

CATIA Tooling Design

Detailed Steps

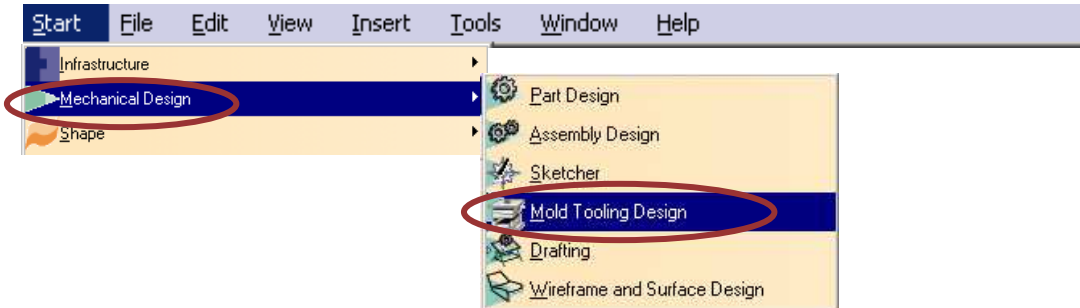
Table of Contents

1. Master Exercise: Phone Handset	3
2. Editing Components	15
3. Creating a Slider mechanism	19
4. Creating a Catalog of User Components	31

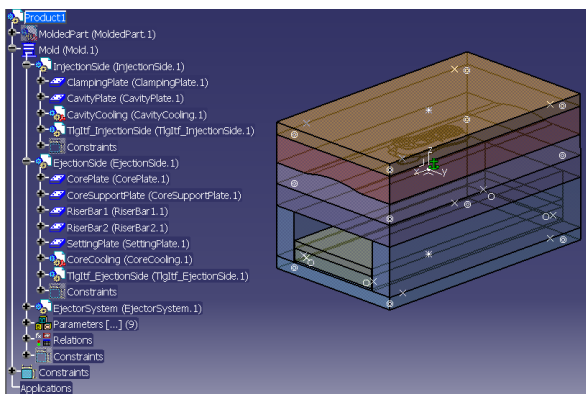
1. Master Exercise: Phone Handset

You will learn how to select and position Components.

1.1 - Enter workbench Tooling Design



1.2 - Select file: Master_Exercise_Step1.CATProduct

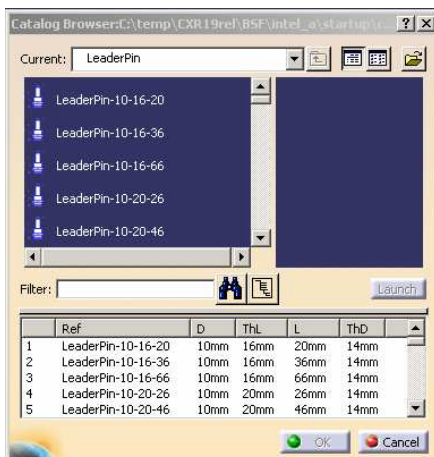


1.3 - Create Leader Pins :

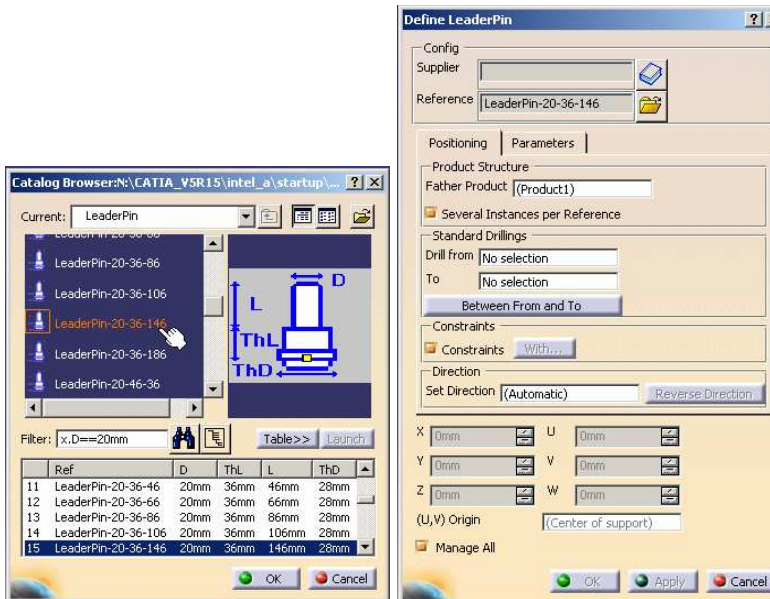
- Click icon *Add Leader Pin*. 
- In panel Define Leader Pin, click icon Catalog Browser



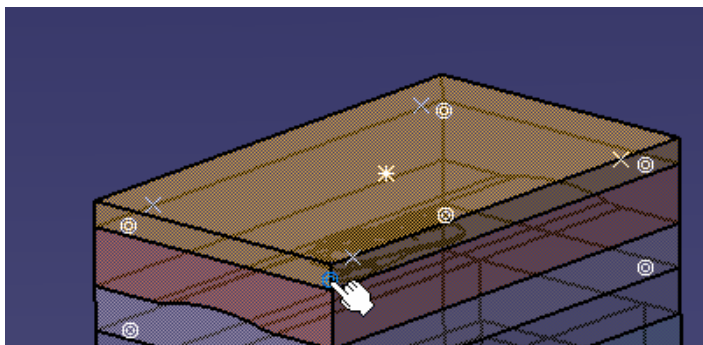
- Check that filter on Diameter is $x.D==20mm$



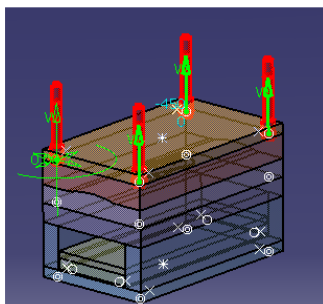
- Select reference LeaderPin-20-36-146 and click OK to validate this selection :



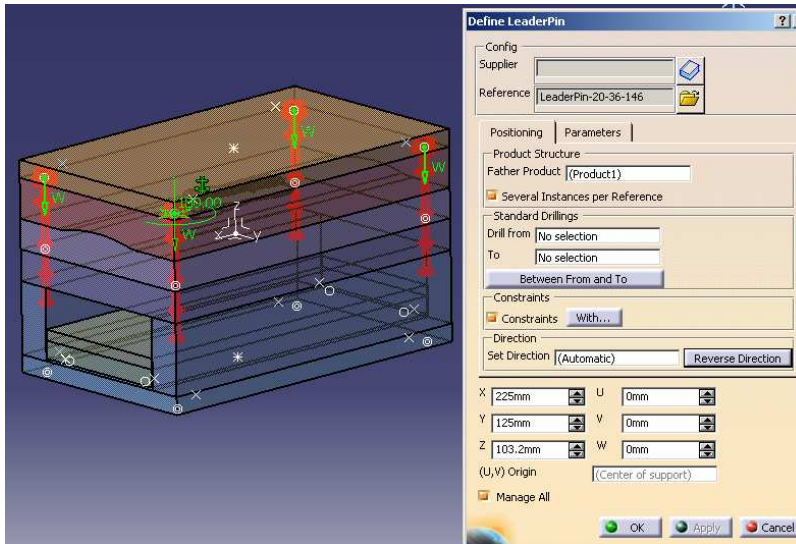
- Pick one of the predefined points located in the corner of the ClampingPlate in order to position the first instance of the Leader Pin :



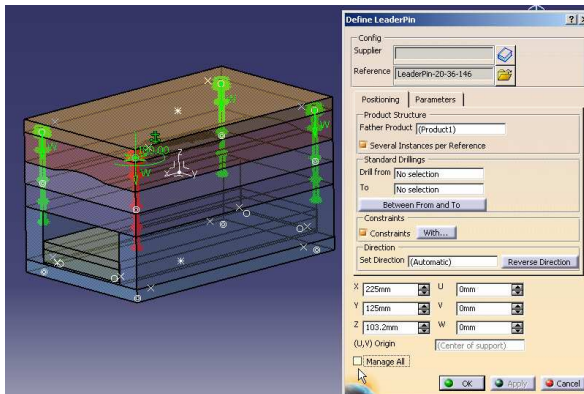
- You can see that the system has automatically recognized all four predefined points intended for positioning the Leader Pins. All instances are highlighted in red, which means that all of them are active : each further positioning operation will therefore apply to all of them simultaneously.



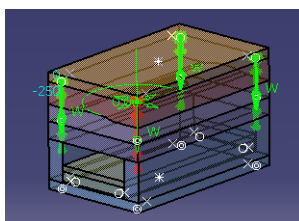
- Click button Reverse Direction to re-orientate the Leader Pins.



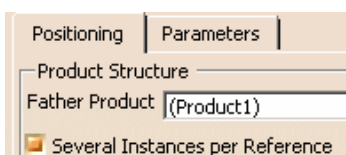
- Uncheck button *Manage All* in the dialog box. Only one instance stays active (red highlight), the other ones become inactive (they turn to green) : any positioning operation will then apply only to the current active instance.



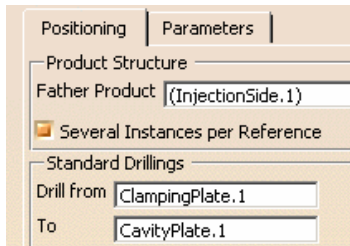
- Move the mouse over the vertical green arrow of another instance : this new instance becomes the active one (red highlight).



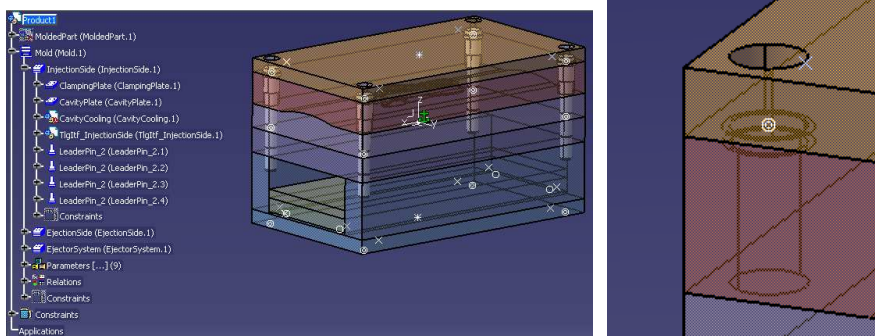
- Activate again button *Manage All* to make all instances active. Note that you can do any number of switch operations and positioning operations in order to reach the desired result for all instances. Note that option *Several Instances per Reference* is active : the four Leader Pins will therefore be created as instances of the same single reference.



- To define the Leader Pin drillings to be applied to the Mold Base, select *ClampingPlate* as plate *Drill From*, and *CavityPlate* as plate *To*. You can select them graphically or in the specification tree. Note that *Father Product* has automatically been set to *InjectionSide*, which is the father of selected *Drill From* plate *ClampingPlate*



- Click *OK* in the dialog box to validate the creation. You can see that all four instances of the Leader Pin have been created, together with their associated holes in the plates. They have been created in the specification tree in father sub-assembly *InjectionSide*.

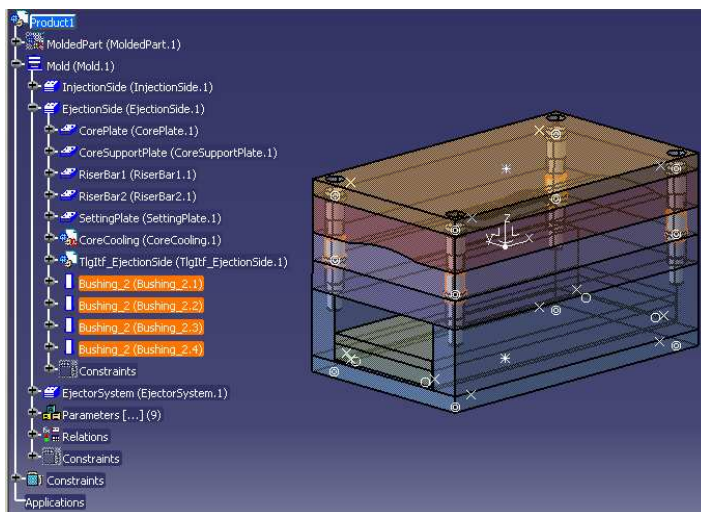
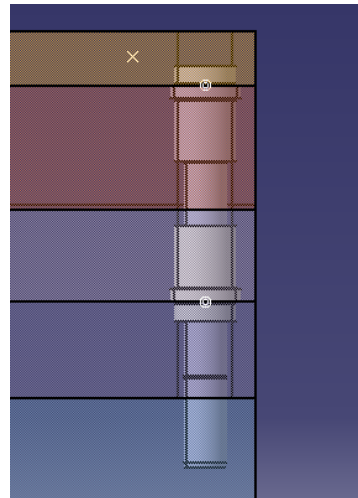
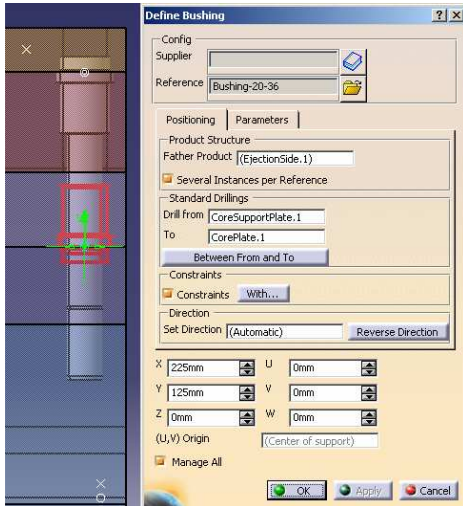
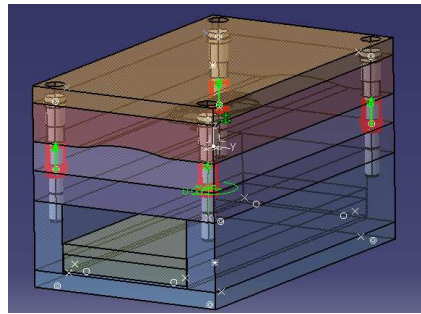
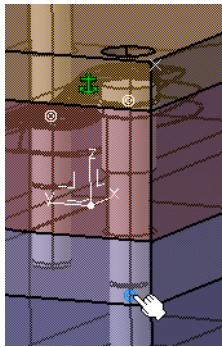


- Note also the automatic creation of Technological Results in the *ClampingPlate* and *CavityPlate*, conveying the technological information of the *DrillHoles* of the *LeaderPins* impacting the plate.



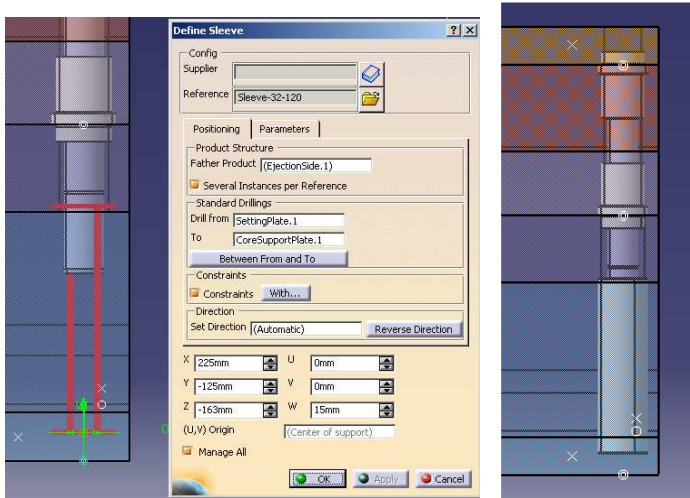
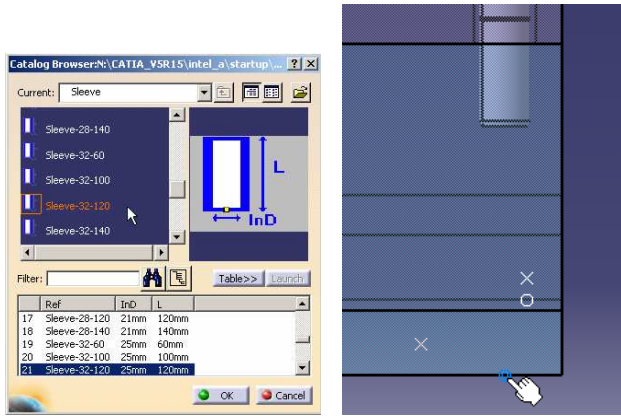
1.4 - Proceed similarly to create four Bushings fitting these Leader Pins, using the predefined points located in the Core Plate for this purpose. Select reference Bushing-20-36. Drill from CoreSupportPlate to CorePlate.



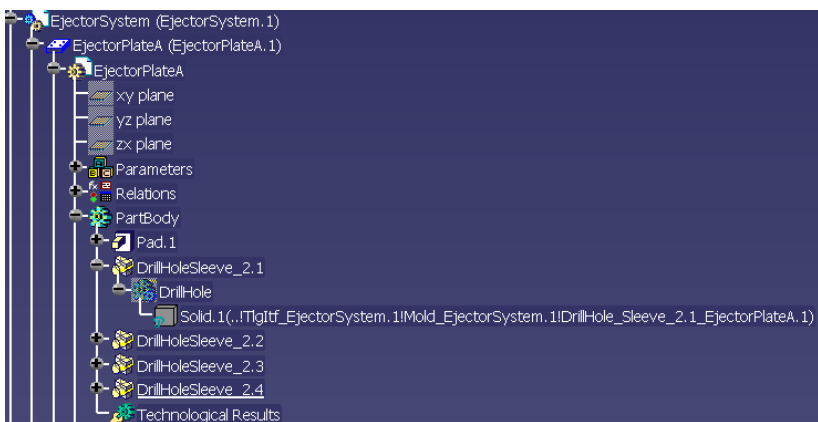
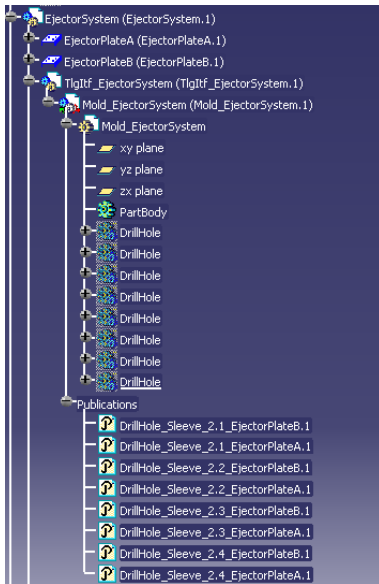


1.5 - Proceed similarly to create the appropriate Sleeves for these Leader Pins. Select reference Sleeve-32-120. Drill from SettingPlate to CoreSupportPlate and adjust W value to 15 mm (note that W position is adjusted in a single operation for all Sleeve instances, thanks to option Manage All).



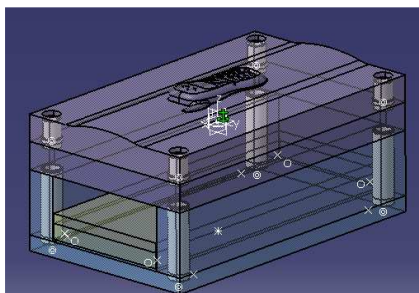


1.6 - Note in sub-assembly EjectorSystem that a new Part called Mold_EjectorSystem has automatically been created in a new sub-product called Tlgtf_EjectorSystem. This Part contains a copy of the drilling Body (DrillHole) of each Sleeve on each of the two plates of sub-assembly EjectorSystem (EjectorPlateA and EjectorPlateB). If you look inside e.g. EjectorPlateA, you can see that these Bodies are used for effectively performing the drilling operation (Assemble feature). This mechanism ensures that sub-assembly EjectorSystem does not have a contextual dependency to the Sleeve Components (located in a different sub-assembly) and can therefore be opened independently if required (no need for the complete Mold assembly).

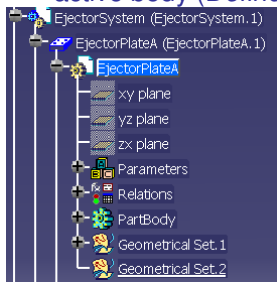


1.7 - Create Ejector Pins :

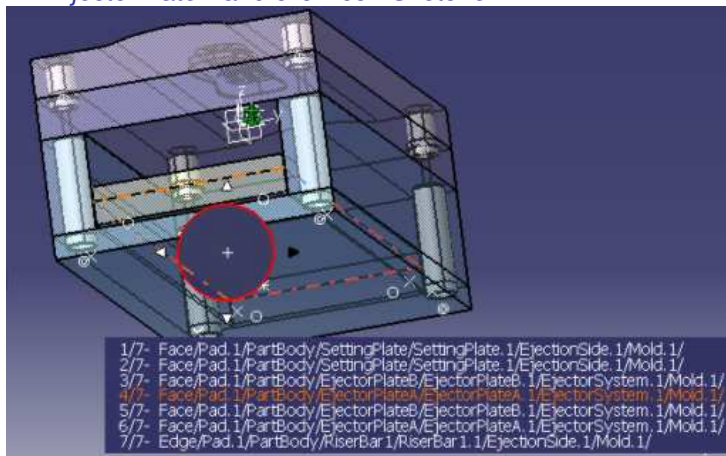
- To facilitate the visualization and the positioning of the Ejector Pins, hide InjectionSide.



- Double-click part *EjectorPlateA* in the tree. Make sure that *Geometrical Set.2* is the current active body (*Define In Work Object*).



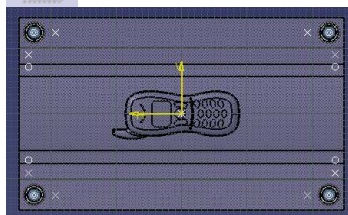
- Using the keyboard up/down arrows to “get through” the lower Plates, select the bottom face of *EjectorPlateA* and click icon *Sketcher*.



- In toolbar Sketch tools, make sure option Snap To Point is de-activated.

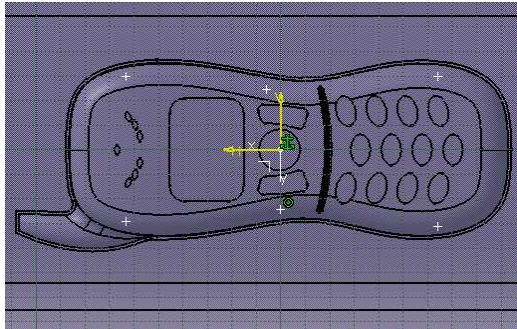


- If necessary, click icon Normal View to have a view from the top of the Mold.

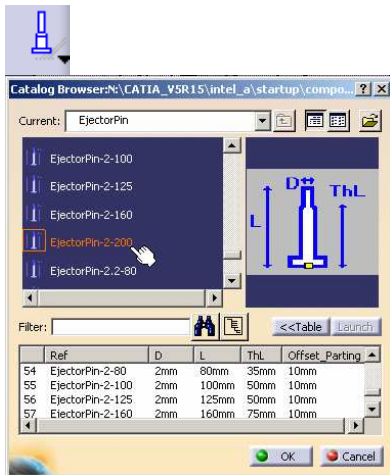


- Double-click icon *Point*, define several points approximately as shown, then exit Sketcher. These points are going to be used to locate Ejector Pins.

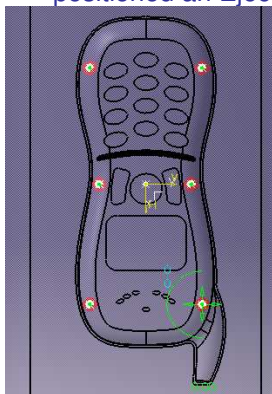




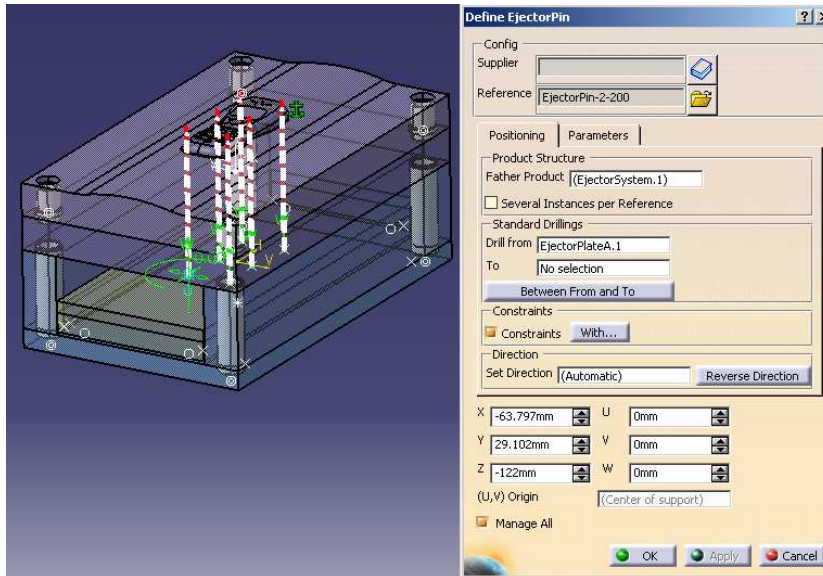
- Double-click top Product to go back to workbench *Tooling Design*.
- Click icon *Add Ejector Pin* and select reference *EjectorPin-2-200* (erase predefined catalog filter on diameter *D* if needed).



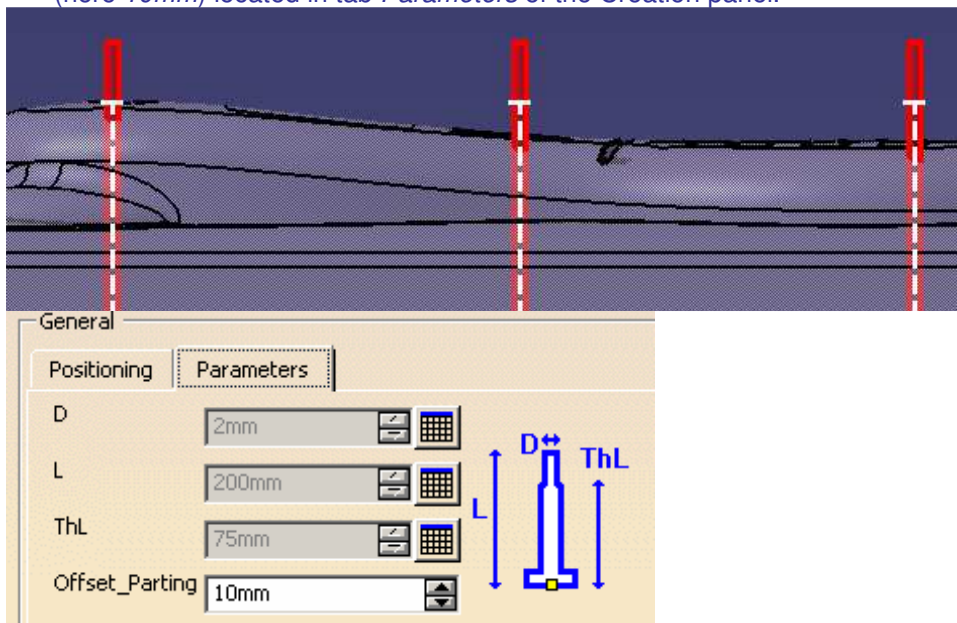
- Pick one of the points of the sketch previously created. You can see that the system has positioned an Ejector Pin on each point of the Sketch.



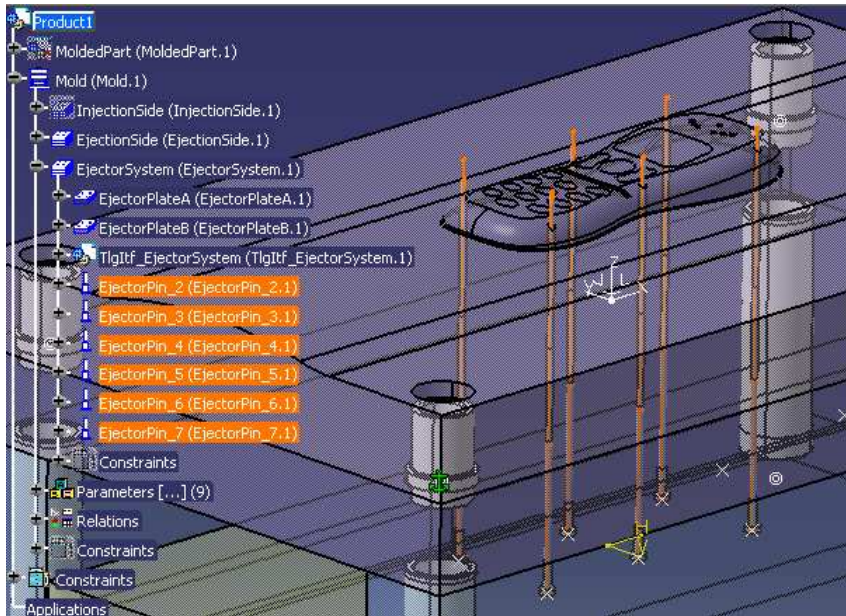
- Note that option *Several Instances per Reference* is not active (the value shown is the one defined in the Settings). This will ensure that each *EjectorPin* is created as a reference and can therefore be handled individually for splitting purposes later on.



- Note that *Drill From* has automatically been set to *EjectorPlateA* because the positioning sketch belongs to this Part. Set by yourself : *To = CorePlate*.
- Zoom onto the top of the Ejectors. The white dashed line in the preview shows the height of the clearance hole associated to each Ejector Pin. The length of this hole is set to a predefined value for the rough, uncut Ejector Pin and therefore needs to be adjusted : the Ejector Pin would not fit in closely with the Core Surface. The length of the fitting hole will later be automatically adjusted by the Split operation to the value defined in parameter *Offset_Parting* (here 10mm) located in tab *Parameters* of the Creation panel.



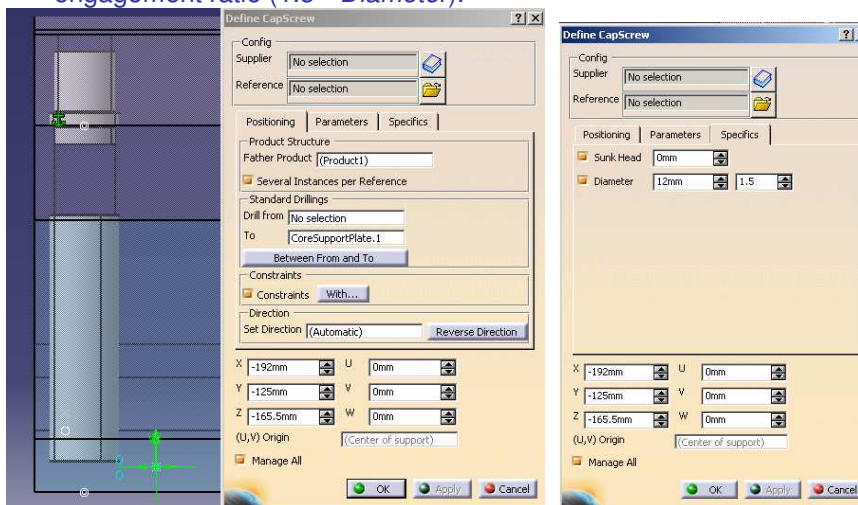
- Validate creation of the Ejector Pins. All the references have been created in sub-assembly *EjectorSystem*.



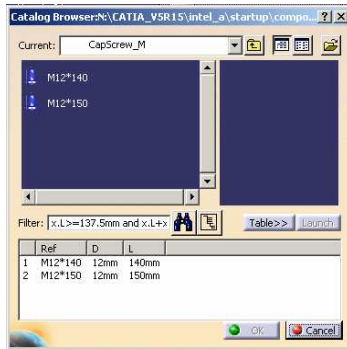
1.8 - Insert Fixing Component : Cap Screw :



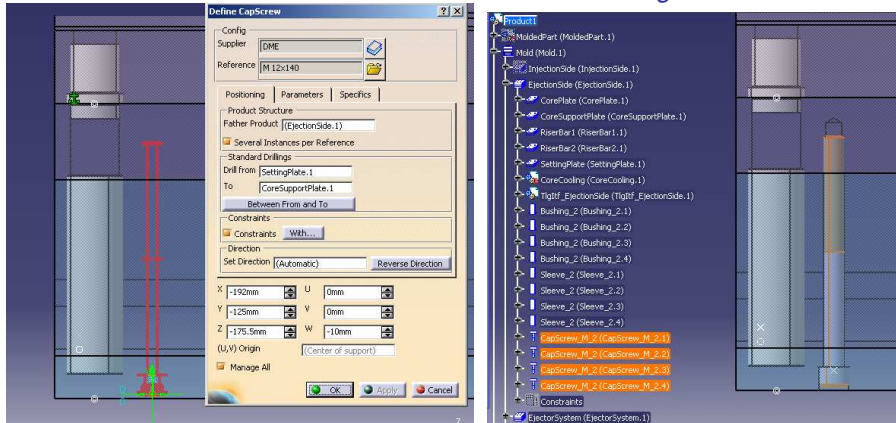
- Click icon Add Cap Screw
- Select one of the predefined 3D points for Cap Screws located in the *SettingPlate*. In panel *Define CapScrew*, select field *To*, then pick *CoreSupportPlate* in the 3D viewer. In tab *Specifics* activate *Diameter* and enter value *12mm*. Keep default value for minimum thread engagement ratio ($1.5 * Diameter$).



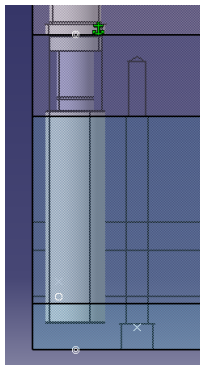
- Click icon Catalog in the dialog box. Double-click provider *Dme*, then type *CapScrew_M*. You can see that the system has automatically pre-selected a subset of Screw variants by using a filter on their length and their diameter. This filter has been computed by using the elements previously defined : positioning point, target drilled plate (*CoreSupportPlate*), diameter, minimum thread engagement ratio.



- Select reference *M12*140*. Define *Drill From = SettingPlate*. Set *W = -10mm*. Validate creation.



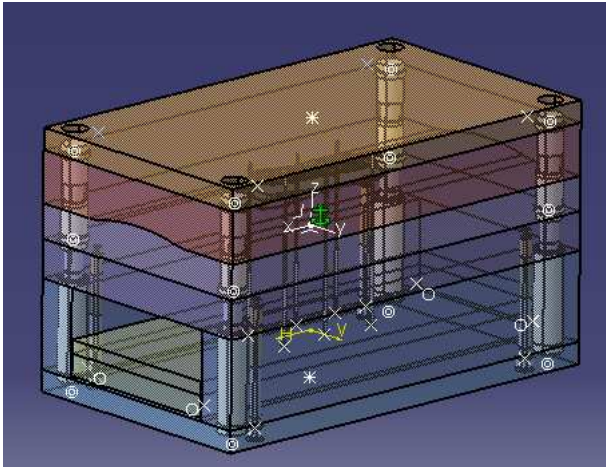
- Hide one of the screws. You can see that *SettingPlate* and *Risers* have been drilled with standard holes, while the last plate (*CoreSupportPlate*) has been bored with a threaded hole.



- Close the document without saving it if you want to use existing prepared data for the next step, or keep it opened if you want to continue with it.

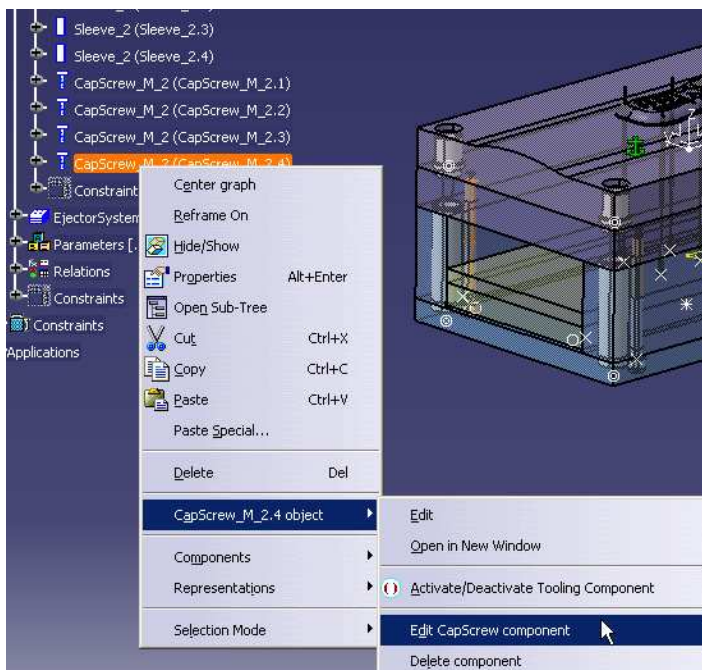
2. Editing Components

2.1 - If you have not kept the result of the previous step, open existing Product : *Master_Exercise_Step2_Start.CATProduct*

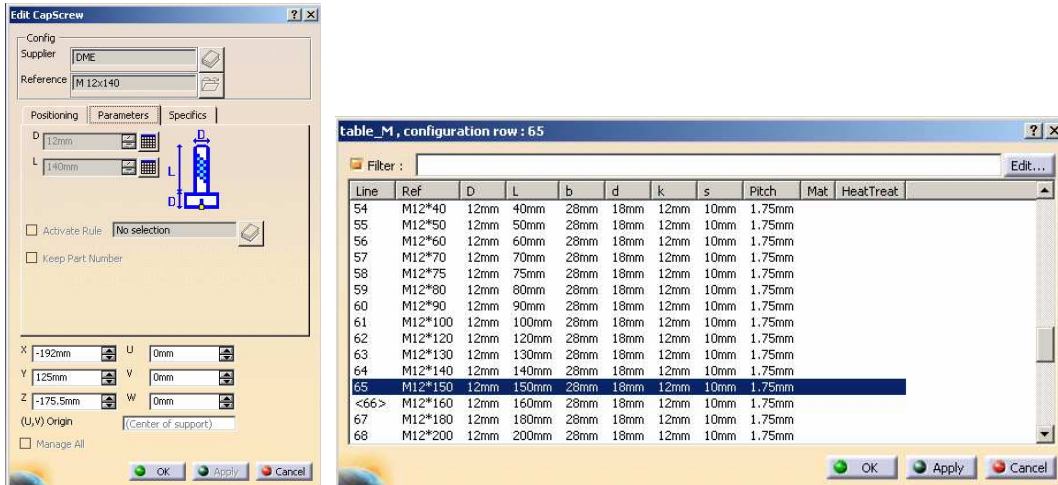


2.2 - Change the size of the Cap Screws located in the Setting Plate :

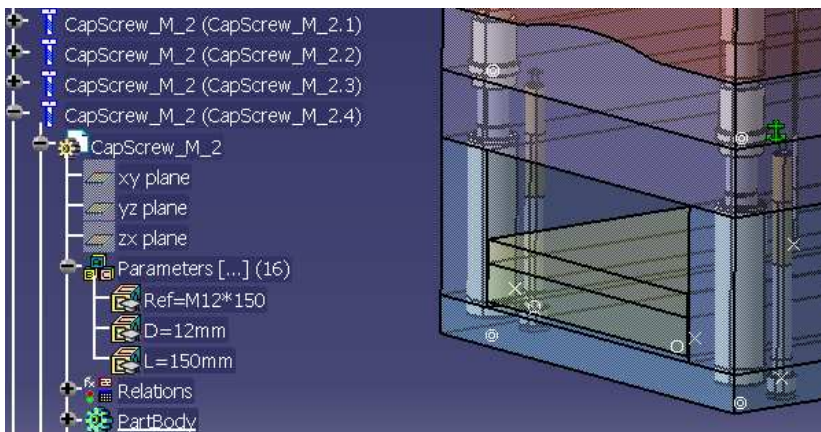
- Select a CapScrew instance in the specification tree (EjectionSide), activate its contextual menu and select function Edit CapScrew component :



- Select tab *Parameters* in dialog box *Edit CapScrew*, activate design table icon corresponding to parameter L (length) and select reference *M12*150*

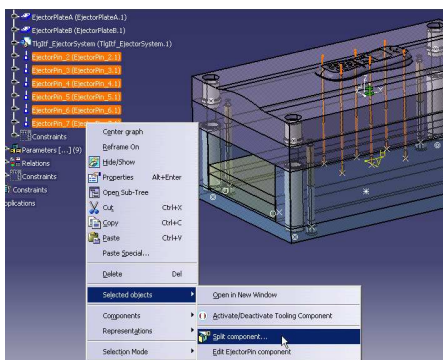


- Validate modification : you can see that all four instances of the Cap Screw have been updated simultaneously.

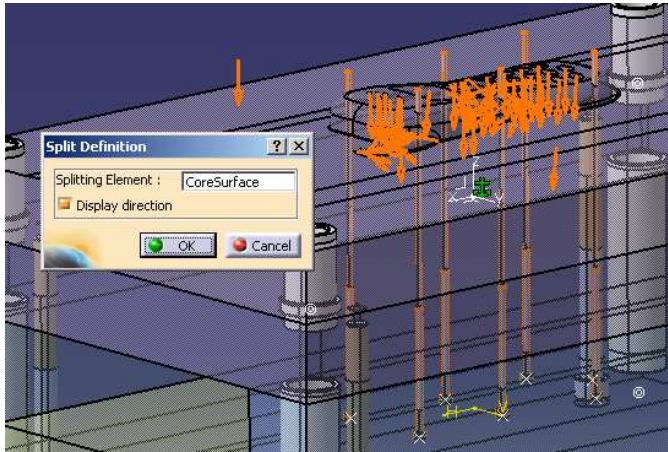


2.3 - Split the Ejector Pins :

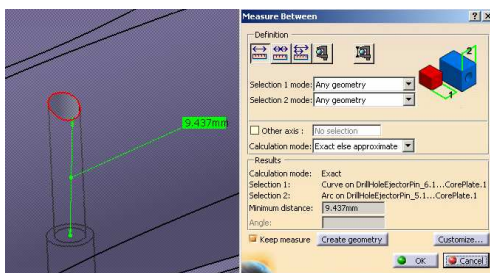
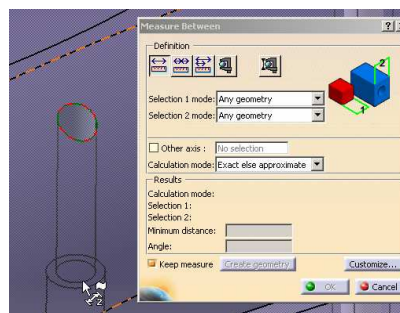
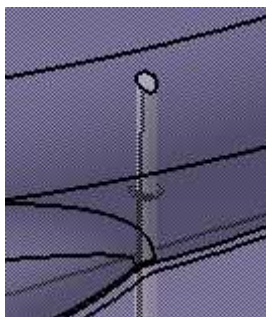
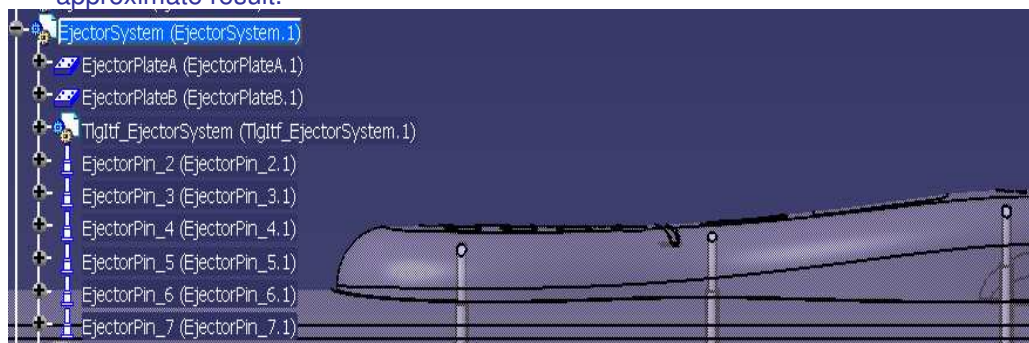
- Multi-select the Ejector Pins in the specification tree and activate function *Split Component* in the contextual menu.



- Keep default value proposed by the system (CoreSurface) as splitting element. If necessary, reverse the splitting orientation by clicking one of the arrows in the 3D viewer in order to have them point downwards to keep the appropriate side of the Ejector Pins after split.

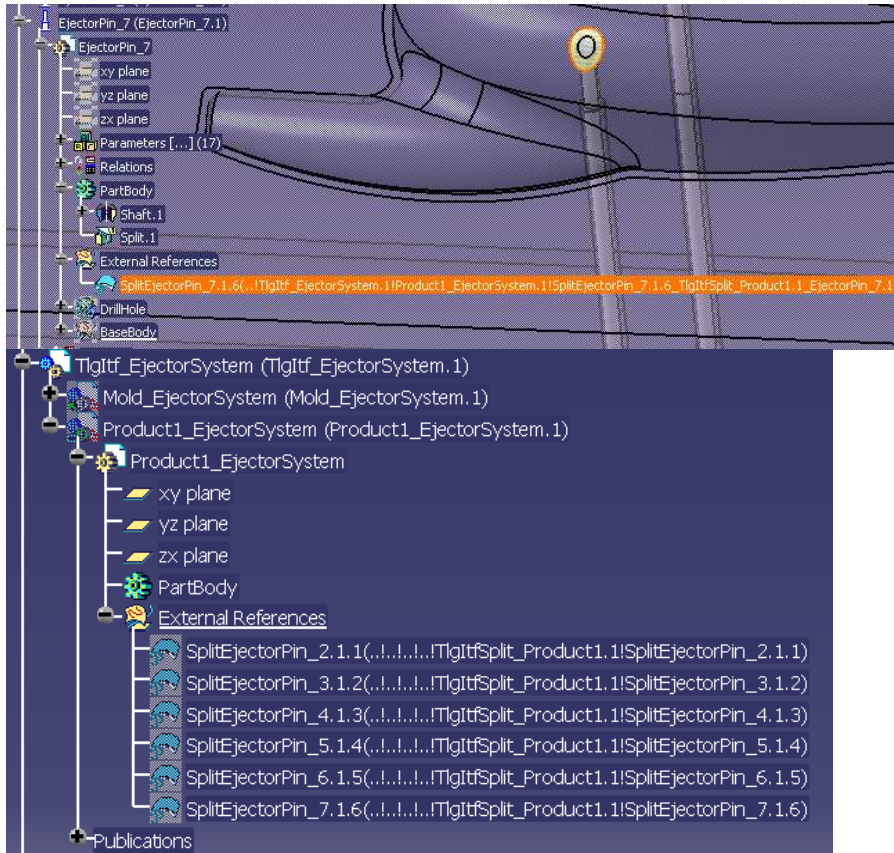


- Validate and zoom onto Ejector Pins area to check the result. Note that Split operation has automatically adjusted the fitting length of each Ejector Pin to take into account the value defined in parameter *Offset Parting*. You can measure it with function *Measure Between*: fitting length is computed using intersection of the center of the Ejector Pin with the Core Plate, but you can select the top edge of the drilling in the Core Plate in order to get an approximate result.



- Show External Reference element in the EjectorPin and note that operation Split Component has automatically restricted the splitting surface of the Ejector to a small area around it (it has not used the full surface). The restricted surface in each Ejector comes from a second Part

automatically generated by the system in sub-assembly Tlgltf_EjectorSystem and containing the reference Split surfaces generated by the system in the new Part TlgltfSplit generated at root level.



- Close the document without saving it.

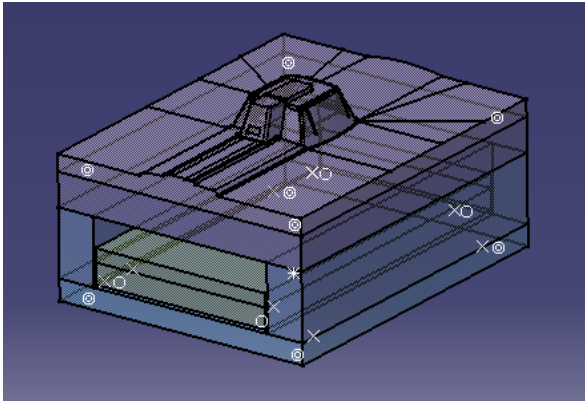
The final result is available in : \MasterExercise\Step2\End\Product1.CATProduct

3. Creating a Slider mechanism

In this step, you will:

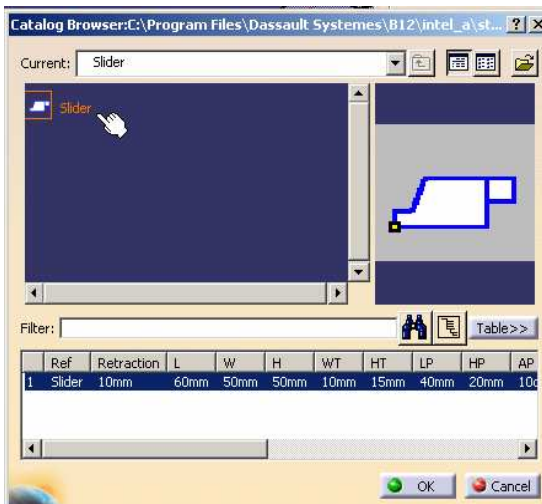
- Create a Slider component
- Create the corresponding Retainers
- Create the corresponding Angle Pin
- Create the fixing screws and perform the relevant drillings

3.1 - Open existing Product: *Master_Exercise_Step3_Start.CATProduct*

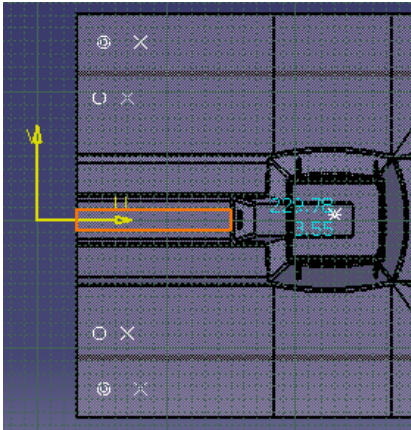


3.2 - Create a Slider :

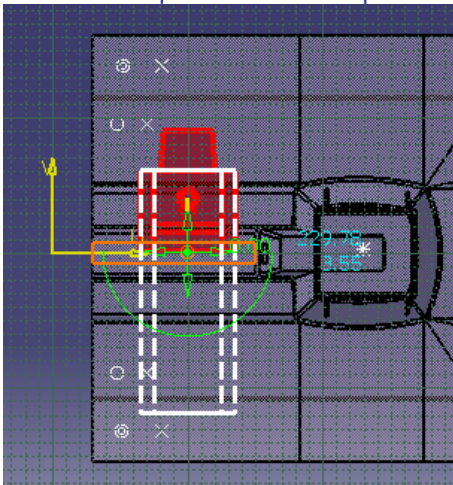
Click icon *Add Slider*, click catalog icon in panel *Define Slider*, then double-click the Slider listed in the catalog window.



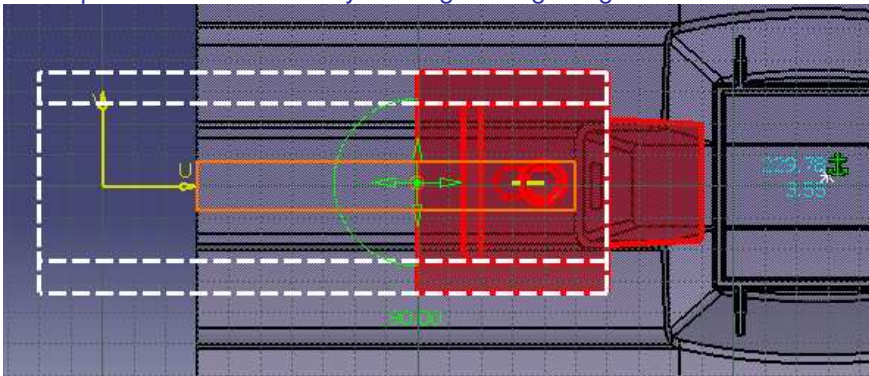
Click the top planar face of the Core Plate located in front of the rectangular hole in the area where the Slider mechanism is required.



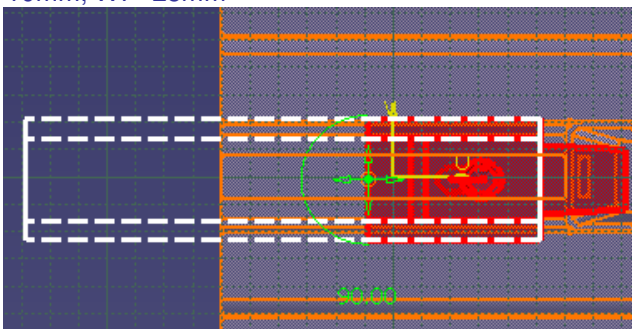
The system has switched to the sketcher-like view for positioning the component. Pick a point anywhere on the previous face : a preview of the Slider is displayed.



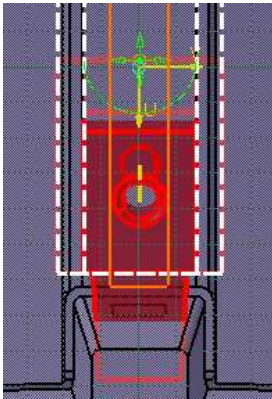
Adjust the position of the Slider by rotating it using the green half-circle.



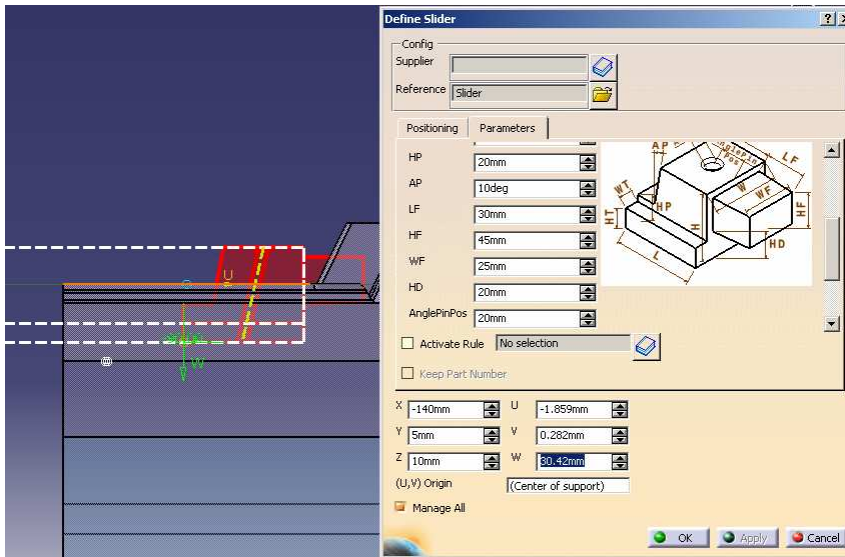
Display tab *Parameters* in dialog box *Define Slider*. Check the meaning of the following parameters in the panel picture and set them to the values mentioned : W=30mm, WT=7mm, HT=10mm, WF=25mm



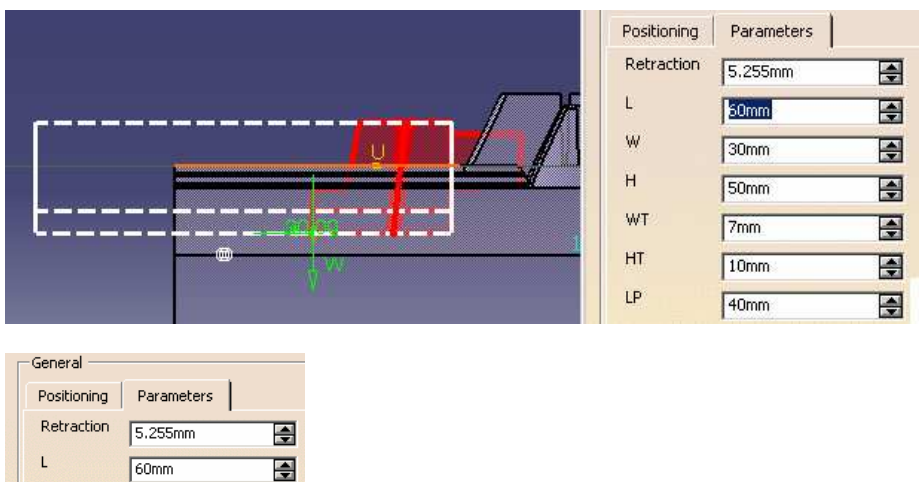
Adjust the position to X=-140mm, Y= 5 mm.



Switch to Left View. Adjust Z=10mm.



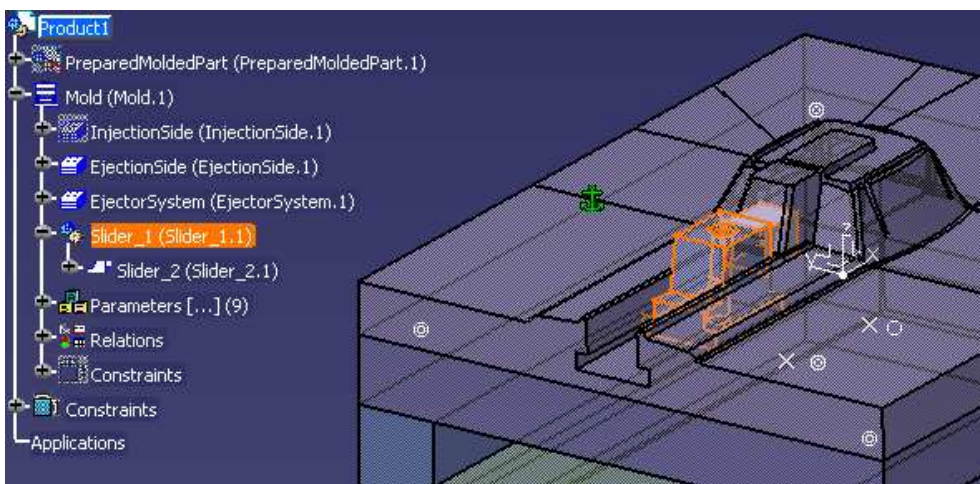
Enter *Retraction* = 5mm. Note that the angle for the Angle Pin hole is automatically modified. Note also that the system has automatically adjusted the value of *Retraction* to the closest appropriate value in order to keep this angle to a rounded value.



In tab Positioning, define the plates to drill : From CorePlate To CavityPlate.
 COPYRIGHT DASSAULT SYSTEMES

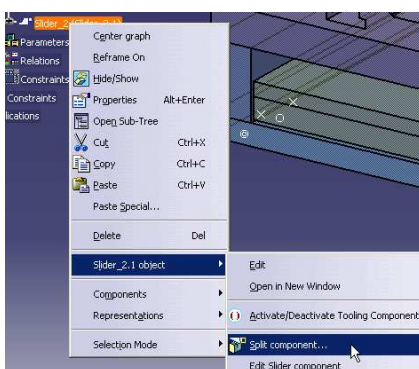


Validate the creation of the Slider. Note that it has been created in a new assembly sub-component located in the tree at the same level as the main sub-assemblies of the Mold.

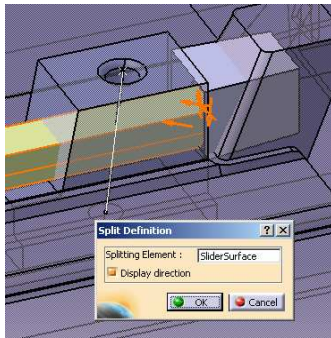


3.3 - Split the form part of the Slider :

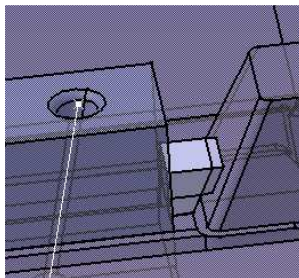
- In *PreparedMoldedPart*, show prepared surface *SliderSurface* located in Geometrical Set *PreparedSurfaces*.
- Right-click *Slider_2.1* object and select *Split component* command.



Select surface *SliderSurface*. Check that the direction arrows are oriented as shown to ensure that the correct side of the form part will be kept after the Split operation.

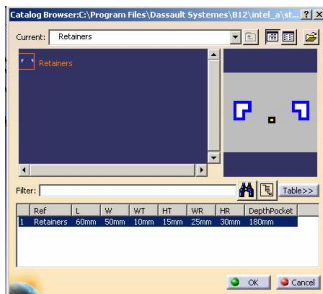


Validate split operation. Hide *PreparedMoldedPart*.

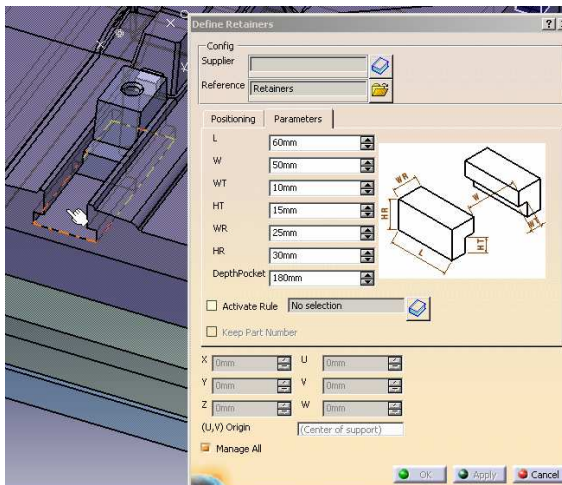


3.4 - Create the Retainers :

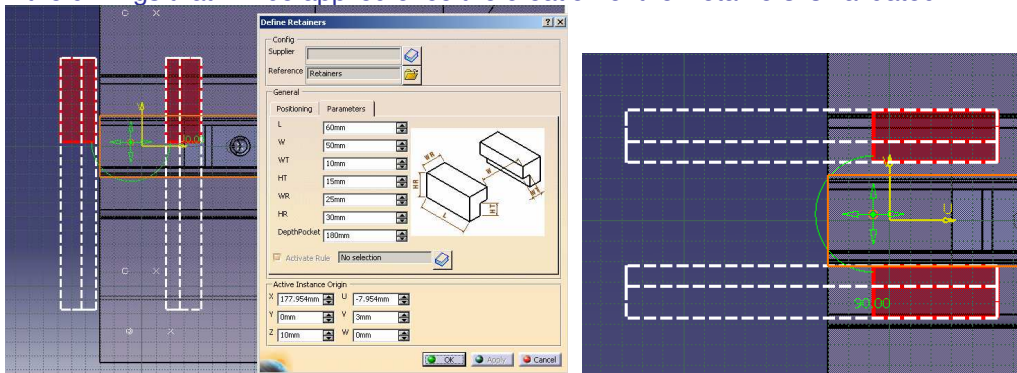
Click icon *Add Retainers*, click catalog icon in panel *Define Retainers*, then double-click the Retainers listed in dialog box *Define Retainers*.



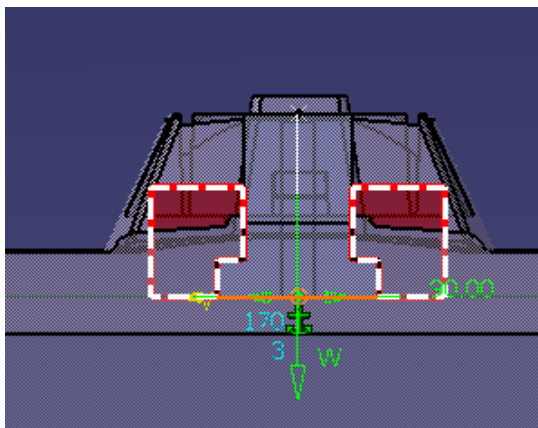
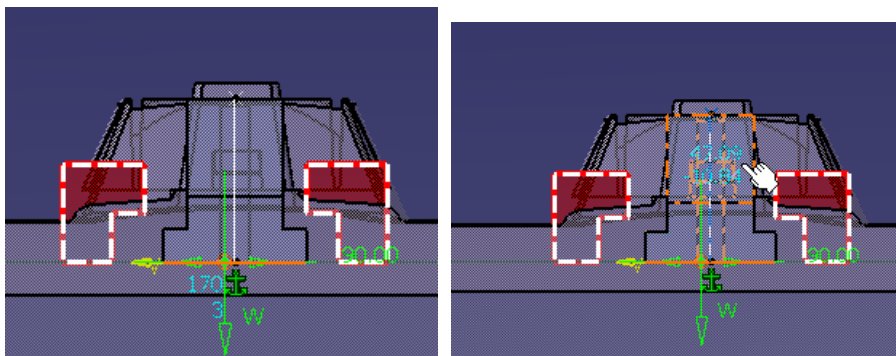
Pick the planar face on the Core Plate where the Slider will move (in contact with bottom of Slider)



Pick a point anywhere on this face to position the Retainers. A preview of the component is displayed. Adjust the position (rotation) by using the green half-circle. The white dashed lines show the drillings that will be applied once the creation of the Retainers is validated.

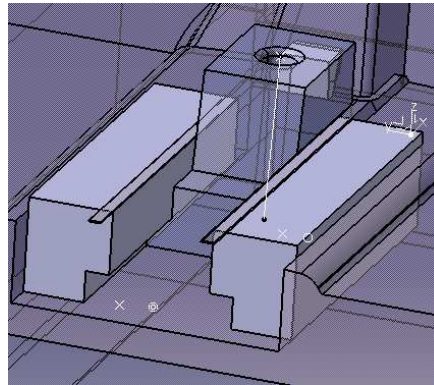
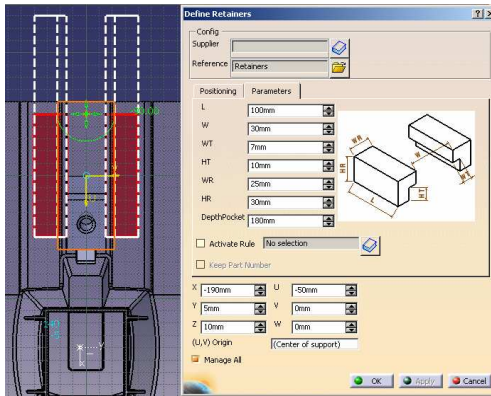


Switch to *Back View*. Make sure that no object is selected (no orange highlight). Make sure also that you will not pick the positioning grid. Then click the Slider in the 3D viewer : some of the dimensions of the Retainers are automatically adjusted to fit the Slider.

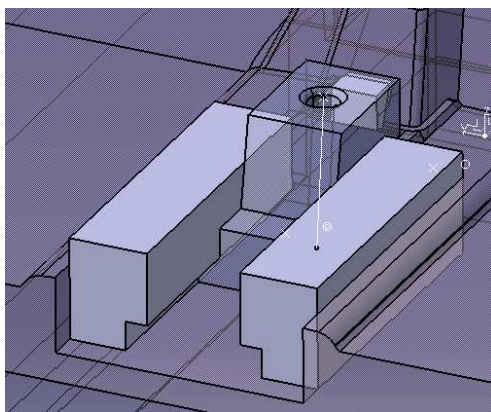


Switch to *Top View*. Define plates Drill from *CorePlate* To *CavityPlate*. Adjust the position and the length of the Retainers to ensure adequate contact length between the Slider and the Retainers. Define for example : L=100mm, X=-190mm, Y=5mm, Z=10mm. Validate creation.



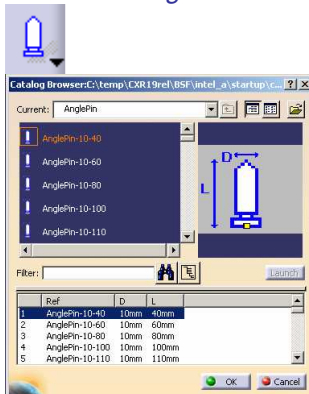


You can see that the height of the *Retainers* is not adequate because some material is left above them. Edit *Retainers* Component and set parameter *HR* = 35 mm for example. Validate modification and check result.

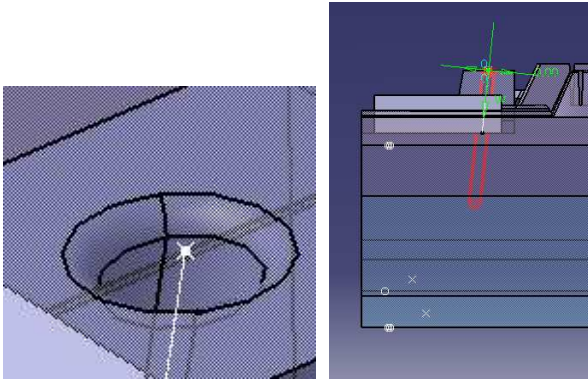


3.5 - Create the Angle Pin :

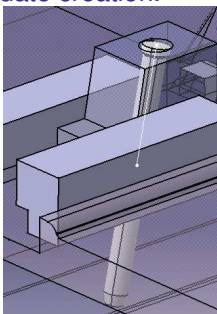
Click icon *Add Angle Pin*. Click catalog icon in panel *Define AnglePin*. Select *AnglePin-10-110*



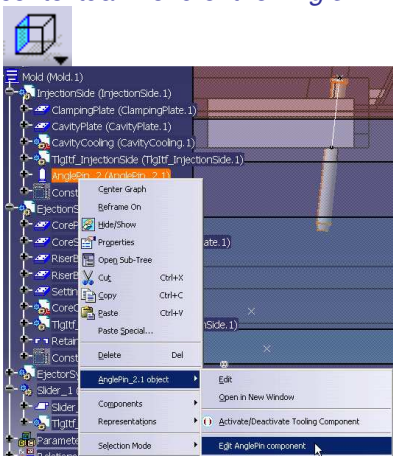
To position the *Angle Pin*, select the line, then the point which were automatically created when the *Slider* was instantiated. Picking the line allows to define the adequate orientation for the Angle Pin instead of positioning it vertically.



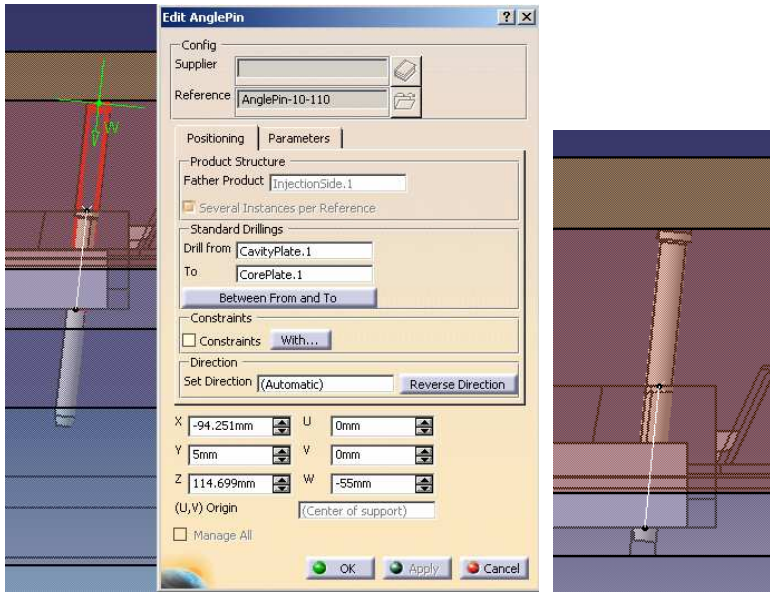
Drill *From CavityPlate To CorePlate*. The *W* position will be adjusted later via *Edit* capability. Validate creation.



Show complete *InjectionSide*, switch to *Left View*. Activate function *Edit AnglePin Component* in the contextual menu of the *Angle Pin*.

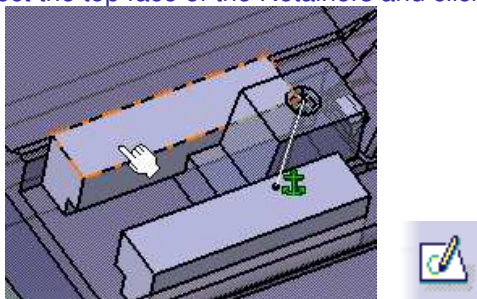


Adjust *W* position of the *Angle Pin*. Set e.g. *W* = -55 mm. Validate.



3.6 - Create the Screws to fix the Retainers :

Double-click Part *Retainers* to make it active and create inside a new Geometrical Set. Select the top face of the Retainers and click icon *Sketcher*.



Create four points to be used as locations for the screws. Define constraints to the sides of the Retainers (choose the value).

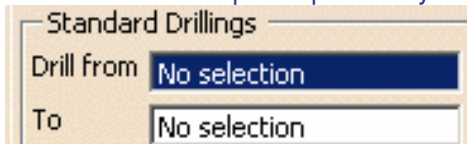
Exit Sketcher.

Double-click top Product in the tree to go back to workbench *Tooling Design*.

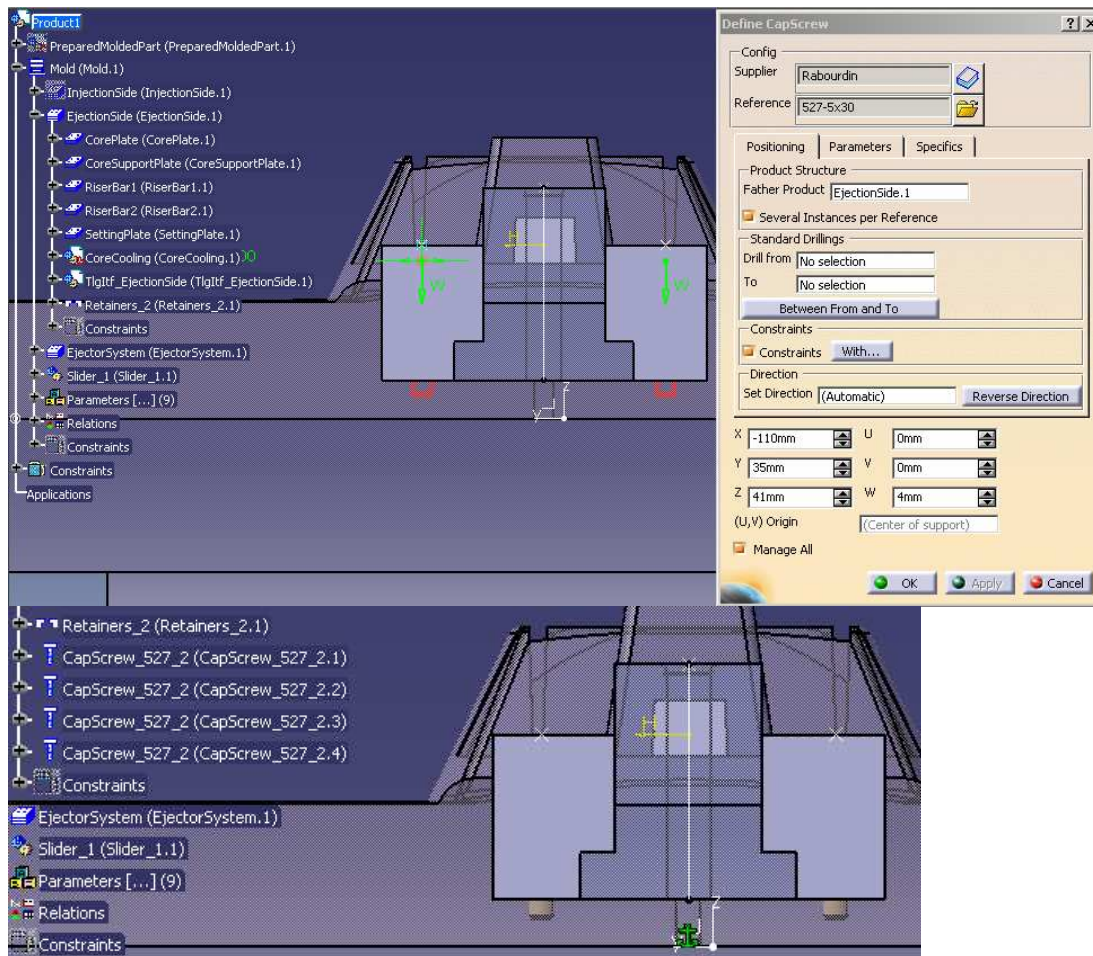
Click icon *Add CapScrew*.



Click one of the sketch points previously defined. Select field *Drill From* to make it empty.



Click icon *Catalog* in the dialog box. Select provider *Rabourdin*, type *CapScrew_527*, variant *527-5x30*. Set *W = 4mm*. Select *EjectionSide* as *Father Product*. Validate creation.

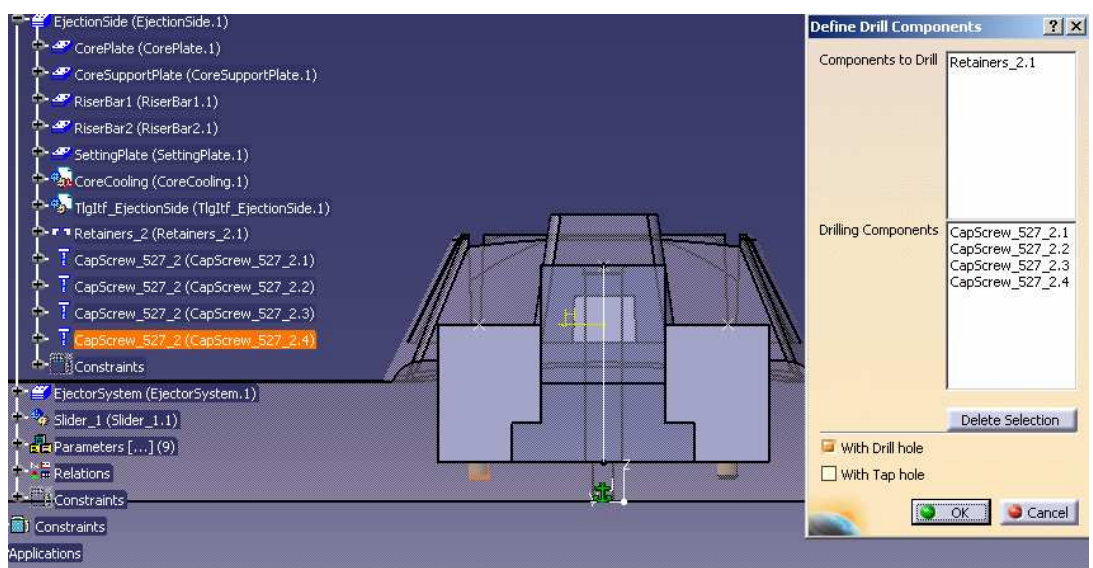


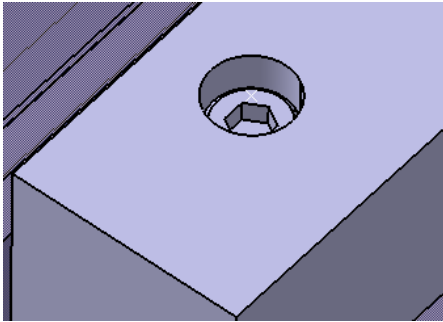
3.7 - Create the drillings for the Cap Screws :

- Click icon Drill a list of Components...

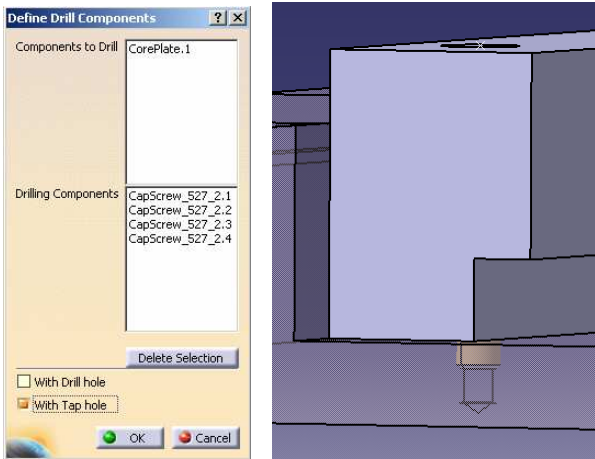


Select *Component to Drill* = *Retainers*. Pick successively all four *Cap Screws* as *Drilling Components*. Check that the active option is *With DrillHole*. Validate.

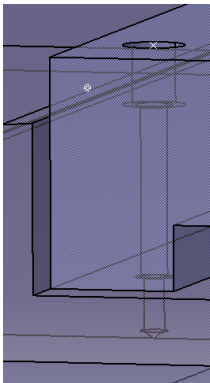




- Similarly, use again *the same function* to drill the *CorePlate* with the *TapHole* of all the *CapScrews*

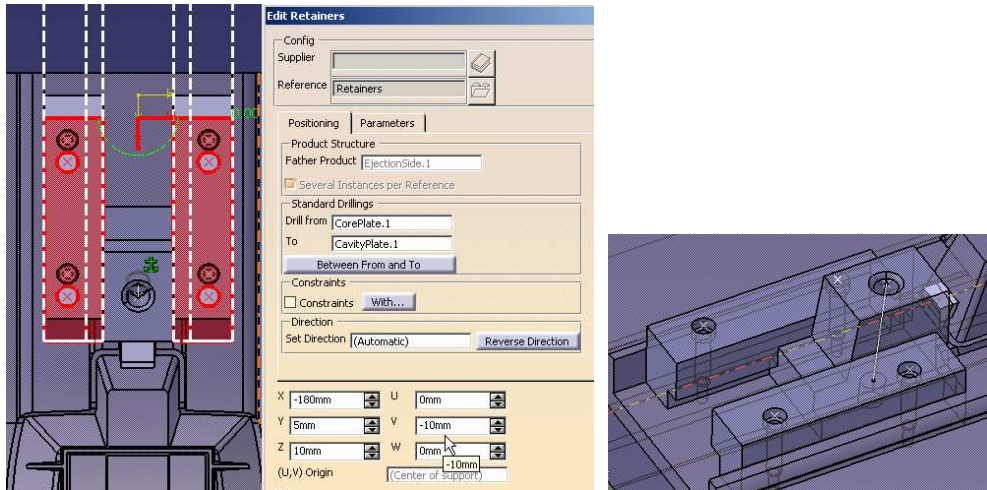


- Put the *CapScrews* in Noshow. Define some transparency in the graphic properties of the *Retainers*. You can now better check the drillings in the *Retainers* and *CorePlate*.



3.8 - Switch to Top View. Select Edit Retainers Component in the contextual menu of the Retainers. Move the Retainers along direction V. Validate the operation. Check that the position of the drillings is up-to-date.





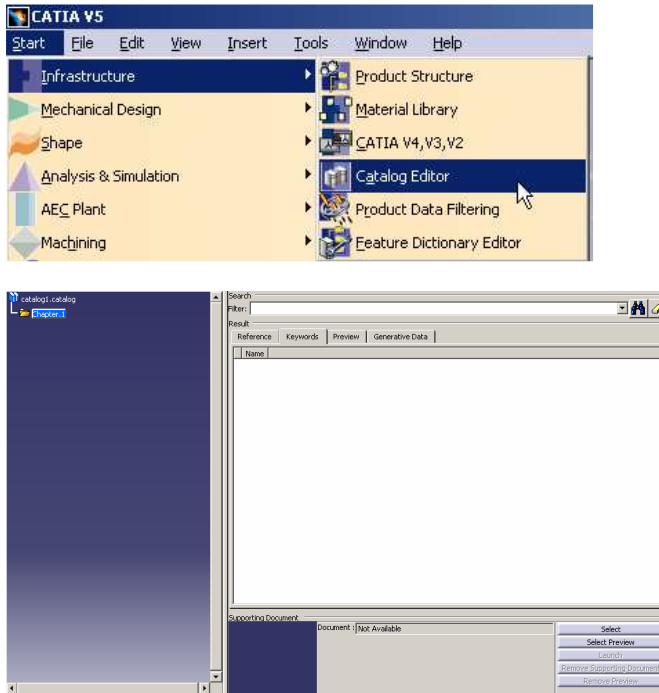
Close the document without saving it.Final result is available in :
 \MasterExercise\Step3\End\Product1.CATProduct

4. Creating a Catalog of User Components

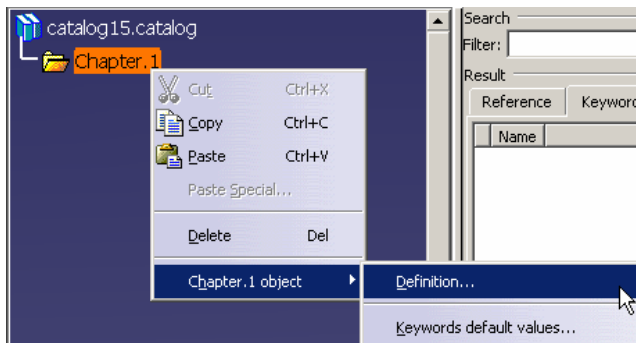
In this step, you will:

- Create a User Catalog with your own type of Sliders
- Link your catalog to the master User Component catalog

4.1 - Enter workbench Catalog Editor. A new empty catalog is automatically created.



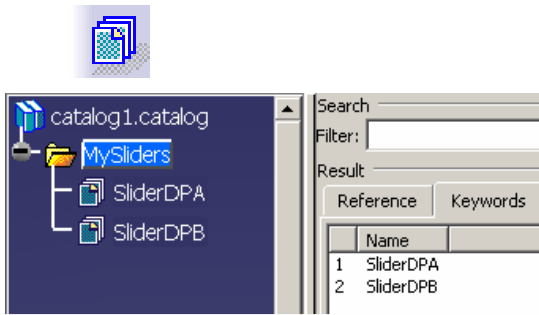
4.2 - Activate function Definition... in the contextual menu of default chapter Chapter.1



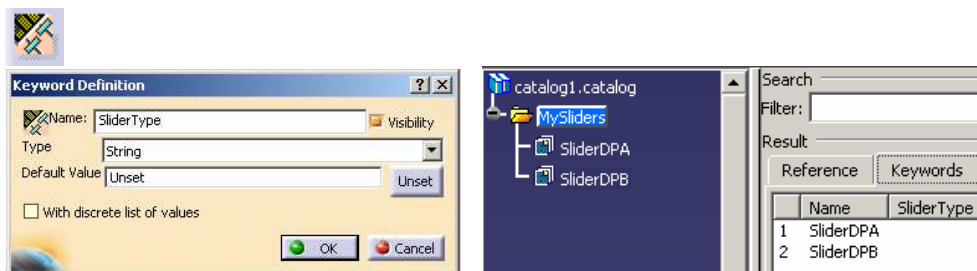
4.3 - Rename the Chapter with the name you wish, e.g. MySliders



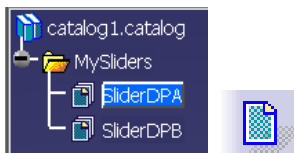
4.4 - Click icon Add Family and name it for example SliderDPA. Validate creation. Repeat these operations to create a second family with another name.



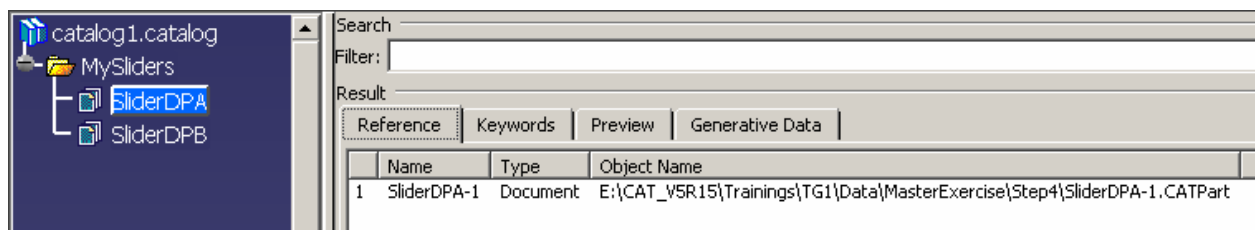
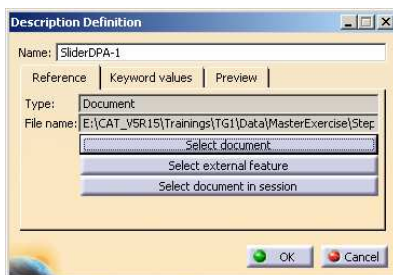
4.5 - Click icon Add Keyword. Create a Keyword of type String, name it for example SliderType. Validate creation. It will contain the user-defined type of object in the family.

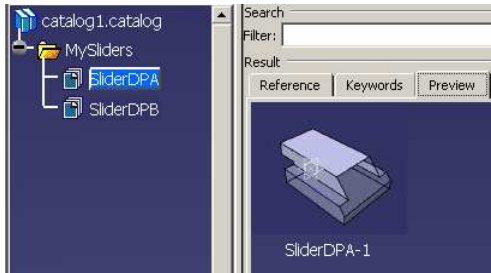


4.6 - Double-click in the tree family SliderDPA to make it active. Click icon Add Component.

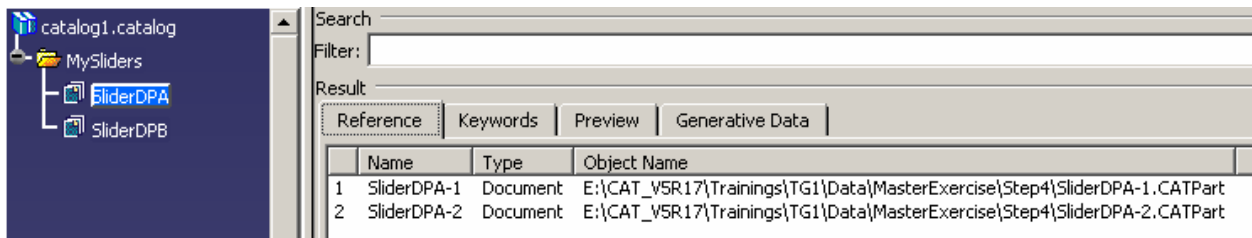


4.7 - Give a name to the component (e.g. SliderDPA-1). Click button Select document in dialog box Description definition. Select document SliderDPA-1.CATPart in folder Step4 (Downloaded to your local machine). Click successively tab Reference, then tab Preview to see the result.

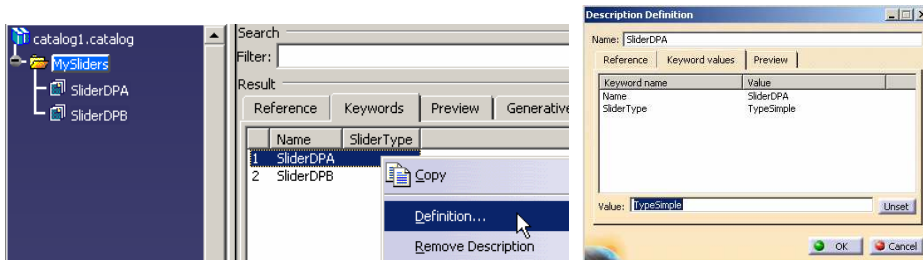




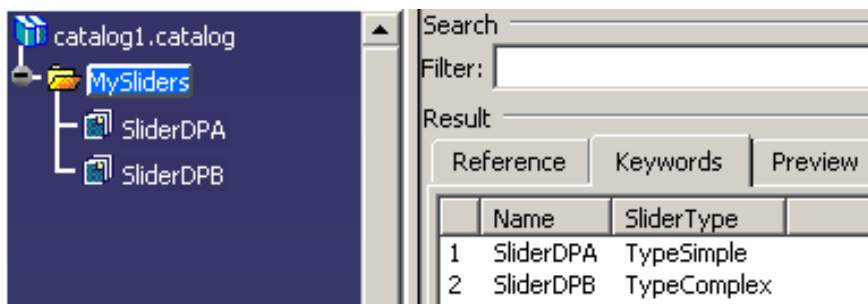
4.8 - Similarly, add in the same family another component, referencing file *SliderDPA-2.CATPart*.



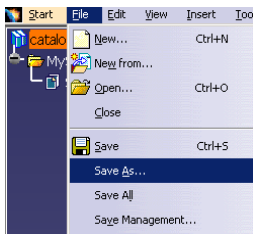
4.9 - Double-click chapter *MySliders* to make it active : Keyword *SliderType* is accessible. Select family *SliderDPA*. Activate its contextual menu and select *Definition...* In tab *Keyword values*, set the value of keyword *SliderType* to e.g. *Type-Simple*. Validate.



4.10 - Similarly, set the value of *SliderType* to *TypeComplex* for family *SliderDPB*.

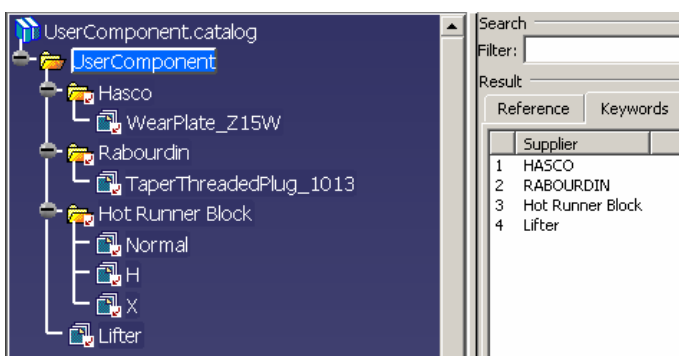


4.11 - Save your catalog, e.g. under name MyOwnCatalog, in any folder you wish. Do not close it..

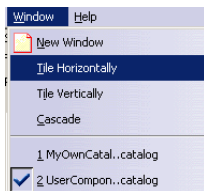


4.12 - Make a copy of TG1 master catalog of user components in the folder of your choice (make sure first it is not in Read-only mode) and open this copy.

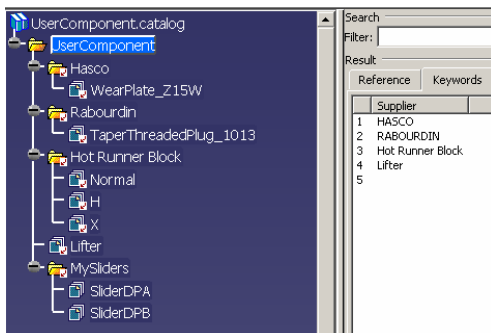
Original Master catalog is : ...intel_a\startup\components\MoldCatalog\UserComponent.catalog



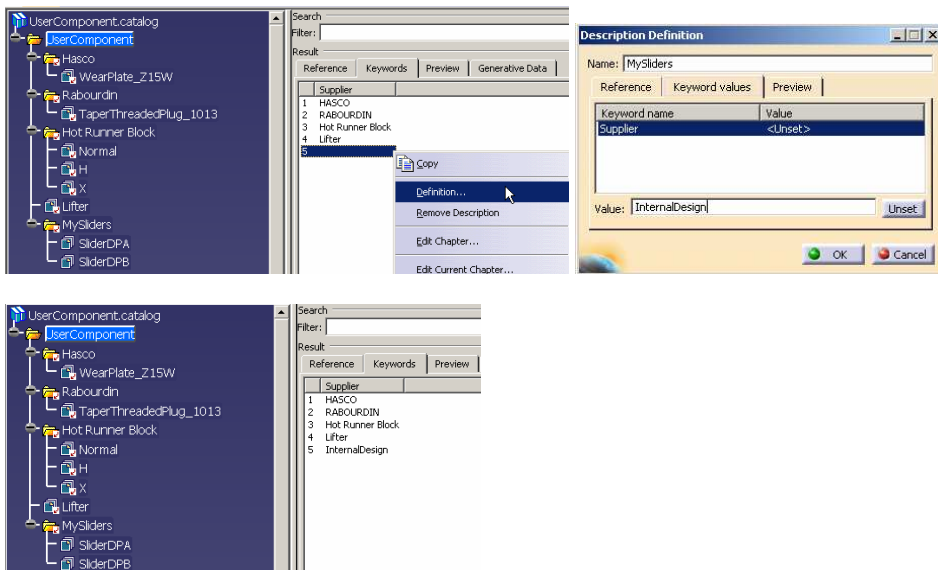
4.13 - Click Window / Tile Horizontally to see both catalogs simultaneously.



4.14 - Make sure that window UserComponent.catalog is active. Click icon Add link to other catalog. Select chapter MySliders in window MyOwnCatalog.catalog : the chapter has been added into the master User Component catalog.

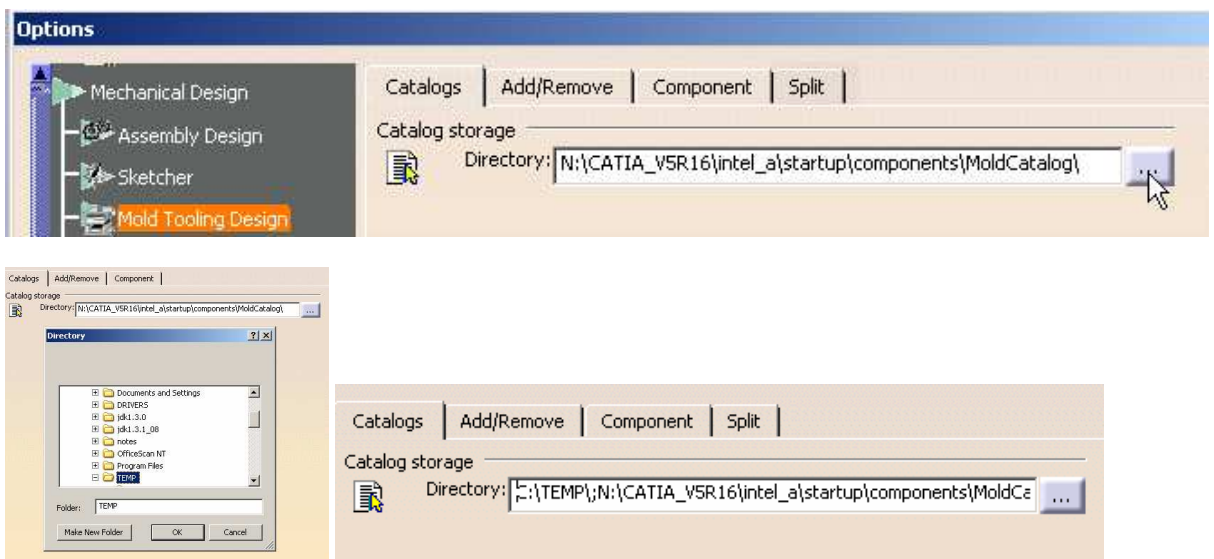


4.15 - Select Keyword Supplier for the new chapter. Activate function Definition... in the contextual menu. Select tab Keyword values. Define a value for keyword Supplier, e.g. InternalDesign. Validate.



4.16 - Save file UserComponent.catalog.

4.17 - Go to Tools/Options/Mechanical Design/Mold Tooling Design. Add your new location as first search location.



4.18 - The new Master user Component Catalog is now ready for use in workbench Tooling Design.