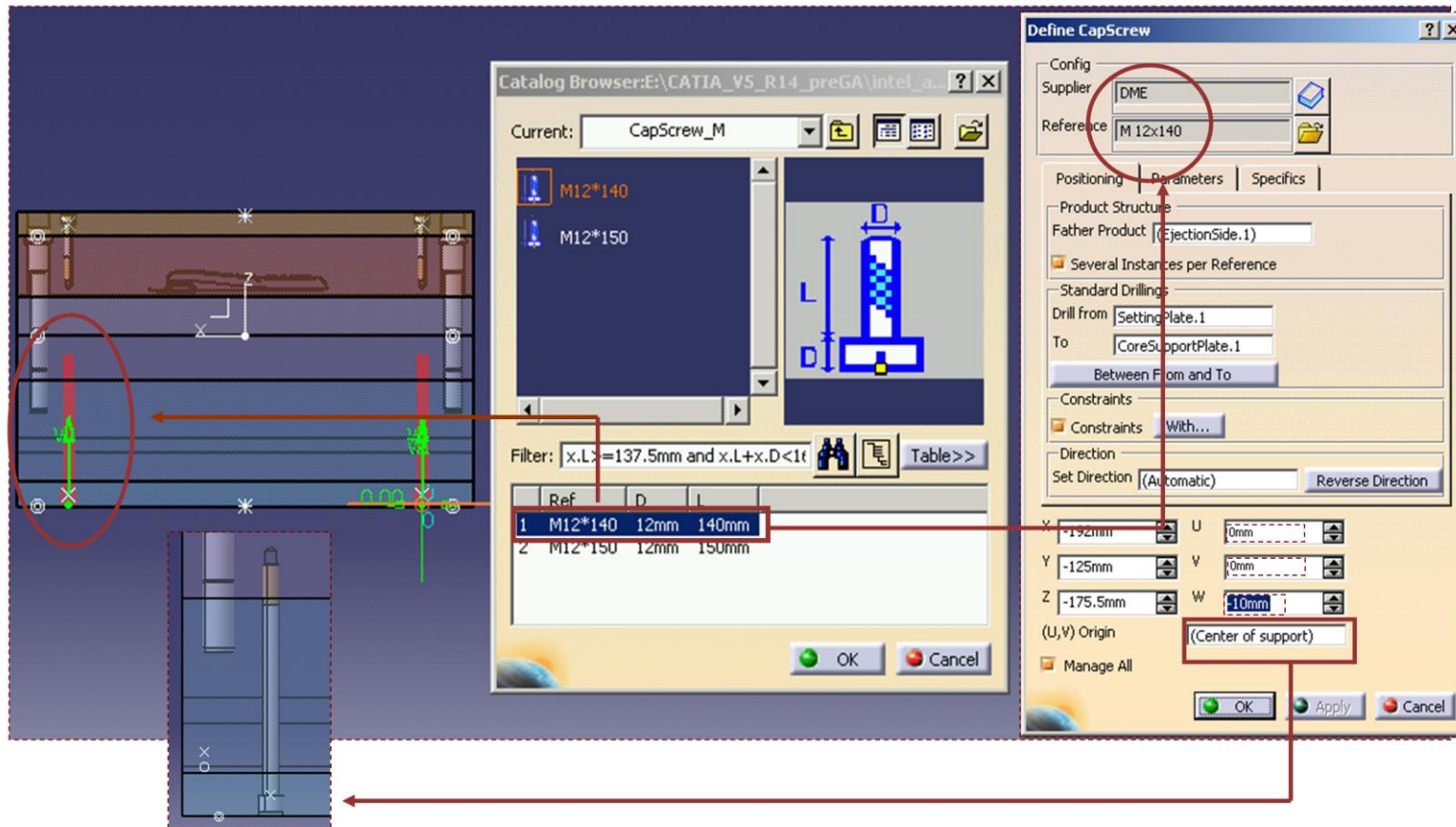
# Creating a Component: Using a Knowledgeware Rule

When inserting a component, you can customize its behavior by picking a rule in an existing catalog. The rule will be inserted in the component created in the Mold or Die.

# Creating a New Instance of an Existing Component

- It is possible to create one or several new instances of a component already existing by using the dedicated function Add New Instance in the contextual menu of this component.

- The benefits are:
  - Minimizing data.
  - Ensuring consistent behavior between copies of the same component.
  - Allowing creation of a more accurate Bill Of Material.

- If the original reference is split, it is not possible to perform action Add New instance

- If option Several Instances per Reference is not active, you will create a new reference instead of a new instance.

- If the original reference is a Product:
  - The result Product will be an instance if option Several is active
  - The result Product will be a new reference if option Several is not active
  - The subparts will be the same as in the original Product, whatever the value of option Several.
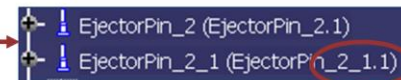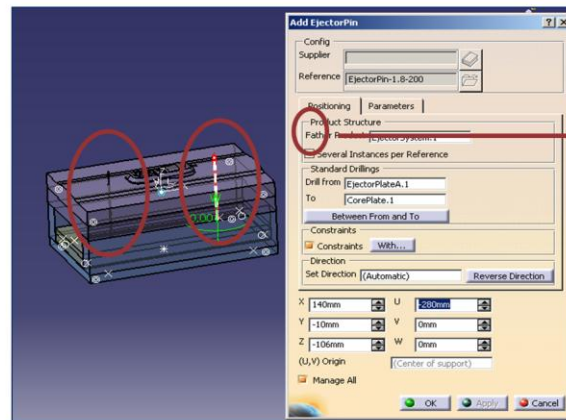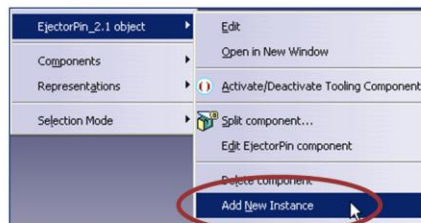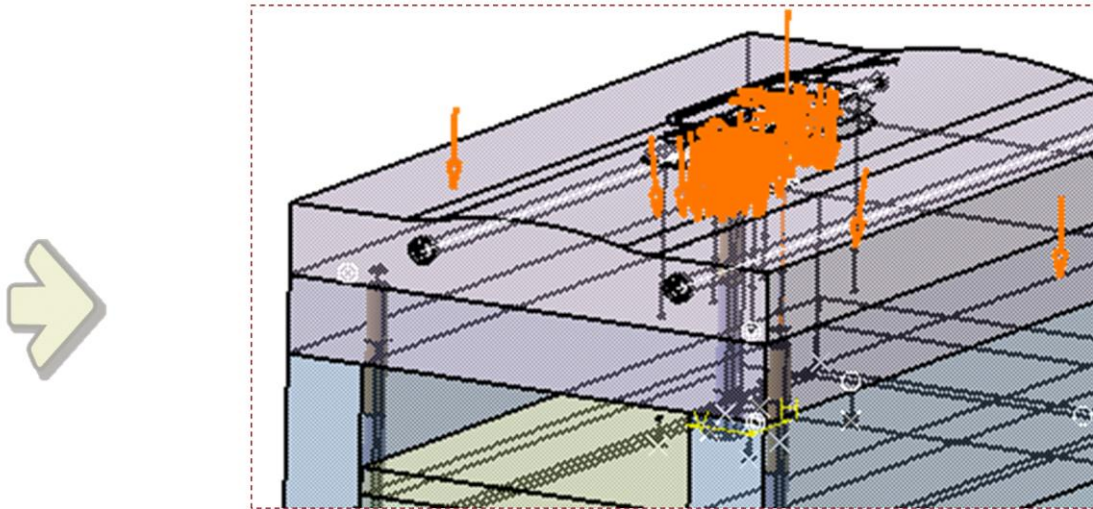


Option Several not active
=> a new reference is created

# Editing Components

*You will see how to modify the definition of a component, how to fit its shape to Core and Cavity requirements if needed, and how to delete or deactivate it in a way that ensures data consistency.*
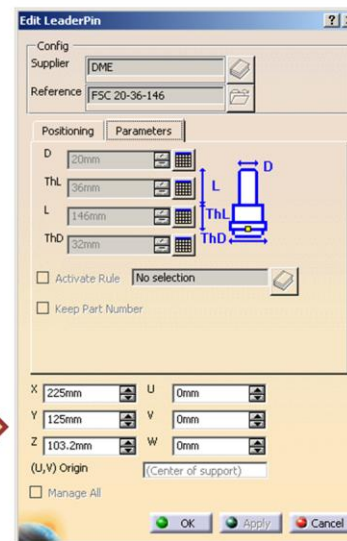
# Editing the Definition of a Component (1/2)

- For each type of component, an Edit Component function is available in its contextual menu.
- This function gives access to the definition dialog box of the component, enabling you to edit most its parameters defined: you can change the component variant to select other dimensions.
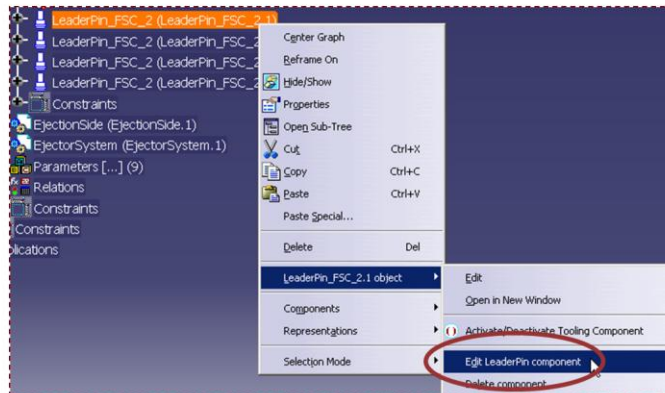


**Tab *Parameters* allows access to the Design Table containing the list of available dimensions**

# Editing the Definition of a Component (2/2)

- **When there are several instances of a given component:**
  - ◆ **Editing the position of one instance (via fields Instance Origin or using the green arrows in the preview) applies to the selected instance only.**
  - ◆ **Changing the dimensions of one instance affects all the other instances too (the common reference of the instances is modified).**
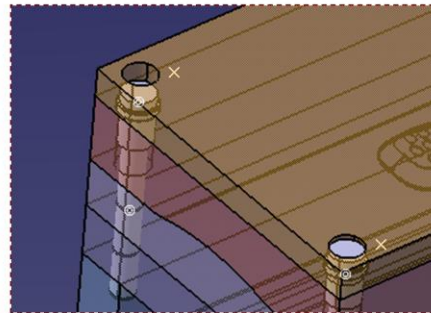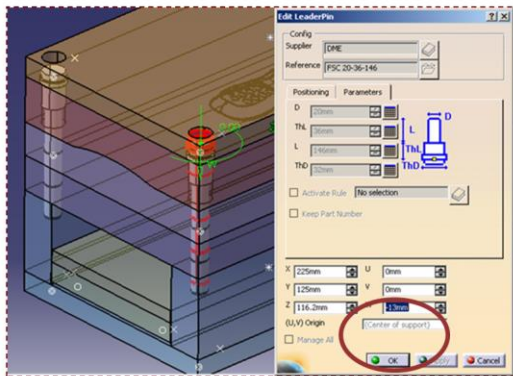


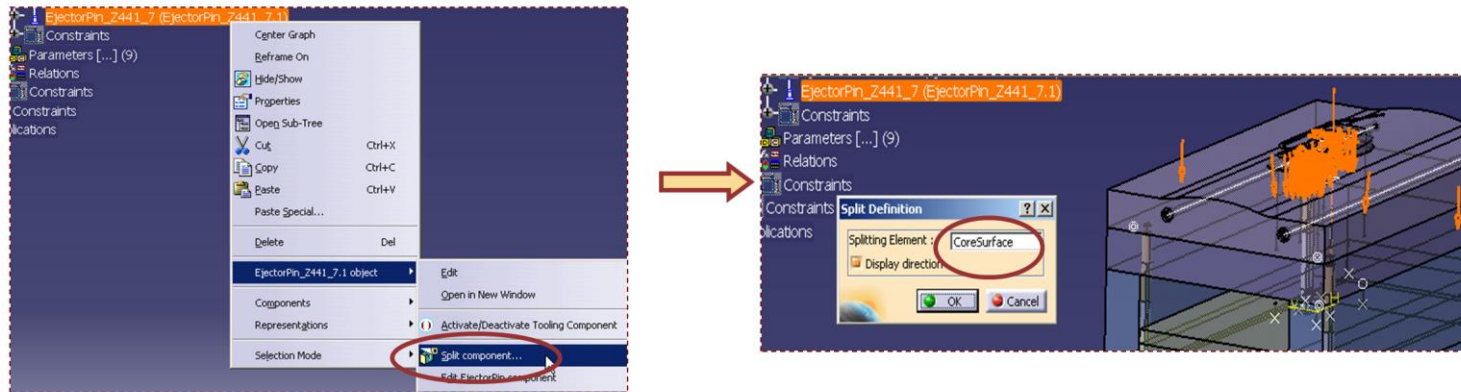**Only the position of the selected instance is modified**

**Selecting a new variant with different dimensions modifies all the instances**
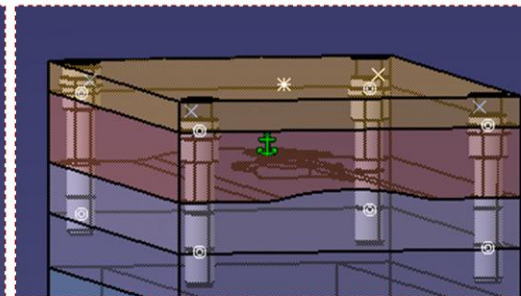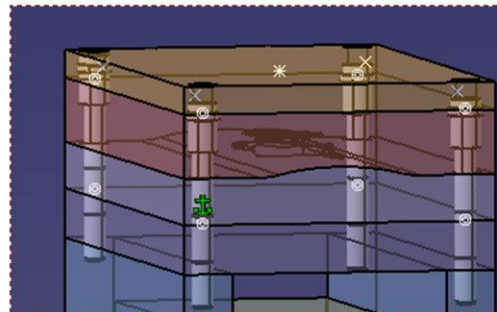
# Splitting a Component (1/5)

🔹 **For the types of Components which can be affected by the form of the Mold or Die Part (Insert, Ejector Pin, User Component ) a dedicated function Split Component is available in their contextual menu.**



🔹 **Multi-selection is available (you can split a list of elements in a single operation).**

🔹 **Note that, for consistency reasons, it is not possible to split a component for which several instances have been created from the same reference. To do so, deactivate the option Several Instances per reference for this component at creation time.**

# Splitting a Component (2/5)

- ▣ **The Split Component function automatically performs the following operations:**

  - ◆ **Copy of the selected Surface.**

  - ◆ **Paste Special As Result With Link into the Tooling Component.**

  - ◆ **Split of the Tooling Component (PartBody and Add/Remove Bodies like DrillHole if their name does not begin with double underscore).**

- ▣ **Before CATIA V5R16, this default behaviour applies to all Tooling Components.**

- ▣ **This default behaviour can lead to a dramatic increase in the overall size of the Tooling Assembly (disk and memory) if the Splitting Surface is big and/or many Components include it.**
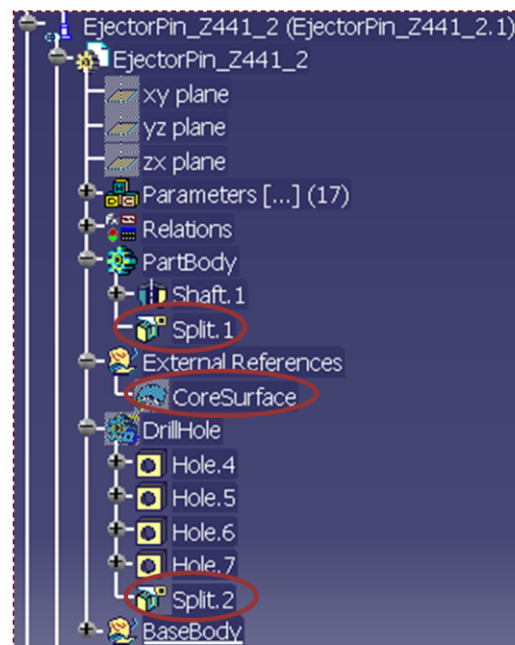
# Splitting a Component (3/5)

- As of V5R16, a new behaviour (Split Reduction) automatically applies in the following cases in order to minimize disk/memory consumption:
  - ◆ Splittable Tooling Components coming from TG1 catalogs provided when installing CATIA (even in pre-V5R16 releases), because they natively include the appropriate Split Reduction Parameter:
    - Ejector, EjectorPin, CorePin, rectangular Insert: parameter D
    - round Insert: max value of parameters L and D
    - FlatEjector: max value of parameters H and G
    - EjectorSleeve: parameter InD
    - SpruePuller: parameter RunD
    - SprueBushing: parameter ShD
  - ◆ User Components including Split Reduction parameter named D_Split.
  - ◆ If the Component is a Product, the appropriate Split Reduction Parameter must be present in all its Parts affected by the Split.
- The Split Reduction parameter is used by application TG1 to automatically define the size and compute a restricted surface to split the Component.
- This restricted Surface is associative to the position of the Component and to the full original Surface if the Component position is changed with function Edit Component.
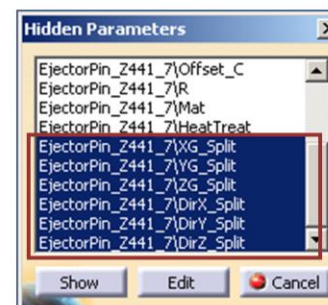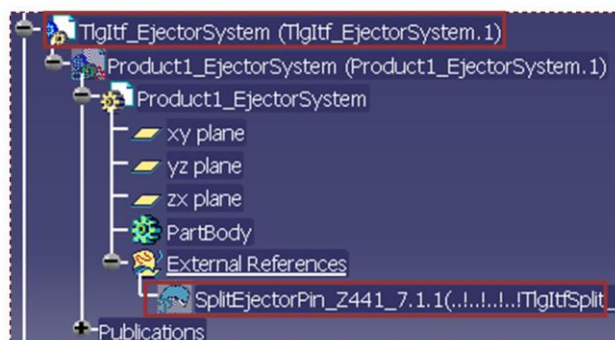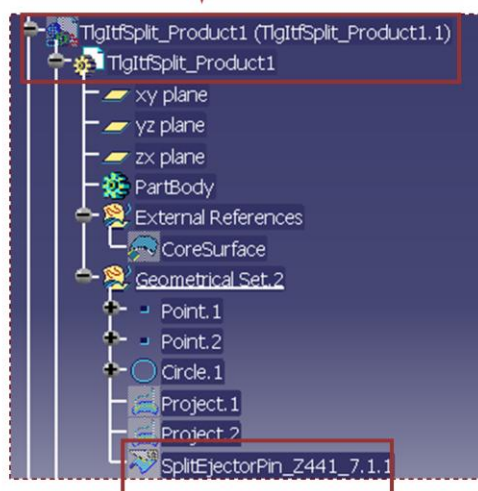
# Splitting a Component (4/5)

**The EjectorPin is split only with a restricted area of the full original surface**

**The restricted area is computed in a dedicated Split Part, automatically created at root level of the Assembly**
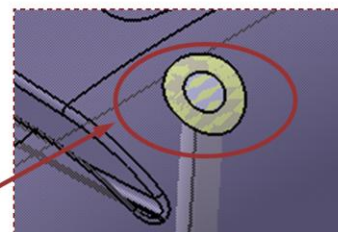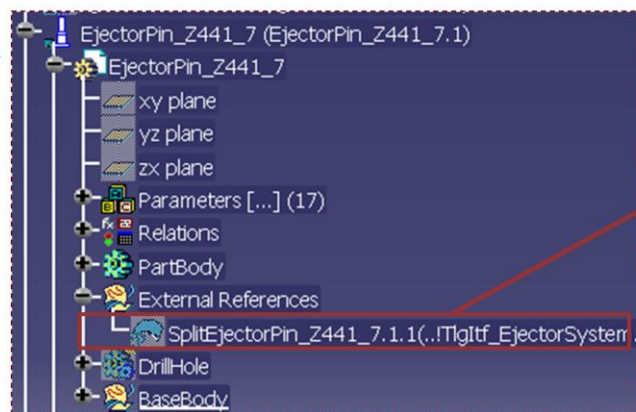
**Application TG1 automatically computes (hidden) parameters describing the position and direction of the Component in the Assembly.**



**The Part Interface mechanism is automatically applied to the restricted area: the restricted area is copied from the dedicated Split Part into the Part Interface of the relevant sub-assembly (EjectorSystem). This copy is then used in the Component (EjectorPin).**
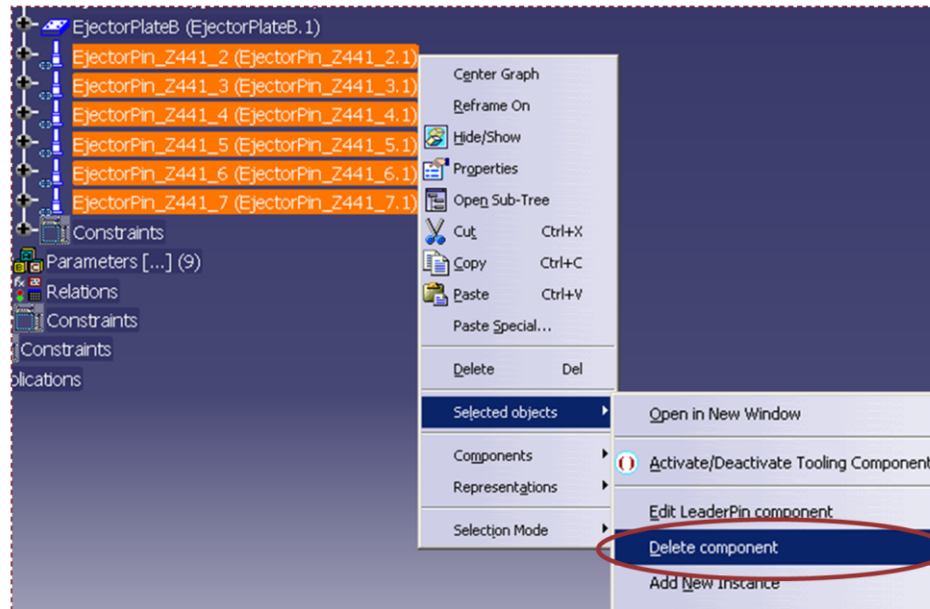
# Splitting a Component (5/5)

- **In case the Component's position is modified by another way than Edit Component function (for example by changing the position of the support Sketch or 3D Point is has been placed onto), the restricted split surface does not "follow" the Component: a synchronization operation has to be performed by the user.**

- **The new command Split Synchronization analyzes all splits with surface reduction which are performed by a given surface and updates these splits (update is performed only on elements needing it).**
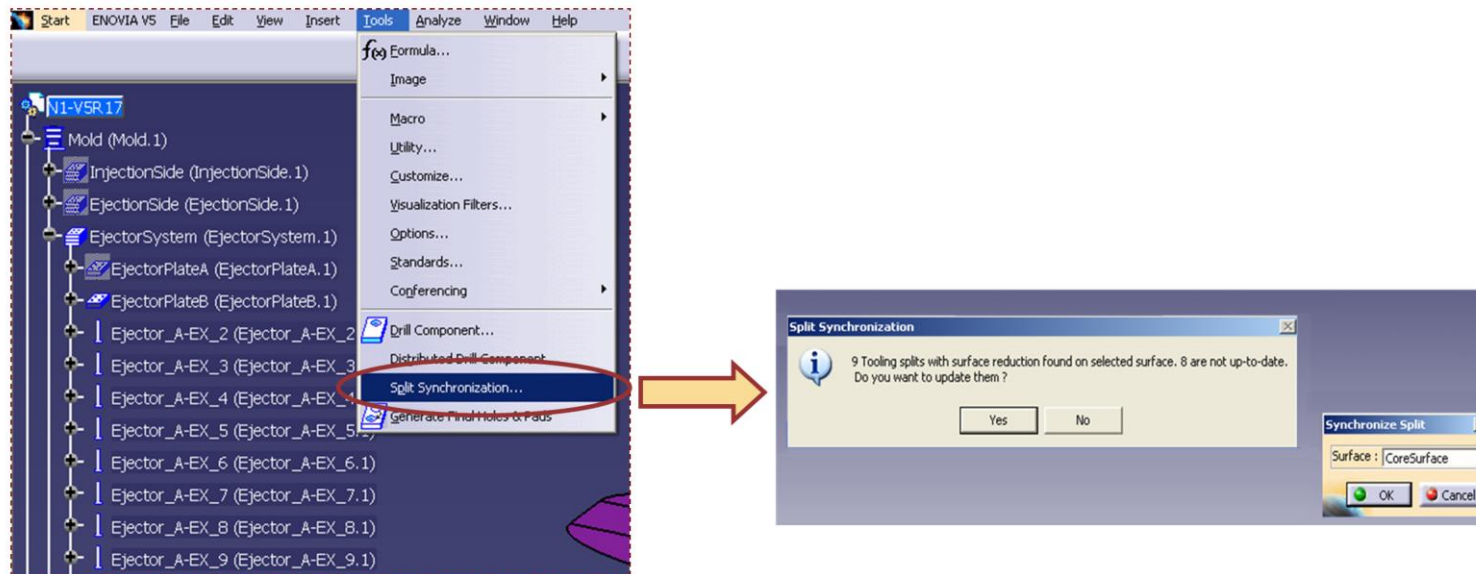
# Activating/Deactivating a Component

- **For all components, there is a dedicated function called Activate/Deactivate Tooling Component in their contextual menu.**

- **This function not only activates/ deactivates the components, but also its associated drillings in the various plates, and relevant constraints. It is mandatory to use this function instead of the standard Activate / Deactivate Component function, which does not ensure such consistency.**

- **Several components can be deactivated/activated simultaneously, by multi-selecting them in the 3D viewer or in the specification tree.**
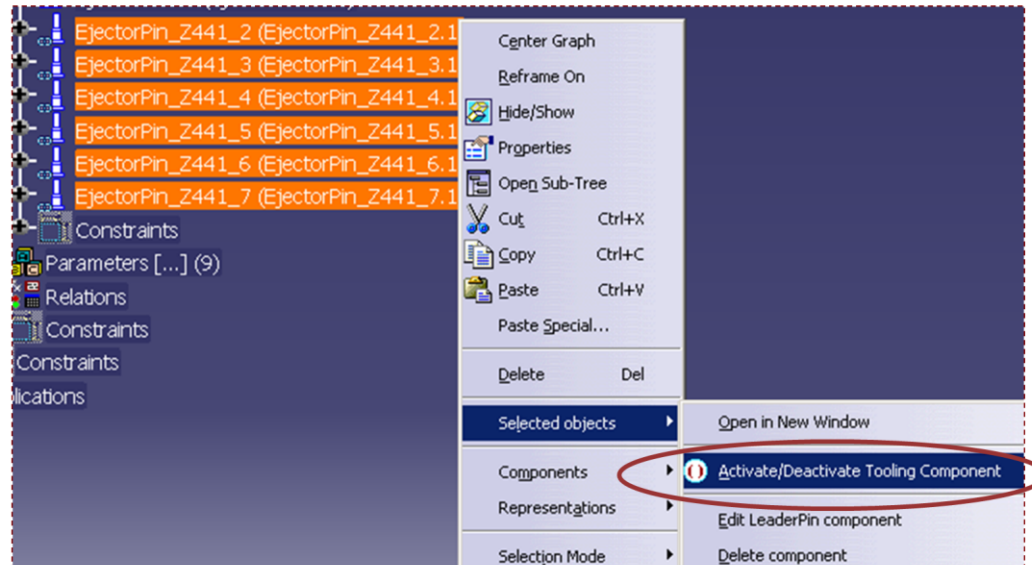
# Isolating an Existing Component

- It is possible to create a Reference of an existing component currently defined as an Instance. This is done by using the dedicated function Isolate Component in the contextual menu of this component.

- This allows to transform an instance of a component in order to edit it independently.
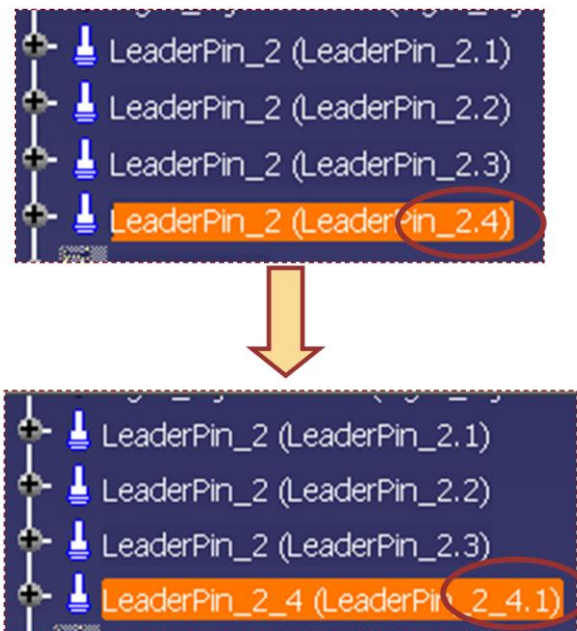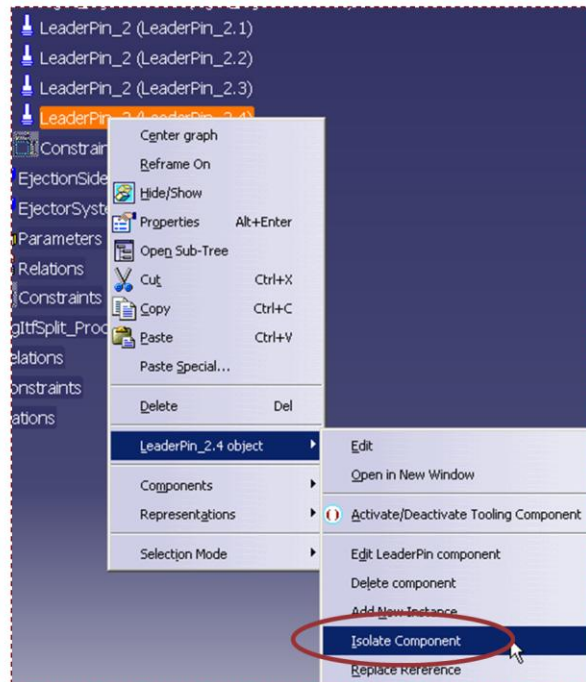
# Replacing an Existing Component

- It is possible to replace an existing Tooling Component by using the dedicated function called Replace Reference in the contextual menu of this component. All the Instances of this Reference will be automatically modified.

- It allows a quick and easy replacement without having to delete and create another element, for example to switch from one provider to another.

- Option Keep part Number (inactive by default) allows to keep the initial Part Number if required (only the Instance Name is then modified).
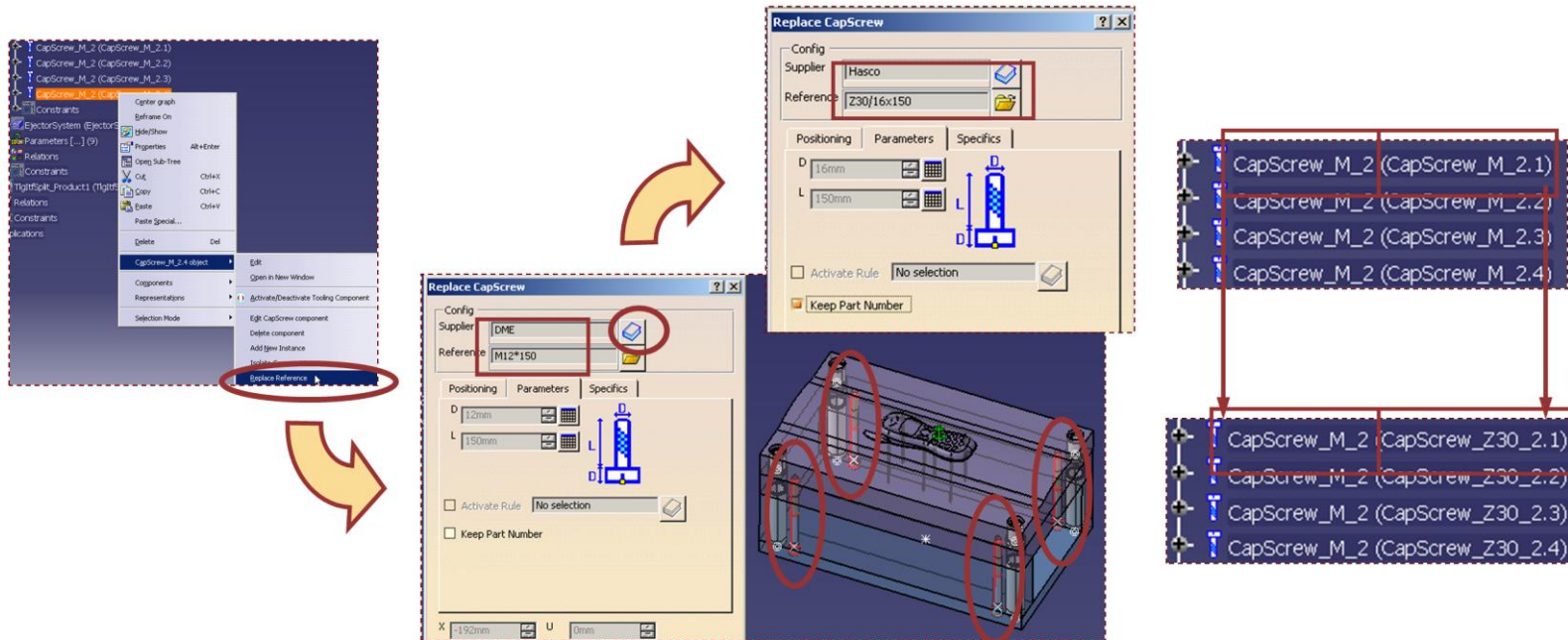
# Specific Components: Inserts & Sliders

*You will learn how to define an Insert and a Slider mechanism.*
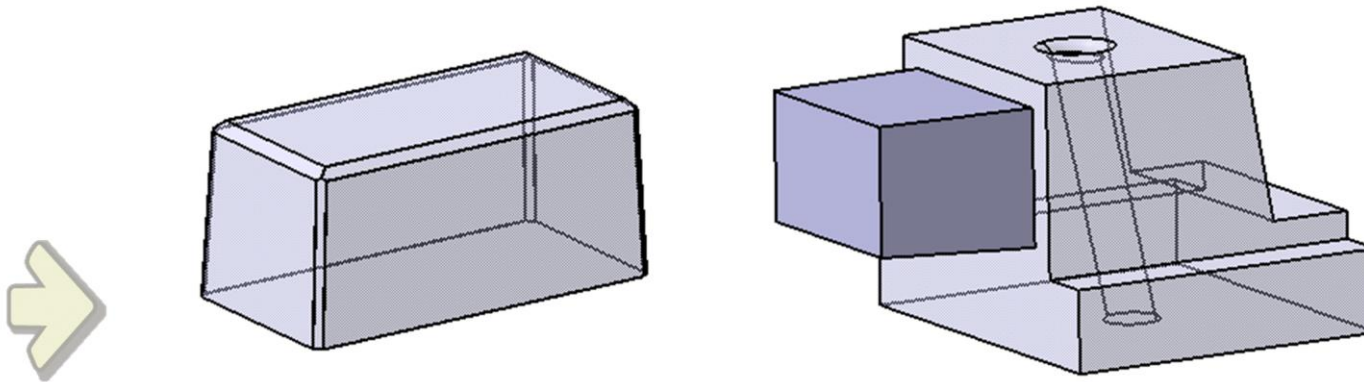
# Defining and Positioning an Insert (1/3)

- **Defining Inserts:**
  - **If necessary, you can create Insert blocks in your Mold or Die Structure, for example in the case of a multi-cavity Mold.**

- **Creating an Insert:**
  - **To create the Insert, click icon Add Insert**



Select one of the predefined types of Insert

# Defining and Positioning an Insert (2/3)

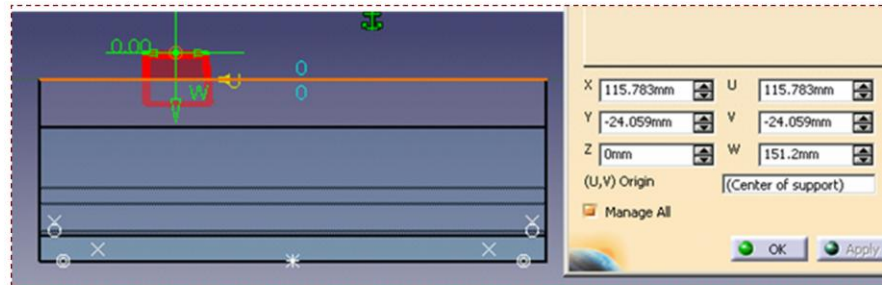- **Creating an Insert:**
    - **Define the position of the insert:**

- **You can pick any of the following elements in order to position the insert:**
    - **A planar face of the plate. The viewpoint is redefined if Sketcher option Position Sketch plane parallel to screen is active. You can then use the grid to define the position of the insert according to Sketcher option Snap to point. These options can be controlled in Tools/Options/Mechanical Design/Sketcher.**
    - **A 3D point**
    - **A 2D point (point in sketch, vertex)**
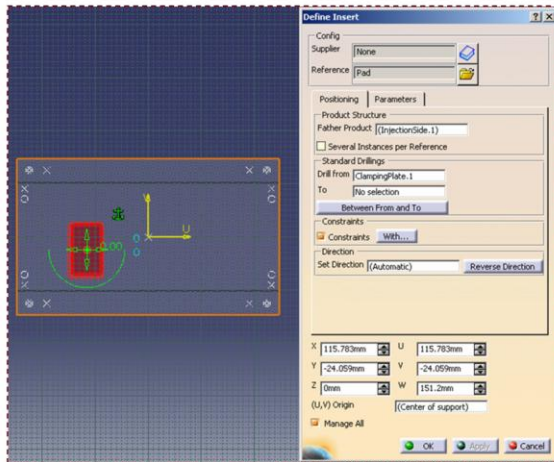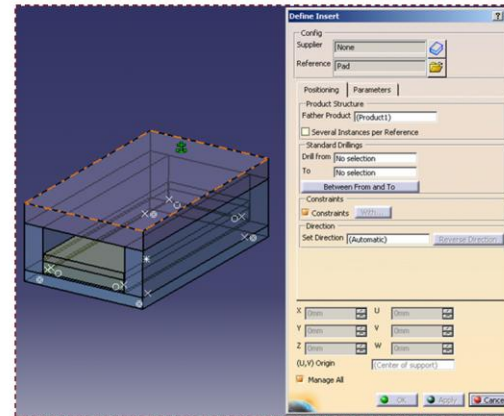


**Pick a position on this face to have a preview of the Insert and adjust its position by using Instance Origin values and/or the green arrows and half-circle attached to the preview of the Insert**
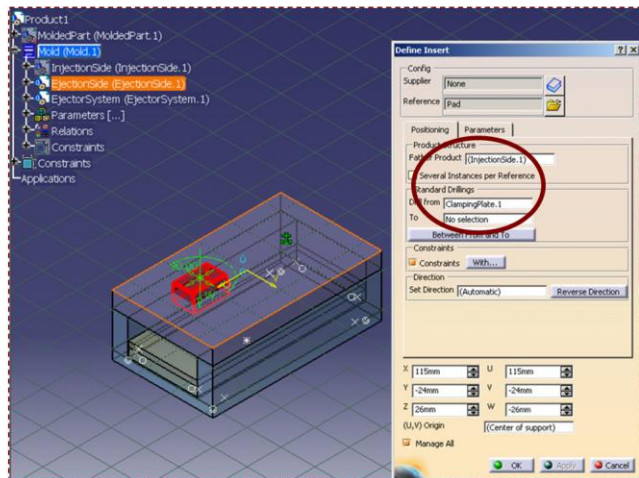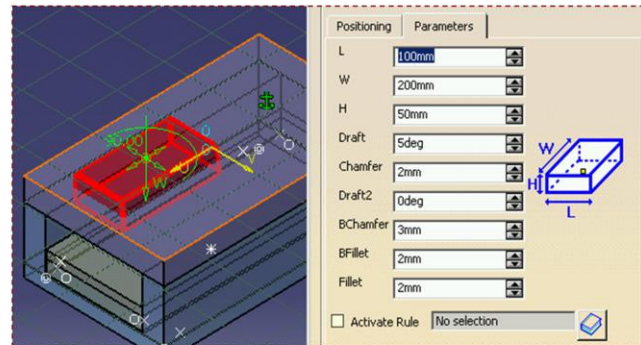
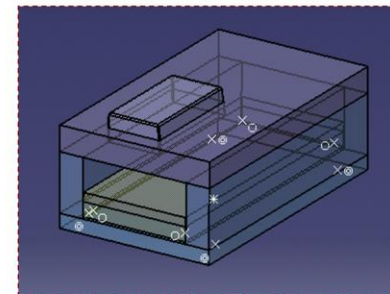# Defining and Positioning an Insert (3/3)

- **Creating an Insert:**
  - ◆ **Define the elements of the Die to be drilled by this insert**
  - ◆ **Adjust the dimension of the insert using tab Parameters**
  - ◆ **Split the Insert using function Split Component in contextual menu**

**Field Drill From is automatically filled with the name of the plate or component picked for positioning it.**
**Select field To and pick the relevant plate or component.**

**Adjust the relevant dimension values and validate creation**

# Defining and Positioning a Slider (1/5)

- The definition of a Slider mechanism can be done by creating the following components:
  - The Slider component itself.
  - The corresponding Retainers component.
  - The Screws components fixing the Retainers to the Mold or Die Structure.
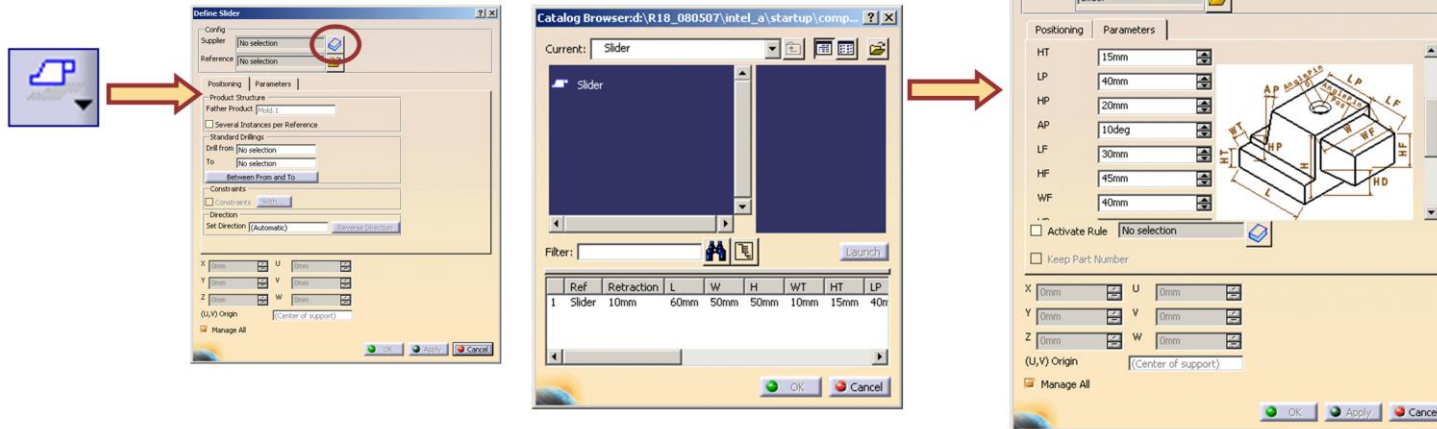  - The Angle Pin.
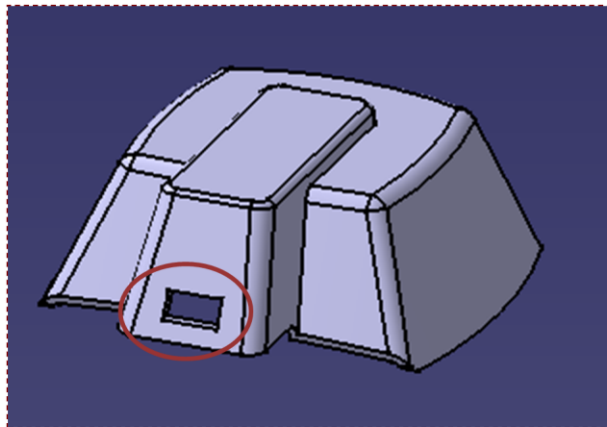


**For example, this Molded Part contains an undercut area requiring the creation of a Slider mechanism**
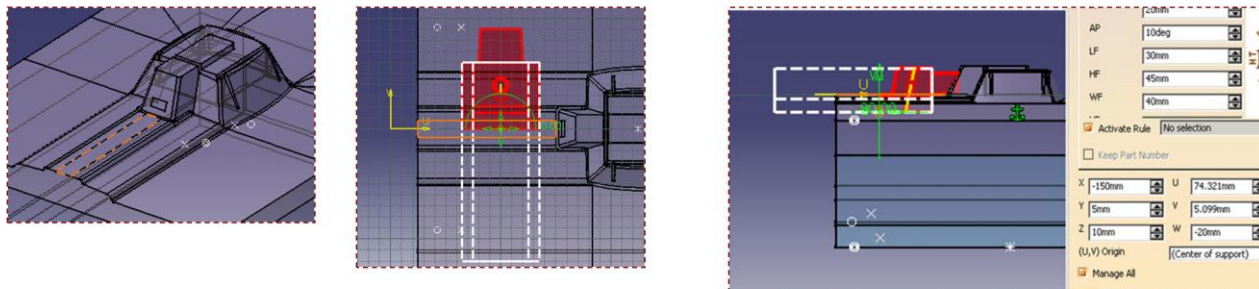
# Defining and Positioning a Slider (2/5)

## Creating a Slider:

◆ **To create a Slider, click icon Add Slider**
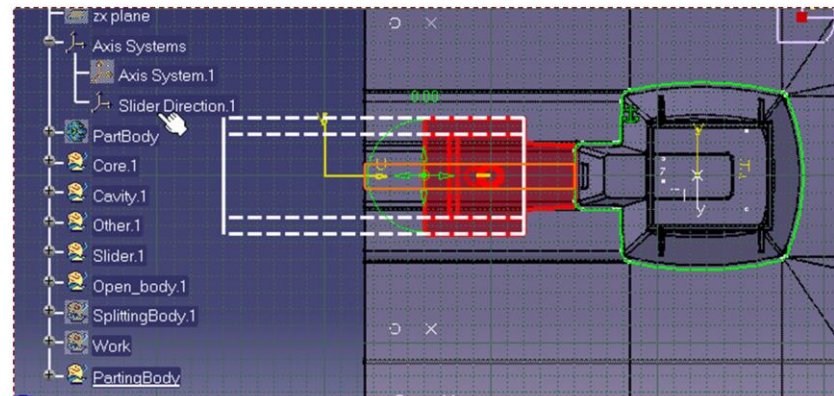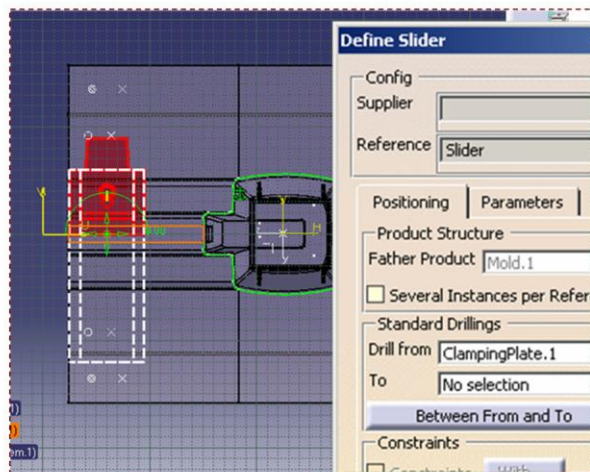


## Define the position of the Slider:



**You can use the same positioning capabilities as for the other types of components, for example by first picking a planar face on the top of your Die Structure Face, then adjusting the position with the green arrows.**

# Defining and Positioning a Slider (3/5)

- **Creating a Slider:**
  - **If the surface preparation of the Part has been done with application Core & Cavity design (CCV), you can align the slider by using the Slider Pulling direction defined in CCV.**
  - **To do so, you just need to select the corresponding Local Axis System in the specification tree or in the 3D viewer.**
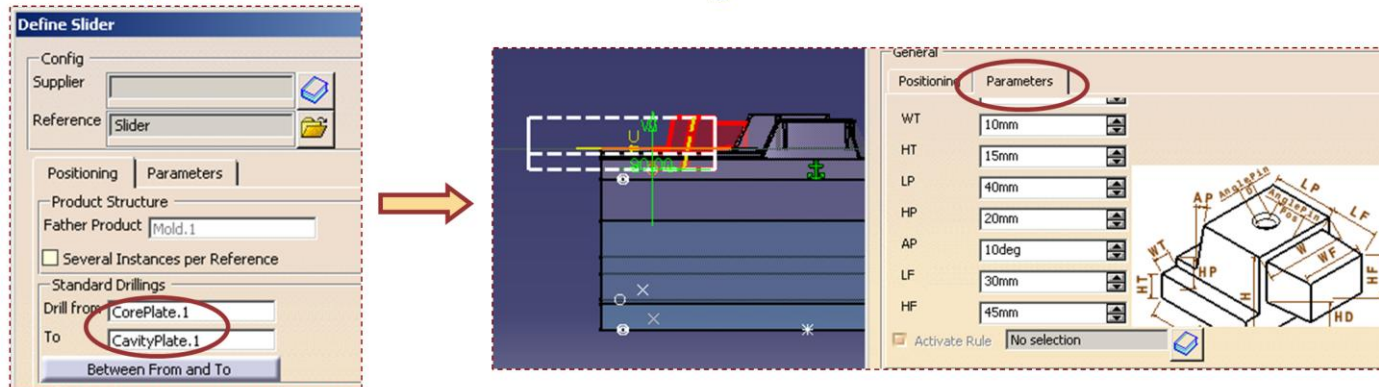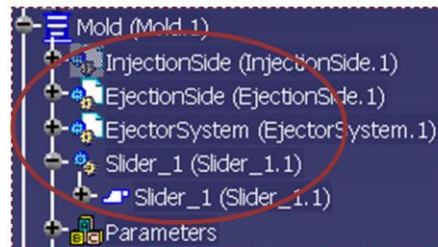
# Defining and Positioning a Slider (4/5)

- **Creating a Slider:**
  - ◆ **Define the elements of the Mold or Die to be drilled by the Slider (for ex: the Core Plate and the Cavity Plate).**
  - ◆ **Adjust the dimensions of the Slider using tab Parameters.**



**Note that the Slider has been created with a location point and a location axis for further reuse when defining the Angle Pin**
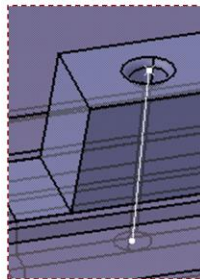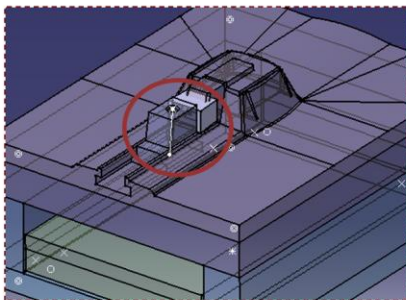


**Note that the Slider has been placed in the tree structure in a new assembly Component which is at the same level as the main sub-products of the Die.**

**This is necessary to ensure proper kinematics simulation.**

# Defining and Positioning a Slider (5/5)

**Creating a Slider:**

- **Split the Slider using function Split Component in contextual menu**

- **The splitting surface must have been created beforehand, for example in workbench Core and Cavity Design**

**Additional information:**

- **Slider parameter Retraction and the orientation of the Angle Pin are correlated. In order to keep a rounded-up value for this angle, the value of the retraction is automatically recomputed by the application to the closest appropriate value.**
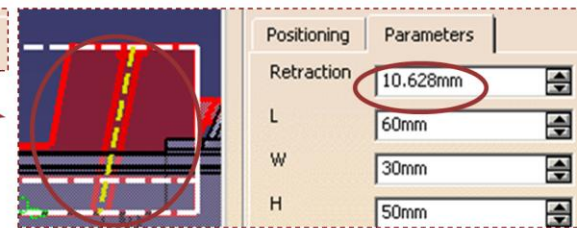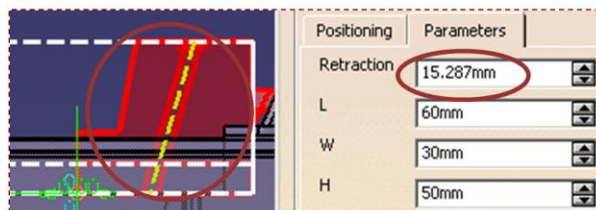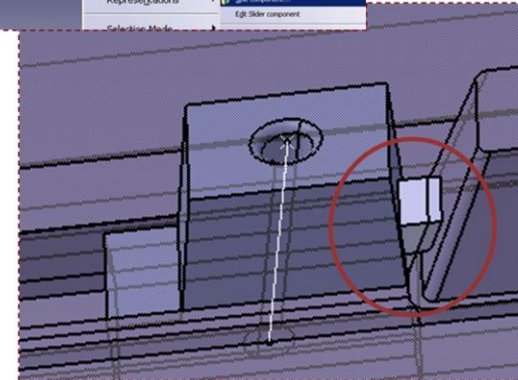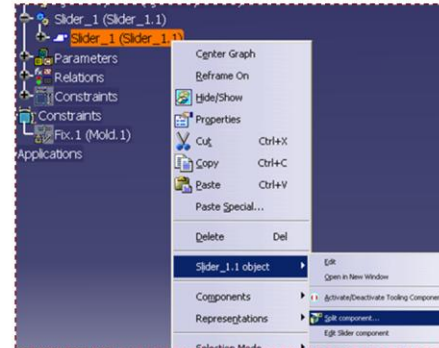
# Defining and Positioning Retainers

**Creating Retainers :**

- **To create the Retainers, use function Add Retainers**
- **Define the appropriate dimensions in tab Parameters**
- **Define the plates to drill (Drill From and To)**
- **Position the retainers using standard positioning functions**
- **Select the Slider in the 3D viewer to automatically fit the Retainers onto the Slider**
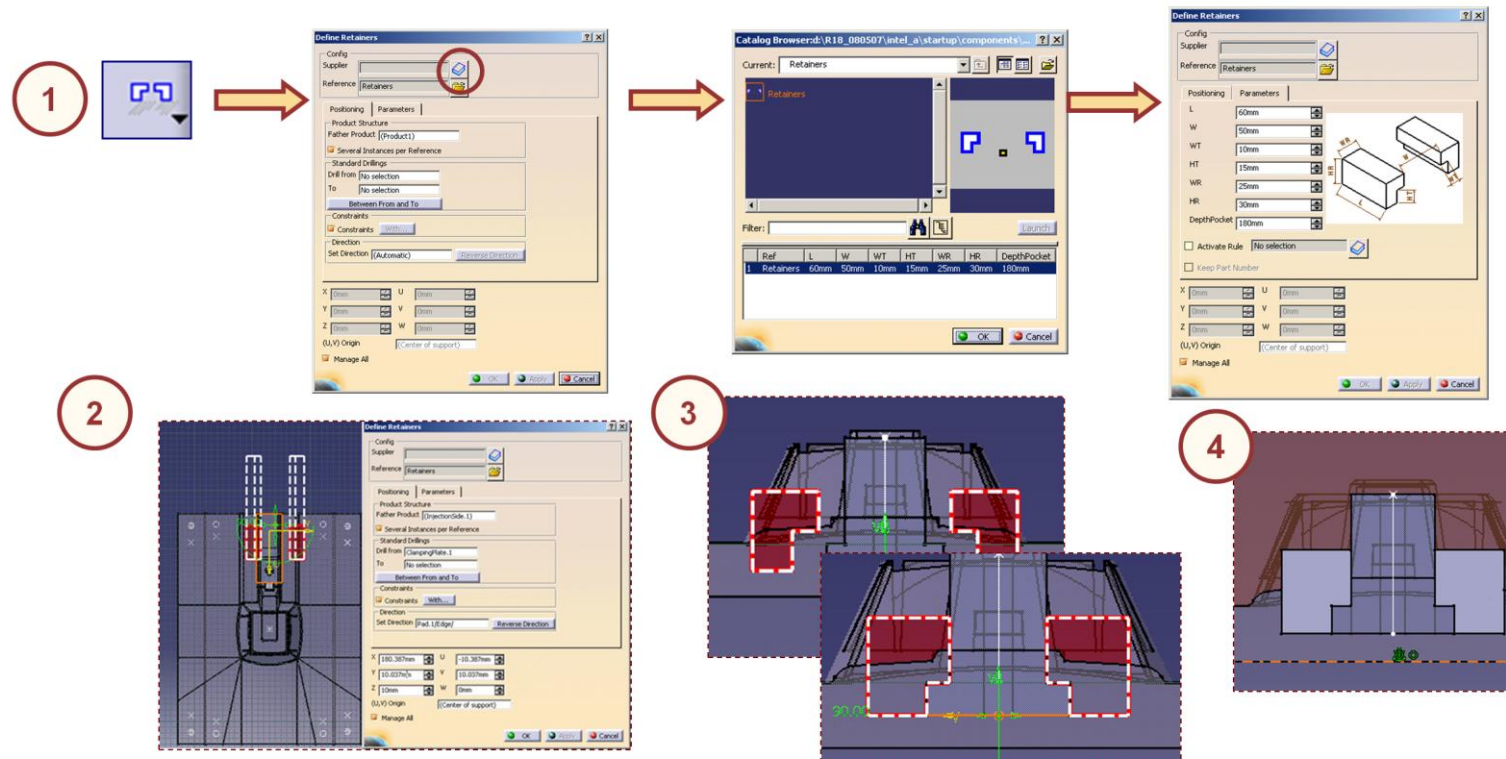
# Defining and Positioning the Angle Pin (1/2)

- **Creating the Angle Pin:**
  - **To create the Angle Pin, use function Add AnglePin**
  - **Select the Angle Pin in catalog**
  - **Position the Angle Pin by picking the location axis and point defined in the Slider**
  - **Adjust the Z position of the Angle Pin**
  - **Define the plates to drill (Drill From and To)**
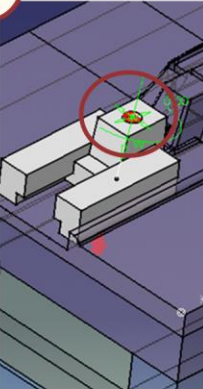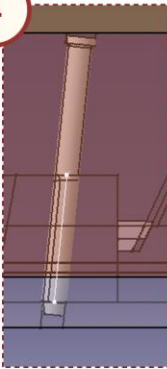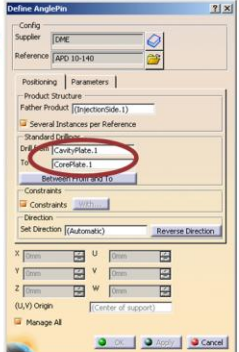
# Defining and Positioning the Angle Pin (2/2)

- **Current limitation:**
  - ◆ **The drilling of the Angle Pin is not correct: it should not simply fit the cylindrical shape of the Angle Pin, because the element will be damaged during the ejection movement.**
  - ◆ **You have to replace this drilling with a larger Hole sufficient to allow for the movement of the Angle Pin (use workbench Part Design).**
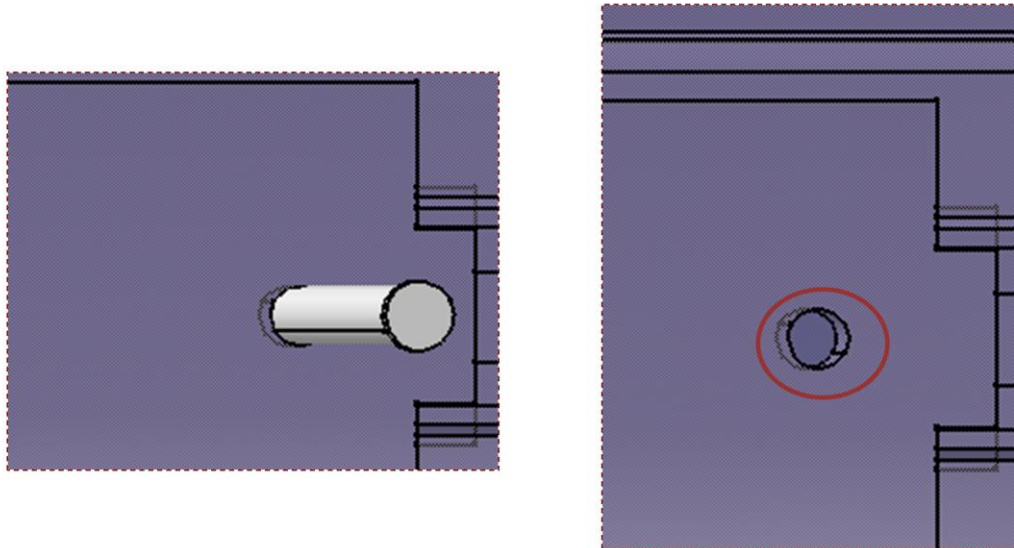
# Instantiating Screws in the Retainers

- **Inserting the Screws :**
  - **To create the Screws, use function Add CapScrew**
  - **Select the Screw in catalog**
  - **Position the screws on the top face of the Retainers**
  - **Adjust the Z position of the Screws**
  - **Define the elements to drill (Drill From and To)**
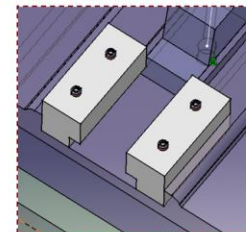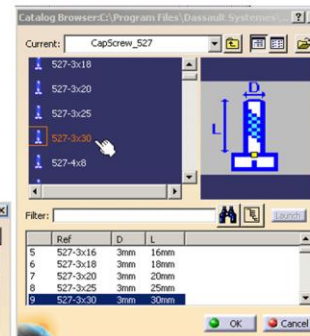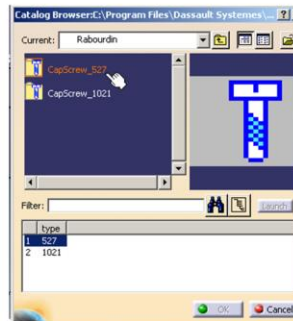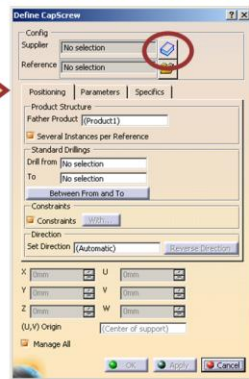


**You can use standard drilling mechanism (Drill From To) to drill from a component (Retainers) to a plate (Core Plate).**

**But it is also possible to insert the screws without defining From/To and use later function Tools/Drill component.**
**In this case, before inserting the screws, you must define the sub-product (e.g. EjectionSide) where the Screw components will be located by making it active.**
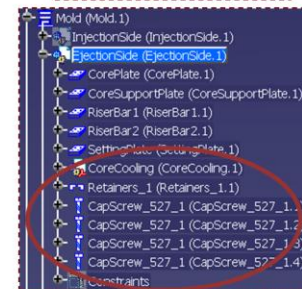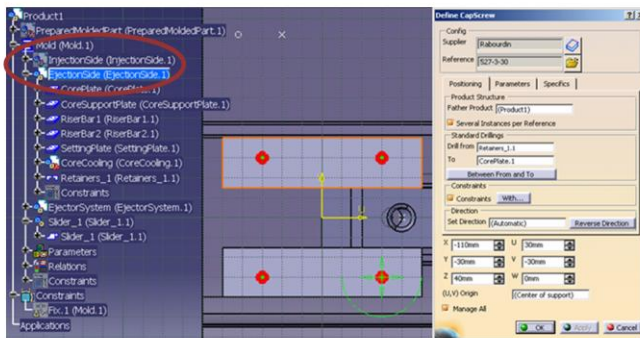
# Advanced Add/Remove Capabilities

*You will learn how to perform complex drillings.*
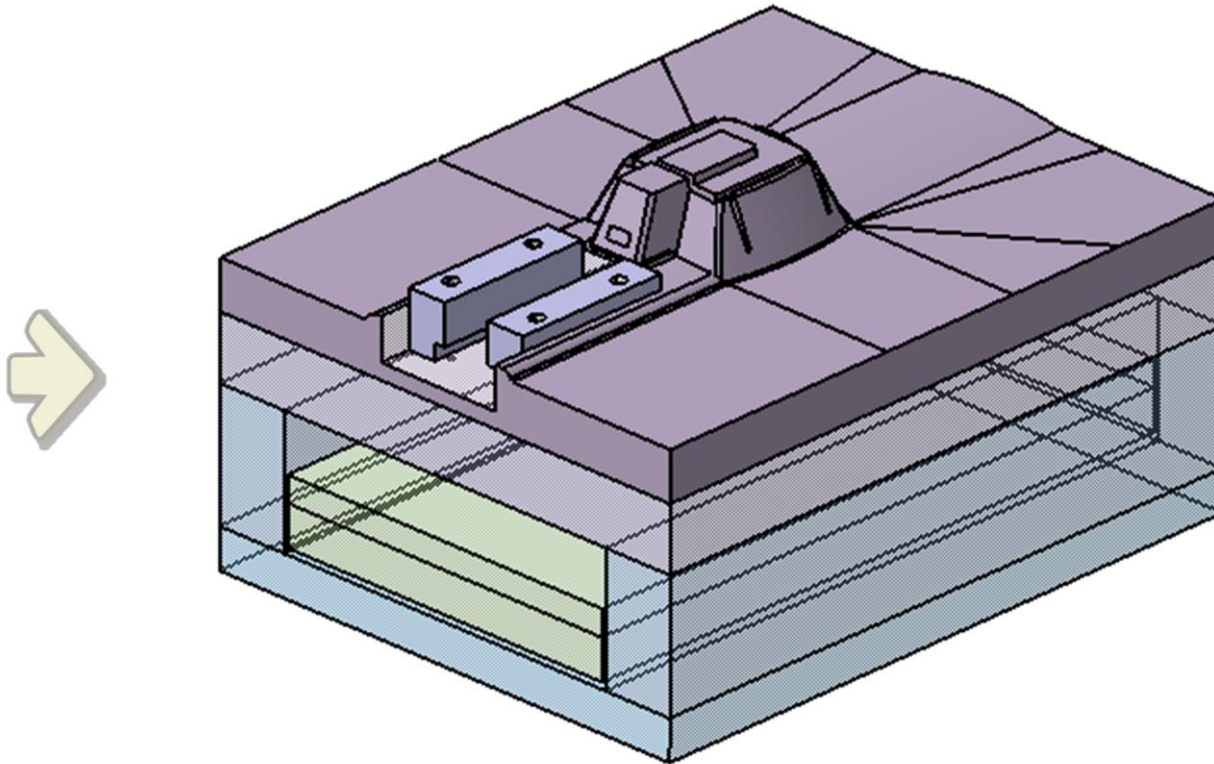
# Assembly Drillings (1/3)

- In some situations, the usual function Drill From / To available in the definition dialog box of each Component, is too simple to provide a satisfactory result.

- A frequent situation is the case of assembly User Components: each part of the assembly can impact the elements of the Mold or Die in a different way, i.e different elements, possibly in different directions. In such a case, the notion of From/To is not meaningful.

- To perform such drillings, use function Tools / Drill Component or click the corresponding icon.

- This function allows to apply drillings defined in the DrillHole or the TapHole of a list of impacting Components to a list of impacted elements.

- It is recommended to proceed in the following way:

  - First, instantiate the Component in the proper Mold/Die sub-assembly without Drill From/To information

  - Then use function Drill Component as many times as as there are groups of sub-components whose drillings affect the same elements. The function could be used once to define in a single operation all impacting sub-components and all impacted mold/die elements, but this would lead to creating unnecessary drillings.
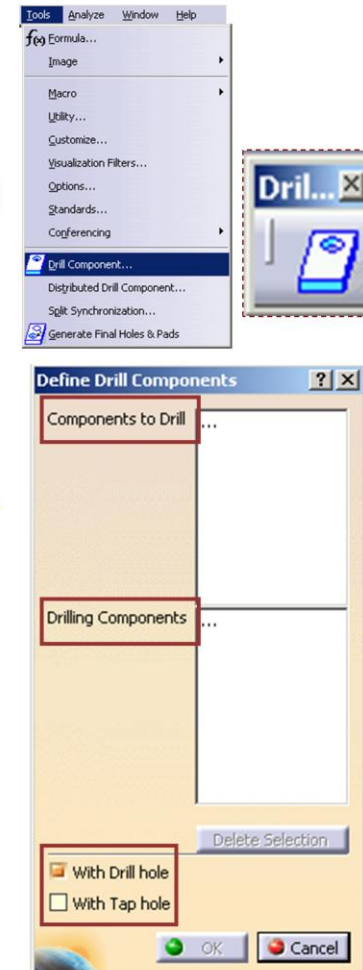
# Assembly Drillings (2/3)
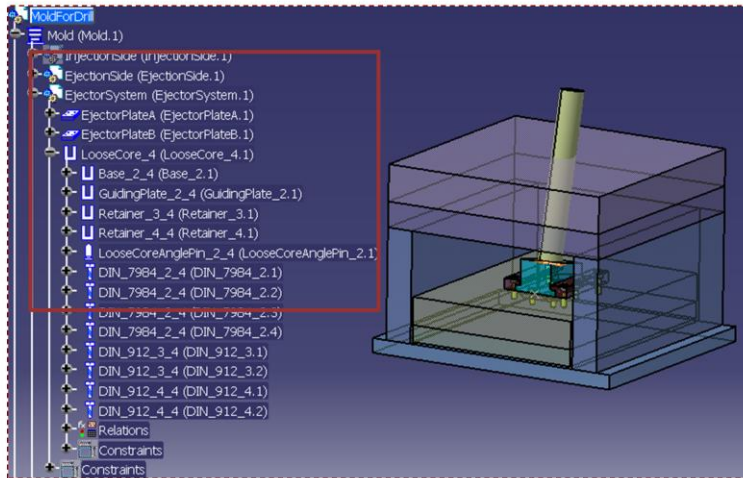


**The Loose Core user Component is an Assembly consistd of several different Parts with different drilling impacts in the Mold Assembly.**
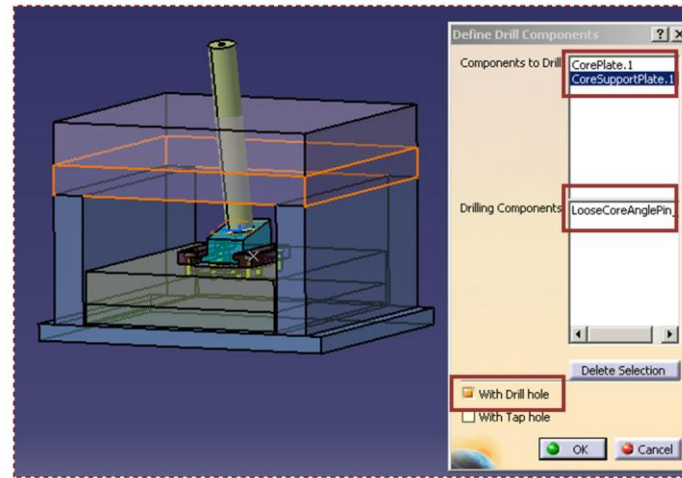
**CorePlate & CoreSupportPlate have similar impact: DrillHole from LooseCoreAnglePin**

# Assembly Drillings (3/3)



**EjectorPlateA is impacted by DrillHole of GuidingPlate**
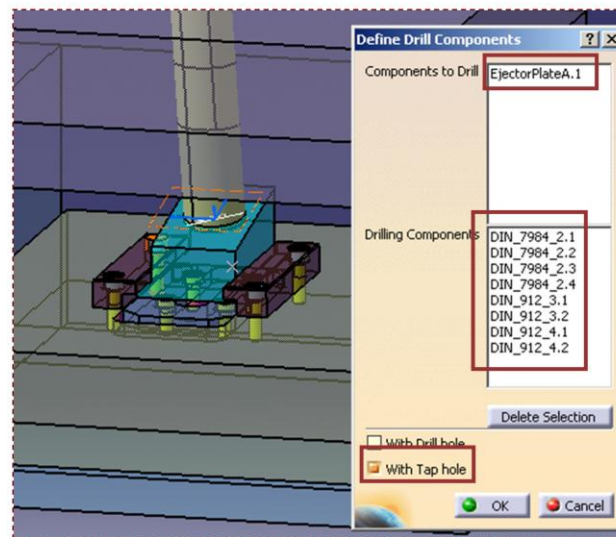


**EjectorPlateA is impacted by TapHole of all CapScrews**

# Extended Definitions (1/7)

Tooling components embed the definition of the machining process together with their own definition. The definition of the machining process is described in terms of material to add or to remove, under the form of Part Bodies inside the component CATPart descriptions. These dedicated Part Bodies include the geometric description of the material to add or remove.
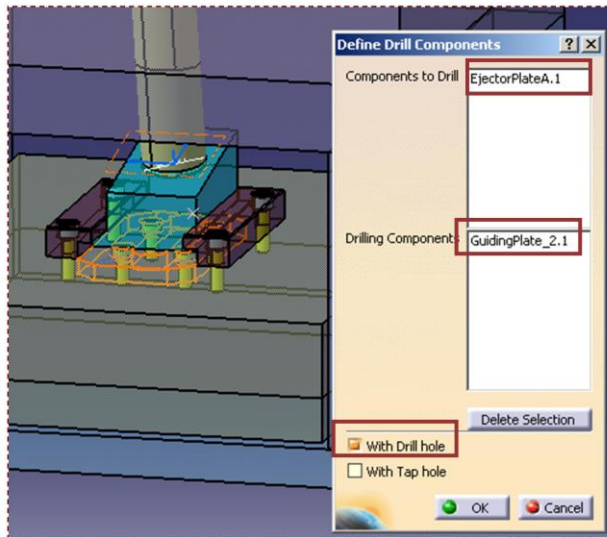
- For some users, the intent of addition or removal of material is:

  - Description of the final machined plate or casting,

  - or description of the rough plate or casting.

- It can also be used to describe plates/castings, as they will be purchased, as opposed to plates/casting as they will be machined. Within a given company:

  - The process for definition and use of add/removes is very structured and includes a lot of company know-how,

  - And it is a repetitive process.

- For these reasons, application TG1 provides extended capabilities in terms of:

  - Naming of add/remove bodies in Tooling Components

  - Structure of the receiving plates/castings (not only the PartBody)

  - Ability to predefine choices and definitions in Settings to minimize the number of user interactions (process productivity) and the risk of errors (process reliability)

- These extended capabilities are provided as:

  - Dedicated Settings capabilities (see before description of Mold Tooling Design tab Add/Remove).

  - Function Tools / Distributed Drill Component.15

# Extended Definitions (2/7)

**Example 1: user-defined add/remove Bodies (Component) and receiving Bodies (Mold/Die Assembly):**
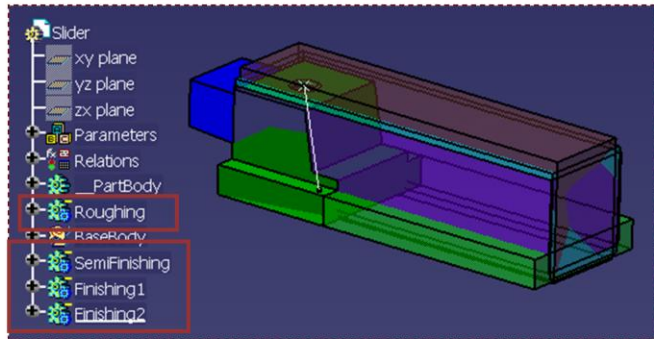


**Slider Component structure:**
- **Body Roughing**
- **Body SemiFinishing**
- **Body Finishing1**
- **Body Finishing2**
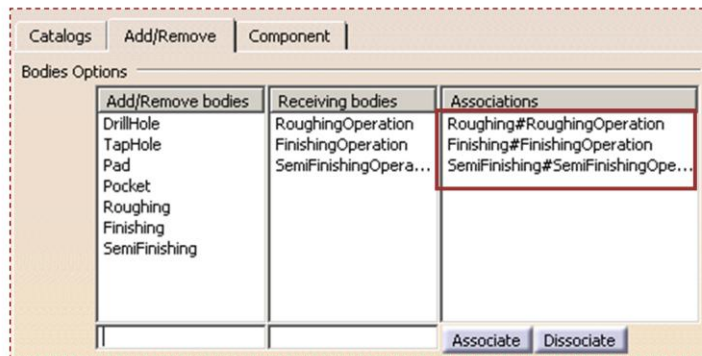
**Mold CorePlate receiving structure:**
- **PartBody**
- **Body RoughingOperation**
- **Body SemiFinishingOperation**
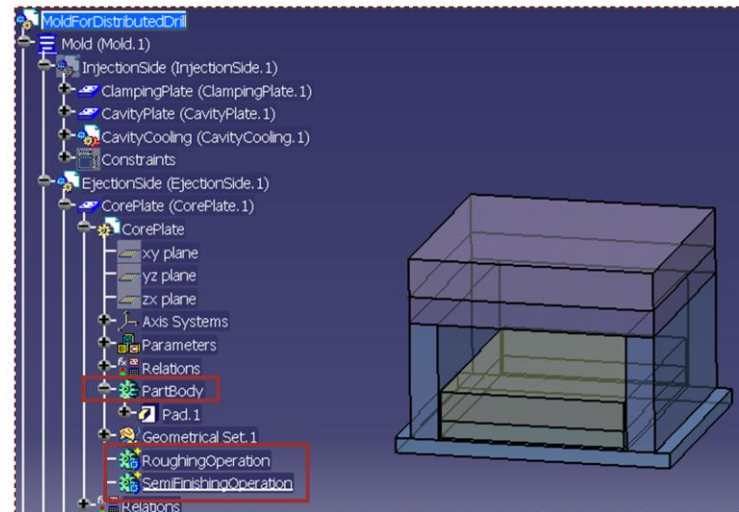


**Settings: Predefined associations**

## Extended Definitions (3/7)

◆ **Slider Component must been inserted in the Mold Assembly without Drill From/To information**

◆ **Drillings are defined using function Distributed Drill Component.**



**When selecting Slider Component, the list of add/remove Bodies present in the component is displayed**

**You must first select the appropriate "empty" line in column Components to Drill, then select the required Component**

# Extended Definitions (4/7)
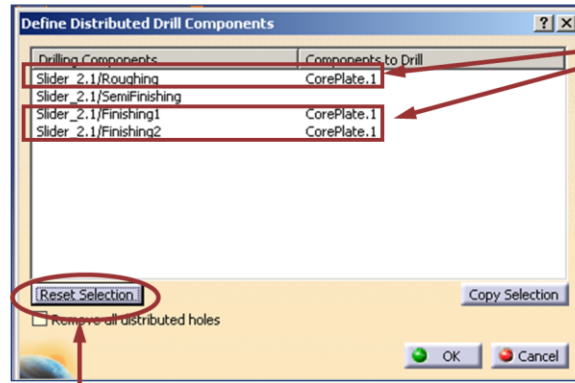
**You have to pick a receiving element (here CorePlate) for required add/remove bodies**

**You can use button Reset Selection to erase the selected receiving element**
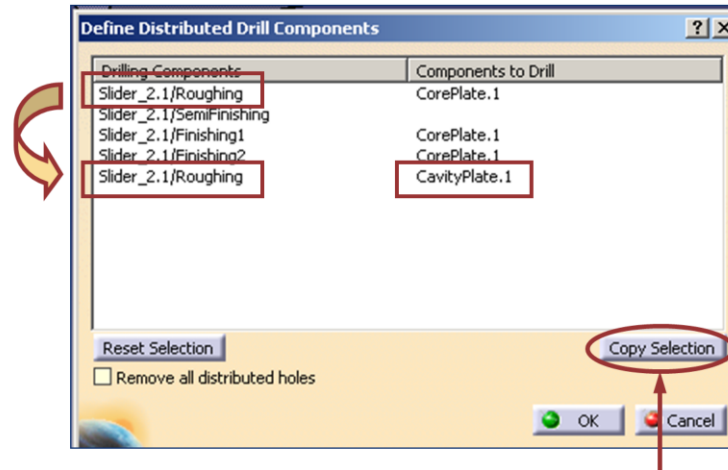
**You can use button Copy Selection to replicate an add/remove Body and affect it to another Component to Drill (here, Roughing is also affected to CavityPlate)**
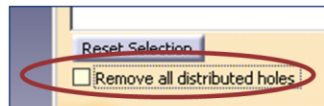
**option Remove all distributed holes allows to erase existing distributed drillings and add new ones if required.**

# Extended Definitions (5/7)

◆ **After validating the Boolean operations are performed on the target Bodies of the elements previously selected in list Components to drill: here CorePlate and CavityPlate**
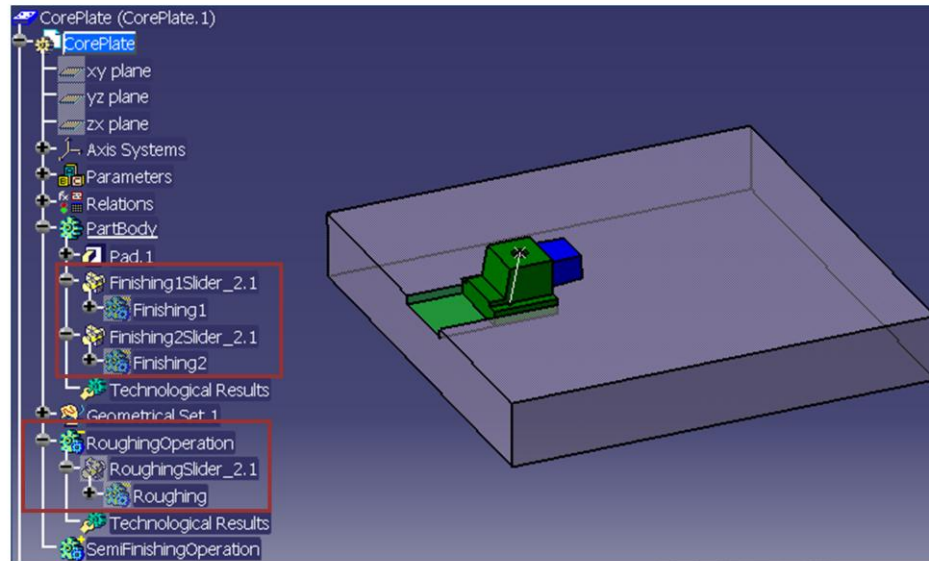


**CorePlate:**
- **Bodies Finishing\* have been applied to the receiving PartBody, because no Body FinishingOperation (defined in Settings) is present in the plate.**

- **Body Roughing has been applied to receiving Body RoughingOperation**

**CavityPlate:**
**Body Roughing has been applied to receiving PartBody, because no Body Roughing Operation (defined in Settings) is present in the plate**

# Extended Definitions (6/7)

- **Example 2: Using pre-selections:**
  - **LooseCore Component must been inserted in the Mold Assembly without Drill From/To information**
  - **Drillings are defined using function Distributed Drill Component.**

Only one TapHole selection is requested from the user, using usage of pre-selection code "-PS1" for the TapHole of all eight CapScrews of the LooseCore assembly
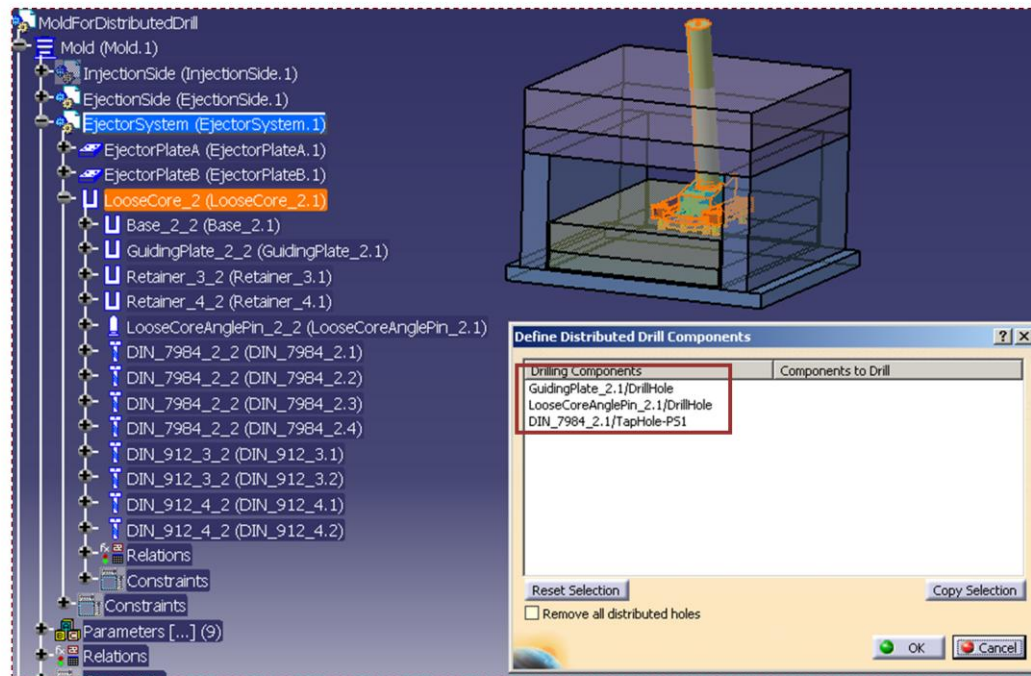
# Extended Definitions (7/7)
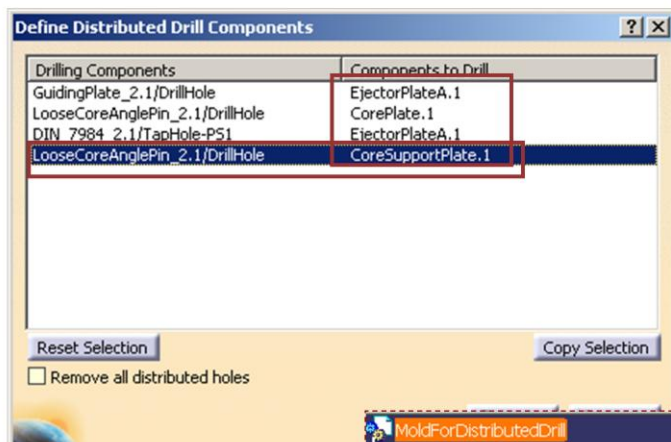
**To define all drillings, only the following user interactions are needed:**

- **Select EjectorPlateA as target for GuidingPlate/DrillHole**
- **Select CorePlate as target for AnglePin/DrillHole**
- **Replicate the previous drilling for CoreSupportPlate**
- **Select EjectorPlateA as target for all CapScrews/TapHole in one shot**

**Result drillings**
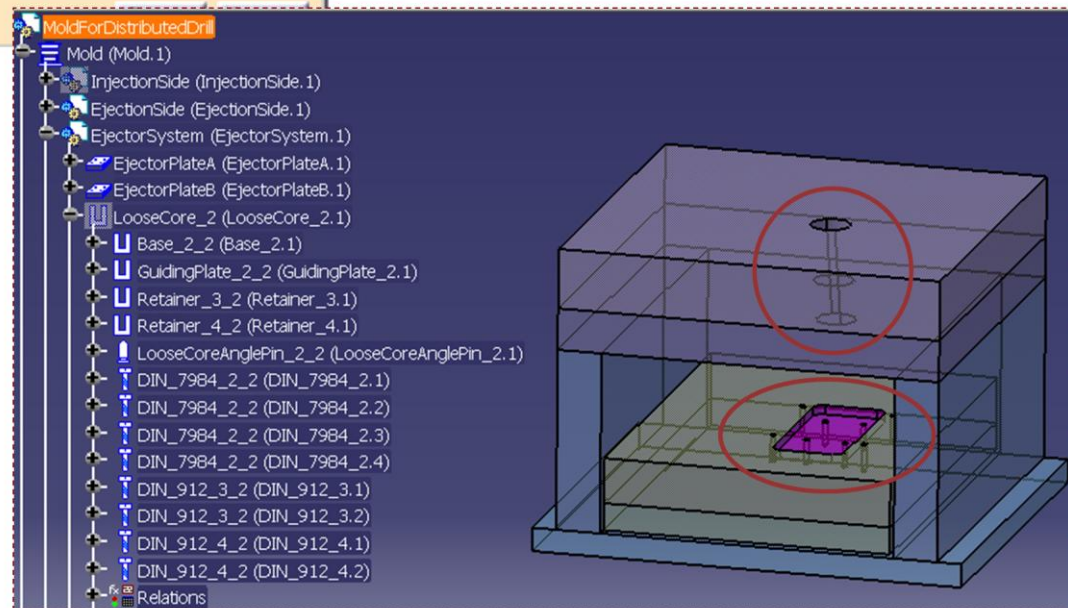
# Managing User Components

*You will see how any User Component is structured, and how to create and instantiate user-defined components.*
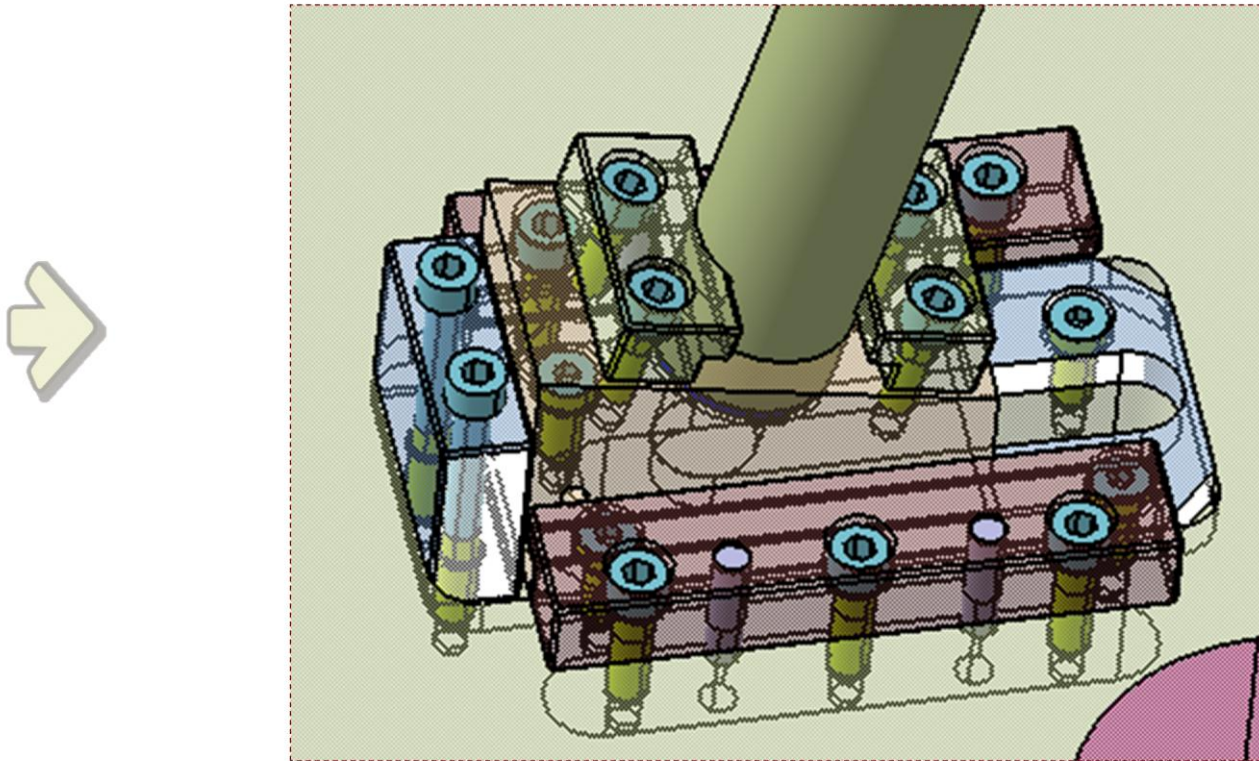
# What is a User Component?

- In addition to the predefined types of components (Leader Pin, CapScrew, Bushing) available, it is possible to create your own types of components (special plates, stamps)

- You can also create your own designs for predefined types of components (e.g. a custom Leader Pin) to extend the libraries offered in the application

- The structure of a user-defined component must follow some rules to ensure it can be managed in application Tooling Design in the same way as the standard components offered in the application

- You can put your user components in your own catalogs and link your catalogs to those known by the application
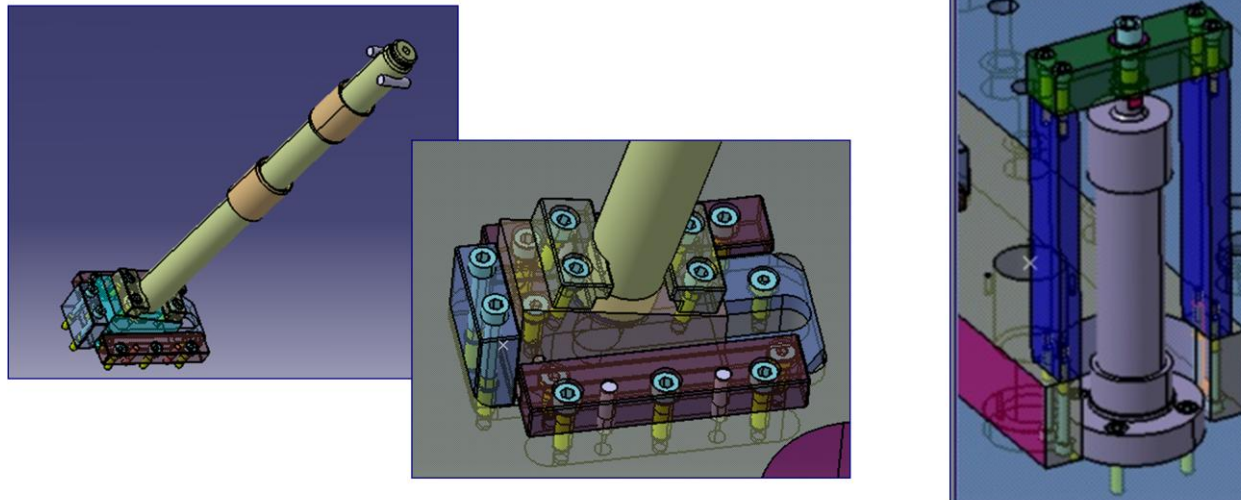
# Structure of a User Component (1/2)

- **A Tooling Design User Component follows the same basic structure rules as Standard Components coming from provided catalogs or template Parts (e.g. a Cap Screw).**

- **It is usually defined as a CATPart file, including potentially a list of variants (defined by a Design Table). It can be put in a dedicated catalog, but the file can also be selected directly in TG1 functions.**

**Color of DrillHole or TapHole is applied to affected faces**
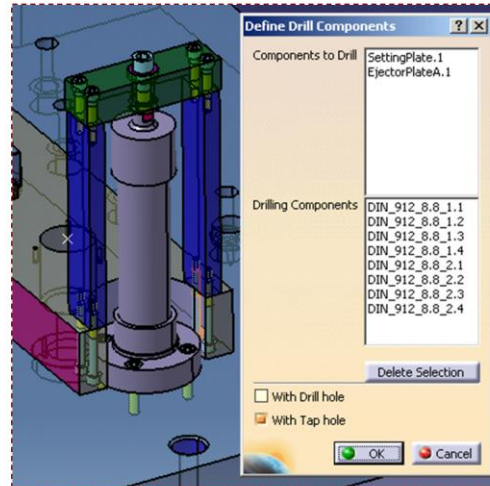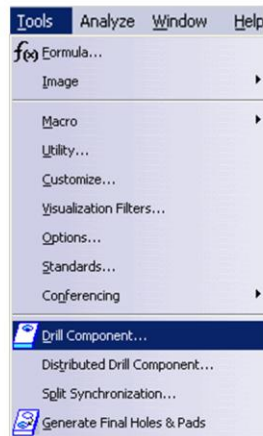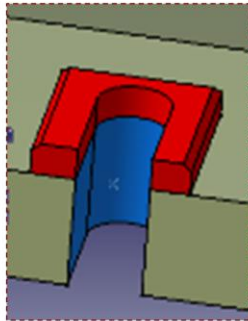
# Structure of a User Component (2/2)

- It is possible to define one or several additional bodies to remove material when inserting the component. Such a body must be named Pocket, e.g. Pocket1, Pocket2, etc.

- Body Pocket can contain any negative-polarity feature (Pocket, Hole). When inserting the component, this body will be used to perform Remove operations only in the plate specified as Drill From.

- It is possible to define one or several additional bodies to add material when inserting the component (e.g. to create a boss). Such a body must be named Pad, e.g. Pad1, Pad2, etc.

- Body Pad can contain any positive-polarity feature (Pad, Shaft). When inserting the component, this body will be used to perform Add operations only in the plate specified as Drill From.

- When inserting the component, the Add and Remove operations generated by all those bodies (Pad, Pocket, DrillHole, TapHole) are created in the order in which the bodies are defined in the component.

- A user component can be a CATProduct made up of several CATParts following the rules described above.

# Creating a Catalog of User Components (1/3)

- Once you have created CATParts or CATProducts containing your user designs following the structure described previously, you can create a new catalog to insert them.

**1** Use workbench Catalog Editor to create a new catalog

| Start | File | Edit | View | Insert | Tools | Analyze | Window | Help |

- Infrastructure ▶ — Product Structure
- Mechanical Design ▶ — Material Library
- Shape ▶ — CATIA V4,V3,V2
- Analysis & Simulation ▶ — Catalog Editor
- AEC Plant ▶ — Product Data Filtering
- Machining ▶ — Feature Dictionary Editor
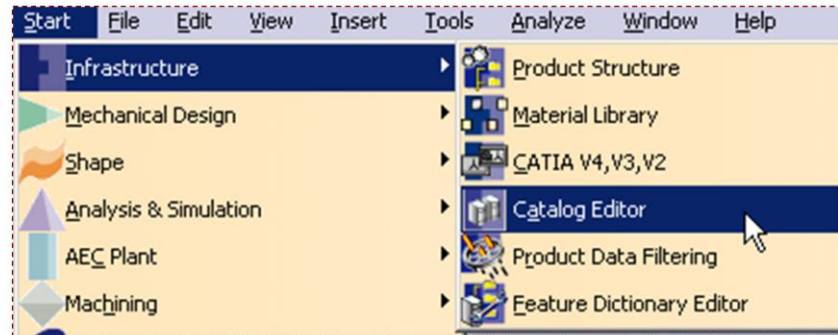
**2** Rename the default chapter to fit the kind of user components it will contain.

You can add other chapters if you want to add other kinds of user components.

catalog10.catalog
Chapter.1

- Cut        Ctrl+X
- Copy       Ctrl+C
- Paste      Ctrl+V
- Paste Special...
- Delete     Del
- Chapter.1 object ▶ — Definition...
- Keywords default values...
- Reorder sub-chapters...
- Reorder keywords...
- Reorder components...
- Copy Keywords...

**Chapter Definition**
Name: MySliders
OK    Cancel
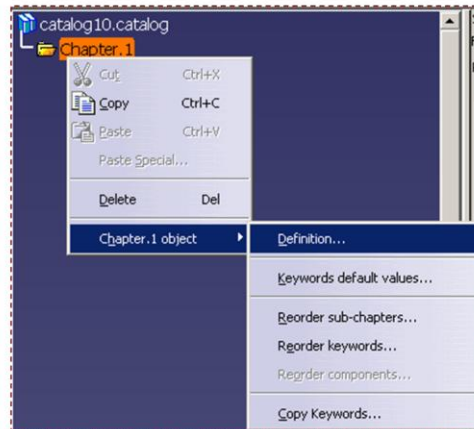
# Creating a Catalog of User Components (2/3)

**Then you can reference your components in the new catalog:**

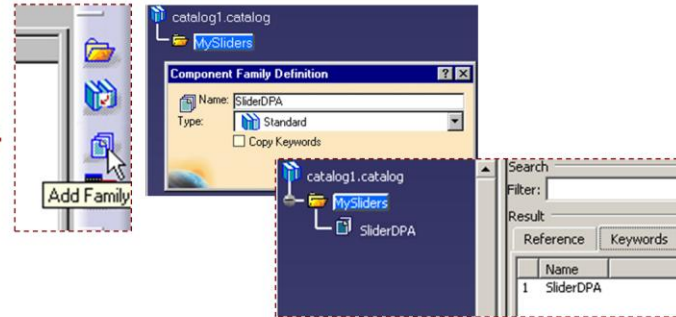**3** Use function **Add Family** to define in the chapter a first family, i.e. a first set of this kind of user components.

You can create other families in the chapter if you want.

If your user-component Part contains a Design Table corresponding to variants of your design, you should use instead function **Add Part Family**.
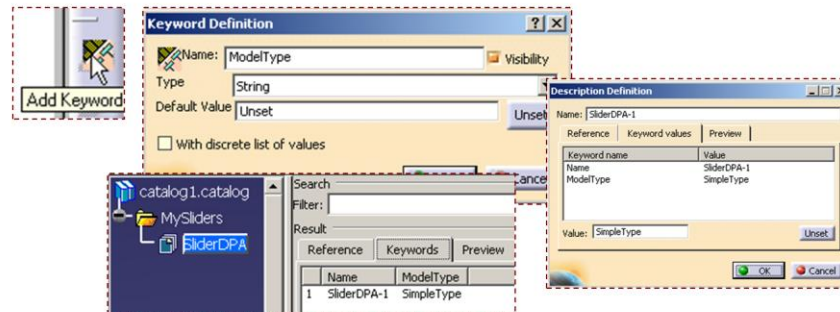
**4** Use function **Add Component** to define the physical location of the CATPart or CATProduct containing your design.

You can create other components in the family if you want.
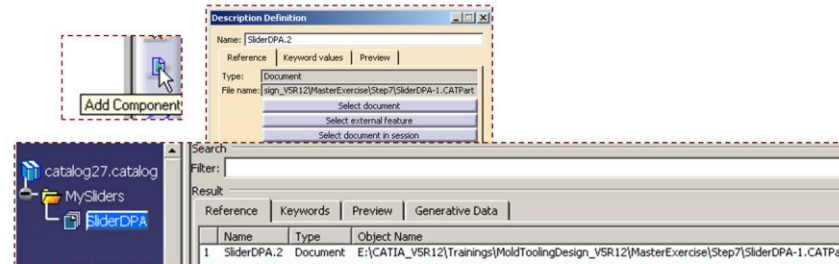
**5** Use function **Add Keyword** to define the relevant parameters for your component. Define the value for these parameters.
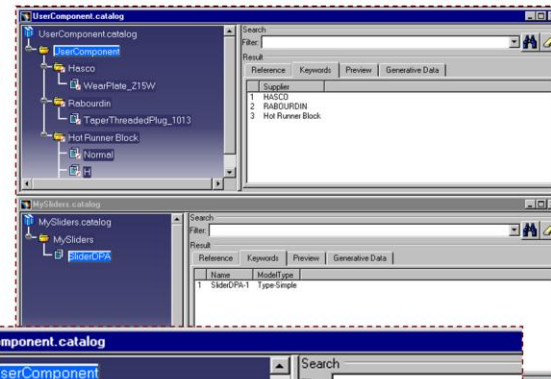
**6** Then save the new catalog file, using the name you want

# Creating a Catalog of User Components (3/3)

- Then you must link you own catalog to the existing User Catalog provided with the application:

**(7)** Open existing master User Component catalog: \intel_a\startup\components\MoldCatalog\UserComponent.catalog

**(8)** Use function Add link to other catalog to make a reference from the master catalog to the chapter of the new user catalog

Add link to other catalog
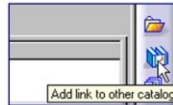
**(9)** Define now a value for Keyword Supplier for the newly added chapter
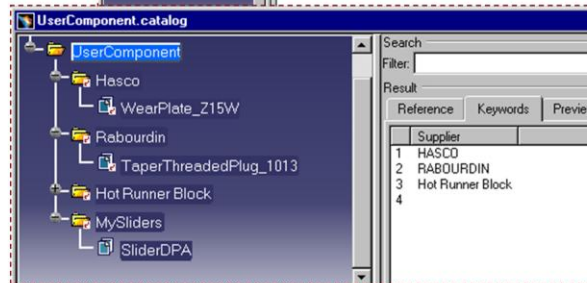
**(10)** Save the new version of UserComponent.catalog

# Instantiating a User Component

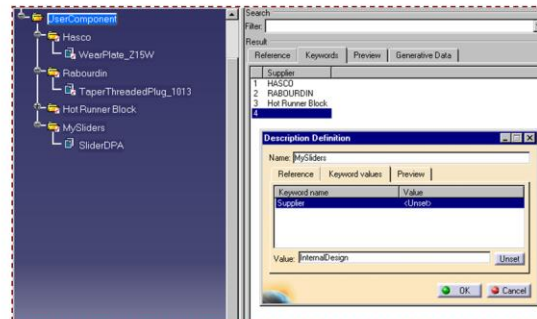- **To instantiate a User Component in a Mold or Die, use dedicated icon Add User Component: you have access to your own components in the same way as for predefined components (preview, positioning capabilities)**



- **You also have access to some specific components delivered with application TG1:**
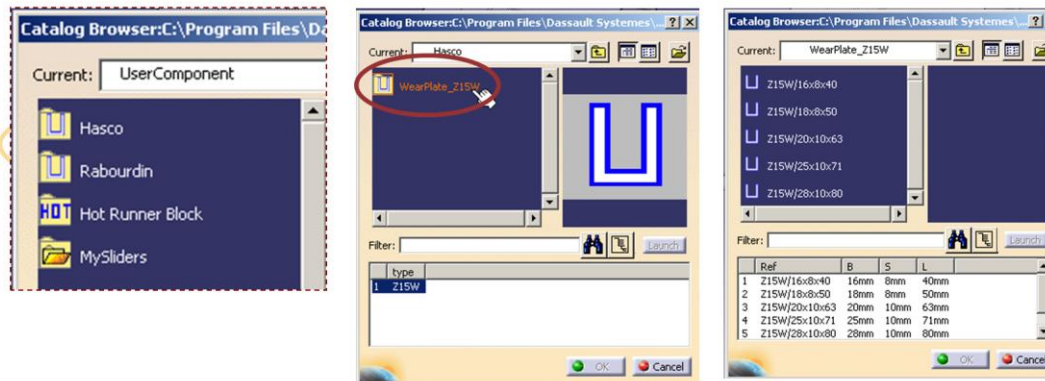
## Additional Information

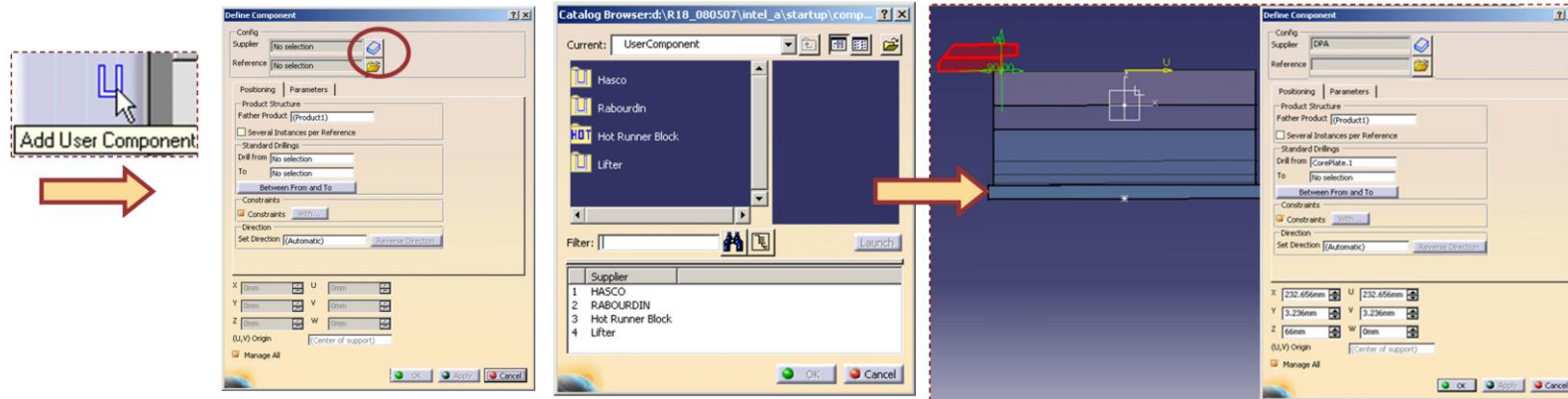- When a User Component is of a predefined type known by the application, you can link your catalog to the one of this type, instead of linking it to UserComponent.catalog:

- For example, if you have created your own type of Spring, you can link your catalog to Spring.catalog, which is located in the same folder as UserComponent.catalog

- It is not mandatory to put a User Component in a catalog. When you want to instantiate it, you can pick directly the CATPart file.
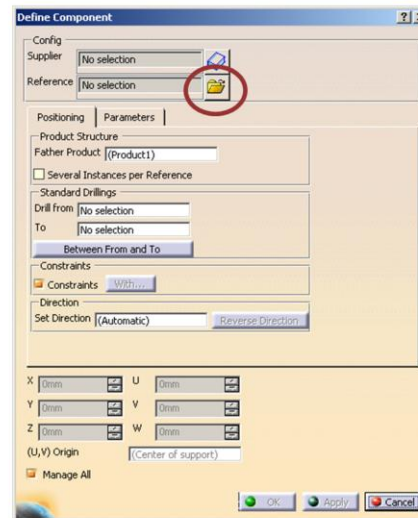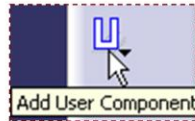
# Mold Design to Manufacturing

*You will become familiar with the tool used for manufacturing.*
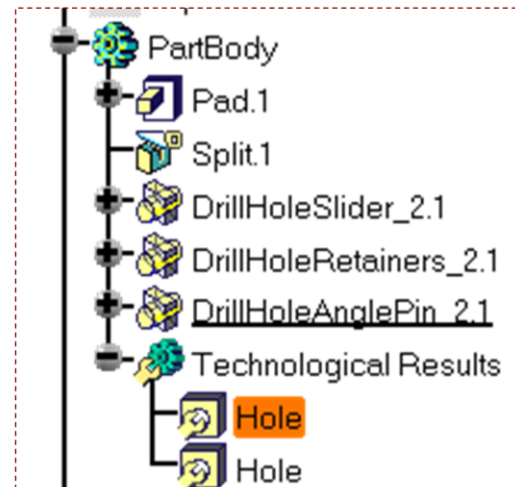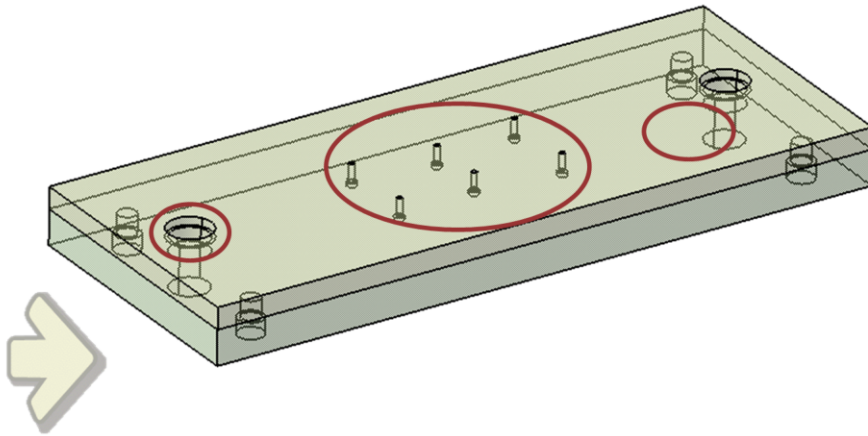
# Tooling Component Drilling

*You will become familiar with the tool used for manufacturing*
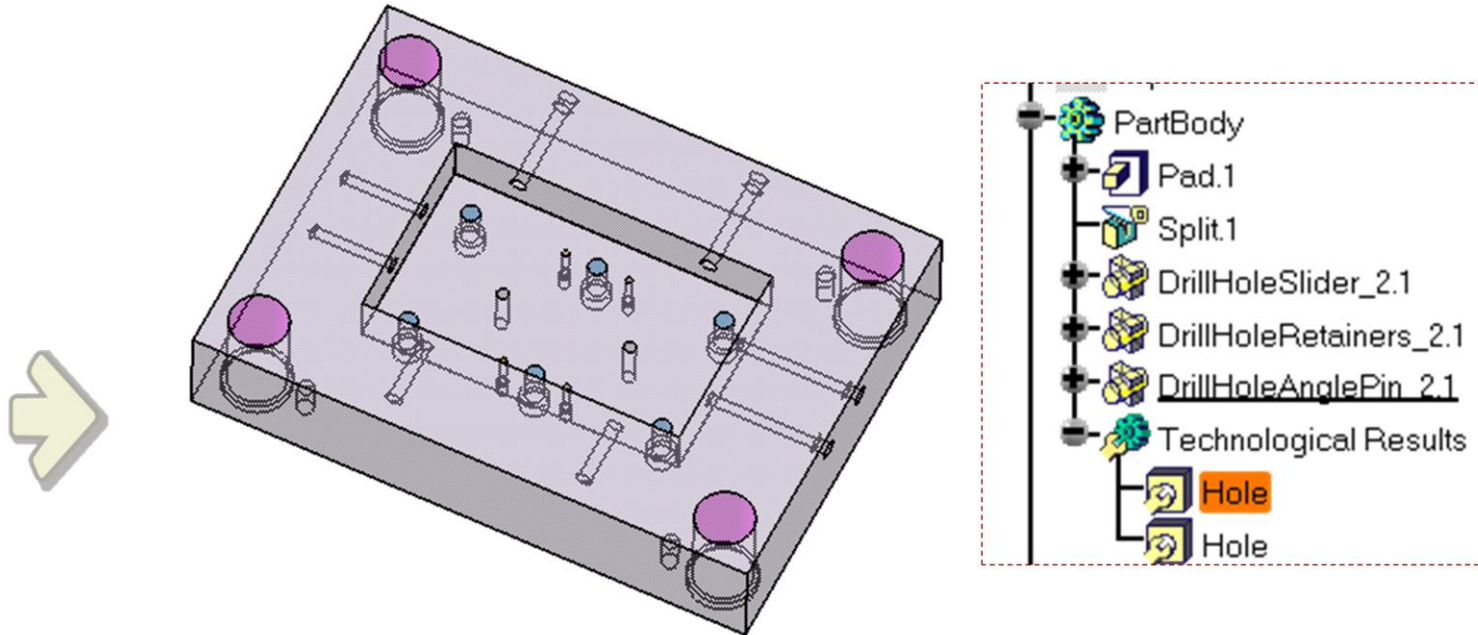
# Technological Results (1/5)

- When Tooling Components are inserted, the technological information of the corresponding drillings (Hole features) is automatically conveyed by TG1 application from the Components into the receiving elements, using CATIA Infrastructure object called Technological Result

- This allows for native propagation of this information for drawing or manufacturing purposes.

- See example:
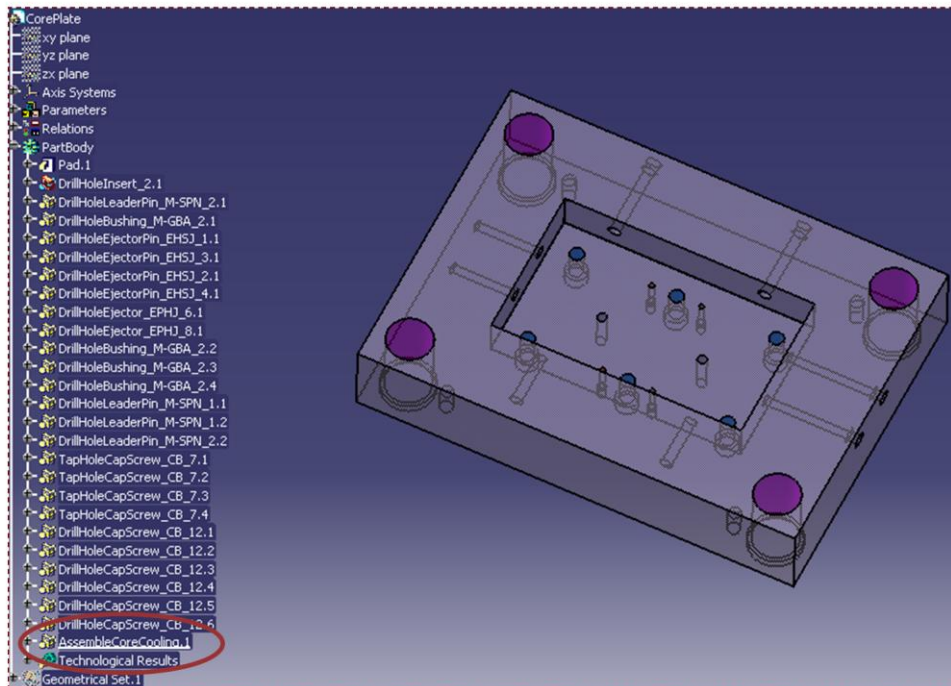


**Mold CorePlate where various types of components have been inserted: LeaderPins, CapScrews.**
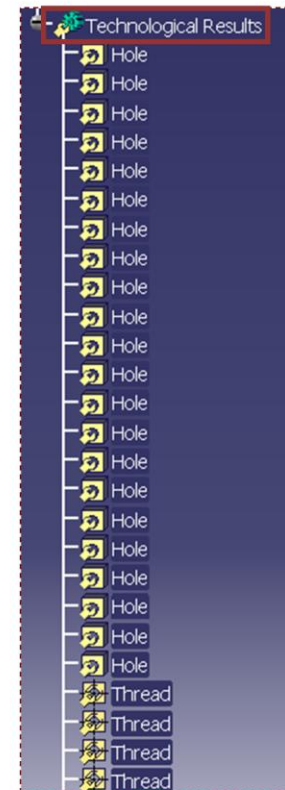
# Technological Results (2/5)

- **When a Drawing is created, Technological Results are automatically used notably to create Hole threads:**



**Thread of CapScrew Hole is automatically created**
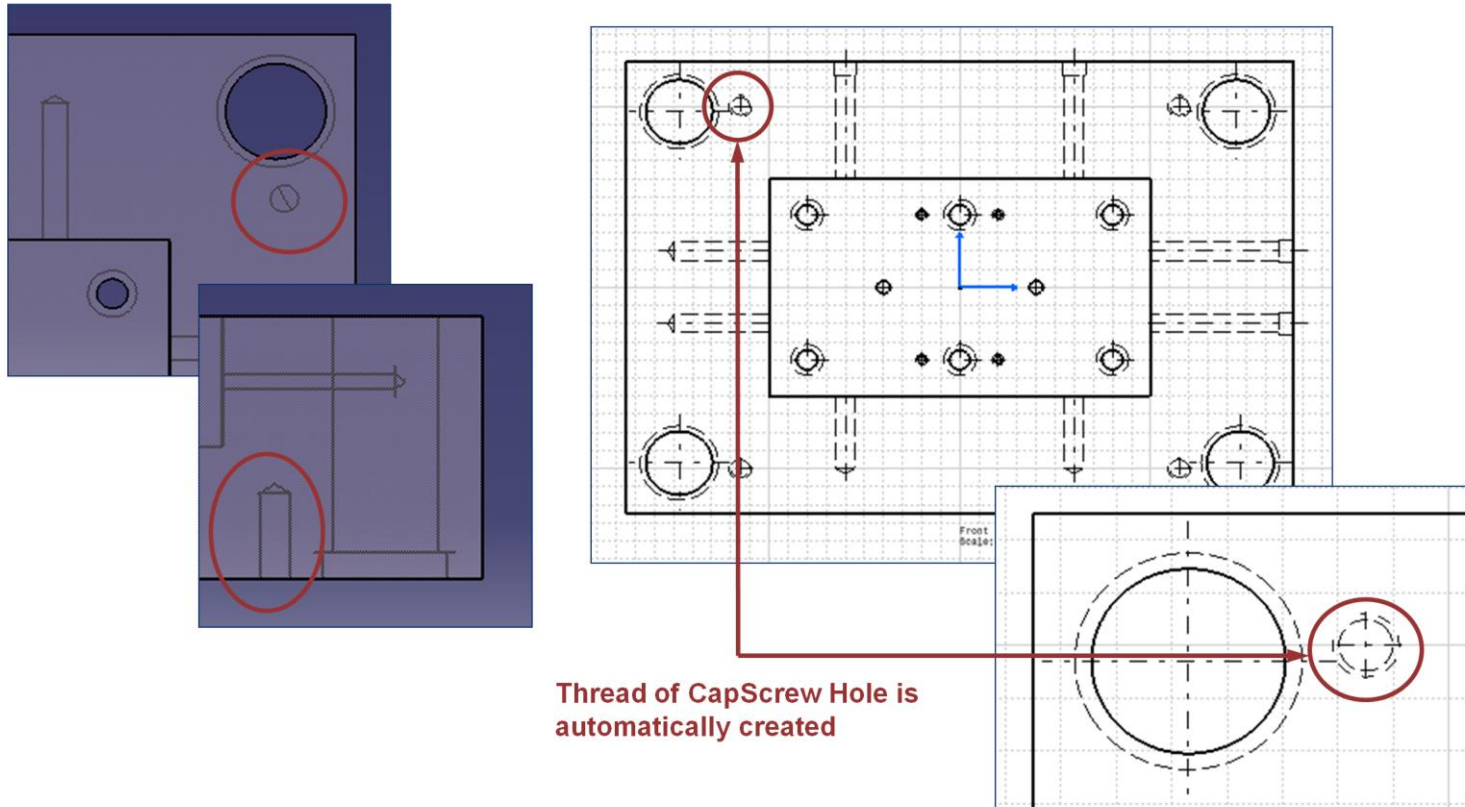
# Technological Results (3/5)

To create 2.5-axis machining operations, use function Global feature recognition in workbench MPA (Machining Prismatic Assistant) to gather technological information and e.g. create machining patterns using required parameters (diameter, thread, color):
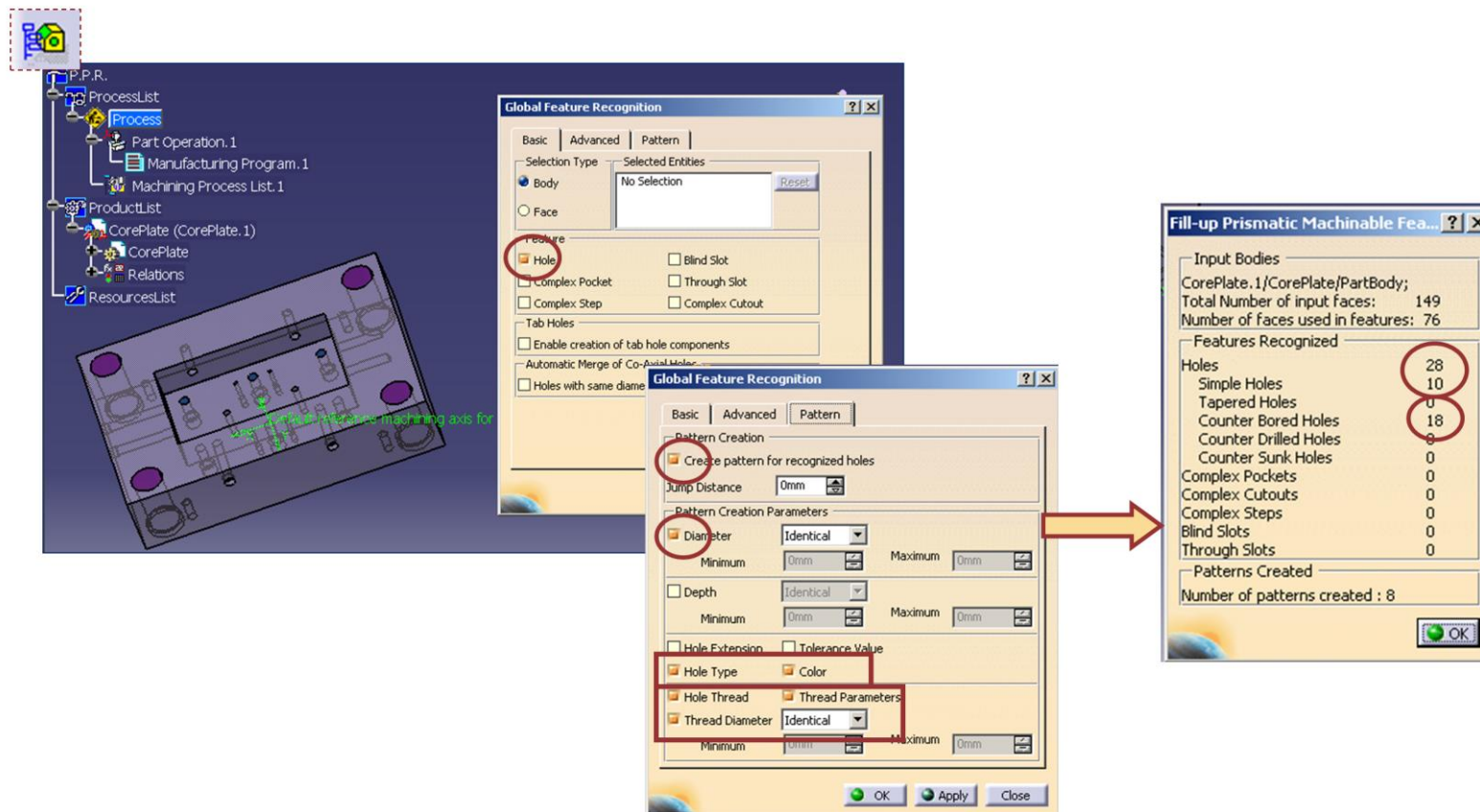
# Technological Results (4/5)

**Machining patterns and Machining Operations can then be used to create appropriate machining processes:**



**See machining operations and machining pattern corresponding to CapScrew holes**
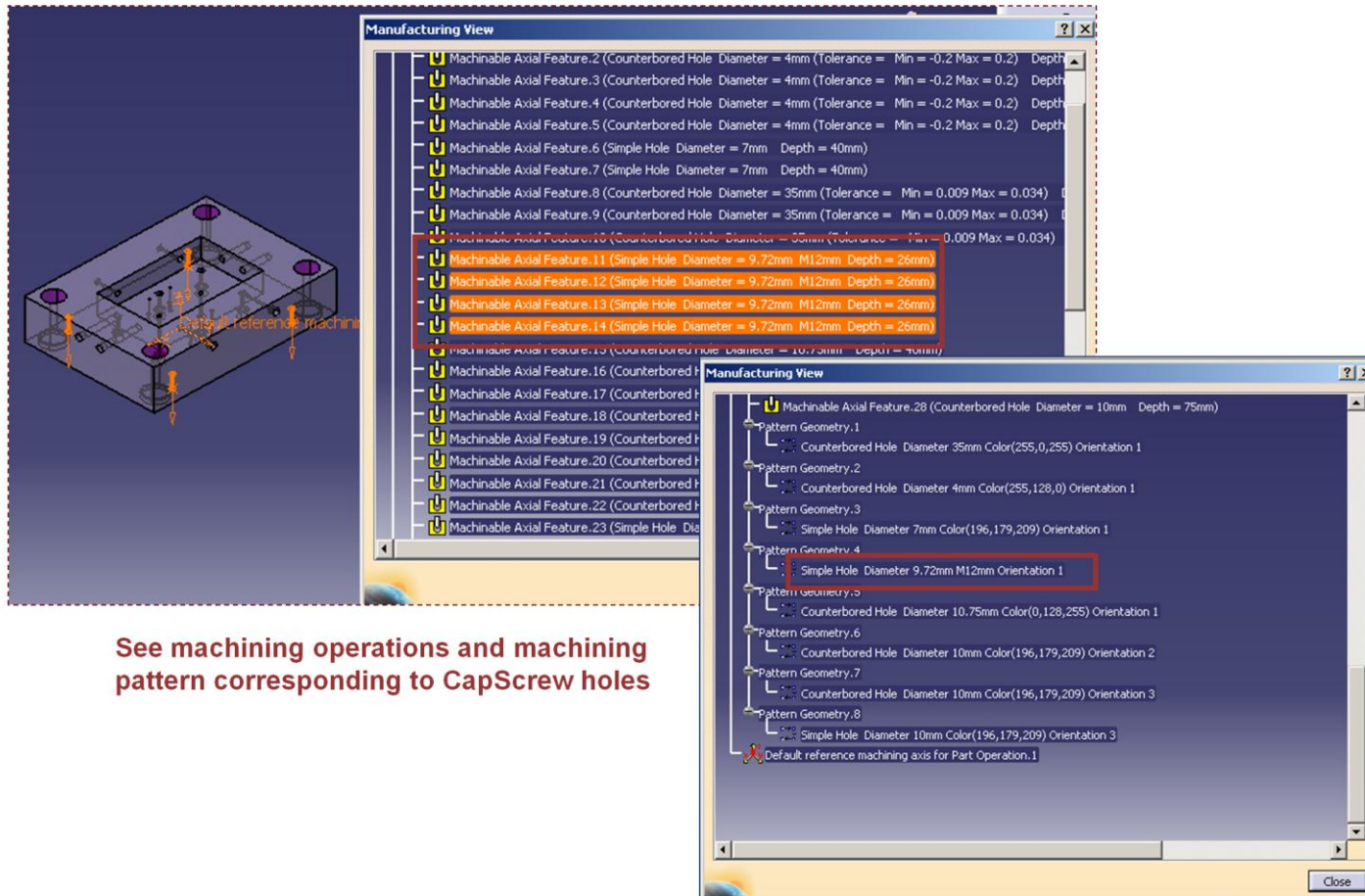
# Technological Results (5/5)

- Technological Results are automatically generated on Tooling components and their associated holes since V5R16.

- A batch program named CATMIdGenerateTRForTooling allows to generate Technological Results for Tooling Products (e.g. Mold Assemblies) created before V5R16:

  - Open a command prompt (MS-DOS) window.

  - Change to folder in <your_CATIA_installation>\intel_a\code\bin

  - Enter command: catstart -run CATMIdGenerateTRForTooling.exe DocumentPath (where DocumentPath is the full path of the documents to process)

    - The output document upgraded with the Technological Results overwrites the input document (writing rights are therefore needed).

**Pre-V5R16 Component**          **Impacted Plate**

**Updated Component**          **Impacted Plate**
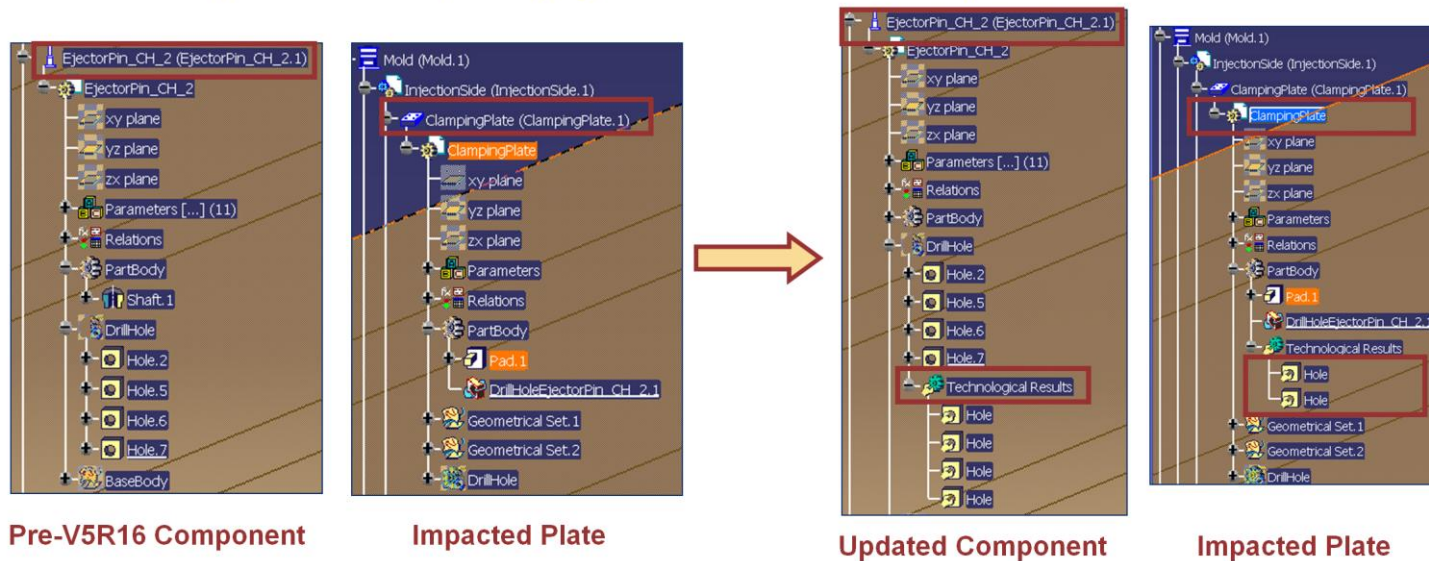
# Explode Holes (1/3)

- **As an alternative option to native Technological Results, it is possible to replace, in the elements containing drillings of Tooling Components, the original Remove or Assemble features by Hole features.**
- **This is done using function Explode Holes:**
  - ◆ **The relevant Remove/Assemble features are deactivated and replaced by the definition of the original Hole features defined in the corresponding components.**
  - ◆ **Note 1: Only features of type Hole are taken into account**
  - ◆ **Note 2: Function Explode Holes does not keep associativity with the design and must be run again if design changes occur.**

# Explode Holes (2/3)

**1** **Select Tools / Explode Holes**
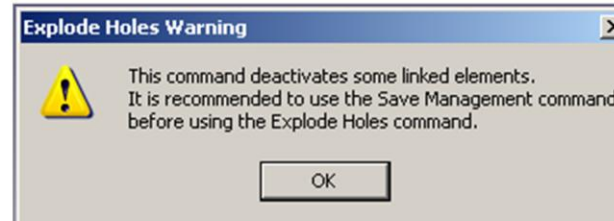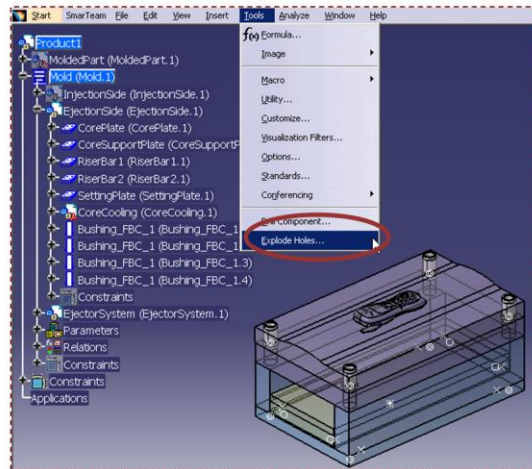
A message is displayed to inform you that some elements will be deactivated

**2** **Select the list of Plates to which operation Explode Holes will be applied**
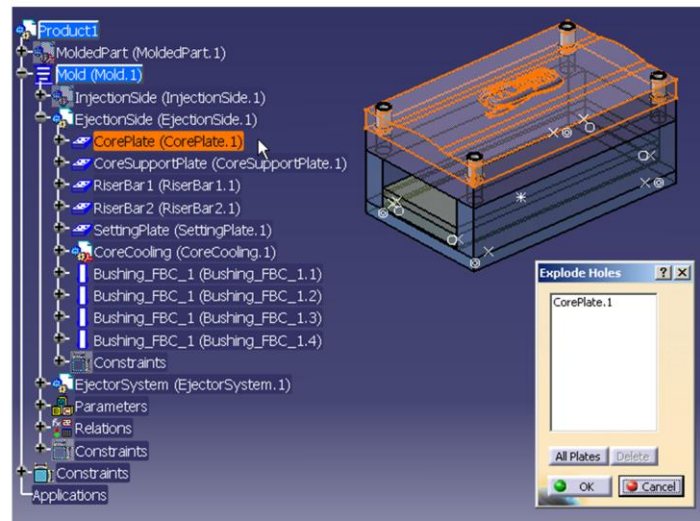
# Explode Holes (3/3)
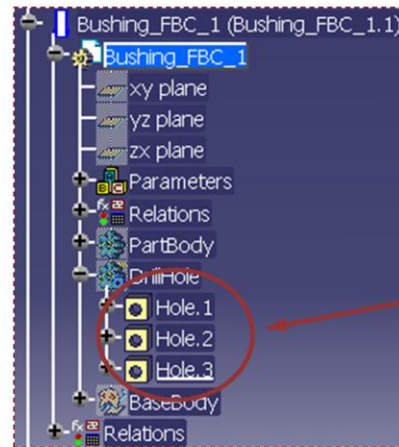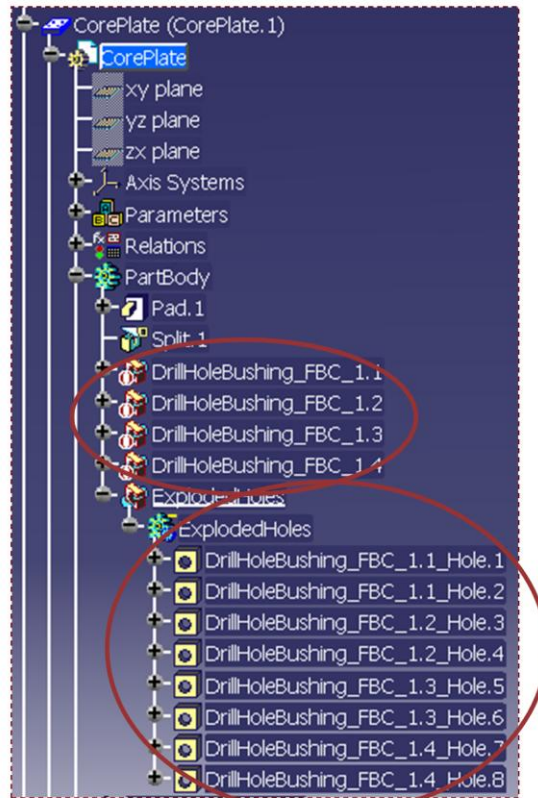
**Note that operation Explode Holes filters out Hole definitions present in the original component, but which do not have a geometric impact on the plate.**
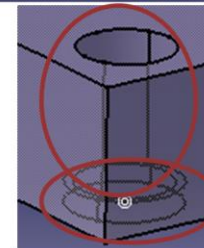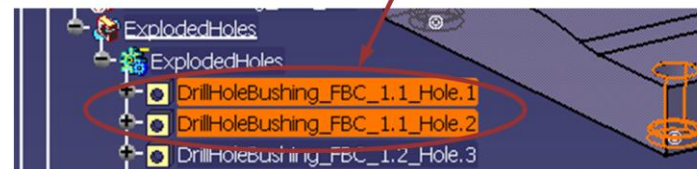
**The original Remove features are deactivated. A new Body named ExplodedHoles is created containing corresponding Hole features. A Remove feature is applied to this new Body.**

# Technological Results vs. Explode Holes

**Technological Results:**
- **No extra feature created**
- **Thread and Tolerance automatically available in drafting and manufacturing.**
- **No need of extra operation for manufacturing and it is easy to machine only one part of the assembly**
- **Covers transformations (Translate, Mirror, Pattern) on Hole and Thread features**
- **UDF supported**
- **No transfer of semantic tolerances and comments**
- **Not available through Knowledge**
- **No API VBScript available**
- **MPA necessary for manufacturing**

**Explode Holes:**
- **Transfer of semantic tolerances and comments**
- **No associativity**
- **Duplication of data (reference data + exploded data) because no rollback possible**
- **Time-consuming operation**

# Creating Injection Features

*You will become familiar with Injection Features of the Mold.*
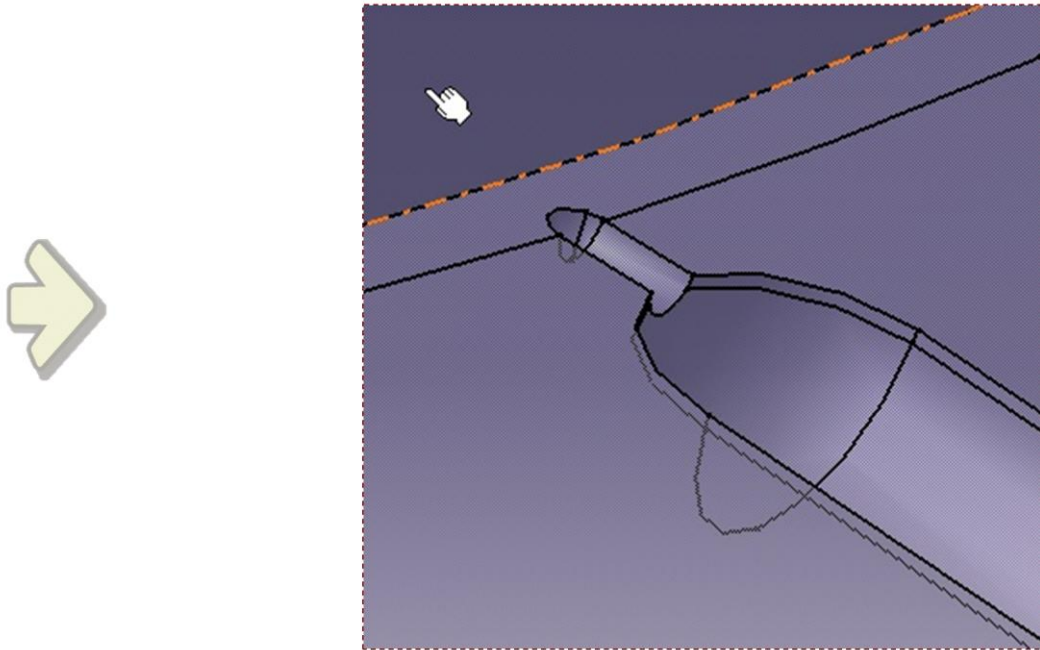
# Creating a Gate (1/2)

- There are 3 different types of gates:
  - ◆ Direct: the Gate is located directly on the Molded Part
  - ◆ Side (2 sub-types are available): the Gate is located on the Parting Line
  - ◆ Submarine (3 sub-types are available): the Gate is located close to the Molded Part, joining it via a small nozzle.

- The creation of a Gate is done in 2 steps:
  - ◆ First step: define the Gate location (point)
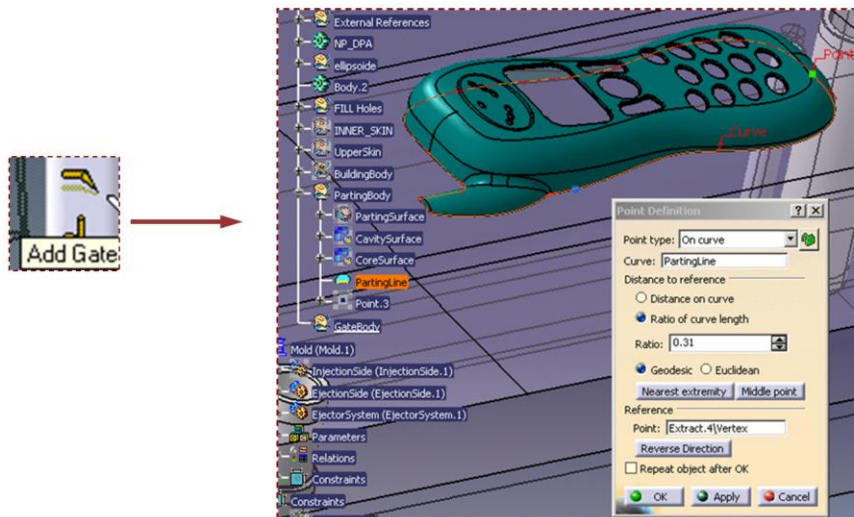
# Creating a Gate (2/2)

◆ **Second step: Once the location point is created (materialized by a yellow square), you can define the characteristics of the Gate:**

- ■ **Select the required type, then subtype of the Gate, using Catalog icon in the Gate definition dialog box:**
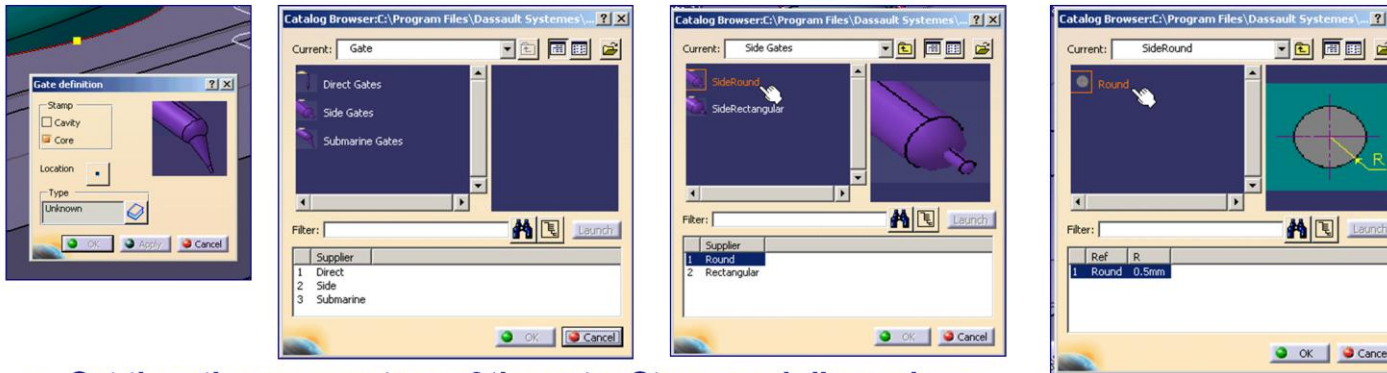


- ■ **Set the other parameters of the gate: Stamp and dimensions:**
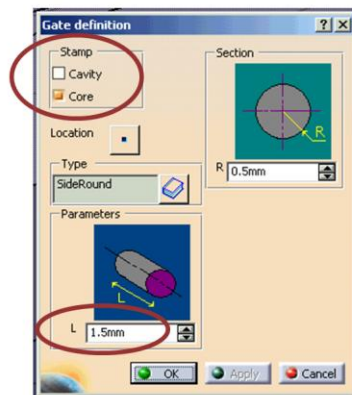


**At the end of these two steps, the Gate is materialized only by a yellow square in the 3D viewer.**
**Its real shape will be created only once the Injection Runner has been created.**

# Creating a Gate: Additional Information

- **Gate creation**
  - ◆ **The application creates in MoldedPart an Open Body called GateBody. It will contain the location (as Point name Gate.N) of all the gates created in this Part.**
  - ◆ **You can modify their location by directly selecting them in the tree.**
  - ◆ **You may create any number of gates.**
  - ◆ **Note that the Gates are picked from catalogs: you can create your own catalogs with gates of one of the predefined types (direct, side, submarine), but with your own sketch.**

- **Gate editing**
  - ◆ **You can edit a Gate via option Edit Gate in its contextual menu.**

- **Gate deletion**
  - ◆ **At this stage, you can delete a Gate using standard function Delete, since it is only a point.**
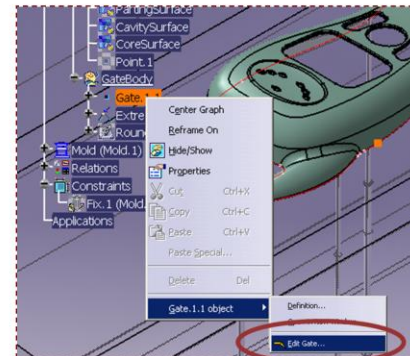
# Creating a Runner



- **The creation of a Runner is done in two steps:**
  - ◆ **Creating its layout using the Sketcher**
  - ◆ **Defining its shape characteristics**

**Step 1: Sketching the Runner layout**

The Gate point location must coincide with one extremity of the layout.
There must be tangency continuity between all elements of the sketch.

This is mandatory to ensure a correct projection of the sketch onto the support surface, operated by the application in the next creation step.



**Step 2: Creating the Runner**

**Click icon Add Runner**
**Define the characteristics of the Runner**
In the case of the Oval section shape, you can define Core or Cavity side as stamp body.
In the case of the Round shape, the stamp may be in Core, Cavity or both simultaneously.

Note that when creating the Runner, the Gate is also created in the 3D viewer

NOTE:
This operation leads to a contextual dependency of Core and Cavity Plates with respect to the Molded Part. Automatic context management is performed by application MTD (Part Interface mechanism) in the same way as it is done in the case of Component drillings.
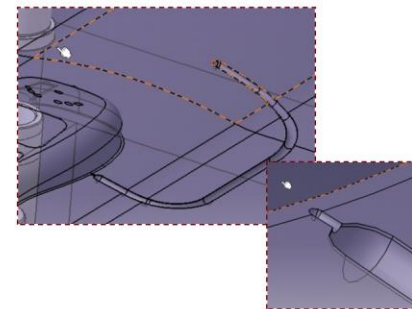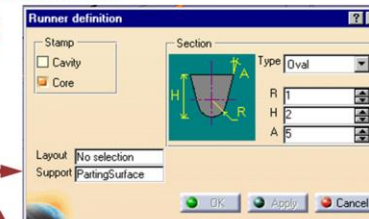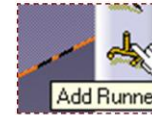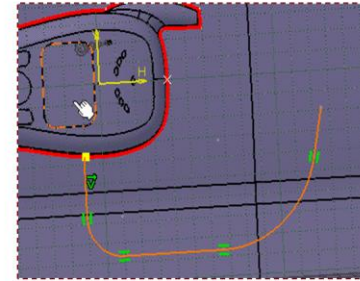
# Creating a Coolant Channel (1/2)

- Coolant Channels can be created in all the plates of the Mold Base, e.g. in CavityPlate, CorePlate, CoreSupportPlate.

- They can be created using various types of support geometry:
  - Two 3D points, which will be used as extremities of the Coolant Channel
  - A 3D line: the application will then automatically use its extremity vertices as extremities of the Coolant Channel
  - A Sketch made up of one or several segments

- The support geometry must be created before entering function Add Coolant Channel, by using for example workbench Wireframe and Surface.

- Note that support geometry must be created in component CoreCooling or CavityCooling, whatever plate the Coolant Channel will be drilled in.

  These components are automatically created by the application when the Mold Base is defined.
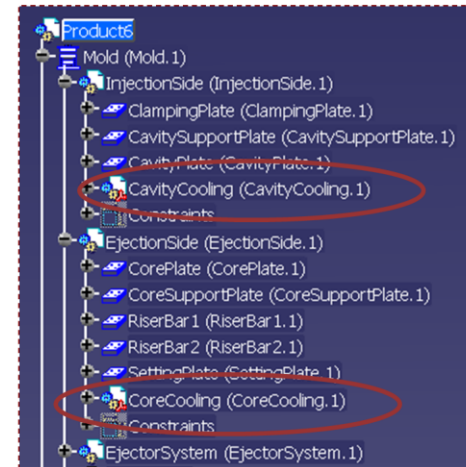
# Creating a Coolant Channel (2/2)

- The creation of a Coolant Channel is done by activating function Add Coolant Channel:

  Add Coolant Channel

  - Pick first its support elements. If you have selected a sketch, a Coolant Channel will be created for each segment of the sketch.
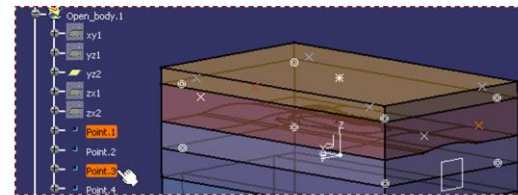
    **Point1 is the start point and Point3 is the endpoint**

  - Then define the parameters of the cooling pipe
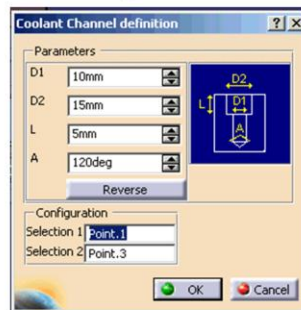
    **D1 = inner diameter**
    **D2 = counterbore diameter**
    **L = counterbore depth**
    **A = V-bottom angle**

  - The Coolant Channel is automatically located in a Body named CoolingBody

# Coolant Channel: Additional Information

- **Coolant Channel creation**
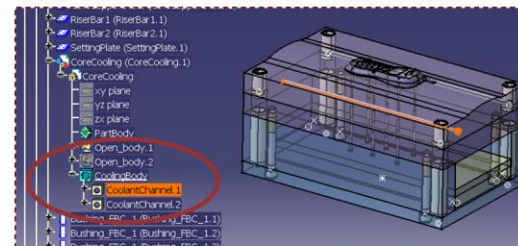  - ◆ **If none of the elements you enter as extremities of the Coolant Channel belongs to the outer planes delimiting the faces of the plate, the application will automatically propose a pipe segment reaching one of these planes: you can use option Reverse to select the opposite pipe segment.**

  - ◆ **If you use a predefined sketch, the system will prompt you, for each segment of the sketch, with the Reverse dialog box, in order to let you invert the pipe created on this segment if you want.**

- **Coolant Channel editing**
  - ◆ **You can edit a Coolant Channel via function Edit Coolant Channel in its contextual menu.**
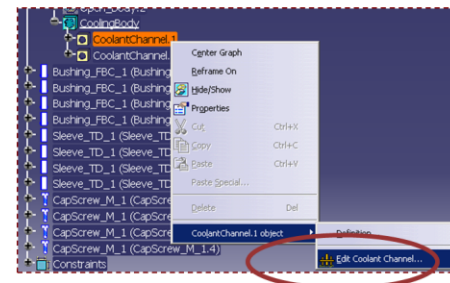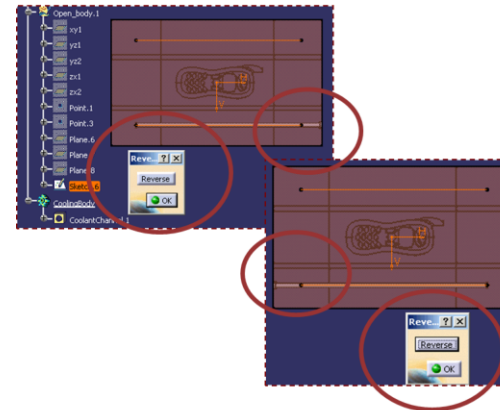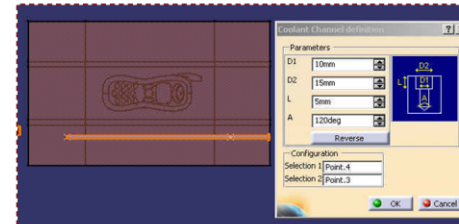
# Additional Information

*You will become familiar with contextual Links, customization of Bill of Material and Cache Management.*

Bill of Material: Product1

| Quantity | Part Number | Type | Nomenclature | Revision |
|----------|-------------|------|--------------|----------|
| 1 | MoldedPart | Part | | |
| 1 | Mold | Assembly | N3050 | |

Bill of Material: Mold

| Quantity | Part Number | Type | Nomenclature | Revision |
|----------|-------------|------|--------------|----------|
| 1 | InjectionSide | Assembly | | |
| 1 | EjectionSide | Assembly | | |
| 1 | EjectorSystem | Assembly | | |

Bill of Material: InjectionSide

| Quantity | Part Number | Type | Nomenclature | Revision |
|----------|-------------|------|--------------|----------|
| 1 | ClampingPlate | Part | N01-3050-26/1 | |
| 1 | CavityPlate | Part | N10-3050-86/1 | |
| 4 | LeaderPin_FSC_2 | Part | FSC 20-36-146 | |

Bill of Material: EjectionSide

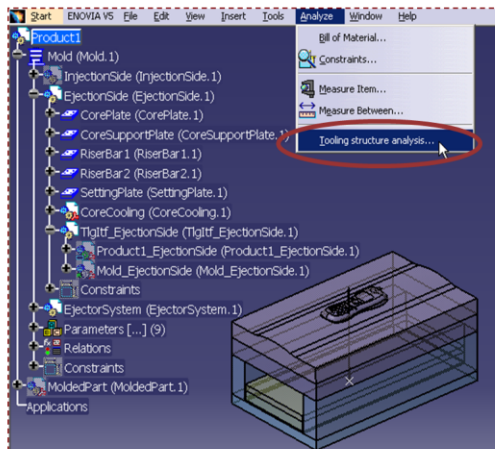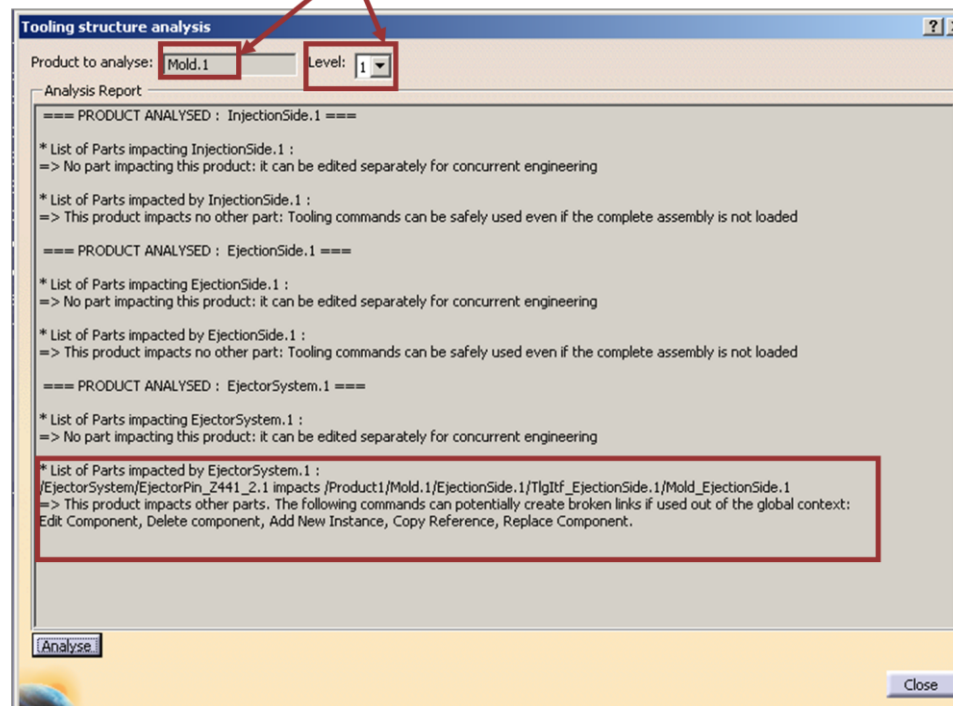| Quantity | Part Number | Type | Nomenclature | Revision |
|----------|-------------|------|--------------|----------|
| 1 | CorePlate | Part | N10-3050-86/1 | |
| 1 | CoreSupportPlate | Part | N20-3050-46/1 | |
| 1 | RiserBar1 | Part | N30-106-3050/1 | |
| 1 | RiserBar2 | Part | N30-106-3050/1 | |
| 1 | SettingPlate | Part | N01-3050-26/1 | |
| 4 | CapScrew_M_4 | Part | M 12x150 | |
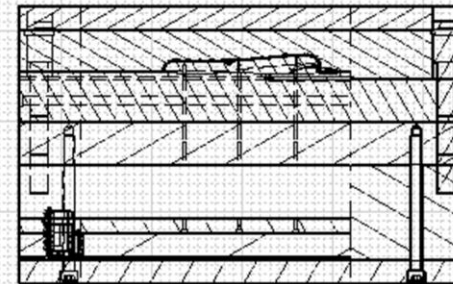
Section view A-A
Scale: 1:3
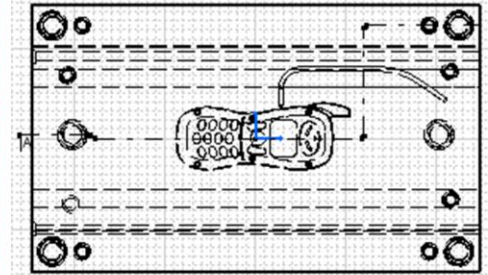
# Analyzing Contextual Links in the Assembly Structure

- **Function Analyze / Tooling Structure Analysis allows to identify contextual links between elements of the Assembly. This information can then be used to best define the structure of the Assembly and organize concurrent engineering tasks.**

**Analysis is performed on Product *Mold.1*, up to depth *Level = 1*, i.e. up to the elements belonging to sub-assemblies *InjectionSide*, *EjectionSide*, *EjectorSystem***



**The contextual impact of Component *EjectorPin* on sub-assembly *EjectionSide* is listed**



Tooling structure analysis

Product to analyse: Mold.1     Level: 1

Analysis Report

=== PRODUCT ANALYSED : InjectionSide.1 ===

* List of Parts impacting InjectionSide.1 :
=> No part impacting this product: it can be edited separately for concurrent engineering

* List of Parts impacted by InjectionSide.1 :
=> This product impacts no other part: Tooling commands can be safely used even if the complete assembly is not loaded

=== PRODUCT ANALYSED : EjectionSide.1 ===

* List of Parts impacting EjectionSide.1 :
=> No part impacting this product: it can be edited separately for concurrent engineering

* List of Parts impacted by EjectionSide.1 :
=> This product impacts no other part: Tooling commands can be safely used even if the complete assembly is not loaded

=== PRODUCT ANALYSED : EjectorSystem.1 ===

* List of Parts impacting EjectorSystem.1 :
=> No part impacting this product: it can be edited separately for concurrent engineering

* List of Parts impacted by EjectorSystem.1 :
/EjectorSystem/EjectorPin_Z441_2.1 impacts /Product1/Mold.1/EjectionSide.1/TlgItf_EjectionSide.1/Mold_EjectionSide.1
=> This product impacts other parts. The following commands can potentially create broken links if used out of the global context:
Edit Component, Delete component, Add New Instance, Copy Reference, Replace Component.
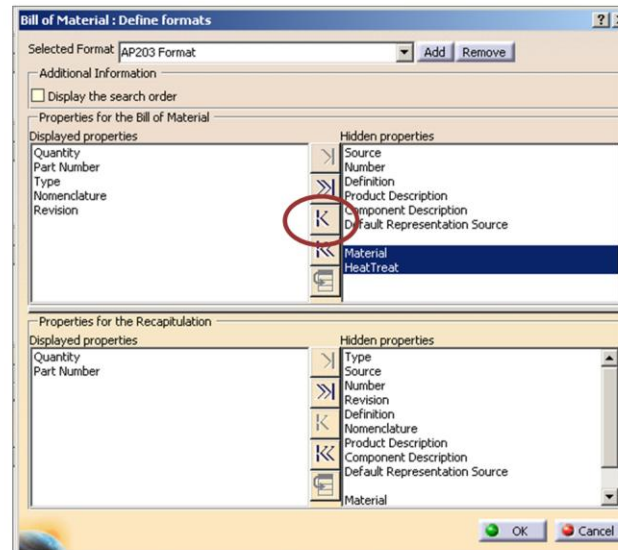
Analyse

Close

# Customizing the Bill of Material of a Mold/Die Product

- You can access the Bill of Material directly from menu Analyze.

- It contains additional properties compared to the standard function accessible in workbench Assembly Design.
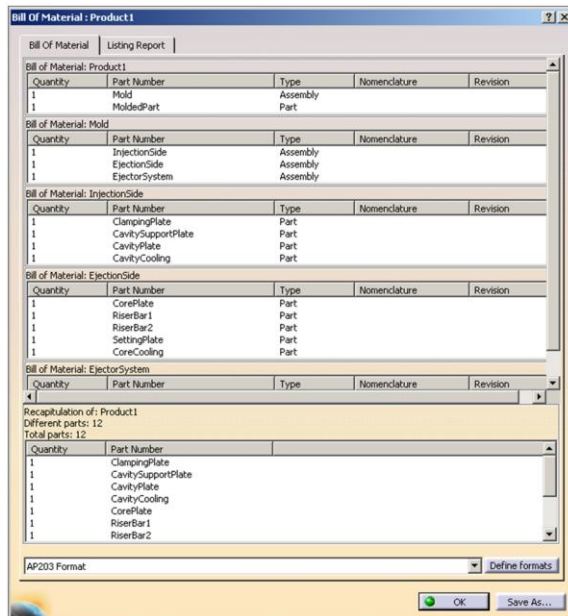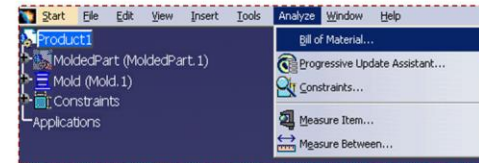
**Standard output format**

**Redefine output format: add as Displayed properties TG1-specific Heat Treatment and Material**

# Working with large Assemblies: Cache Management

- When working with very large assemblies, it is possible to greatly improve performance by using the cache system in order to decrease memory consumption: activate option Work with the cache system in Tools/Options/Infrastructure/Product Structure/Cache Management
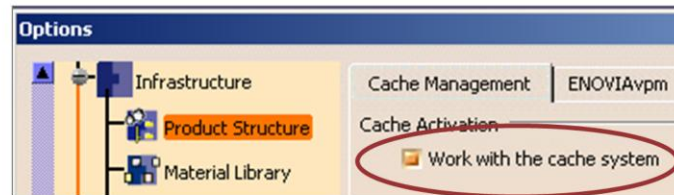


- When a TG1 operation (Create Component, Edit Component, Split Component ) involves the modification of receiving elements (Plates being drilled), application TG1 will automatically switch these elements (and only these elements) to Design Mode. These objects will be created (or recreated) as CGR objects only when the user performs a Save operation.

# What's New in Mold Tooling Design

*You will become familiar with the new and enhanced features that are now available in Mold Tooling Design.*

# New Mold Tooling Design Capabilities

🔹 **The following list summarizes what is new in Mold Tooling Design:**

◆ **Display Technological Results (V5R19)**

◆ **Create Holes Table in Drawing (V5R20)**

◆ **Component Instantiation: Keep As Reference (V5R20)**

◆ **Tooling Components Bill Of Material (V5R21)**

◆ **Tool Viewer (V5R21)**

◆ **Split Untrimmed Ejectors (V5R21)**

# Display Technological Results (1/2)

🔷 **Full report of existing Technological Results (TR):**

- ◆ **Automatic generation of TRs is controlled by dedicated Setting**
- ◆ **New dedicated command Display Technological Results**
- ◆ **Result file (.csv) can be output (with MPA license)**



- ◆ **Enables to define company-specific processes between design and manufacturing stages of the process.**
- ◆ **Enables better control of the integrity of the Tool from design to manufacturing**
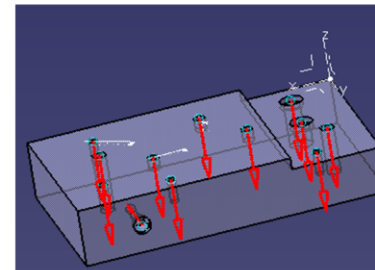
# Display Technological Results (2/2)

**Display of Technological Results (TR):**

- **When selecting a plate in the graphic area, the list of TR characteristics is displayed in the panel.**

- **When picking a line in the panel or a face of a hole in the graphic area, the associated TR is highlighted.**

- **If user parameters have been defined for holes, they are displayed also and output in the file.**
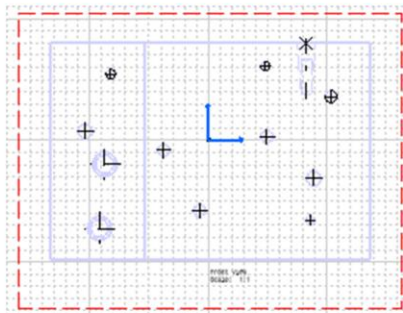
# Create Holes Table in Drawing (1/2)

**New Tooling command in Drawing (TG1 or MTD license active)**

- **Creates Holes Table and Labels**
- **One label associated to each hole**
- **Label number corresponds to Ref in Hole Table**



Text Properties
Tooling to Drawing
Tools
Update

My drilling hole table
(DrawingTestASI.CATPart; Axis System:DrawingTestASI.CATPart: (Default)); Friday, March 20, 2009 4:23:26 PM)

| REF. | TRType | Thread | Name | Diameter | Diameter Min Toler | Diameter Max Toler | Thread Description | Thread Nominal Diameter | X | Y | Z | Geometrical Depth | DX | DY | DZ |
|------|--------|--------|------|----------|--------|--------|--------|--------|------|------|------|------|------|------|------|
| | | | | mm | mm | mm | | mm | mm | mm | mm | mm | | | |
| A123-100 | Hole | | Hole.49 | 10 | -0.1 | 0.1 | | | -7.469 | -57.784 | 50 | 10 | 0 | 0 | -1 |
| A123-101 | Hole | | Hole.50 | 12 | 0 | 0 | | | 89.152 | -31.52 | 50 | 50 | 0 | 0 | -1 |
| A123-102 | Hole | | Hole.51 | 12 | 0 | 0 | | | -103.799 | 7.318 | 40 | 40 | 0 | 0 | -1 |
| A123-103 | Hole | | Hole.52 | 15 | 0 | 0 | | | -91.796 | -72.44 | 40 | 40 | 0 | 0 | -1 |
| A123-104 | Thread | | Thread.17 | 22 | 0 | 0 | | 22 | -30 | 70.279 | 20 | 10 | 0 | 1 | 0 |
| A123-105 | Hole | | Hole.53 | 15 | 0 | 0 | | | -87.394 | -19.618 | 40 | 24.33 | 0 | 0 | -1 |
| A123-106 | Hole | | Hole.54 | 10 | 0 | 0 | | | -37.552 | -8.189 | 50 | 10 | 0 | 0 | -1 |
| A123-107 | Hole/Thread | Thread.18 | Hole.55 | 8.917 | 0 | 0 | M10x1 | 10 | 103.488 | 36.052 | 50 | 22.574 | 0 | 0 | -1 |
| A123-108 | Hole/Thread | Thread.19 | Hole.56 | 6.917 | 0 | 0 | M8 | 8 | -82.397 | 54.641 | 40 | 21.997 | 0 | 0 | -1 |
| A123-109 | Hole/Thread | Thread.20 | Hole.57 | 6.917 | 0 | 0 | | 8 | 48.125 | 61.123 | 50 | 21.997 | 0 | 0 | -1 |
| A123-110 | Hole | | Hole.58 | 6.917 | -0.2 | 0.1 | | | 85.915 | -65.856 | 50 | 21.997 | 0 | 0 | -1 |
| A123-111 | Hole/Thread | Thread.21 | Hole.59 | 8.5 | 0 | 0 | | 9.35 | 82.845 | 80.279 | 23.437 | 42.148 | 0 | -1 | 0 |
| A123-112 | Hole | | Hole.60 | 10 | 0 | 0.015 | | | 49.548 | 3.022 | 50 | 10 | 0 | 0 | -1 |

Front view
Scale: 1:1

# Create Holes Table in Drawing (2/2)

- **The title and format of the table are user defined. You can:**
  - **Merge the threads into Holes with an option.**
  - **Define the axis system.**
  - **Define the units ( Tools/Options ).**
  - **Define the prefix and numbering.**

- **The values available in the table are all the Technological Results associated to the holes.**

- **It is possible to associate one or several user parameters to the holes, for added customization possibilities.**
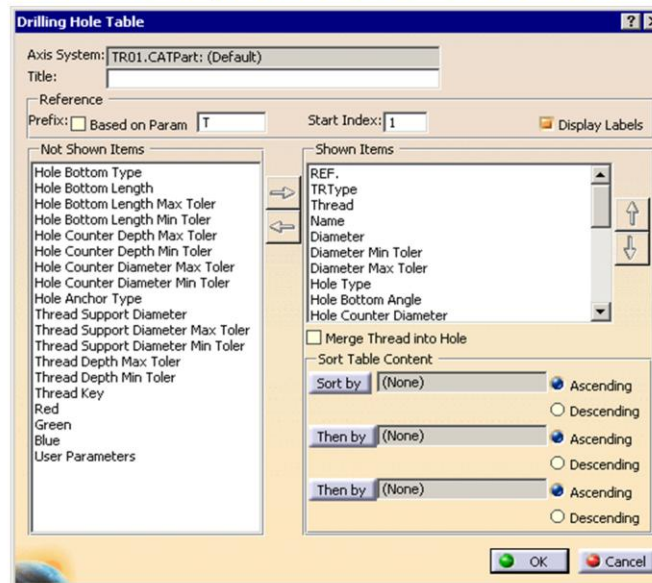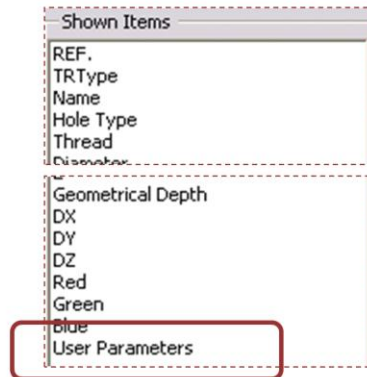
# Component Instantiation: Keep As Reference

- You can now instantiate a component as an instance of the original reference. It is now possible to:
  - Either instantiate a component by cloning the original reference (as in V5R19 or before).
  - Or instantiate a component as an instance of the original reference: new mode "Keep As Reference".
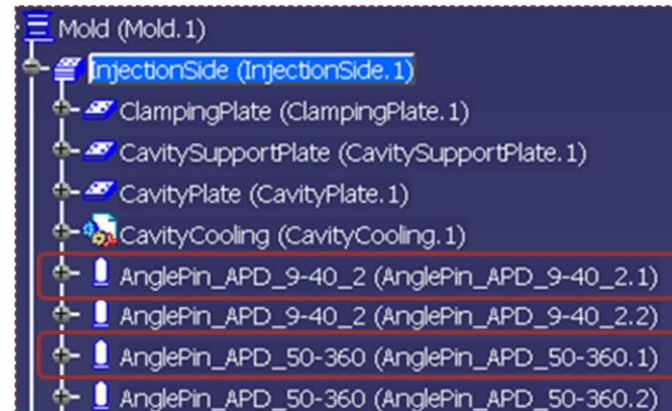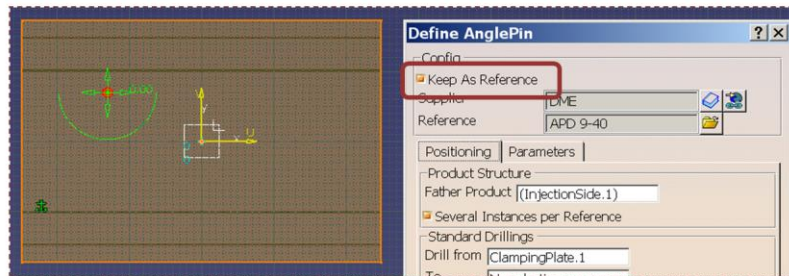


- Advantages:
  - For better compliance with PDM/PLM system requirements for part management.
  - To simplify BOM management.
  - To enable certain Component lifecycle management capabilities.
  - To improve resource management.

# Bill of Material (1/2)

- **Customizable by using a Reference Profile, persistent numbering.**
- **All Tooling components with same supplier reference appear on one BOM line as instances.**
- **Management of columns: hide, show, add user-defined column.**
- **Management of component price.**
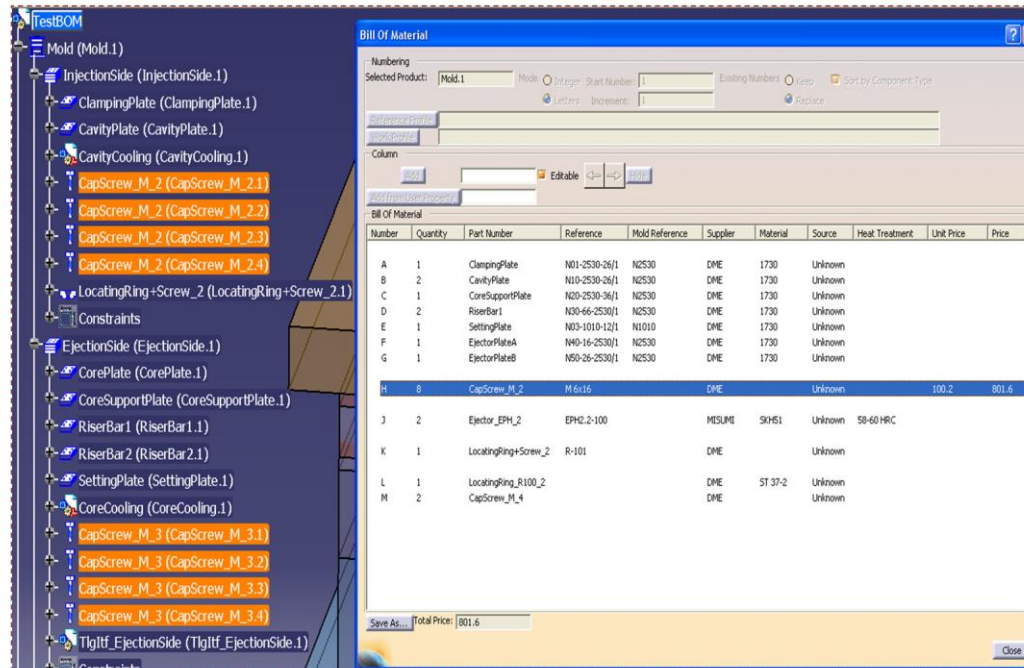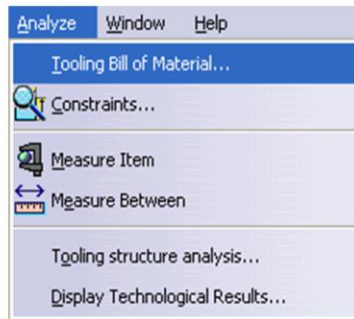
# Bill of Material (2/2)

- **Output is created in *.csv file format.**

Compute Date: Wednesday, December 02, 2009 6:12:54 PM

Project: /1Owner:    LCH

| Number | Quantity | Part Number | Reference | Mold Reference | SuppShonan | Material | Source | Heat Treatment | Unit Price | Price | Comment |
|--------|----------|-------------|-----------|----------------|------------|----------|--------|----------------|------------|-------|---------|
| B | 1 | ClampingPlate | N01-2530-26/1 | N2530 | SHONAN | | 1730 Unknown | | | | |
| C | 2 | CavityPlate | N10-2530-26/1 | N2530 | | | 1730 Unknown | | | | |
| D | 1 | CoreSupportPlate | N20-2530-36/1 | N2530 | | | 1730 Unknown | | | | hello |
| E | 2 | RiserBar1 | N30-66-2530/1 | N2530 | | | 1730 Unknown | | | | |
| F | 1 | SettingPlate | N03-1010-12/1 | N1010 | | | 1730 Unknown | | | | |
| G | 1 | EjectorPlateA | N40-16-2530/1 | N2530 | | | 1730 Unknown | | | | |
| H | 1 | EjectorPlateB | N50-26-2530/1 | N2530 | | | 1730 Unknown | | | | |
| J | 8 | CapScrew_M_2 | M 6x16 | | | | Unknown | | 12 | 96 | |
| J | 2 | Ejector_EPH_2 | EPH2.2-100 | | | SKH51 | Unknown | 58-60 HRC | | | |
| K | 1 | LocatingRing+Screw_2 | R-101 | | | | Unknown | | | | |
| L | 1 | LocatingRing_R100_2 | | | | ST 37-2 | Unknown | | | | |
| M | 2 | CapScrew_M_4 | | | | | Unknown | | | | |

# Tool Viewer

**Navigation in the Mold by component type, functional family (customizable), properties, …**
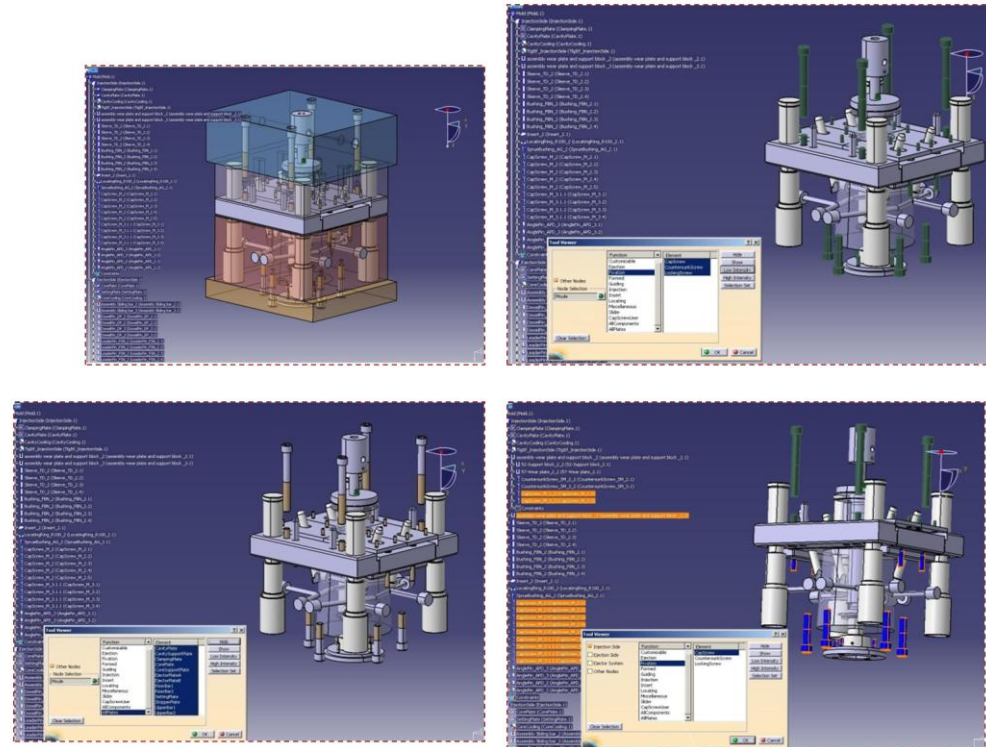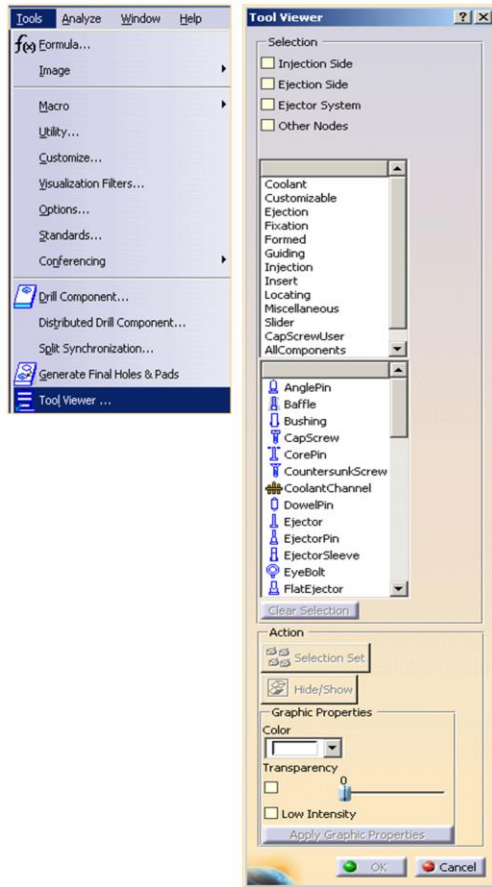
# Split Untrimmed Ejectors

- **Automatic detection of all untrimmed ejector-type components (Core Pin, Ejector, Ejector Pin, Ejector Sleeve, Flat Ejector)**
- **Direct modification of Offset Parting value**
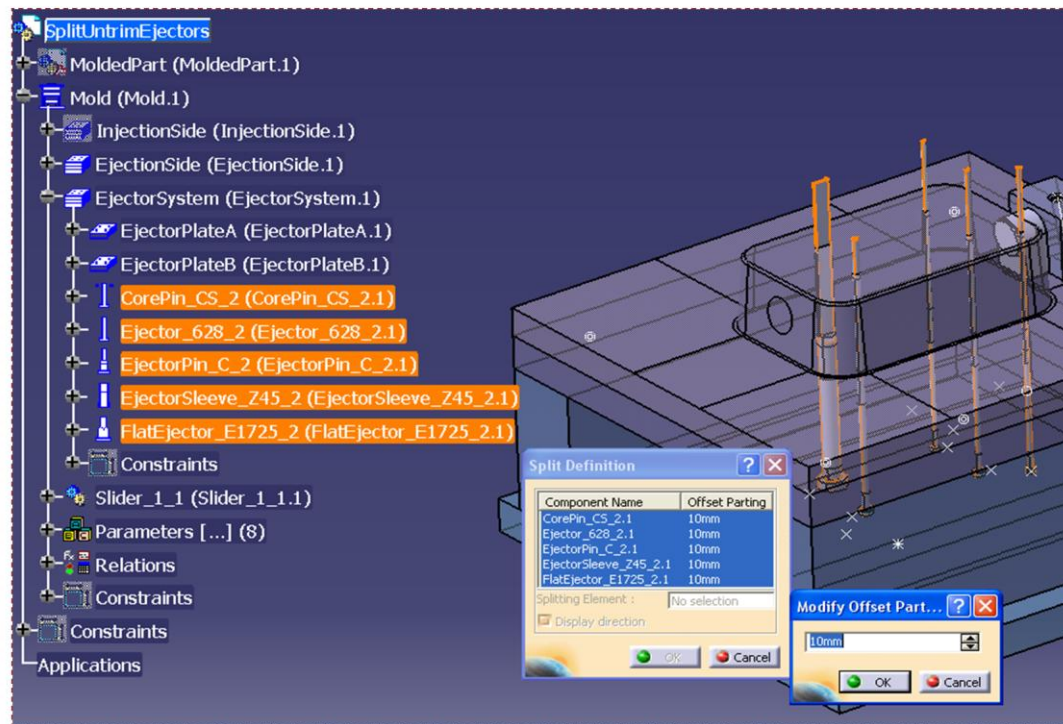
# Enhanced Mold Tooling Design Capabilities

🔹 **The following list summarizes what is new in Mold Tooling Design:**

◆ **Simplified Holes structure in Components (V5R19 SP6)**

◆ **Concurrent Engineering: Impacts Management (V5R19)**

◆ **Component Design: Customized Icons (V5R19)**

◆ **Component Design: Additional Tools (V5R20)**

◆ **Mold Tooling Design Catalogs: User Components (V5R19 SP6)**

◆ **Smart Ejector Template: Ejector with Auto-adjustable anti-rotation (V5R19 SP6 / V5R20 SP2)**

◆ **Smart Insert Template: Insert with escape holes (V5R19)**

◆ **Mold Base catalogs: Equipped Mold Bases (V5R19 SP6 / V5R20 SP2)**

# Simplified Hole Structure in Components

- **Mold to CAM : DrillHoles redefined to optimize connection with MPA:**
  - ◆ **Cap Screws**
  - ◆ **Countersunk Screws**
- **Before :**
  - ◆ **Two Part Design simple Hole features (screw head and screw body)**
- **After :**
  - ◆ **One single Part Design Hole feature (Counterbored or Counterdrilled)**
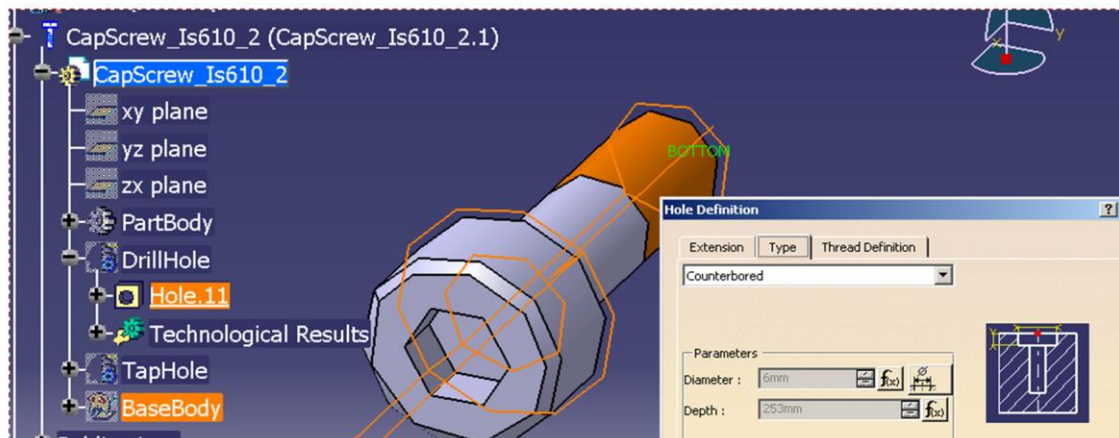
# Impacts Management

🔹 **Concurrent Engineering : Manage editability of impacts:**

   🔷 **Pre-create and assign Tooling Interface Parts to each user when defining the structure of the Mold/Die assembly.**

   🔷 **Existence of needed Tooling Interface Part is checked to display warning messages when necessary.**

   🔷 **Uses Read/Write status resulting from check-out in order to filter access right to Interface Part.**

   🔷 **To avoid risk of interference between users for assembly context definition.**

   🔷 **To minimize risk of losing work if necessary context information has not been loaded.**
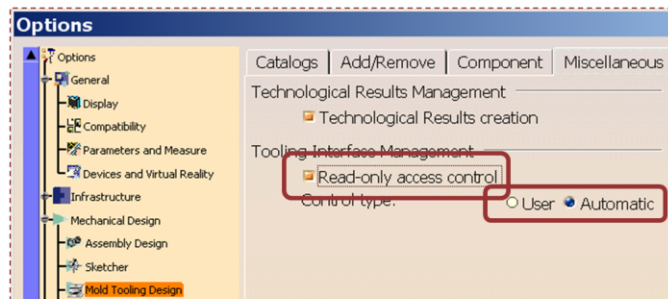


🔹 **Option to activate control access status.**

   🔷 **User Warnings are displayed**

     ▪ You can either take action or let MTD/TG1 make decisions.

   🔷 **Automatic (default)**

     ▪ MTD/TG1 automatically makes decisions.
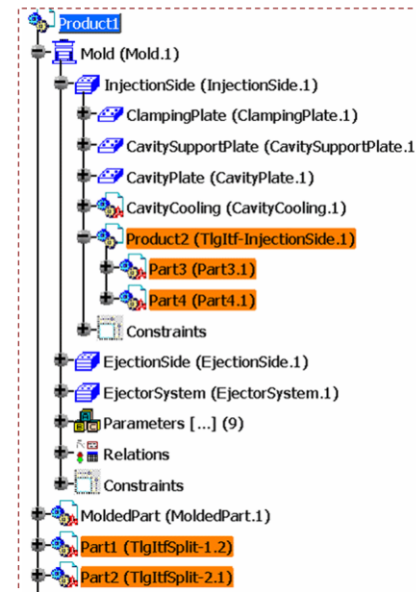
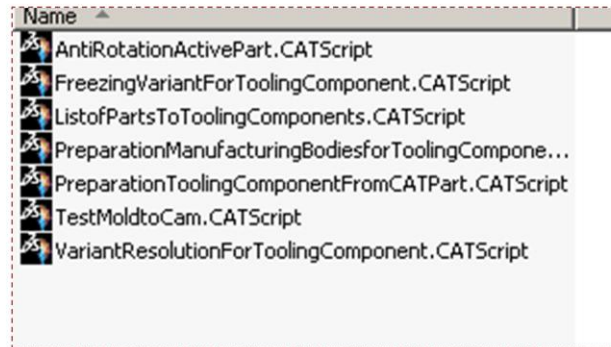# Customized Icons



🔹 **User-defined icons for User Components :**

◆ **A new parameter IconRef (type String) can be added in the list of User Component Parameters : it contains the name of the bitmap file of the icon.**

▪ If the parameter is not defined, the default icon is selected when instantiating the component.

◆ **For better customization and understanding of the specification tree.**
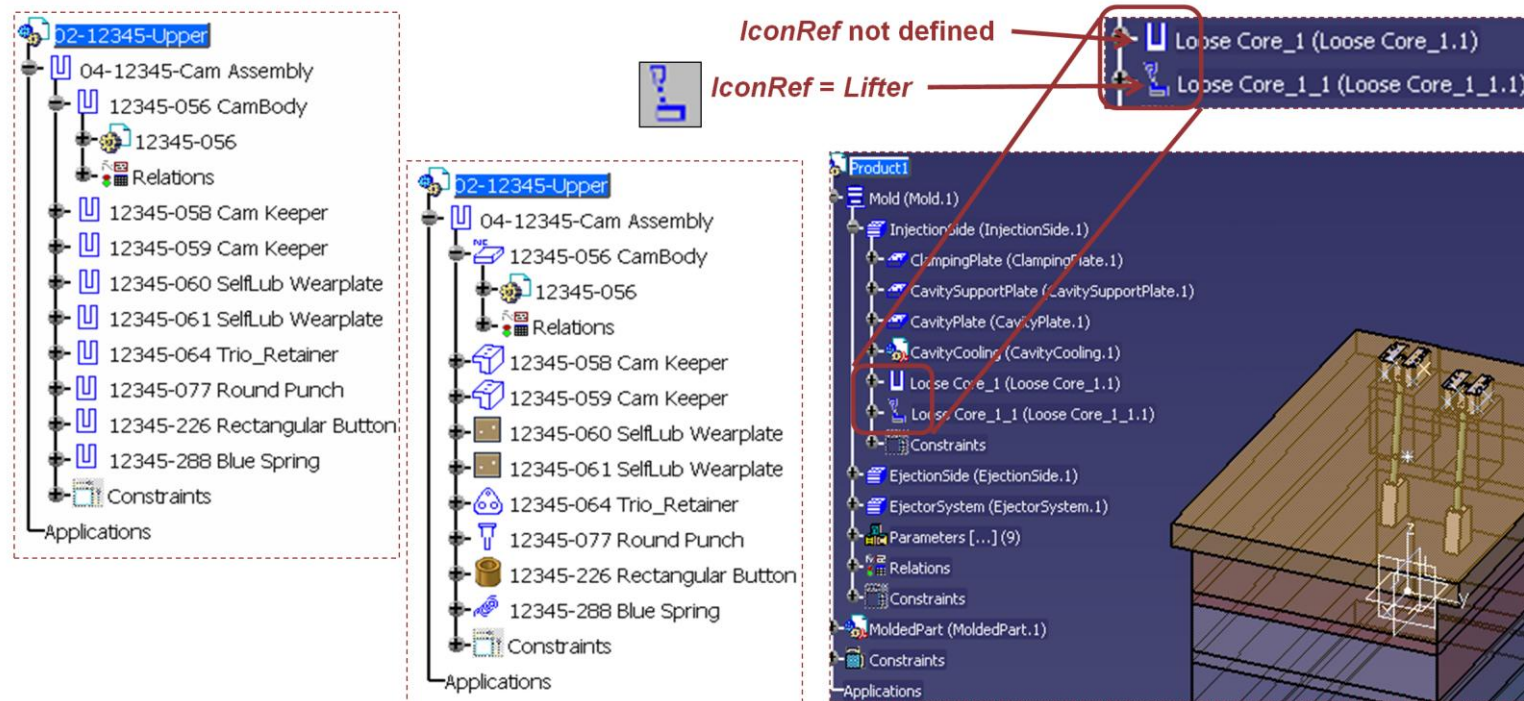
## Component Design (1/5)

### Additional Tools

- **New utilities are available to enhance productivity.**
  - They can be adapted to fit more closely company's practices.
- **Tooling components are CATParts which follow some basic conventions.**
- **Several macros automatically update a CATPart to make it a recognized Tooling Component.**
  - They can apply to the active part or else to a list of CATParts

```
📁 Macros
📁 UDF
```

| Name ▲ |
| --- |
| AntiRotationActivePart.CATScript |
| FreezingVariantForToolingComponent.CATScript |
| ListofPartsToToolingComponents.CATScript |
| PreparationManufacturingBodiesforToolingCompone... |
| PreparationToolingComponentFromCATPart.CATScript |
| TestMoldtoCam.CATScript |
| VariantResolutionForToolingComponent.CATScript |

**Located in  … \intel_a\startup\Tooling\MTD**
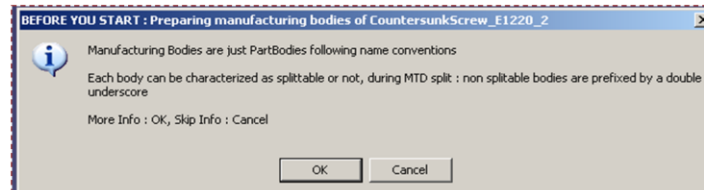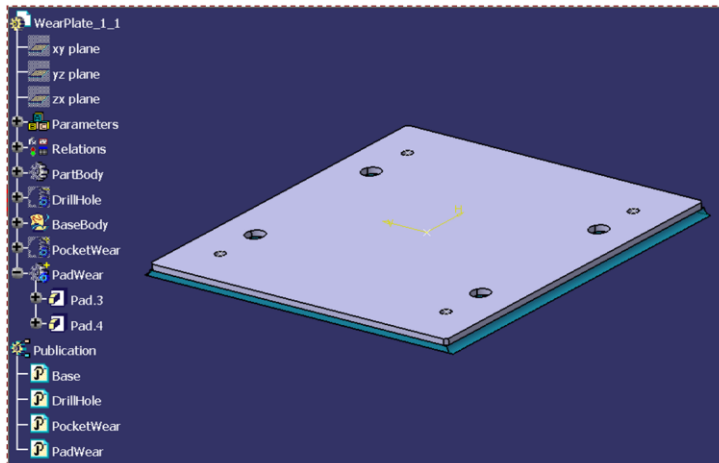
# Component Design (2/5)

## 🔹 Define Position

◆ **The macro *ListofPartstoToolingComponents* takes a list of parts and automatically creates and publishes positioning information**
- Geometrical Set BaseBody
- Point Base
- Axis System BaseAxisSystem and associated dialog parameters

◆ **Macro *PreparationToolingComponentFromCatPart* does the same for the active Part**
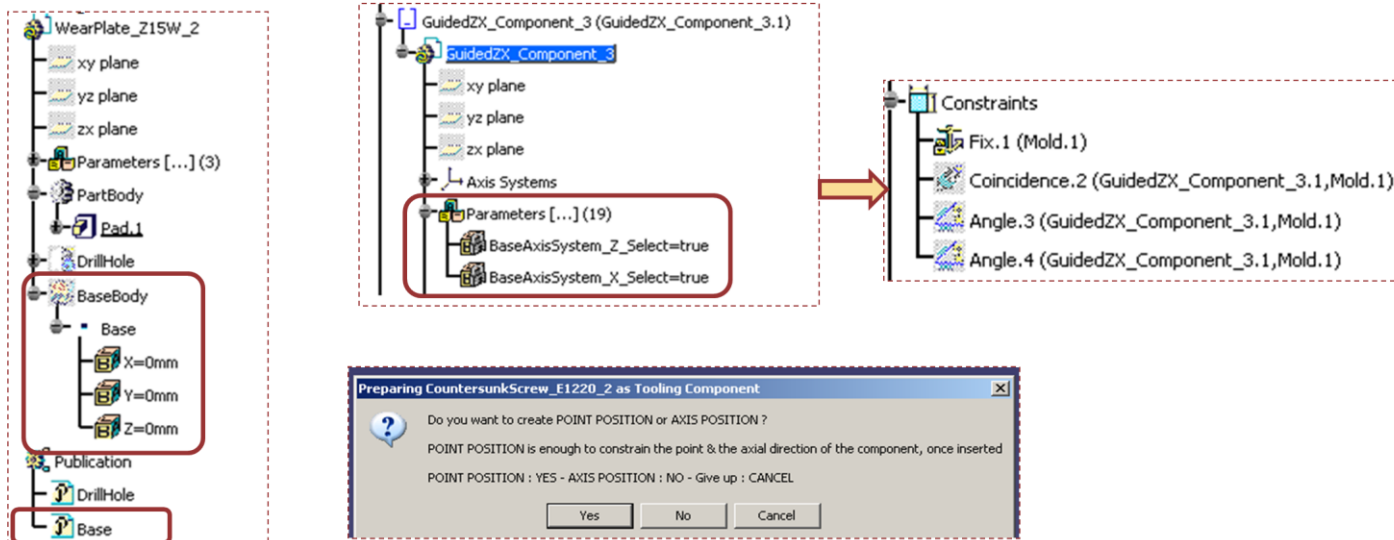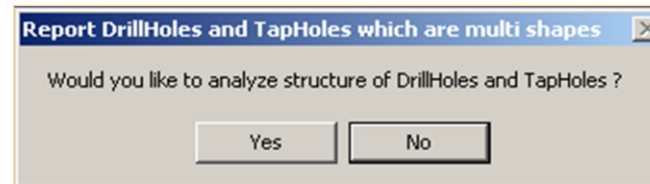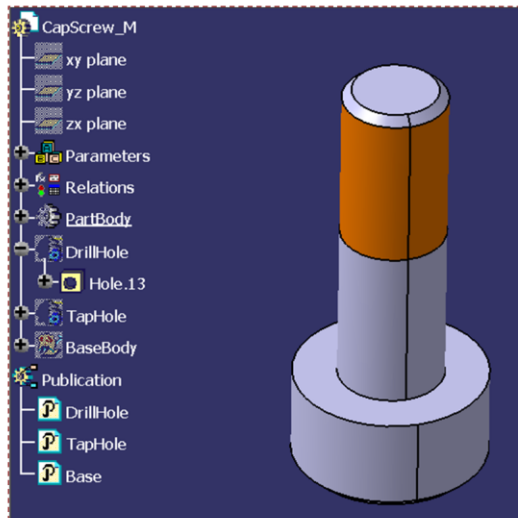
# Component Design (3/5)

### Define Impacts

- #### The macro *PreparationManufacturingBodiesForToolingComponents* creates and publishes impact information:
  - Drill Holes and Tap Holes
  - Pads and Pockets
- #### Additional possibilities:
  - create the geometry of the DrillHole automatically as the negative geometry of *PartBody*.
  - Tag non-splittable impacts (naming convention "initial double-underscore").
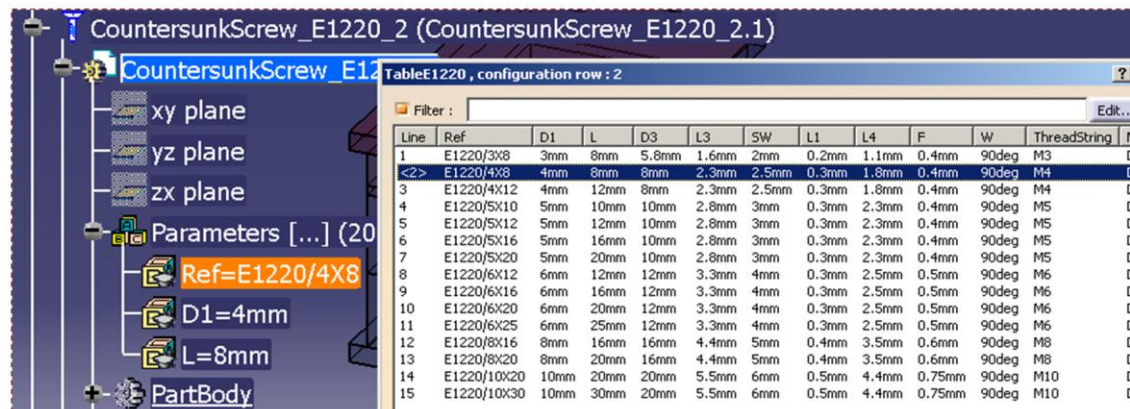
# Component Design (4/5)

### ◆ Check Structure

- ◆ **The Mold to CAM process is enhanced if the Drill Holes end Tap Holes are designed with as few Hole features as possible.**

- ◆ **The macro** *TestMoldToCAM* **analyzes a list of CATParts and indicates if there are Drill Holes or Tap Holes with several Hole features inside.**

- ◆ **It can be applied :**
  - To all the CATParts of an existing Assembly.
  - To all the CATparts corresponding to a catalog content.

# Component Design (5/5)

### Resolve

- **A V5 component is often represented by a CATPart including a Design Table.**
- **It can be useful to resolve the component to avoid unwanted variant changes .**
  - Create a separate CATPart for each design table line.



**The macro *VariantResolutionForToolingComponent* resolves and saves the variants for a list of Tooling components (Parts only).**
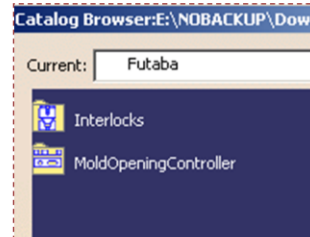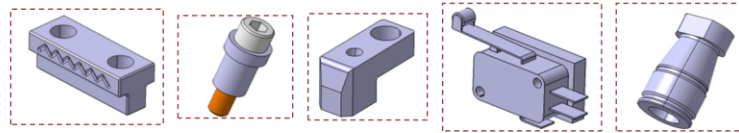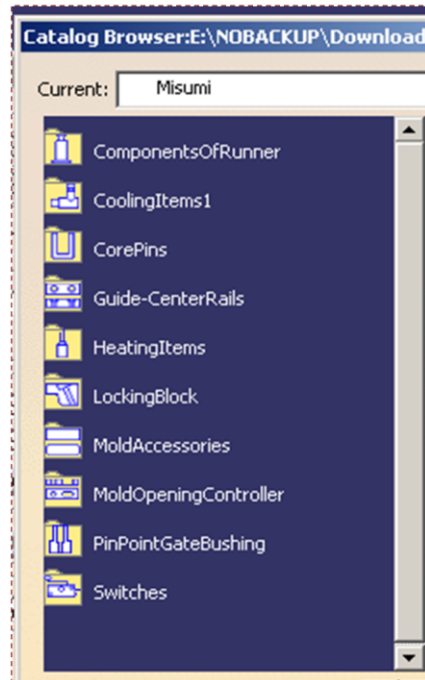
**The macro FreezingVariantForToolingComponents removes the design table, sets the Part Name and its property Nomenclature according to the current variant value.**

# Mold Tooling Design Catalogs – User Components (1/3)
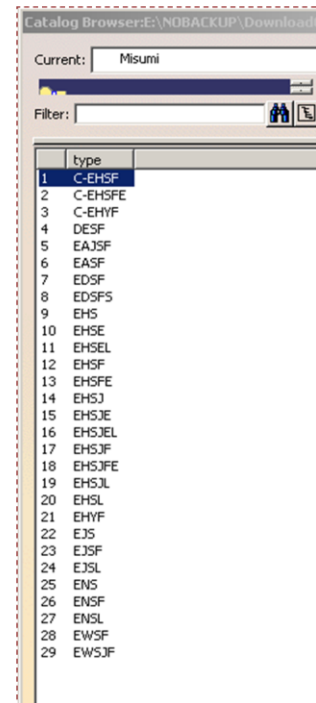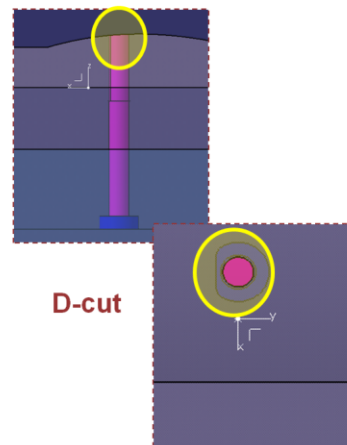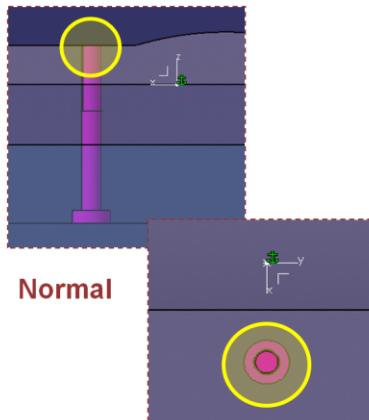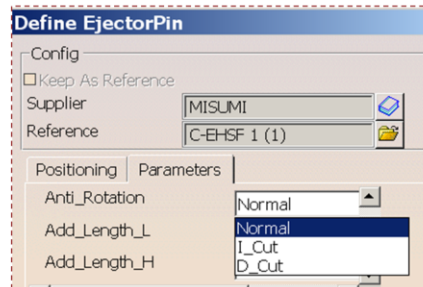
**Misumi / Futaba :**

◆ **New user components**

# Mold Tooling Design Catalogs – User Components (2/3)

❖ **Enriched contents :**

◆ **For instance there are now 29 ejector pins in Misumi catalogs.**
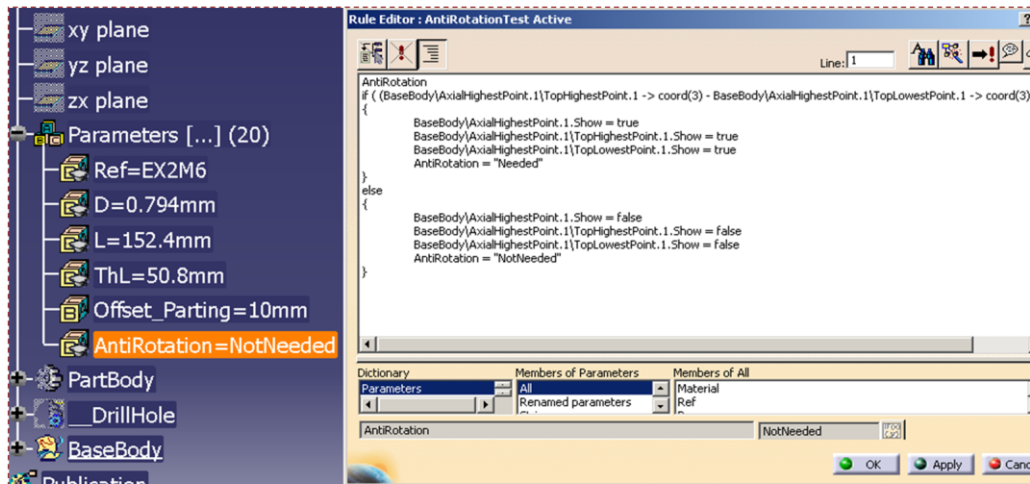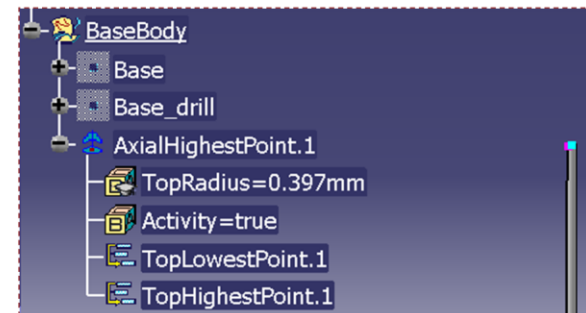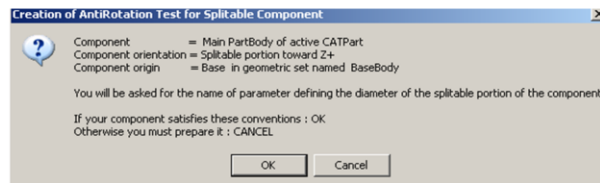
❖ **Ejectors : anti-rotation option :**

◆ **Normal/DCut/ICut for all Ejector and Ejector Pins.**



**Normal**

**D-cut**

# Mold Tooling Design Catalogs – User Components (3/3)

**The macro *AntiRotationActivePart* automatically creates in the active part the geometry (UDF), the rules and the parameter which states if anti-rotation is needed.**
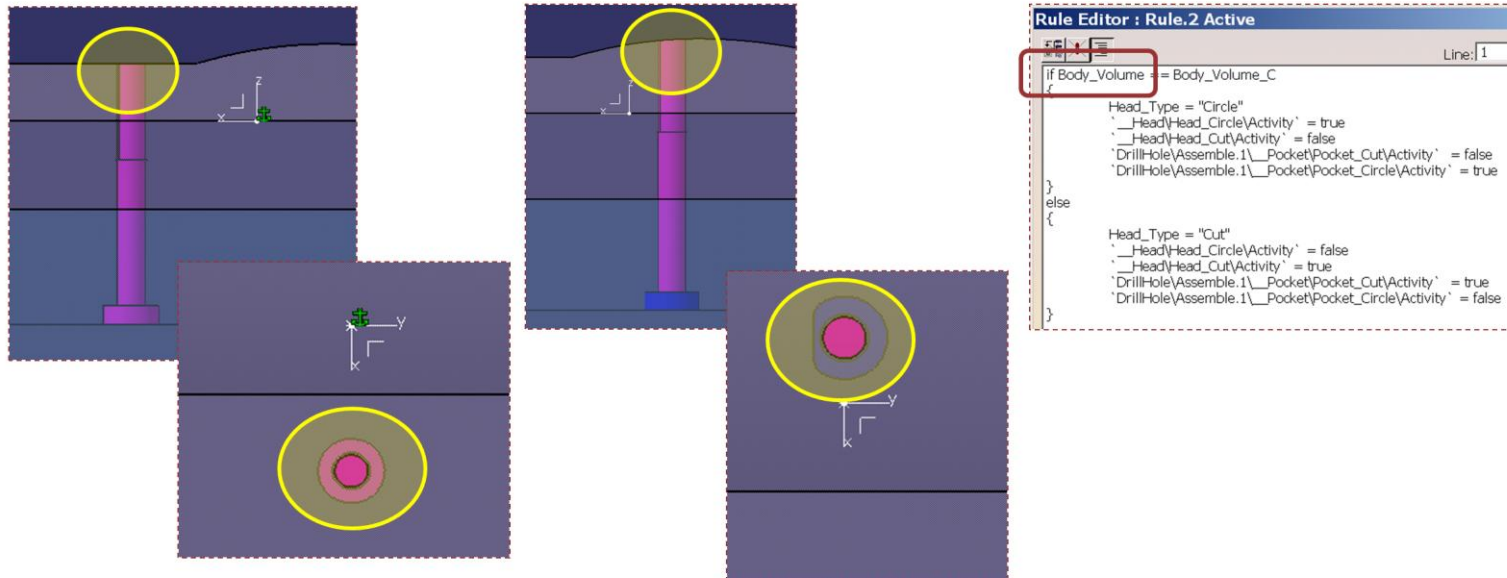
◆ **A KT1 license is needed.**







**Located in  … \intel_a\startup\Tooling\MTD**

# Smart Ejector Template

🔹 **Ejector with Auto-adjustable anti-rotation:**

◆ **Available in:**

- *…\intel_a\startup\components\MoldCatalog\UserComponent\Tooling\Samples.*

◆ **Configuration switch is automatic.**

◆ **Based on a Knowledgeware Rule.**

◆ **For enhanced component flexibility and tool design productivity.**



**Planar form area => no anti-rotation needed**

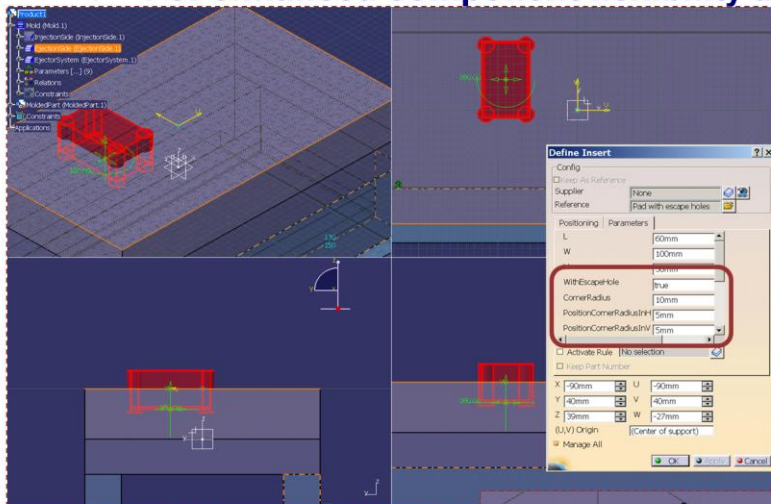**Move Ejector Non-planar form area => Ejector automatically switches to anti-rotation configuration**

# Smart Insert Template

**Insert with escape holes :**

- **Available in:**
  - *...\intel_a\startup\components\MoldCatalog\Insert\Block_EscapeHoles.*
- **Configuration switch on user decision (Component Parameter).**
- **Based on Knowledgeware Rule.**
- **For enhanced component flexibility and tool design productivity.**



• Escape Holes are defined
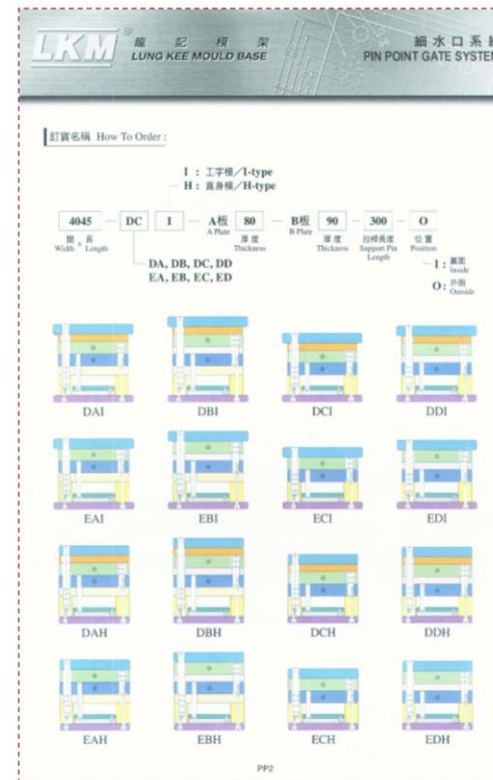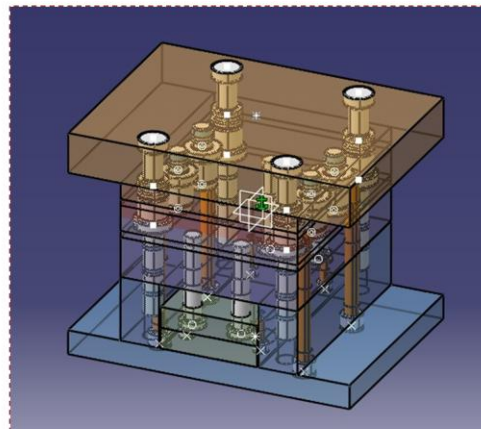• Corresponding dimensioning parameters are accessible
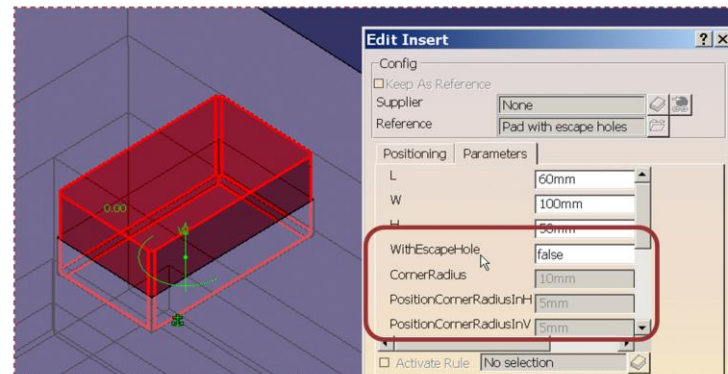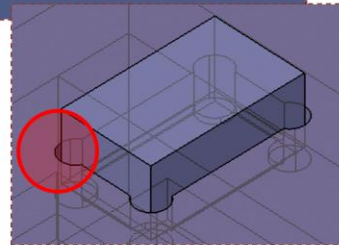
• Escape Holes are NOT defined
• Corresponding dimensioning parameters are NOT accessible

# Mold Tooling Design Catalogs – Equipped Mold Bases

♦ **The following Mold Bases are available:**

- ◆ **LKM**
- ◆ **Futaba**
- ◆ **DME, DME America**
- ◆ **HASCO**
- ◆ **Rabourdin**
- ◆ **Meusberger**
- ◆ **Pedrotti**
- ◆ **PCS**

♦ **Options equipped : YES/NO are available.**

# To Sum Up

**In this course you have learnt how to use workbench Mold Tooling Design to create a Plastic Injection Mold**

## CONGRATULATIONS !

**You have completed the training on Mold Tooling Design workbench**