

# **CATIA Core and Cavity Design**

## **Detailed Steps**

# Table of Contents

Master Exercise: The Pan Handle .....	3
Master Exercise Step 1: Importing the Part to Mold.....	3
Step (2): Defining the Main Pulling Direction.....	6
Step (3): Defining a Slider Direction .....	10
Step (4): Working on Mold Areas .....	12
Step (5): Creating the Parting Line .....	19
Step (6): Creating the Parting Surface .....	23
Additional Exercise: The Support.....	35
Step 1: Checking & Reworking the Orientation of Faces .....	35
Step 2: Defining the Mold Areas.....	37
Step 3: Creating the Bounding Box .....	39
Step 4: Comparing Two Versions of the Part.....	41

# Master Exercise: The Pan Handle

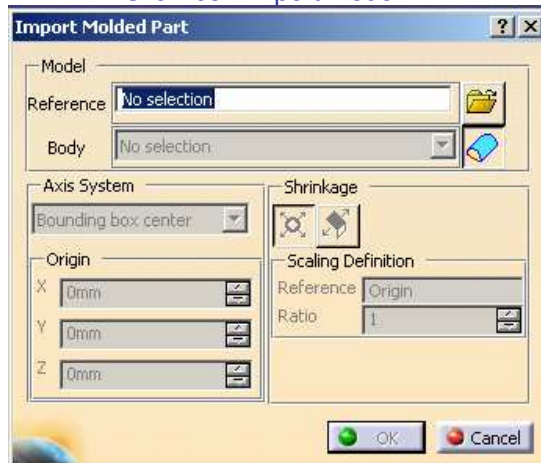
## Master Exercise Step 1: Importing the Part to Mold

In this step, you will:

- Create a new Product
- Access workbench Core & Cavity Design
- Select the Part to import
- Select the relevant surface inside this Part
- Define an Axis System
- Apply a shrinkage factor
- Enter workbench *Assembly Design* to create a new Product.
- Make sure that this Product is active (double-click it in the specification tree) and enter workbench *Core & Cavity Design* :

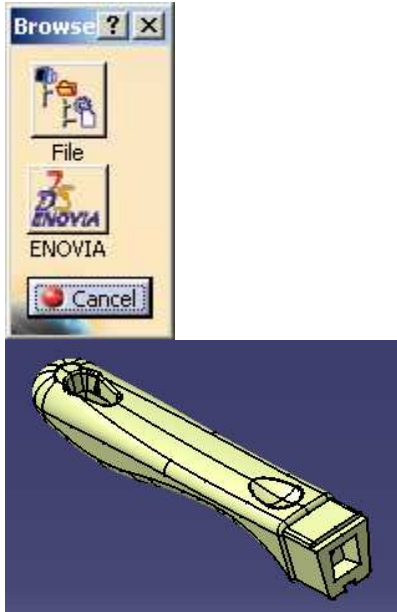


- Click icon Import Model 

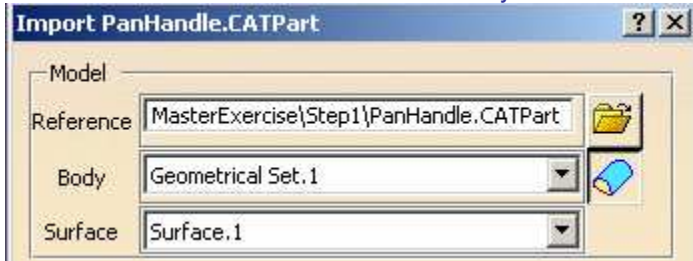


- Select file \MasterExercise\Step1\PanHandle.CATPart. 

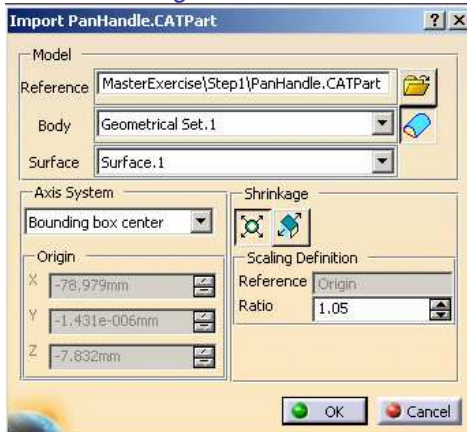
If you have a connection to a Smarteam or Enovia database choose File option:



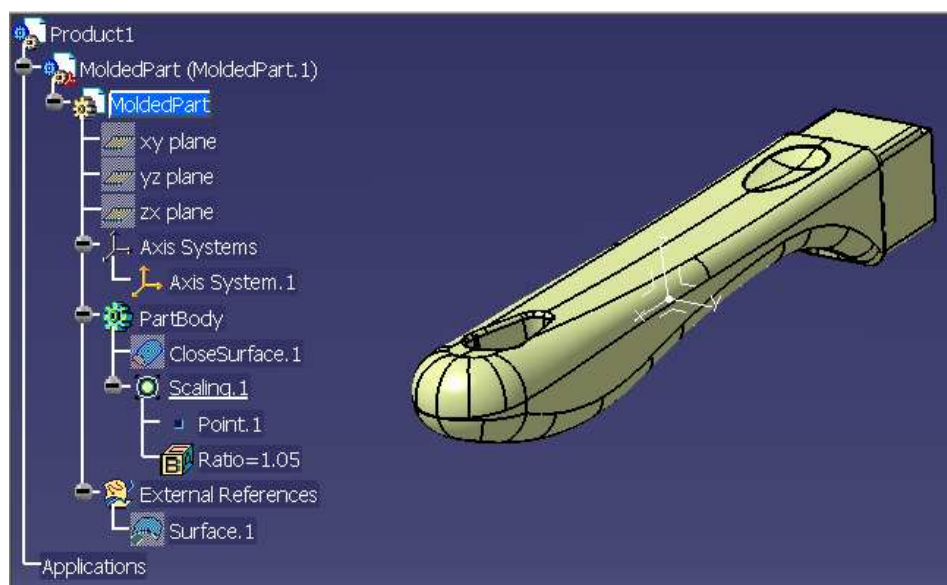
- Make sure that the selected body is *Geometrical Set.1* and selected surface is *Surface.1*



- Select *Bounding box center* as origin of the Local Axis System and define shrinkage as *Scaling with Ratio = 1.05*



- Click *OK* to validate Import operation. Check the result in the tree.

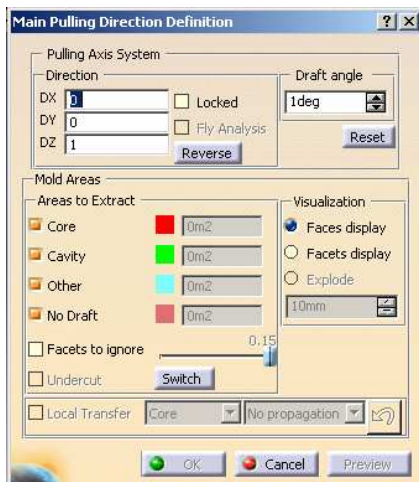


- Close the document without saving it if you want to use existing prepared data for the next step.

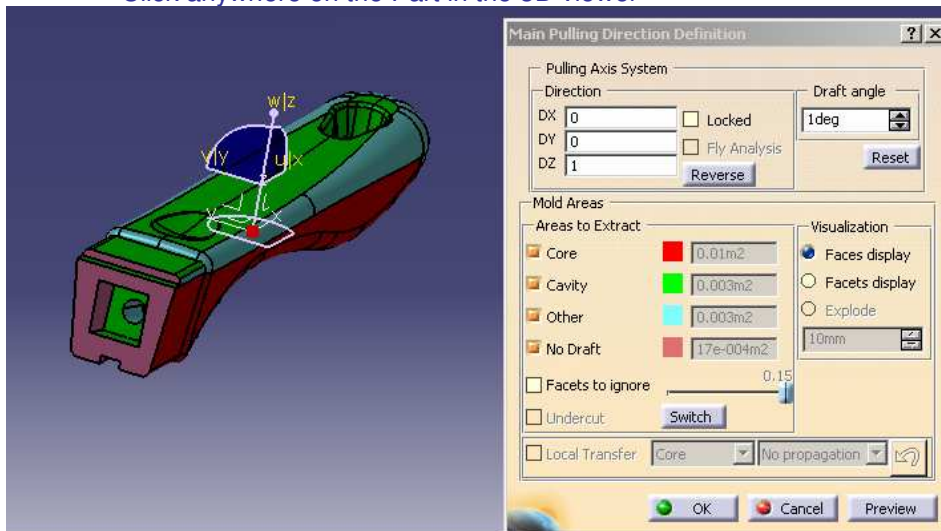
## Step (2): Defining the Main Pulling Direction

In this step, you will define the Main Pulling direction for the Part.

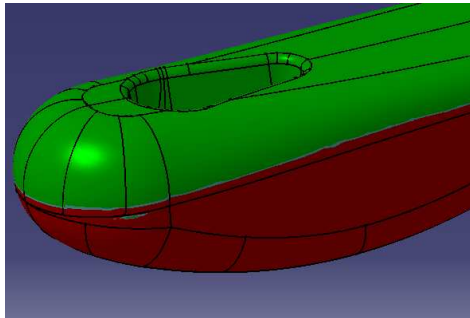
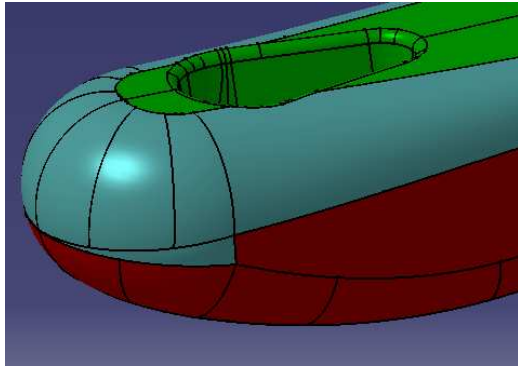
- If you have not kept the results of the previous step, open existing Product \MasterExercise\Step2\Master\_Exercise\_Step\_2.CATProduct, make its Part active (double click part in specification tree) and select workbench Core & Cavity Design.
- Click icon Define pulling direction 



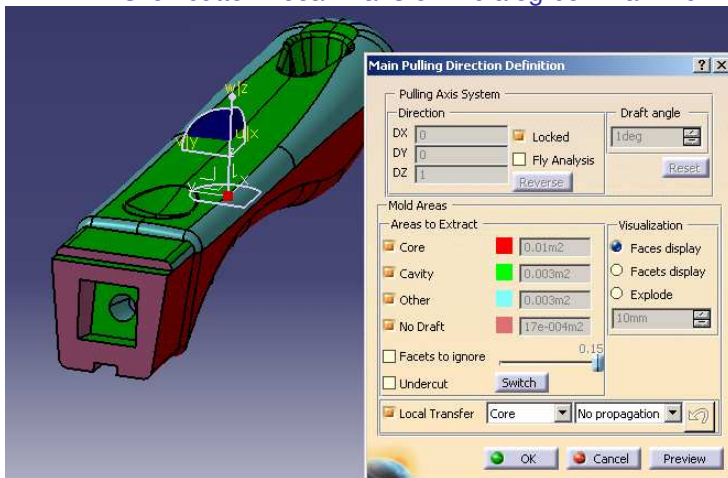
- Click anywhere on the Part in the 3D viewer



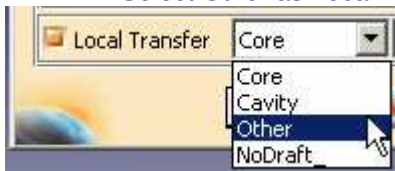
- Move the compass in various directions to see the effect on the result.
- Change the value of the draft angle to see the effect on the result.
- Activate option Facets to ignore to see its effect, and then turn it off.
- Activate option *Facets display* and check the location of the Reflect Line.



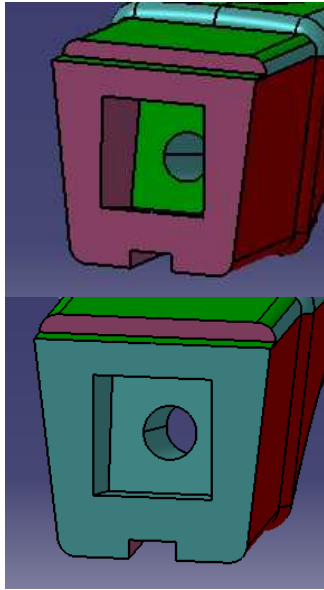
- Activate option *Faces display*.
- Click button *Locked* in dialog box Main Pulling Direction Definition.
- Click button *Local Transfer* in dialog box Main Pulling Direction Definition.



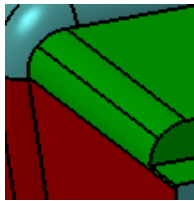
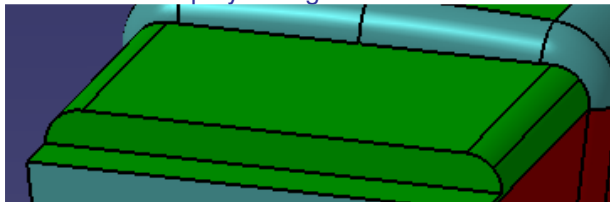
- Select *Other* as Local Transfer destination.



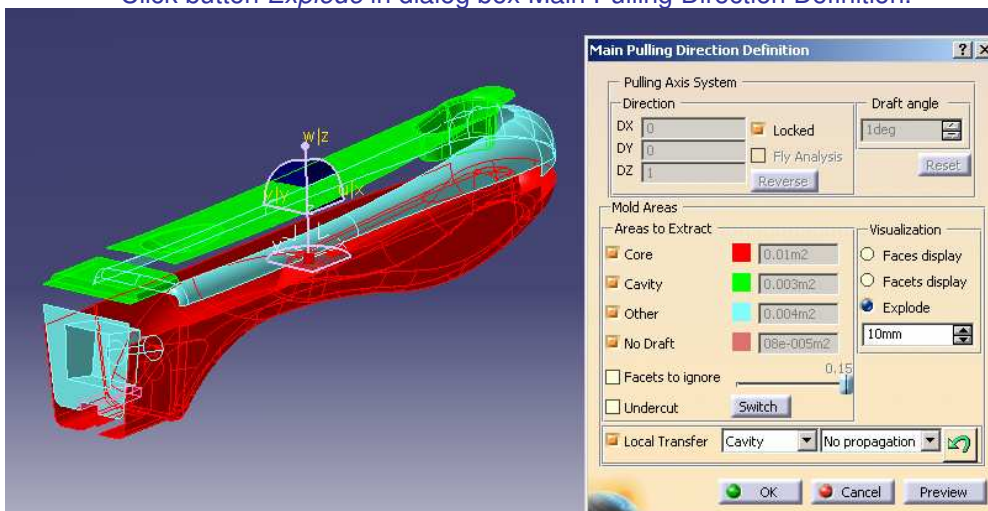
- Pick individually the pink, green and red faces located in the undercut area (pocket) at the extremity of the Part. These faces are now displayed in blue.



- Do a little more rework: now select *Cavity* as target destination.
- Pick the pink and blue faces located on the top of the pocket as shown. These faces are now displayed in green.

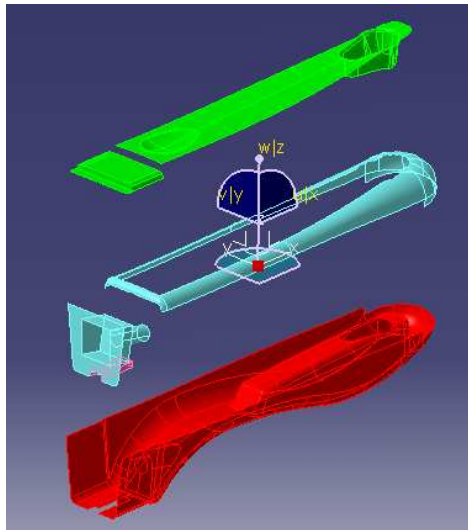


- Click button *Explode* in dialog box *Main Pulling Direction Definition*.

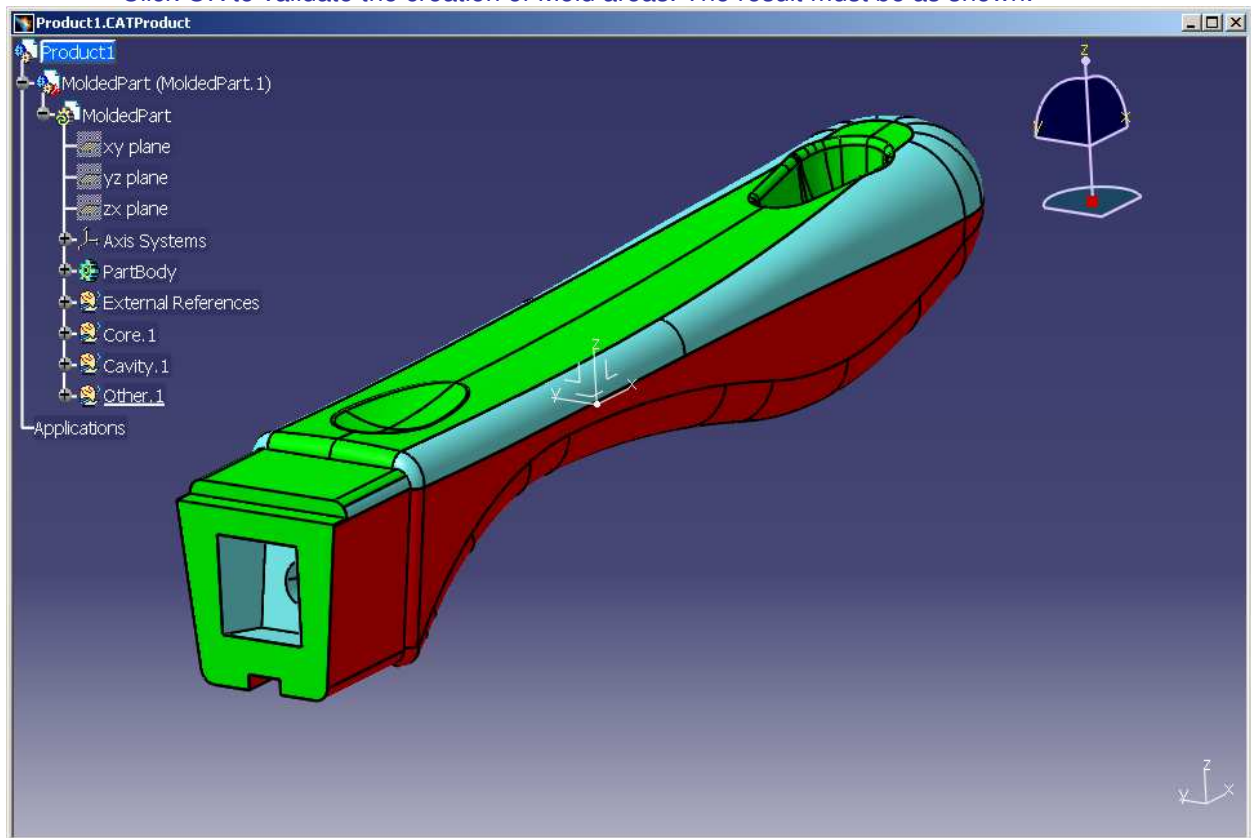


- Set *Explode* distance to 100mm. Click keyboard button *Tab* to validate this input. You can see that further rework will be necessary to tune the definition of *Core* and *Cavity* areas.





- Click OK to validate the creation of Mold areas. The result must be as shown:



- Close the document without saving it if you want to use existing prepared data for the next step.

## Step (3): Defining a Slider Direction

In this step, you will:

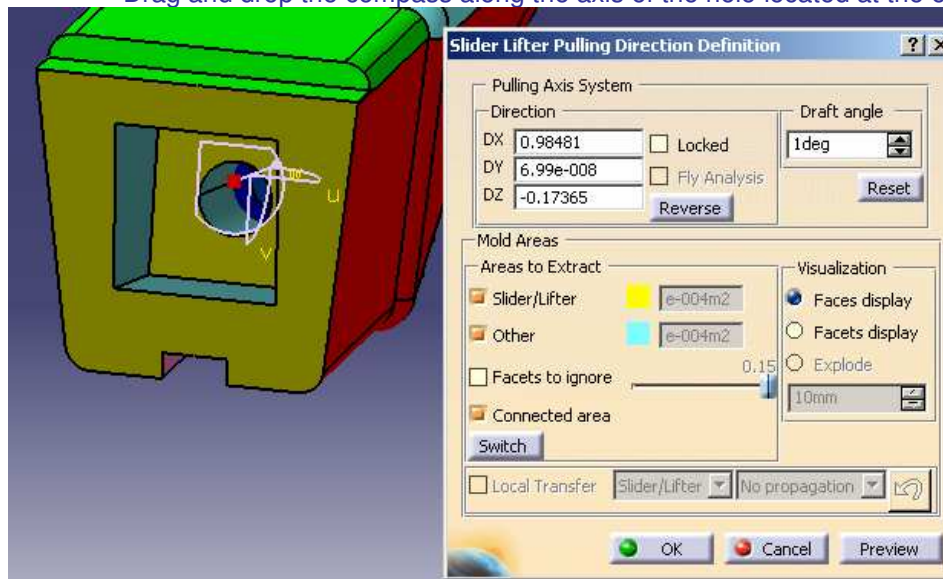
- Create an additional Pulling Direction to locate a Slider at the extremity of the handle.

If you have not kept the result of the previous step, open existing Product `\MasterExercise\Step3\Master_Exercise_Step_3.CATProduct`, make its Part active and select workbench *Core & Cavity Design*.

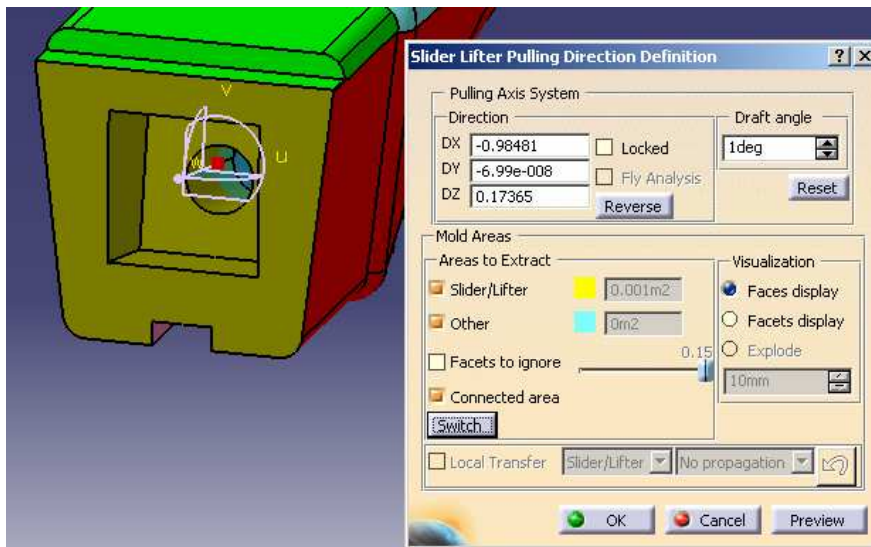
- Click icon Define slider lifter direction.



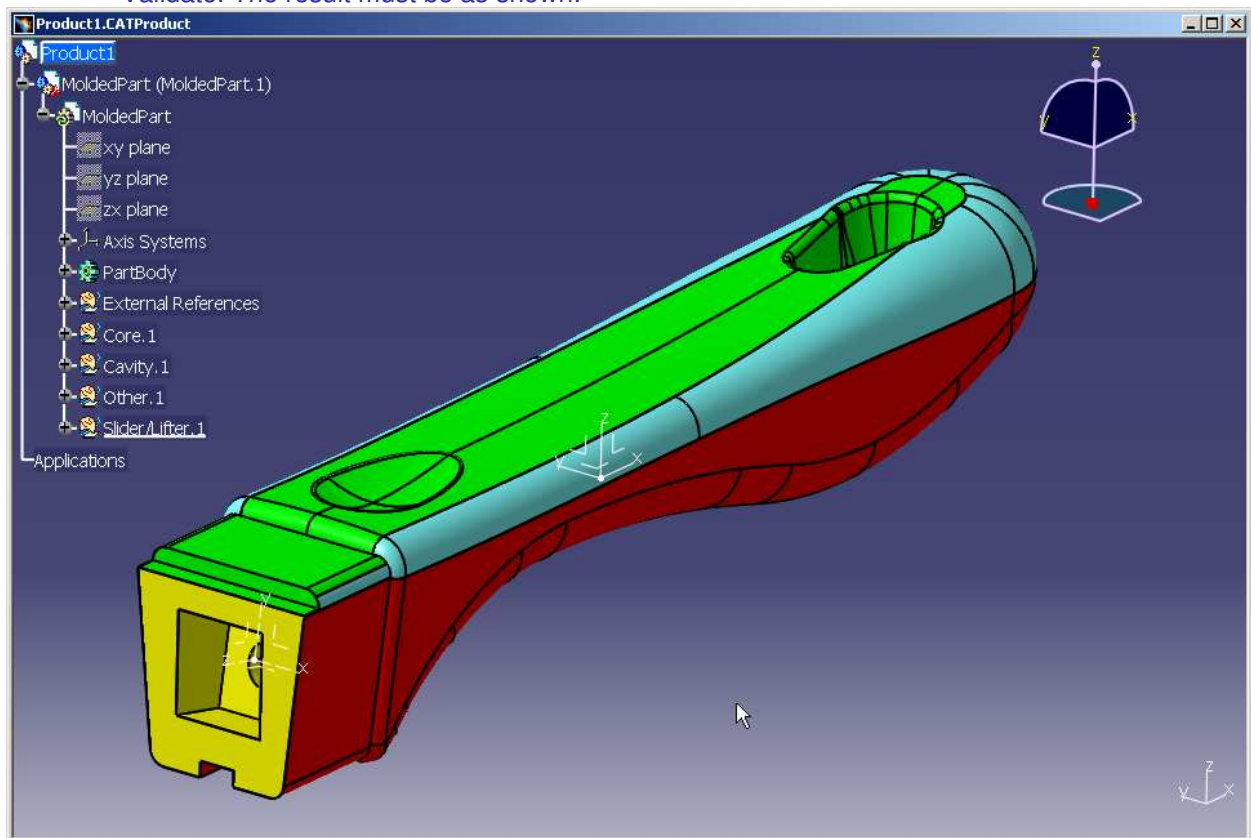
- Drag and drop the compass along the axis of the hole located at the extremity of the part.



- Click button *Reverse* and button *Switch*, if necessary, to ensure that the compass is oriented towards the outside of the Part and that the faces are colored in yellow as shown.



- If necessary, transfer the elements to Slider/Lifter.1.
- Validate. The result must be as shown:



- Close the document without saving it if you want to use existing prepared data for the next step.

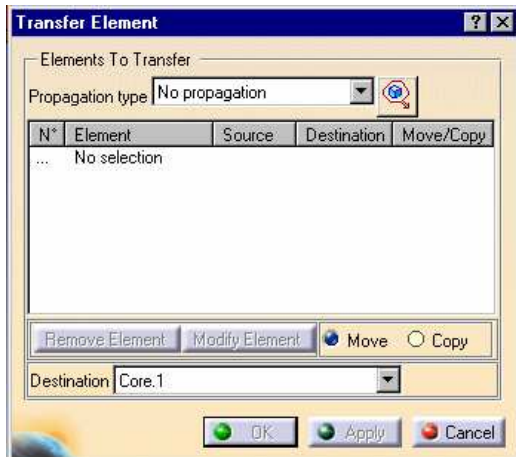
## Step (4): Working on Mold Areas

In this step, you will:

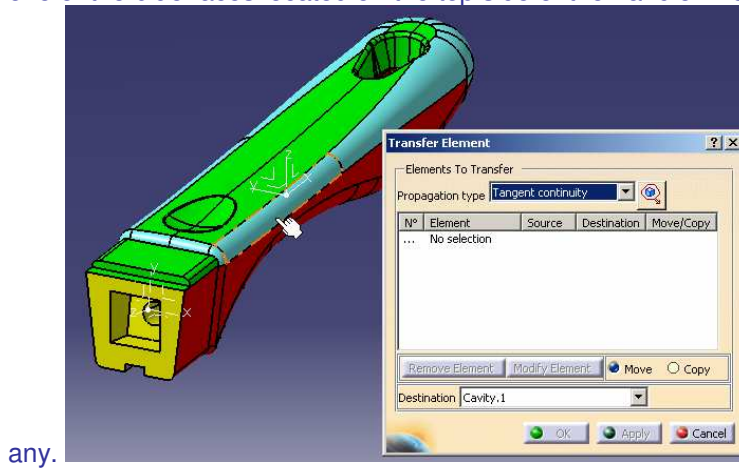
- Transfer faces between mold areas.
- Aggregate faces
- Split mold areas
- Check the mold areas with Explode View

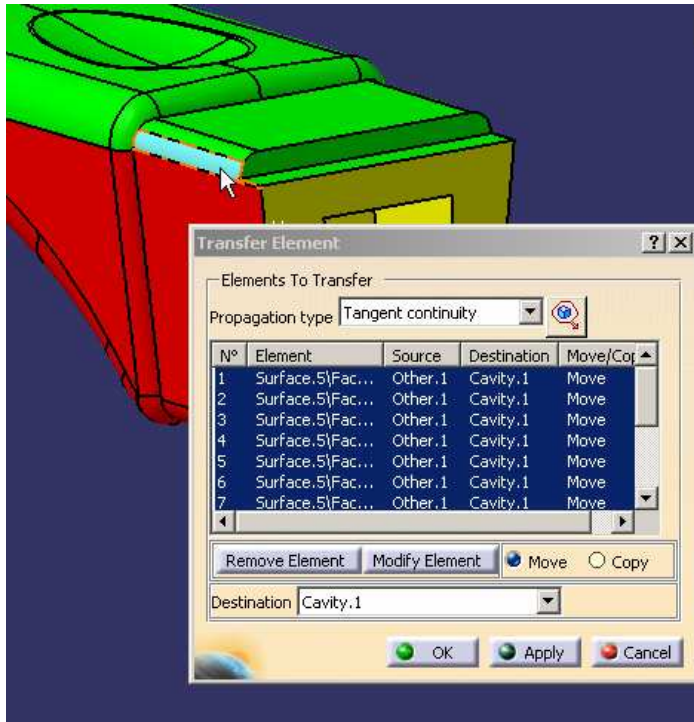
If you have not kept the results of the previous step, open existing Product `\MasterExercise\Step4\Master_Exercise_Step_4.CATProduct`, make its Part active and select workbench *Core & Cavity Design*

- Click icon Transfer an element. 

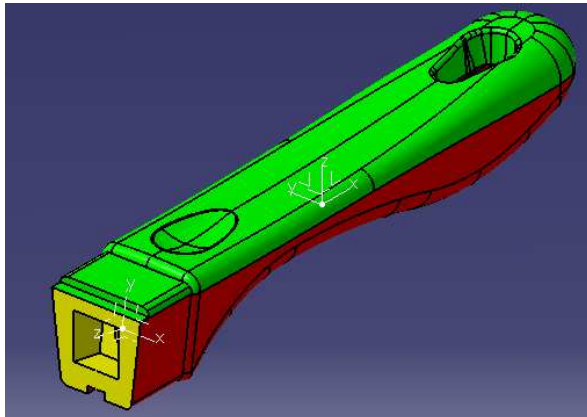


- Select *Cavity* as Destination, activate option *Propagation type : Tangent continuity* and pick one of the blue faces located on the top side of the handle. Pick the remaining blue surface if





- Validate.

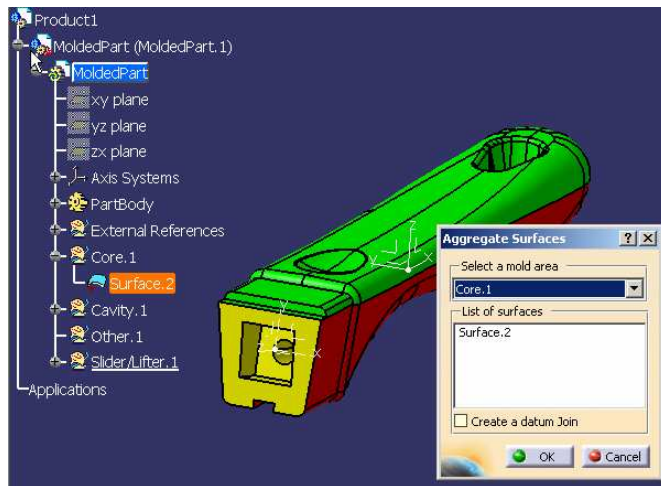


- Click icon *Aggregate Mold Area*.

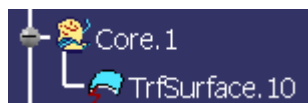


- Select Geometrical Set *Core* in the specification tree or pick anywhere in the red area in the 3D viewer.

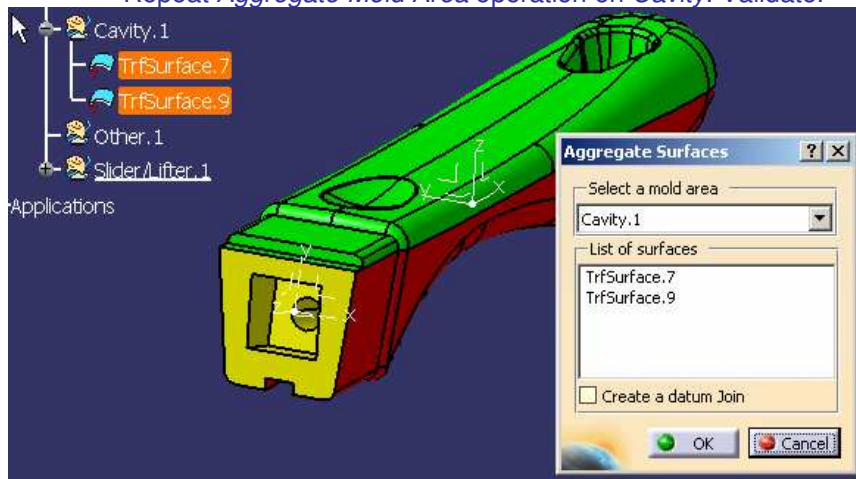




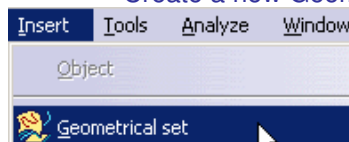
- Click OK to validate.



- Repeat *Aggregate Mold Area* operation on *Cavity*. Validate.

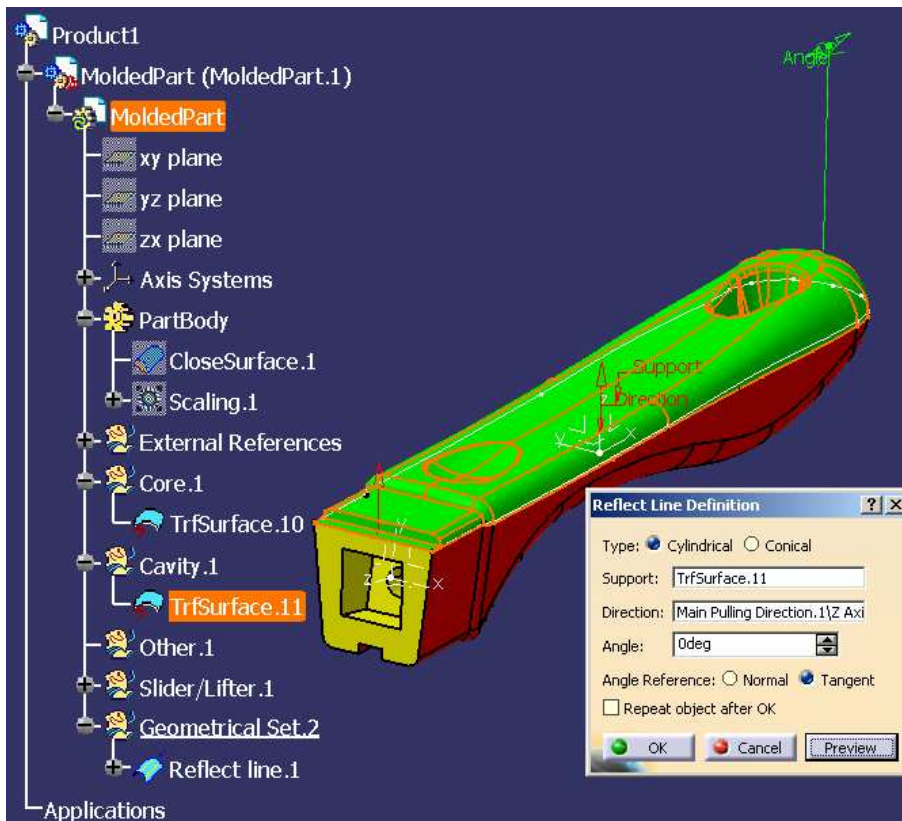


- Now we need to refine the separation between the Core and Cavity sides of the part to fit the Reflect line, which will be used to split the *Cavity* area.
- Create a new Geometrical Set in the part.

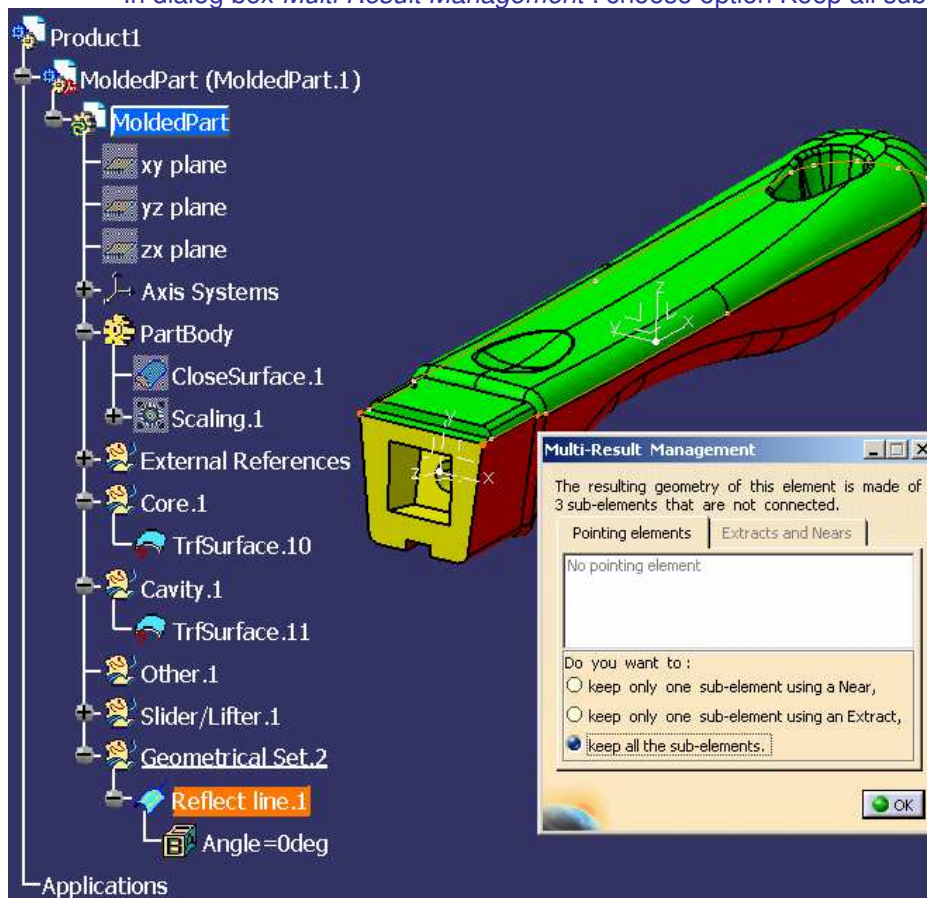


- Click icon *Reflect Line*. Select *Direction: Z axis* of Main Pulling direction axis system. Select *Angle: 0 deg*. Select *Support: green surface*. Check *Tangent* option. Validate. Ignore the warning if any.

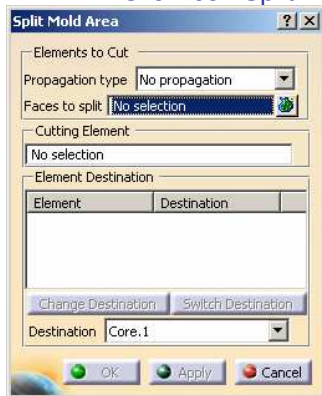




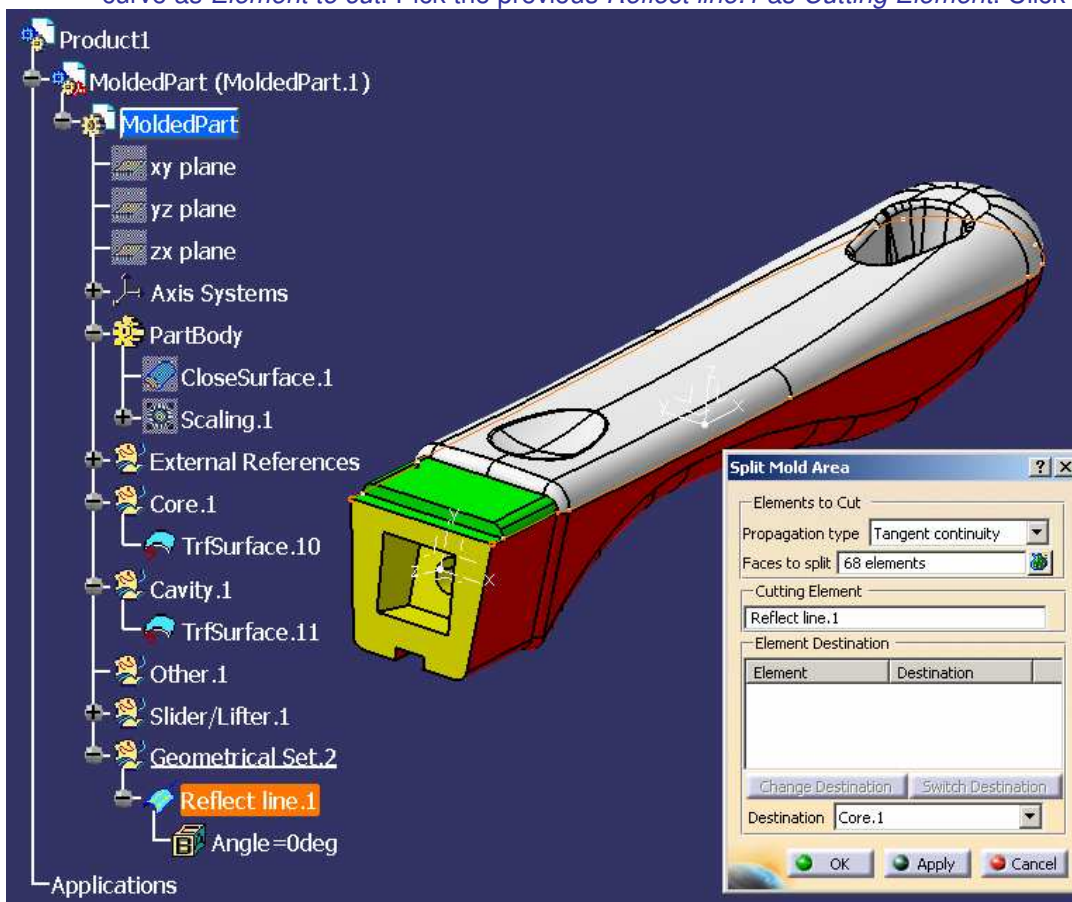
• In dialog box *Multi Result Management* : choose option *Keep all sub-elements*



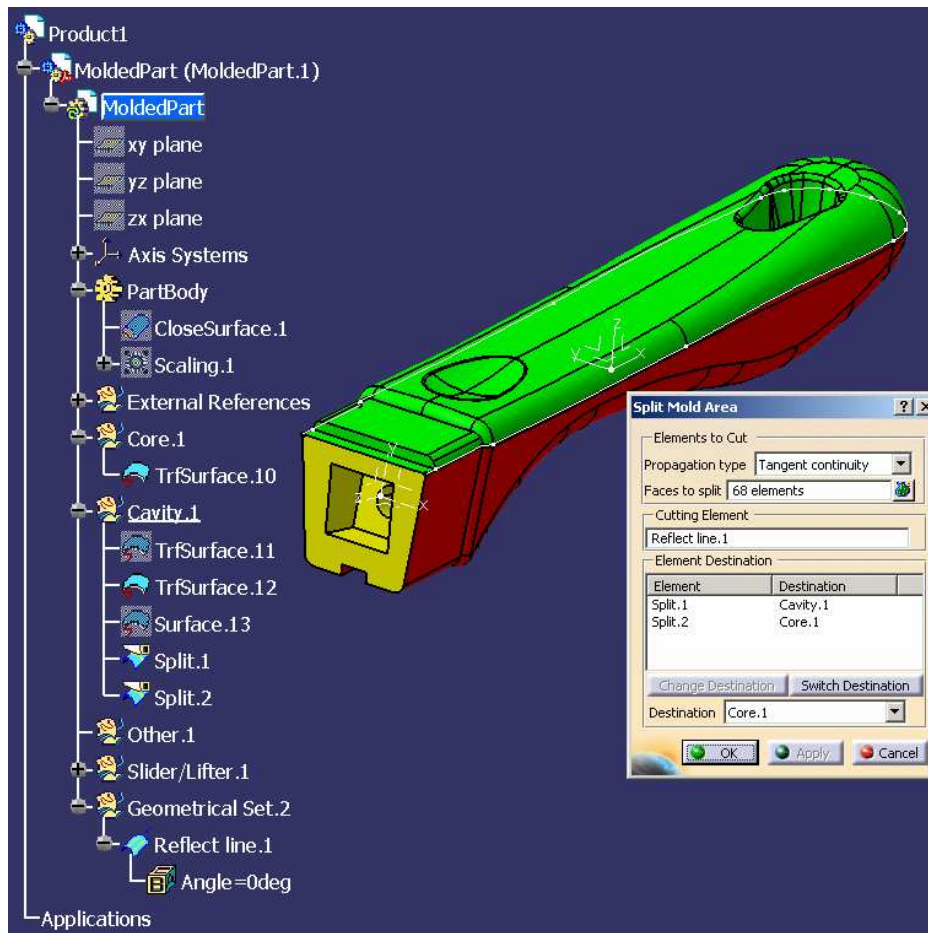
- Click icon Split Mold Area. Select Destination : Core



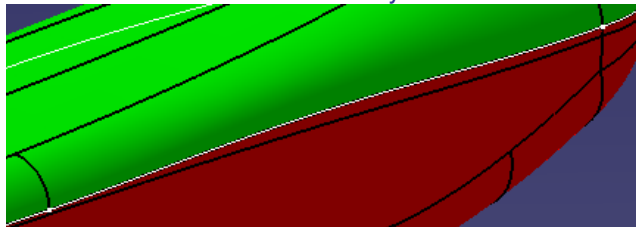
- Select *Propagation type* : Tangent continuity. Pick one of the faces impacted by the splitting curve as *Element to cut*. Pick the previous *Reflect line.1* as *Cutting Element*. Click *Apply*.

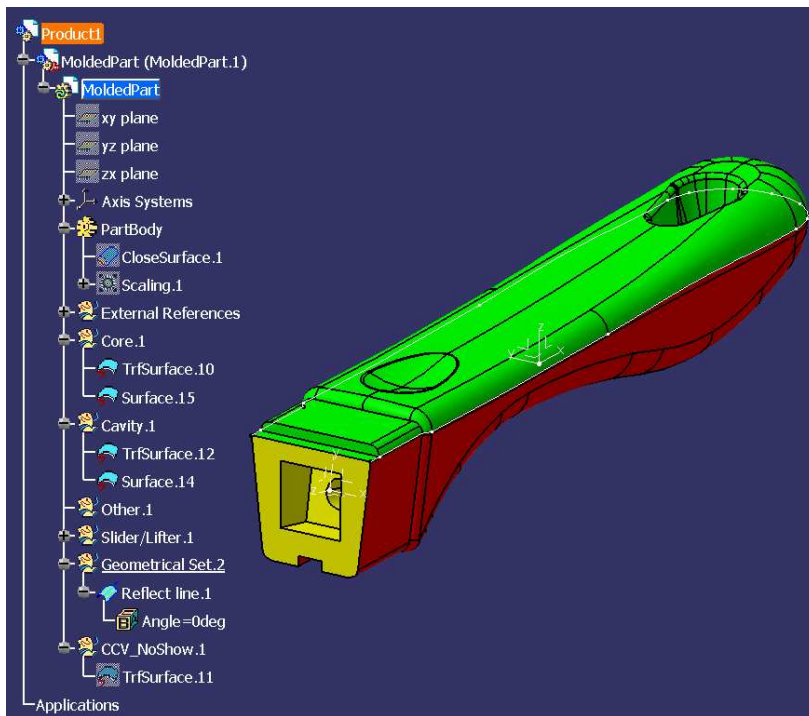




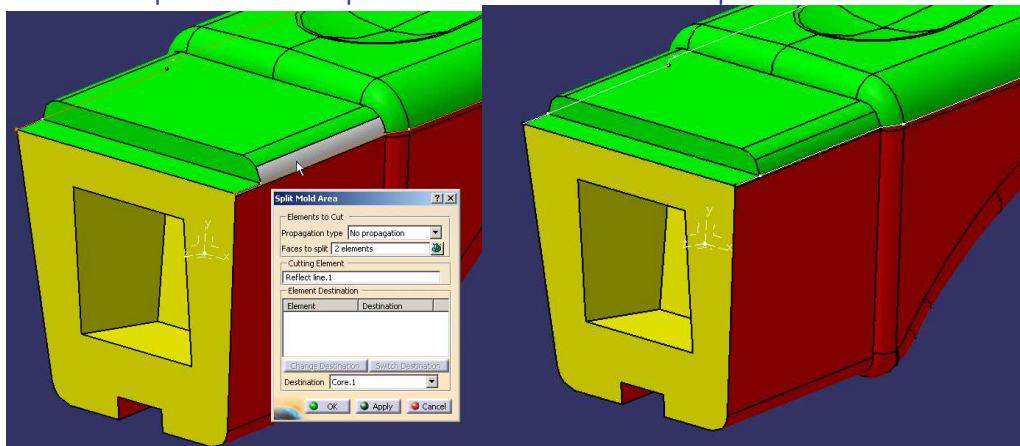


- If necessary, click button *Switch Destination* to revert the distribution of resulting split faces between Core and Cavity areas. Validate.





- Repeat the same operation for the last 2 faces to split



- Close the document without saving it if you want to use existing prepared data for the next step.

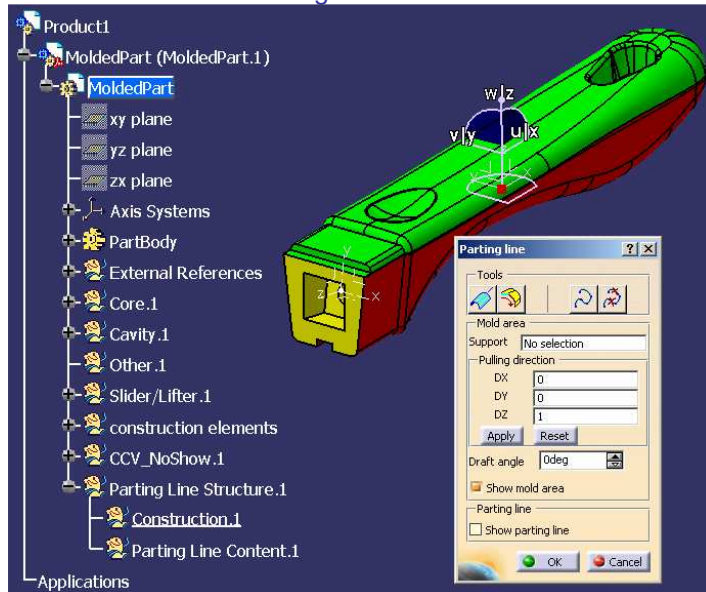
## Step (5): Creating the Parting Line

In this step, you will:

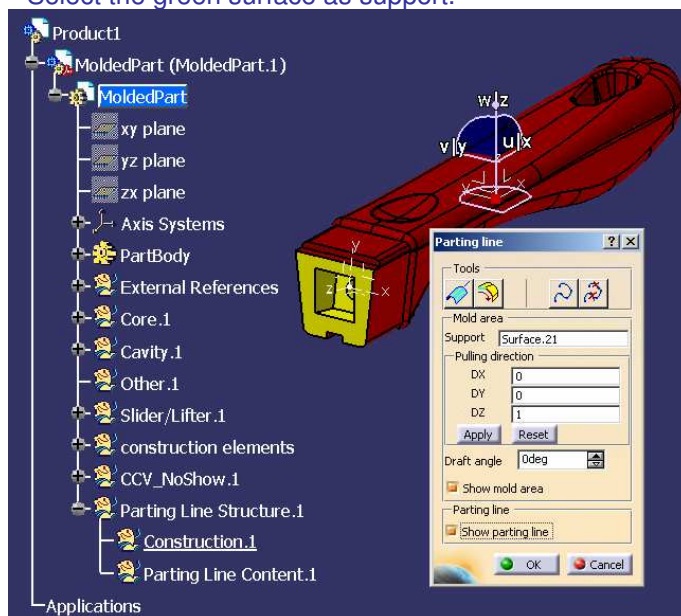
- Create the parting line.

If you have not kept the results of the previous step, open existing Product `\MasterExercise\Step4\Master_Exercise_Step_5.CATProduct`, make its Part active and select workbench *Core & Cavity Design*

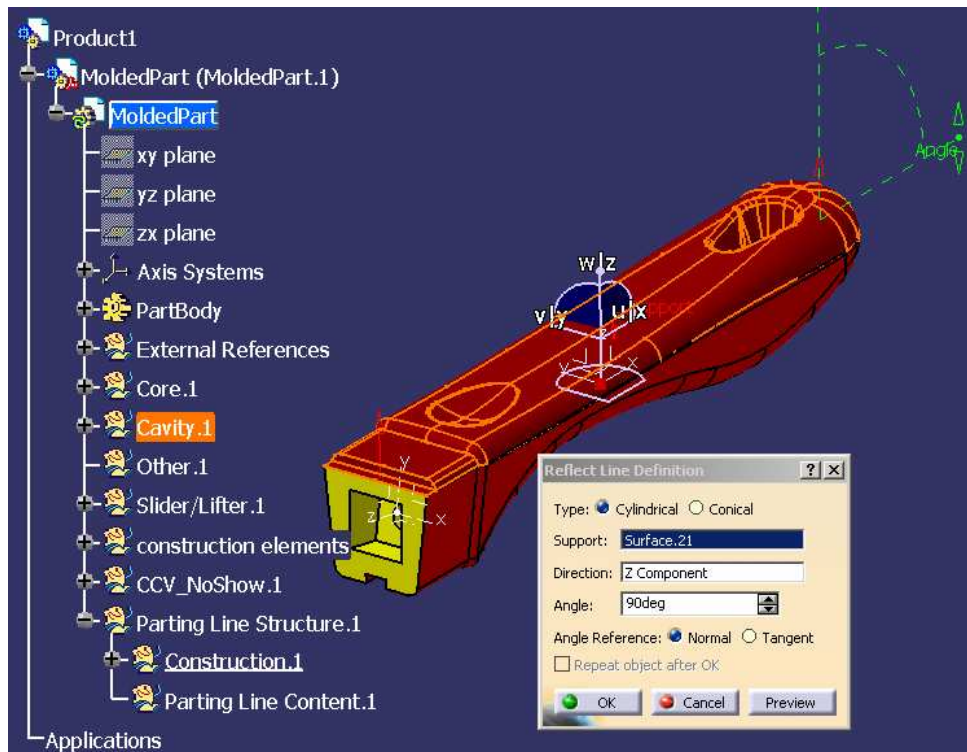
- Click icon Parting line.



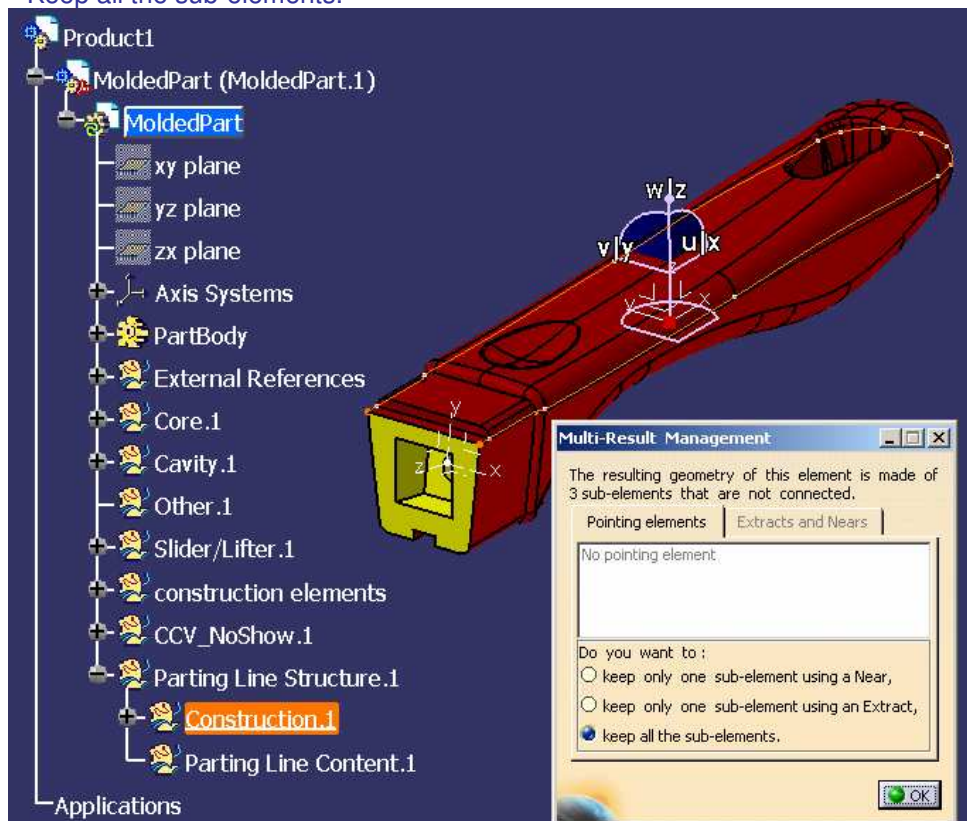
- Select the green surface as support.



- Click the Reflect line icon in the Parting line dialog box and create the Reflect line.



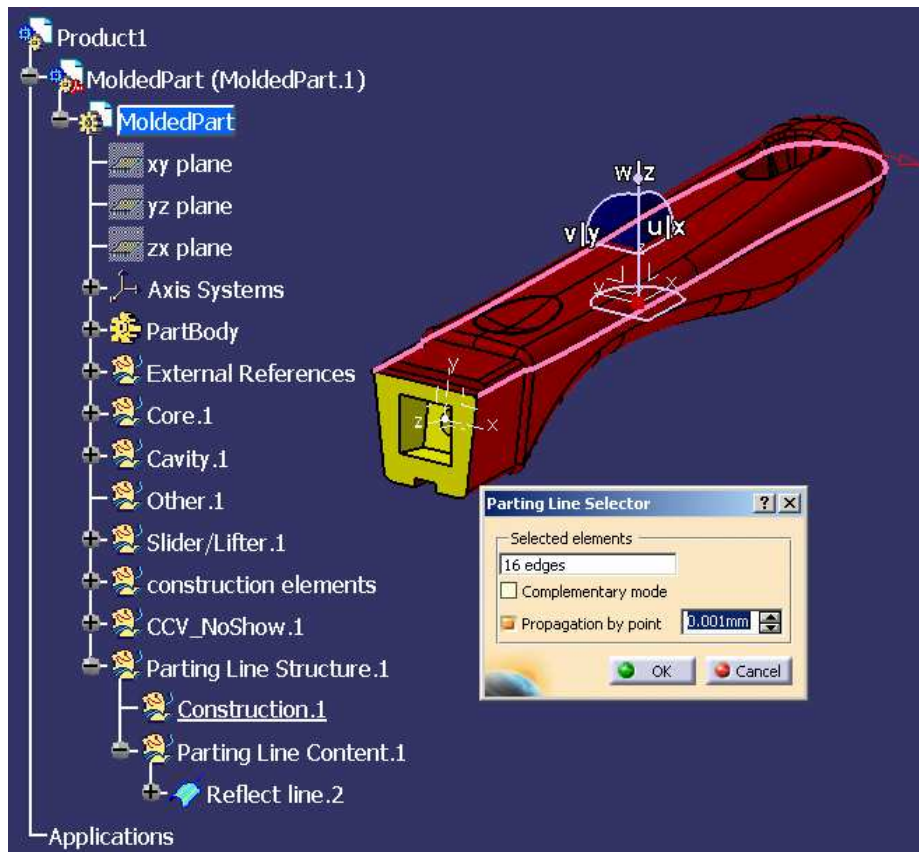
- Keep all the sub-elements.



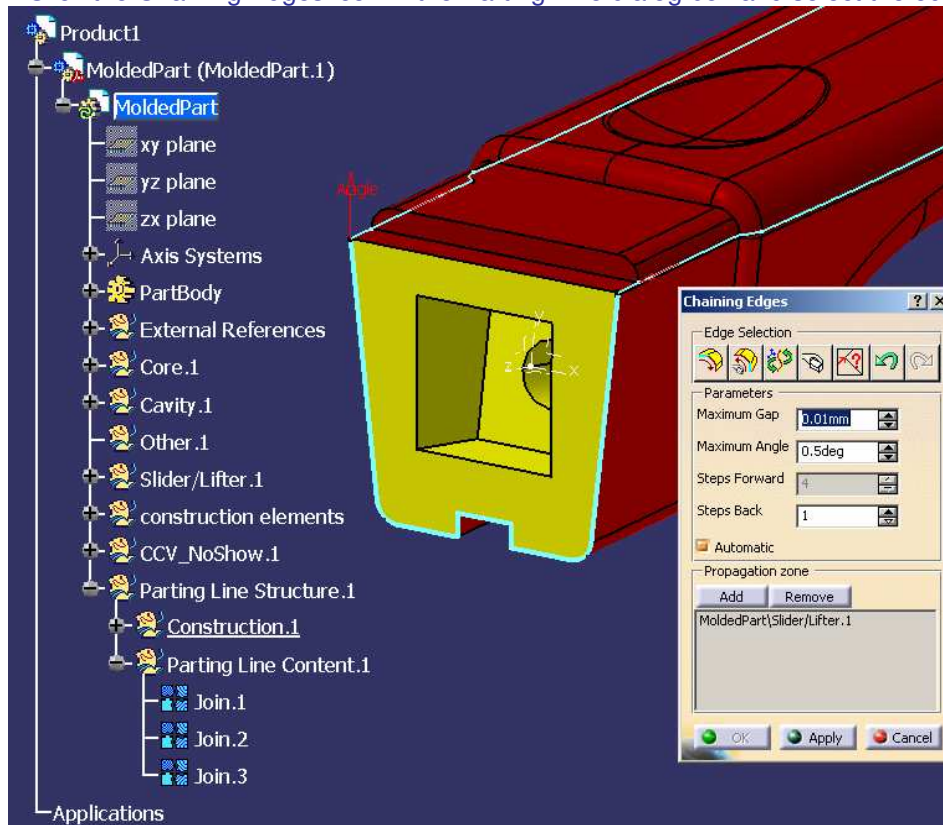
- Click the Parting Line Selector icon and select shown edges.

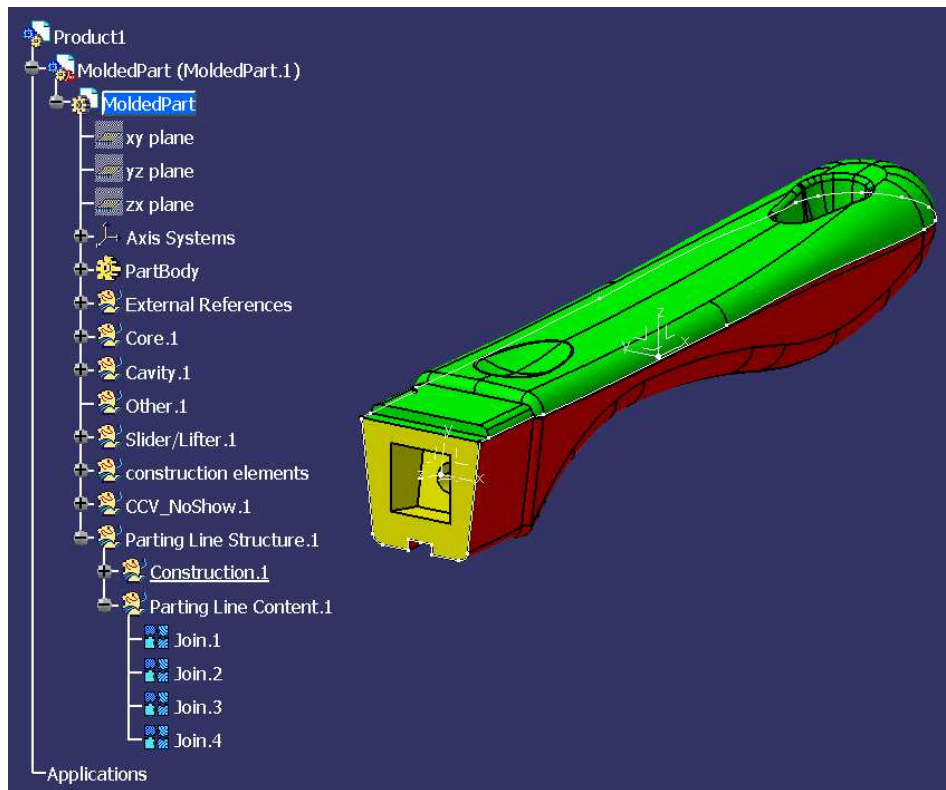






- Click the Chaining Edges icon in the Parting Line dialog box and select the edges as shown.





## Step (6): Creating the Parting Surface

In this step, you will:

- Create the surface elements constituting the outer Parting surface for the Cavity Side of the Mold.

If you have not kept the results of the previous step, open existing Product `\MasterExercise\Step6\Master_Exercise_Step_6.CATProduct`, make its Part active and select workbench *Core & Cavity Design*.

- We will first create the bounding sketch of the external Parting Surface.
- Create a new *Geometrical Set* in the Part.



- Hide all mold areas except *Cavity*. Hide the splitting Join used previously.
- Select plane XY of axis system *Main Pulling Direction*.

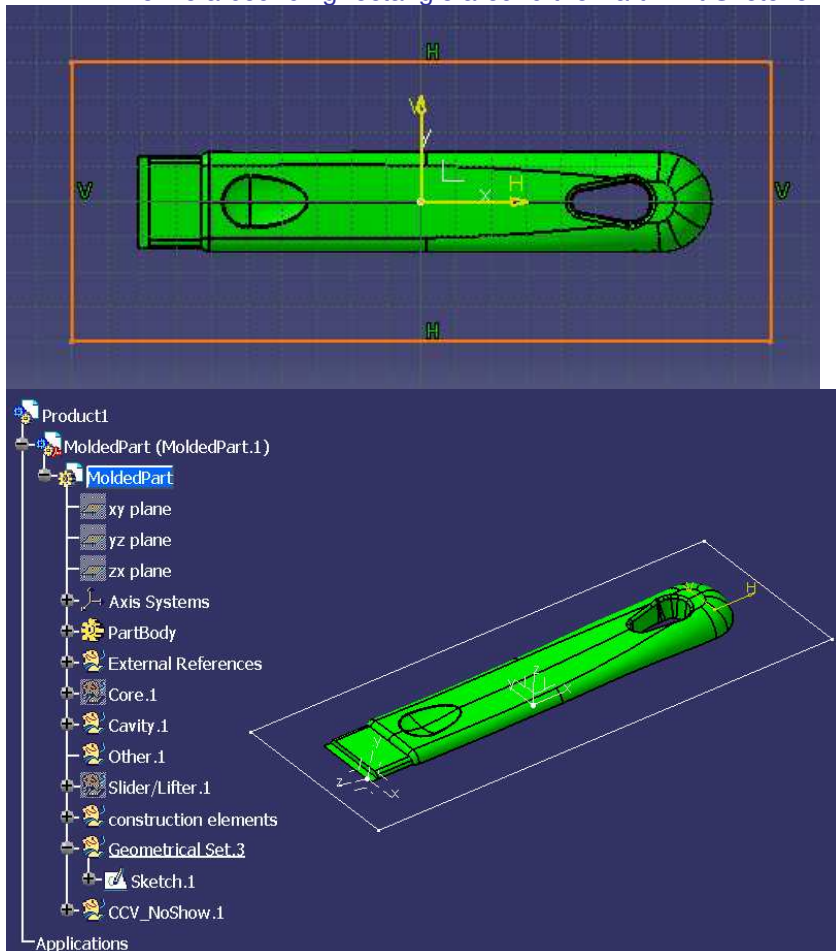
- Click icon *Sketcher*



- Click icon *Rectangle*.

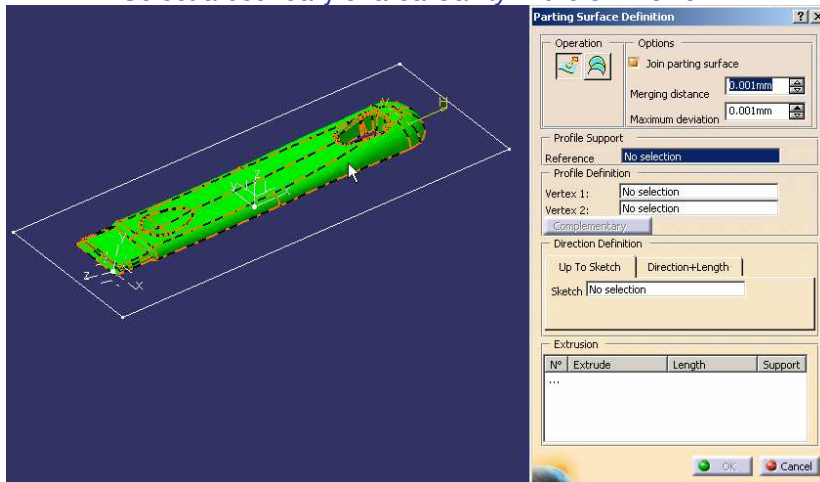


- Define a bounding rectangle around the Part. Exit Sketcher.

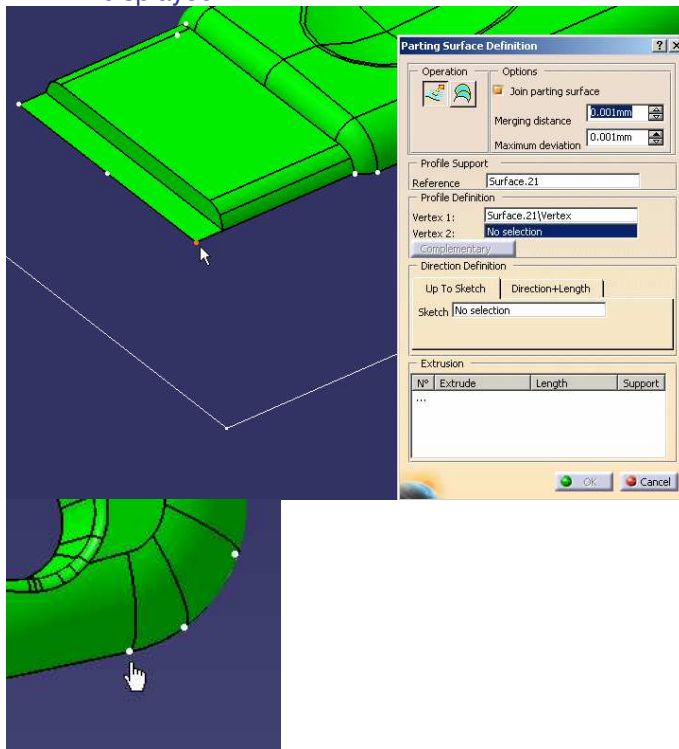




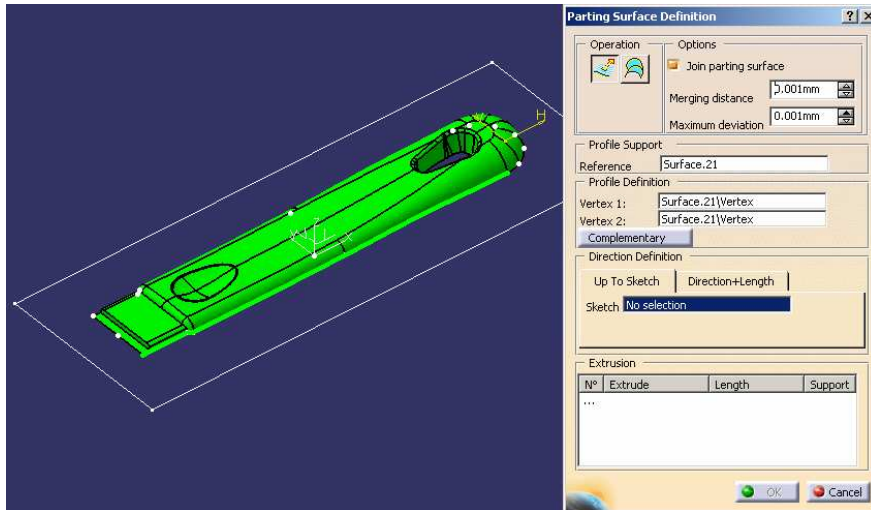
- Click icon Parting Surface.
- Select a boundary of area *Cavity* in the 3D viewer.



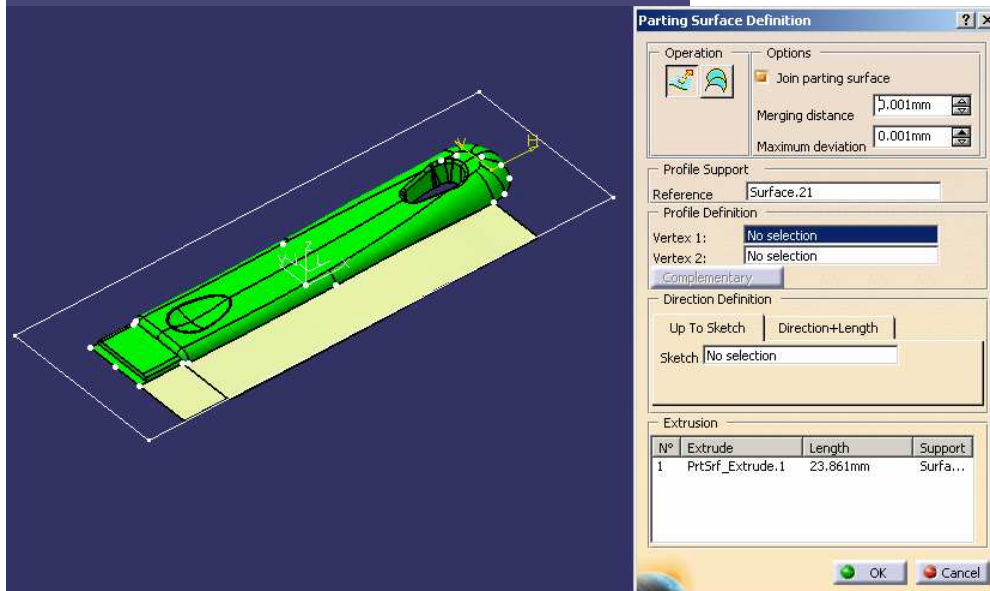
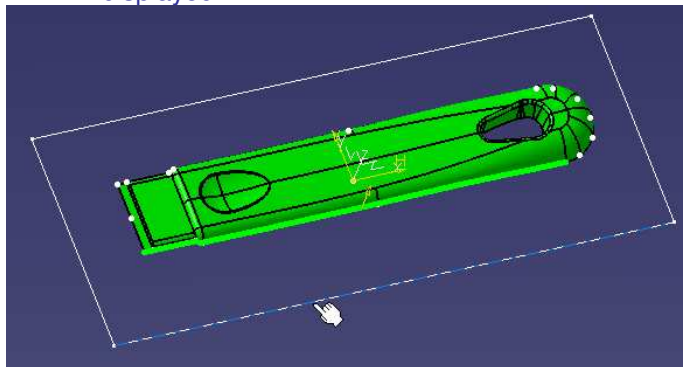
- Pick two vertices as shown. The corresponding portion of the Cavity boundary is then displayed.



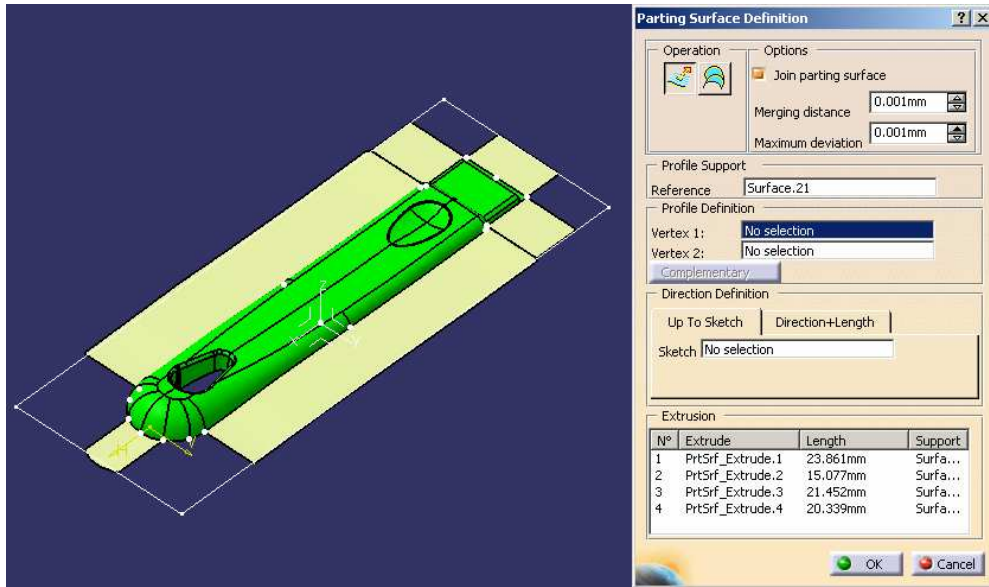




- Select the side of the bounding Sketch as shown. The boundary to be extruded is then displayed.



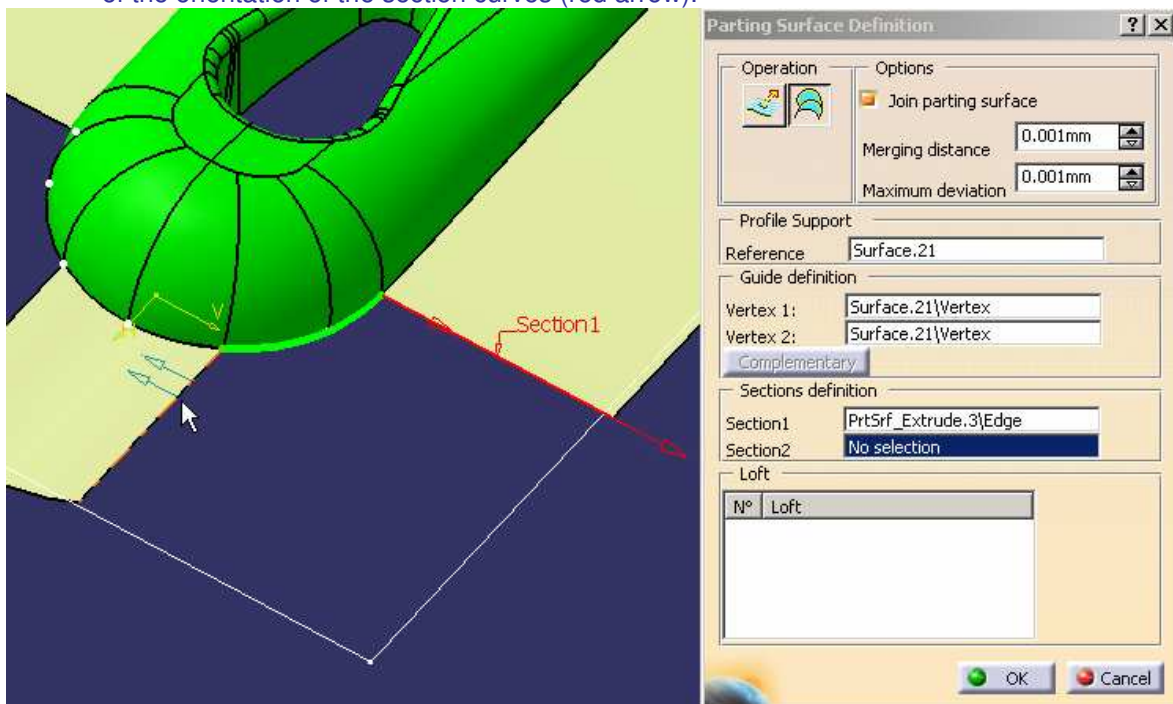
- Proceed similarly to create three other *Extrude* surfaces as shown. Use button *Complementary* if necessary to define the right portion of boundary.

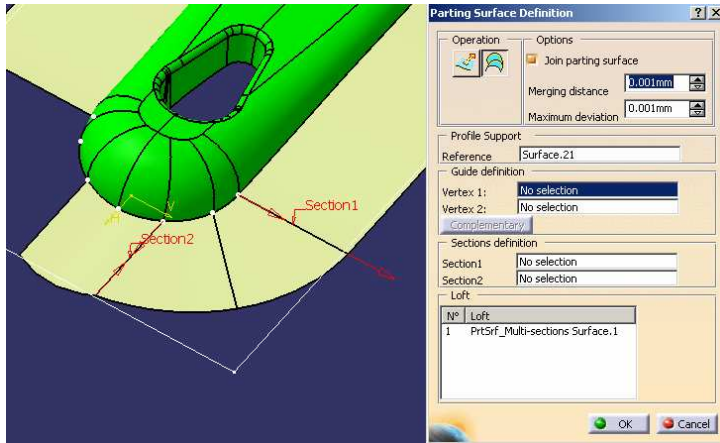


- Now activate icon *Loft* in dialog box *Parting Surface Definition*.

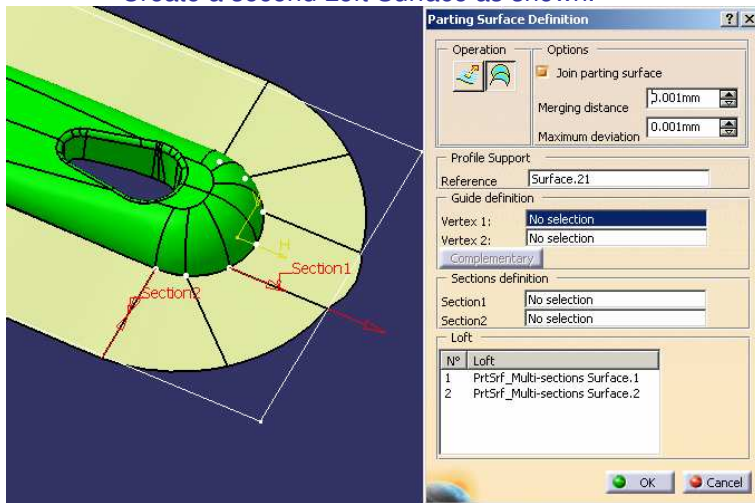


- Successively select the two vertex points, then the two Section curves as shown. Take care of the orientation of the section curves (red arrow).

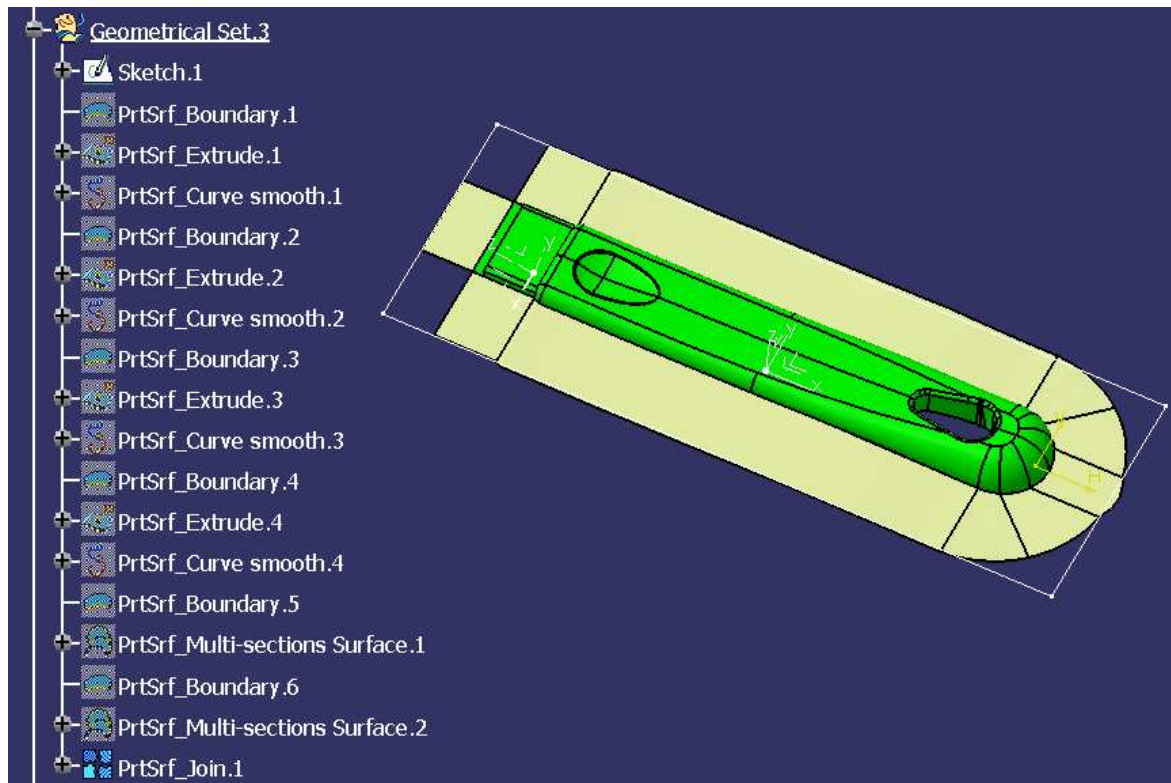




- Create a second Loft Surface as shown.

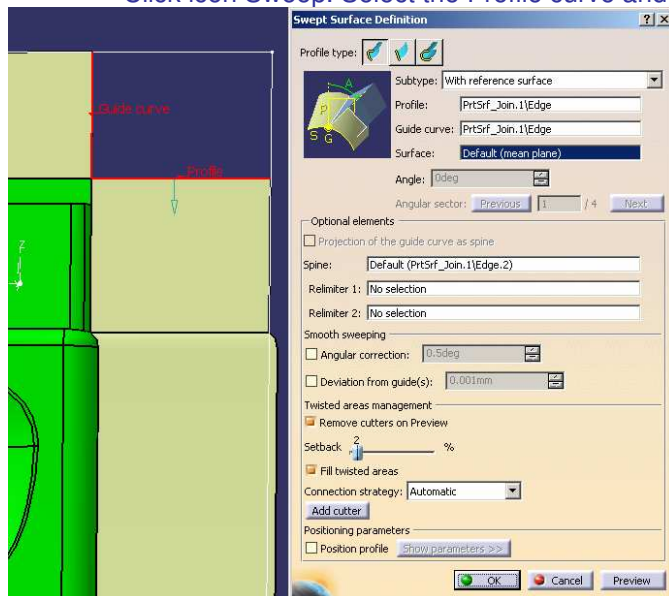


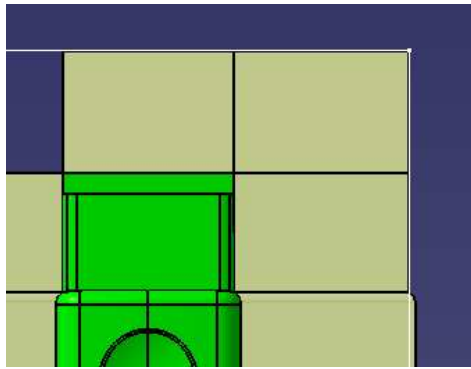
- Click OK in dialog box *Parting Surface Creation* to validate the creation of the Extrude and Loft elements. You can see that all surface elements have been created as features and automatically put into a *Join*.



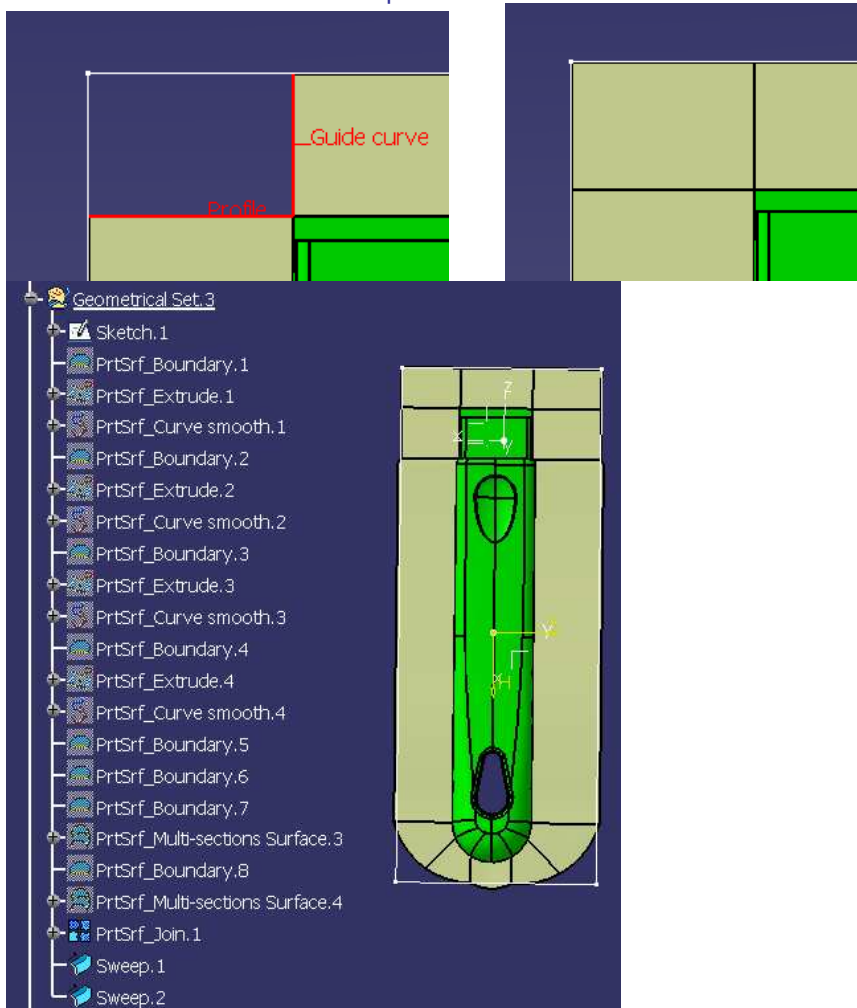
- Now we will complete the definition of the Parting Surface by using standard Sweep operations.


- Click icon Sweep. Select the Profile curve and Guide curve as shown. Validate.



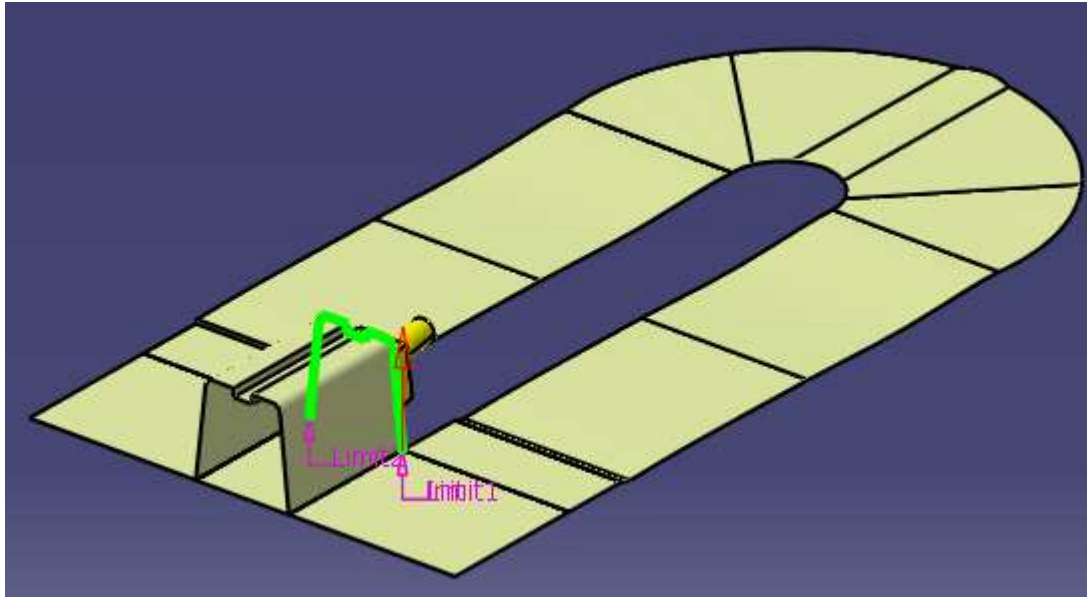


- Create a second sweep as shown.

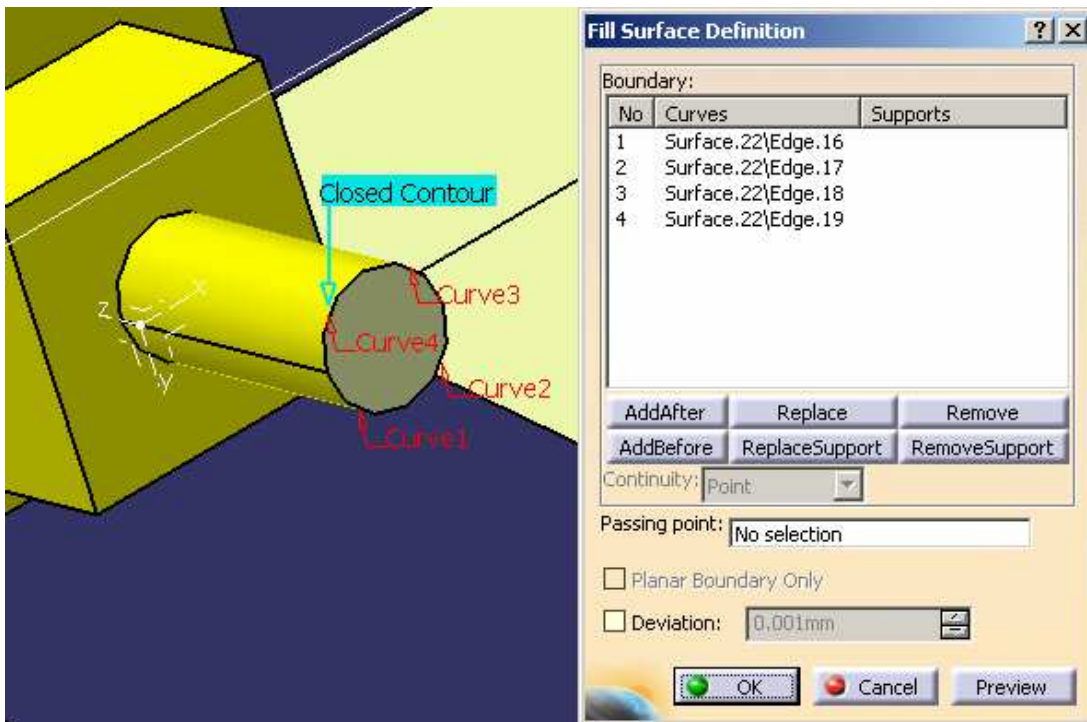


- Use the Parting Surface command again to create an extrusion on the boundary of the slider surface. 



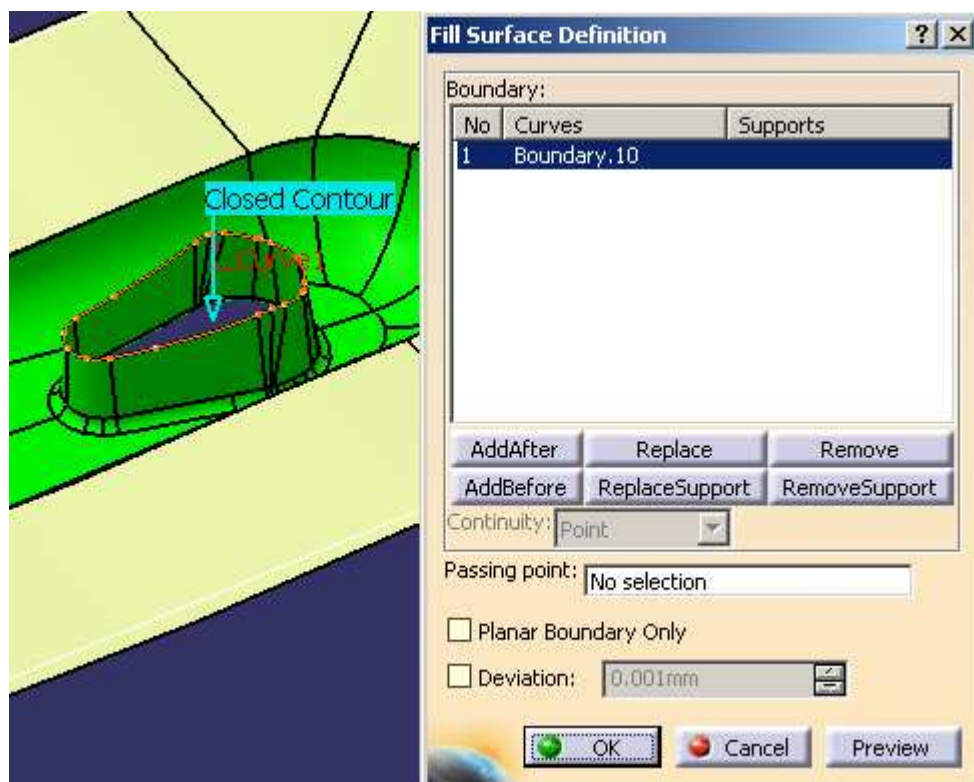



- Create the bottom of the hole in the slider by Fill



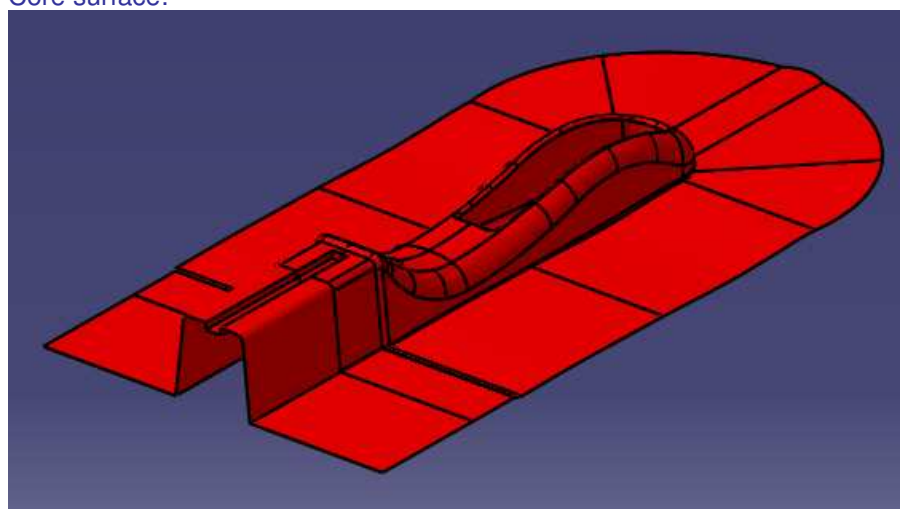
- Create the boundary of the hole in the cavity and use Fill to create the corresponding surface



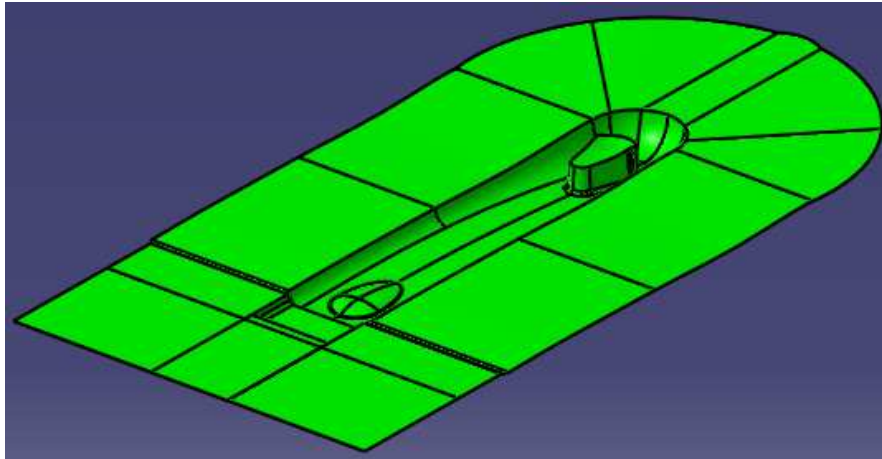


- Use Join to assemble the surfaces as shown (release the tolerance to avoid unwanted internal boundaries). 

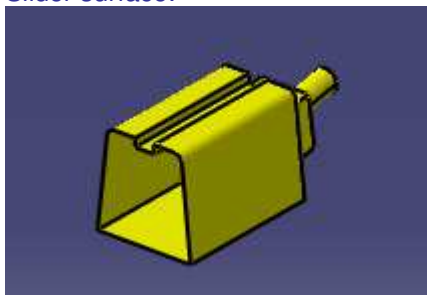
Core surface:



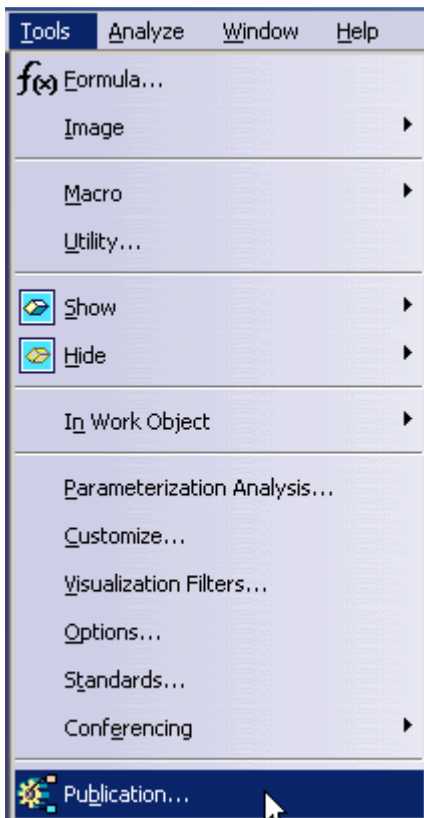
Cavity surface:



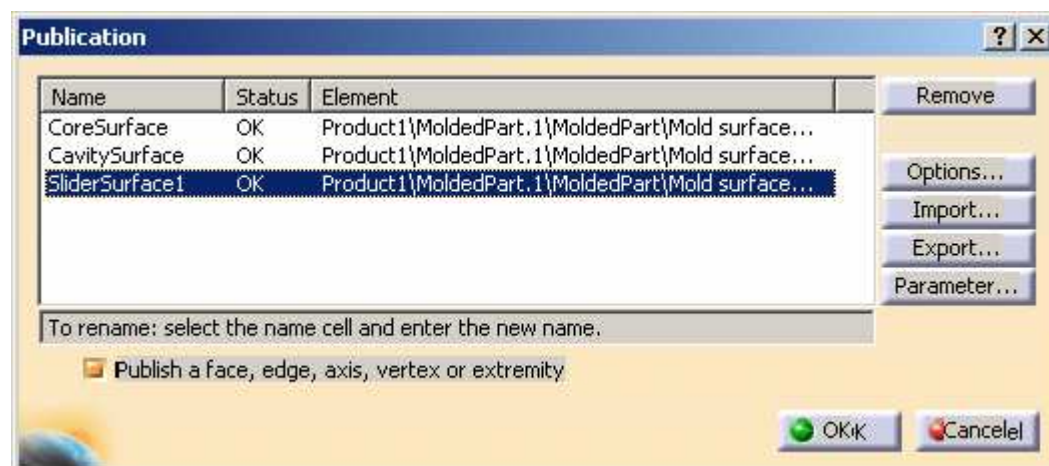
Slider surface:



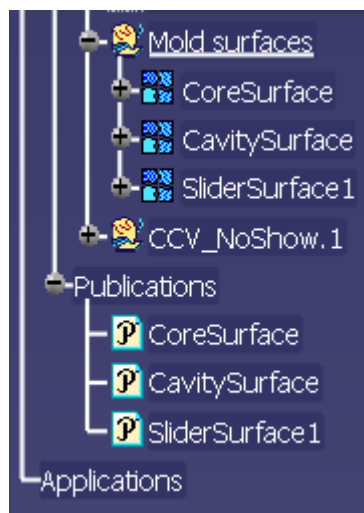
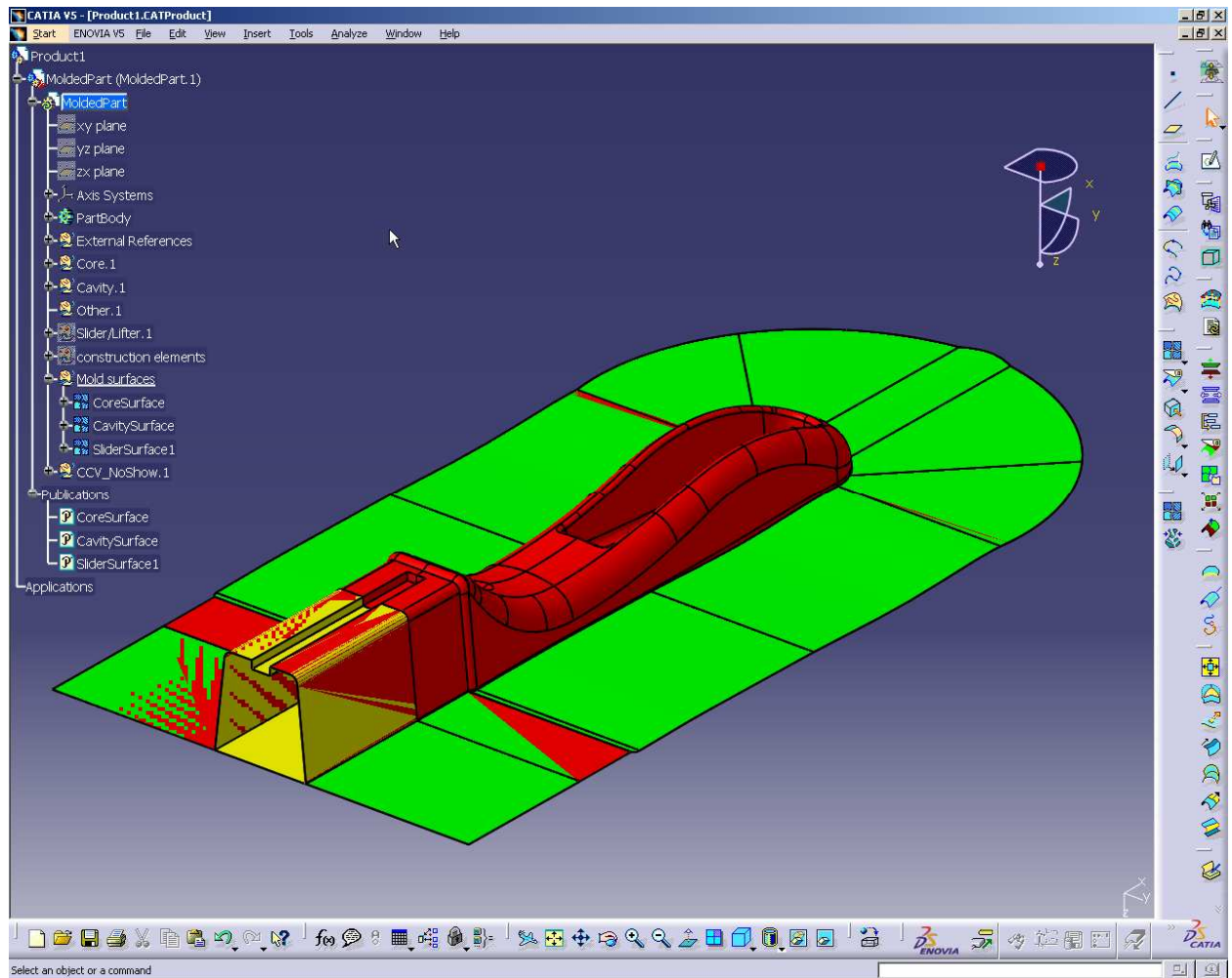
- Rename the surfaces as CoreSurface, CavitySurface, and SliderSurface1.
- Select Tools/Publications, then select the 3 surfaces in the specification tree; confirm to publish the final mold surfaces.









- The final result is available as  
\\MasterExercise\End\Master\_Exercise\_Step\_End.CATProduct.



## Additional Exercise: The Support

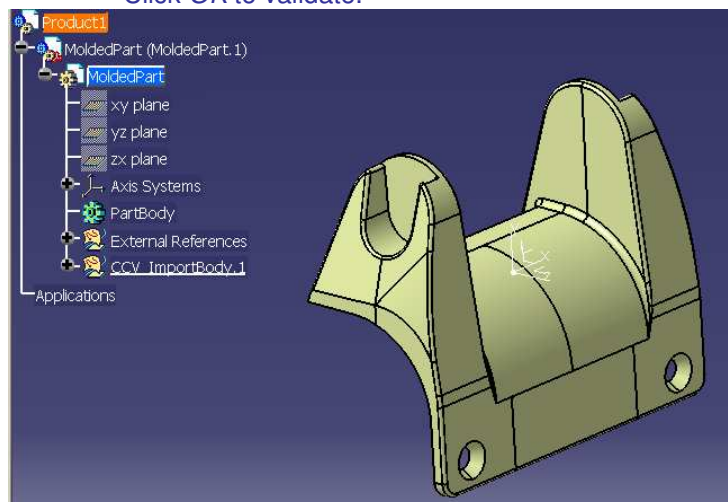
### Step 1: Checking & Reworking the Orientation of Faces

In this step, you will:

- Import a part as a set of loose surfaces
- Define a consistent orientation for the surfaces
- Create a Part in a Product and enter the workbench *Core & Cavity Design*
- Click icon *Import Model*  and select `\AdditionalExercise\Step1\Support.CATPart` as a set of surfaces: 
- Define Shrinkage as *Scaling* with *Ratio = 1.1* and keep *Bounding box center* as the Axis system

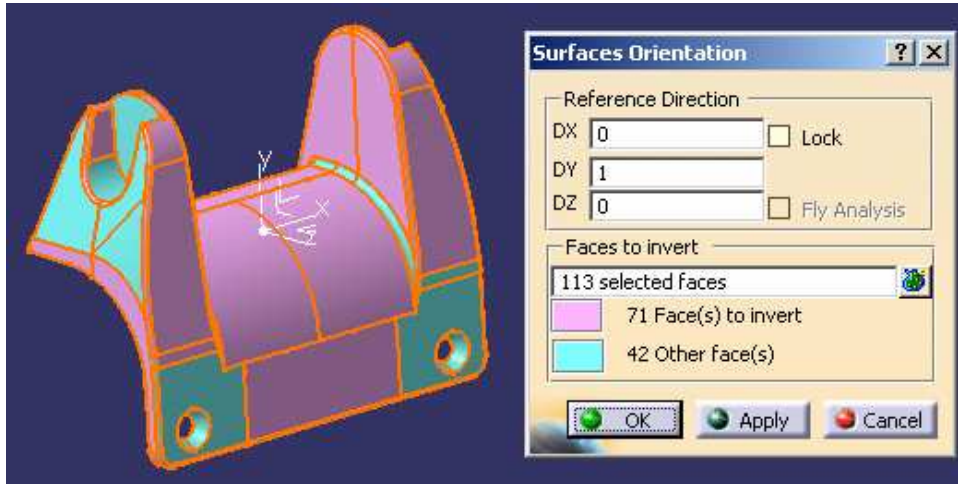


- Click *OK* to validate.

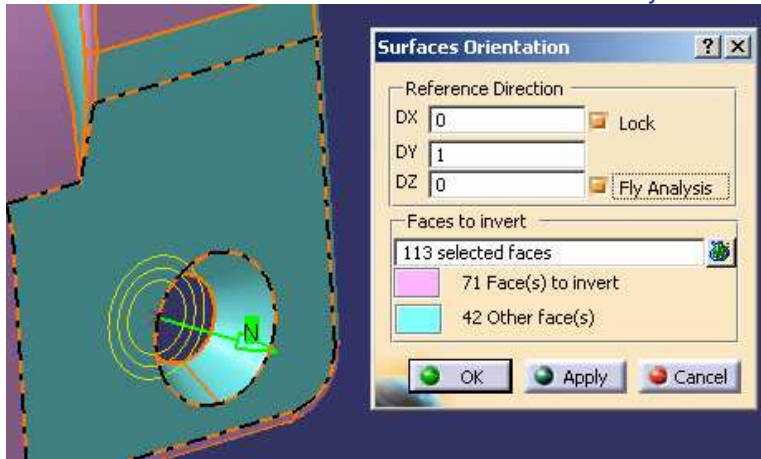


- Click icon *Face Orientation* 

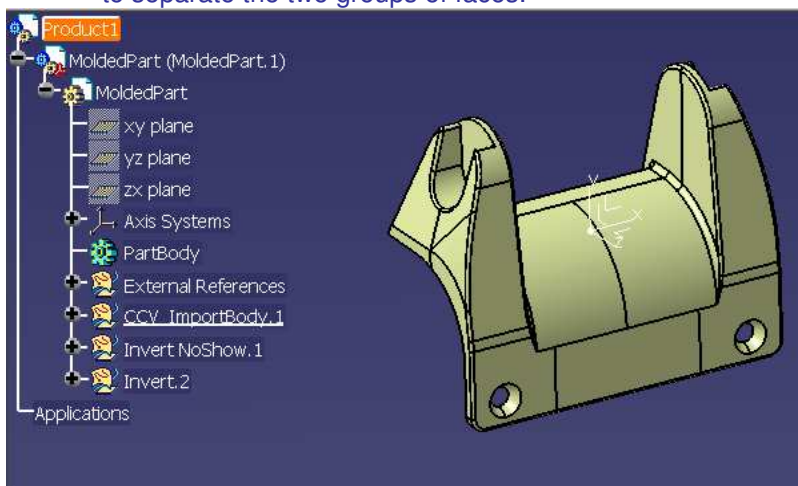
- Select all the surfaces with a trap and set the Reference Direction as shown below (it will be the pulling direction). Click *Apply*. The two groups of faces (faces to invert and other faces) are identified.



- You can check the orientation of each face by activating options *Lock* and *Fly Analysis*



- Confirm with *OK*. Now the faces have a consistent orientation with regard to the pulling direction to be used later. Notice that two Geometrical Sets have automatically been created to separate the two groups of faces.



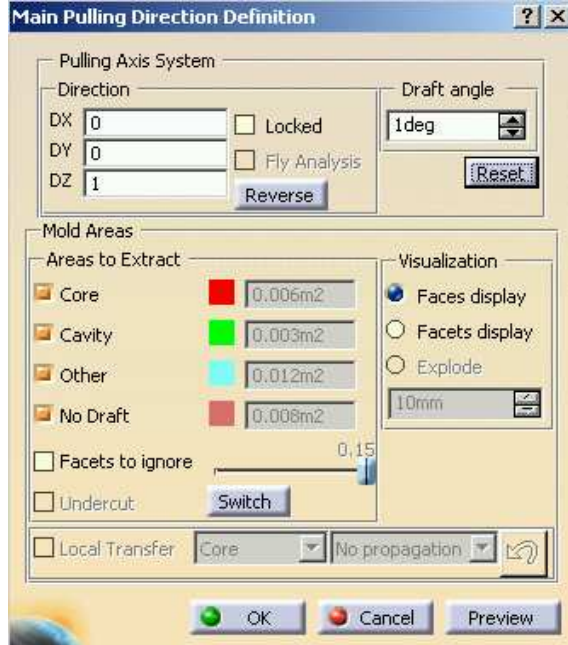
- Close the document without saving it if you want to use existing prepared data for the next step.

## Step 2: Defining the Mold Areas

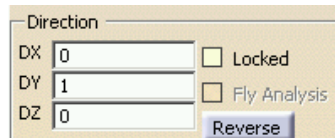
In this step, you will define the main pulling direction of the part in order to separate the different areas.

If you have not kept the results of the previous step, open existing Product *\AdditionalExercise\Step2\Additional\_Exercise\_Step\_2.CATProduct*, make its Part active and select workbench *Core & Cavity Design*.

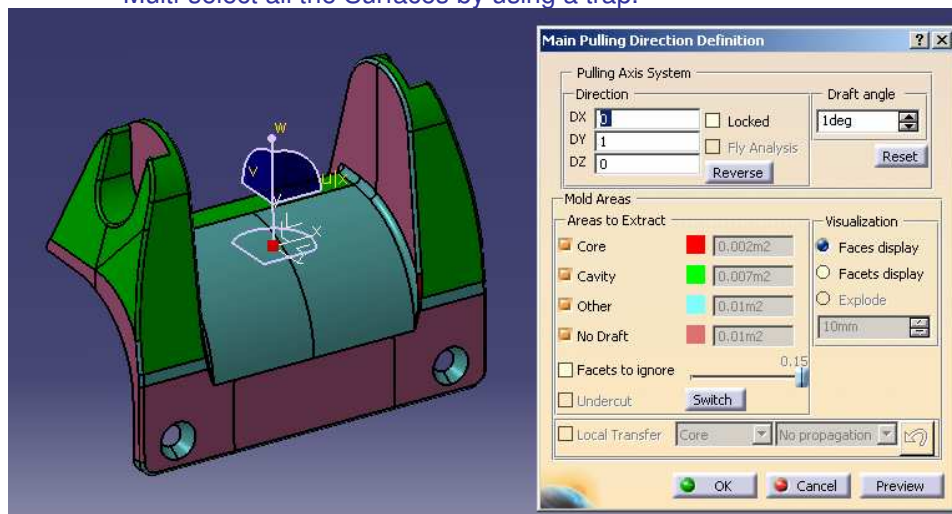
- Click on the icon Main Pulling Direction Definition 



- Set the Main Pulling Direction to  $(DX = 0, DY = 1, DZ = 0)$ .

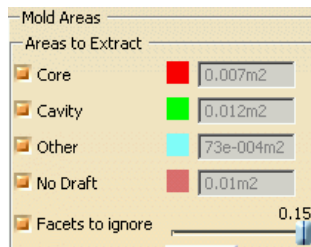


- Multi-select all the Surfaces by using a trap.



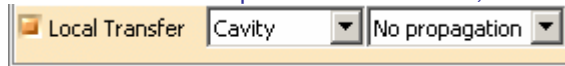
- Activate option *Facets to Ignore* and keep maximum value = 15%.





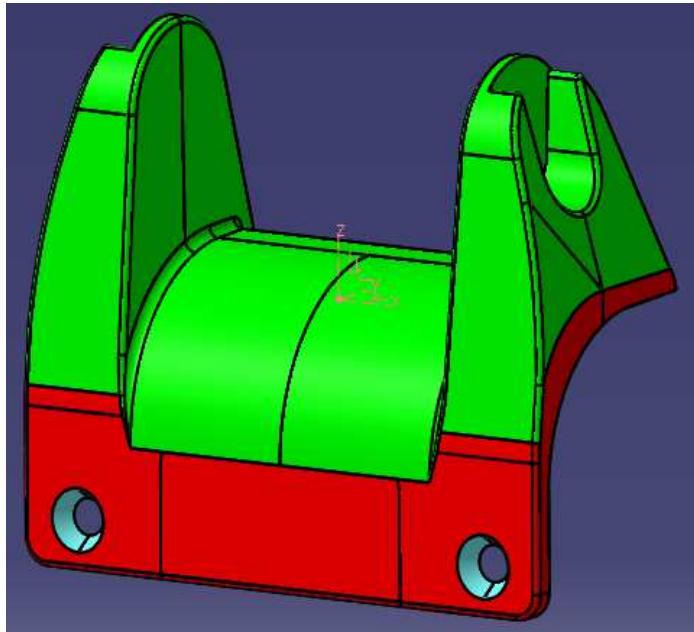
Now we have to transfer the remaining surfaces into the relevant areas. First, activate the *Lock* option.

- Activate option *Local Transfer*, select Destination = *Cavity* and *No propagation*.



- Pick all the faces to transfer to the cavity side.
- Select Destination = *Core* and pick all faces to transfer to the core side.
- Idem for area Other

The result must be as shown:



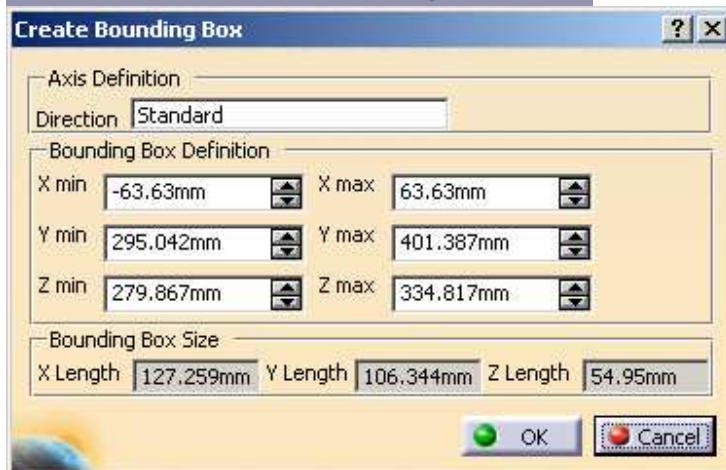
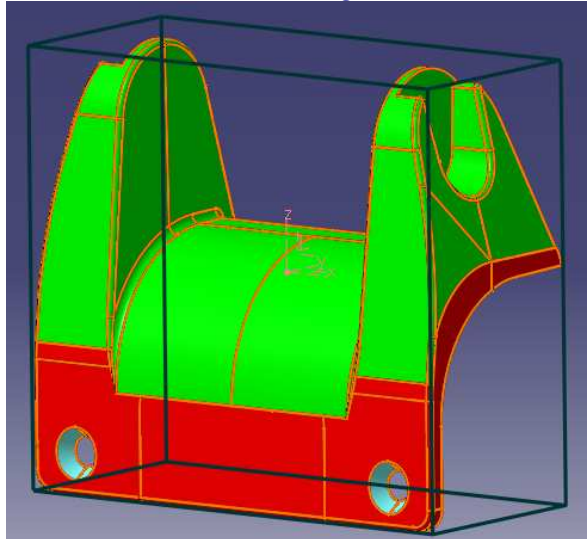
- Close the document without saving it if you want to use existing prepared data for the next step.

## Step 3: Creating the Bounding Box

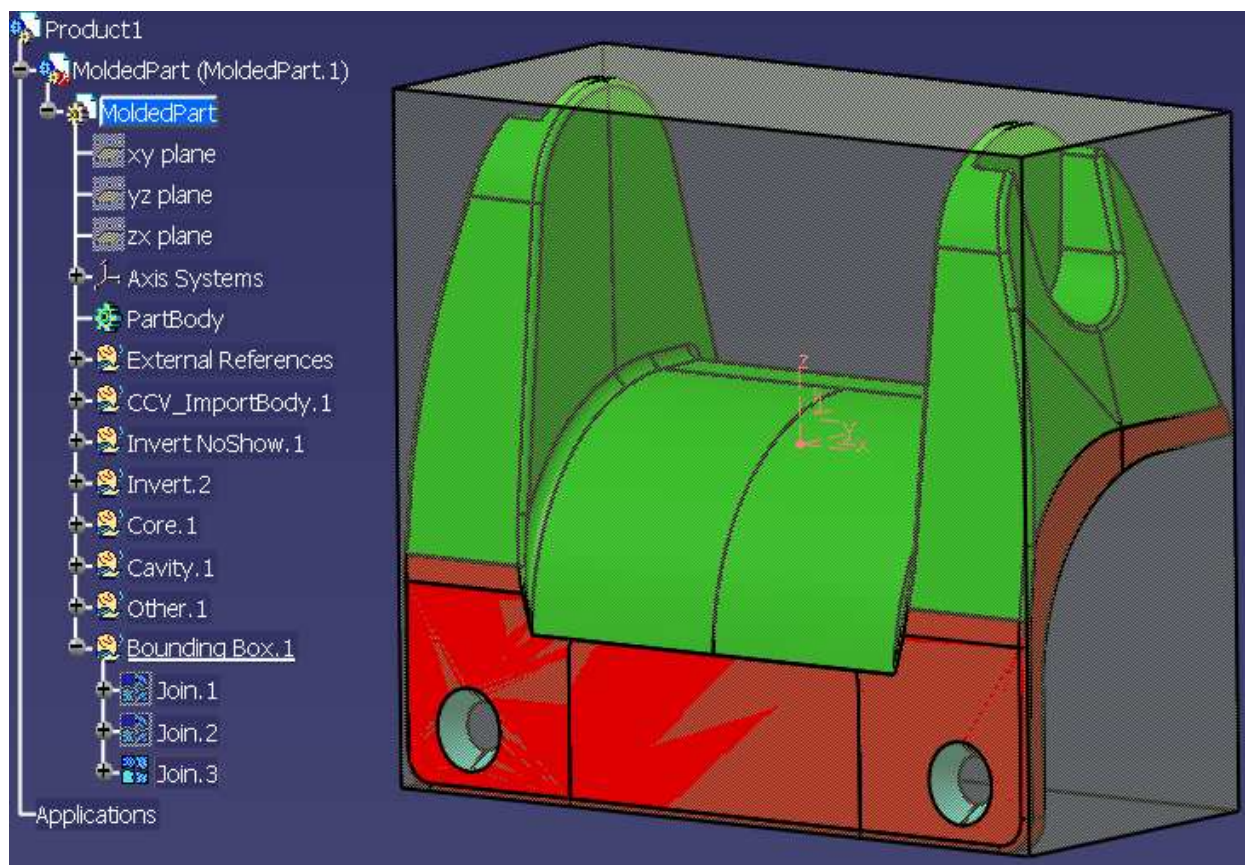
In this step, you will create a Bounding Box around the part

If you have not kept the results of the previous step, open existing Product *\AdditionalExercise\Step3\Additional\_Exercise\_Step\_3.CATProduct*, make its Part active and select workbench *Core & Cavity Design*.

- Click icon *Bounding Box*. Select all the Surfaces by using a trap. 



- Click *OK* to confirm. The bounding box is created (*Join* feature).



- Close the document without saving it if you want to use existing prepared data for the next step.



## Step 4: Comparing Two Versions of the Part

In this step, you will compare two versions of a Part to identify the geometric differences.

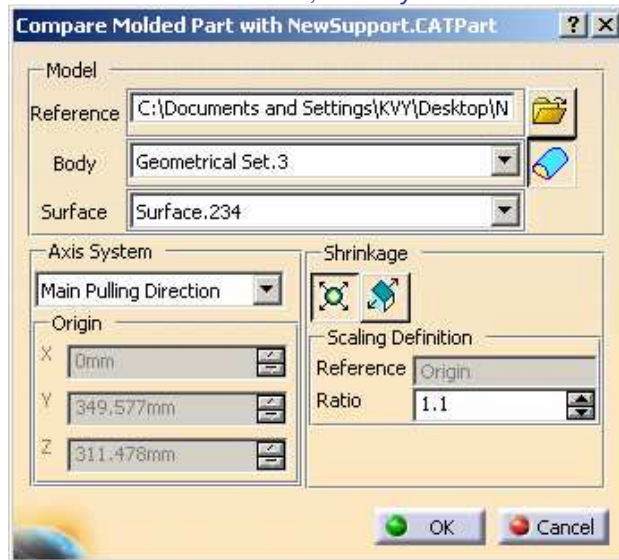
If you have not kept the results of the previous step, open existing Product *\AdditionalExercise\Step4\Additional\_Exercise\_Step\_4.CATProduct*, make its Part active and select workbench *Core & Cavity Design*.

- Hide the Bounding Box.

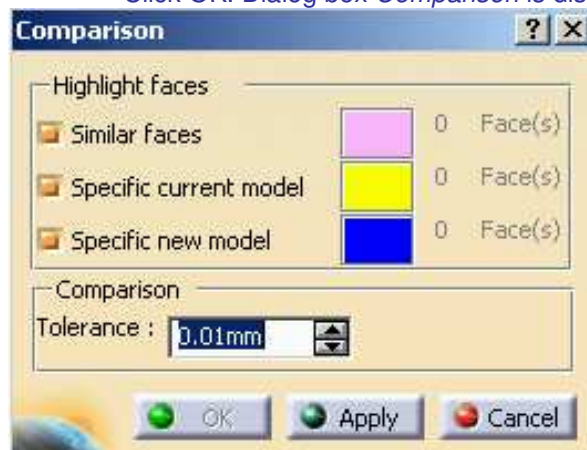


- Click icon *Compare*. Select file *\AdditionalExercise\Step4\NewSupport.CATPart*. Click in the dialog box the type icon to select type "Set of loose surfaces". Check that selected Body =

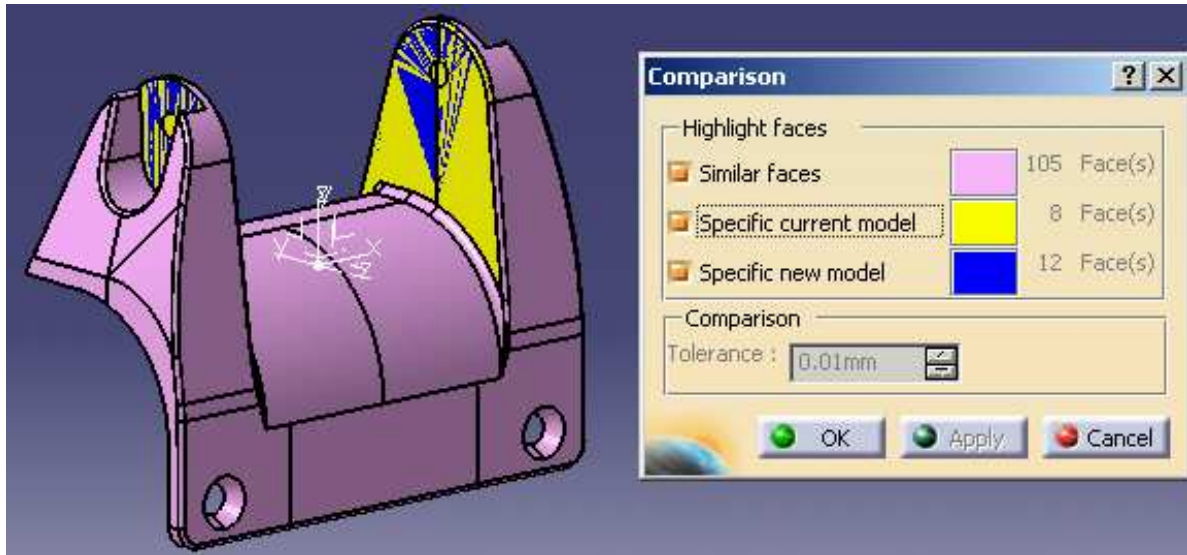
*GeometricalSet.3*, Axis System = *Main Pulling Direction* and Scaling Ratio = 1.1.



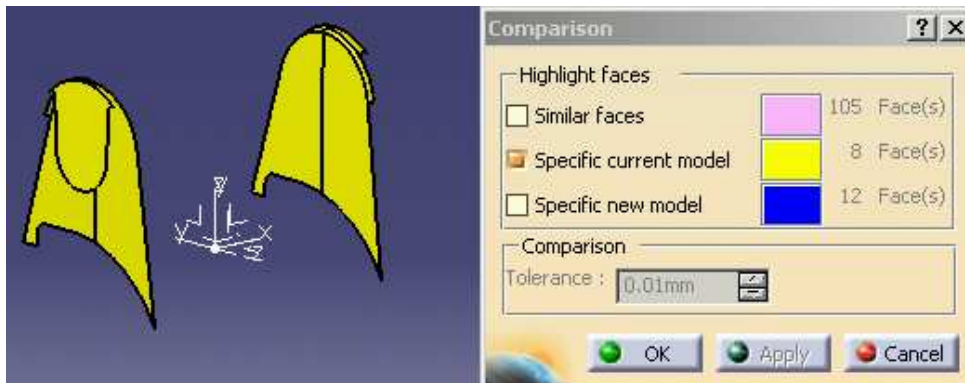
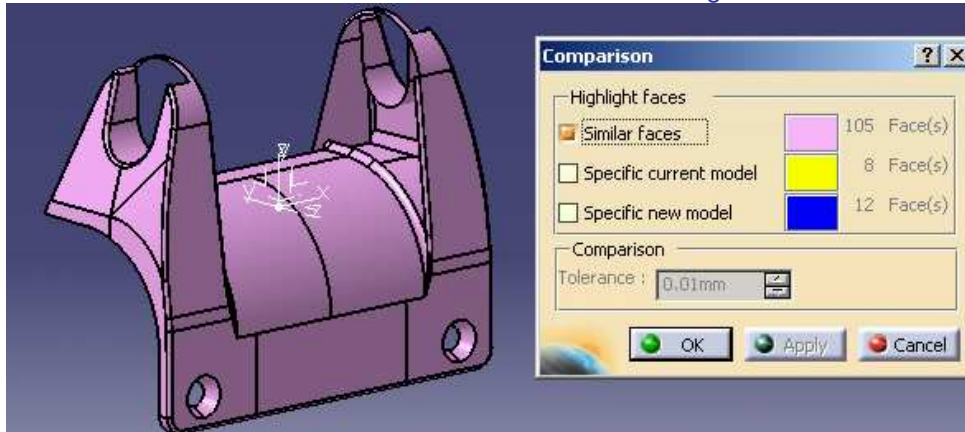
- Click OK. Dialog box *Comparison* is displayed. Keep tolerance 0.01mm and click *Apply*.



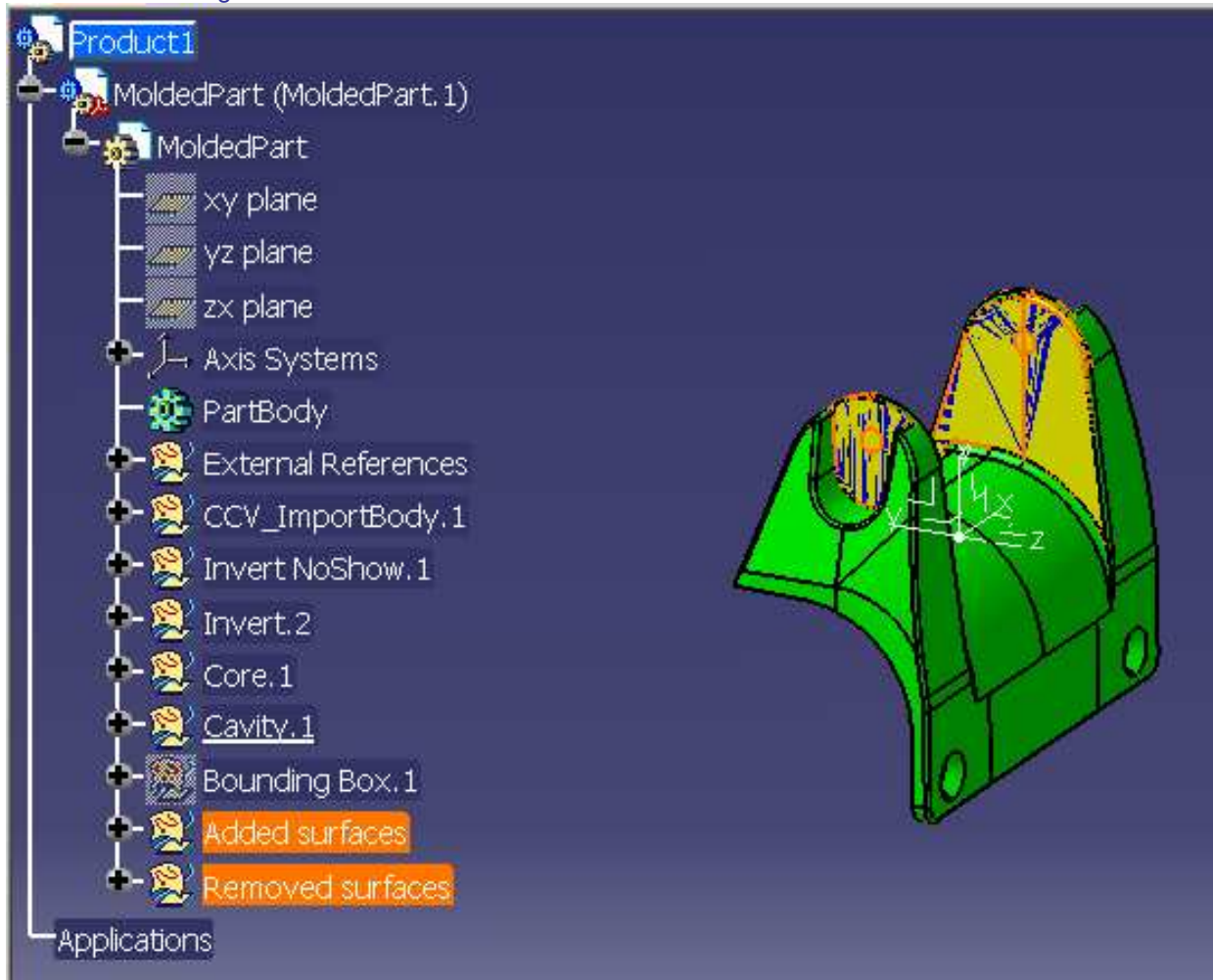
- The faces of the first and of the second Part are compared and classified into 3 groups: identical faces, faces specific to version no.1, faces specific to version no.2.




- Check/uncheck the buttons to visualize the 3 categories in the 3D viewer.

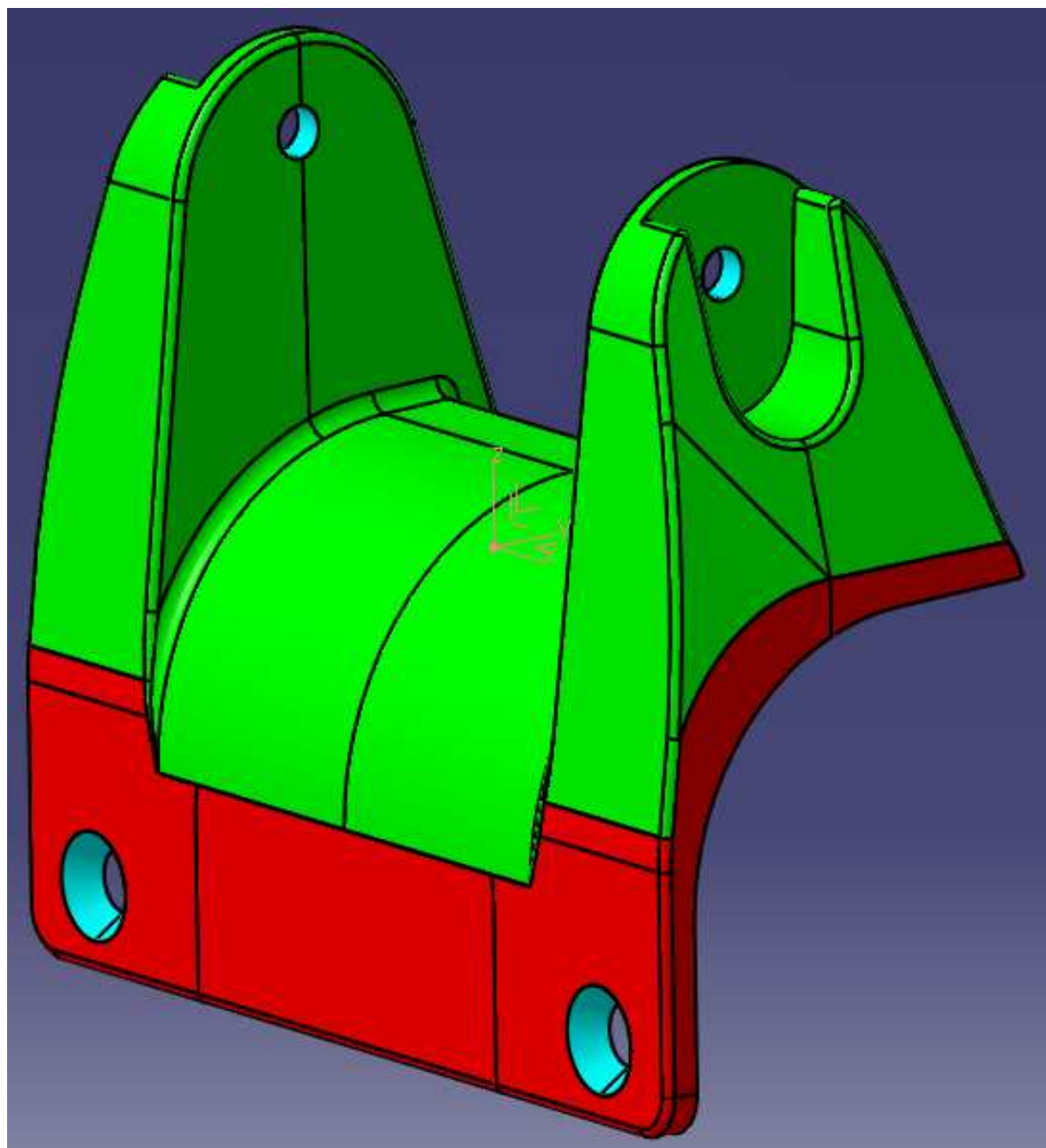



- Click OK to validate. Two new Geometrical Sets have automatically been created: *Added surfaces* and *Removed surfaces*. Geometrical Set *Removed surfaces* can be deleted. Surfaces in Geometrical Set *Added surfaces* should be analyzed to complete the definition of the existing Mold Areas.



- User Transfer an Element to send added surfaces to their proper destination (cavity and other). 

The result must be as shown:



- Now Aggregate Core, Cavity and Other with datum joins. 
- The final result is available as  
\\AddedExercise\End\Master\_Exercise\_Step\_End.CATProduct.