

Part Design

Part 2

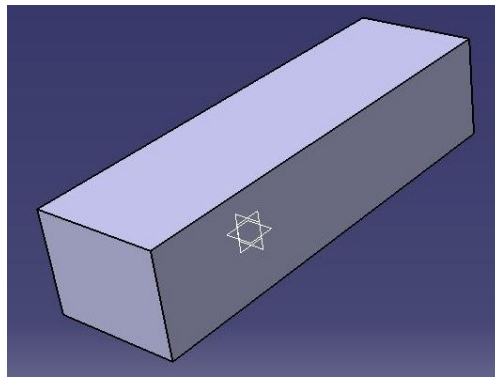
Modifications Part

The Dress-Up Features

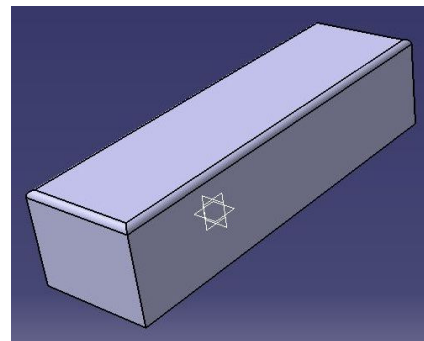
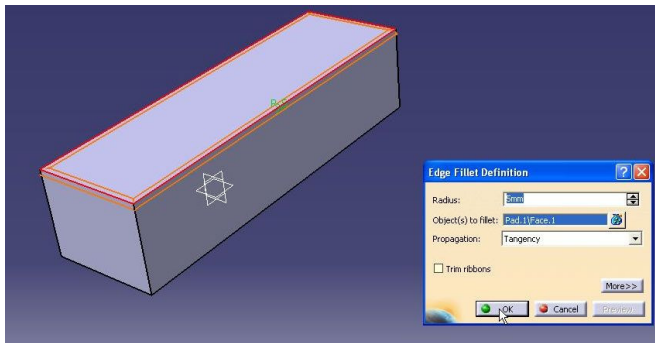
Edge Fillet

Example No. 1.


1. Create a simple pad.

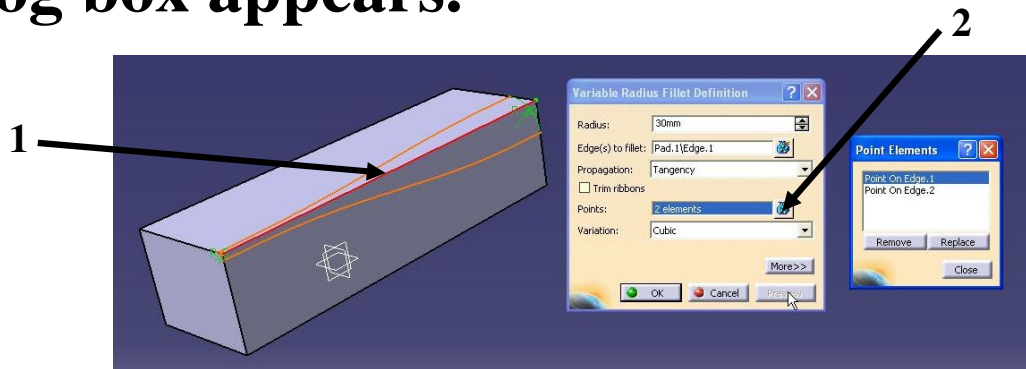


2. Select Edge Fillet icon . Then, select surface on solid or select four edges to be filleted. Enter the Radius value and confirm by clicking Preview button and select OK.



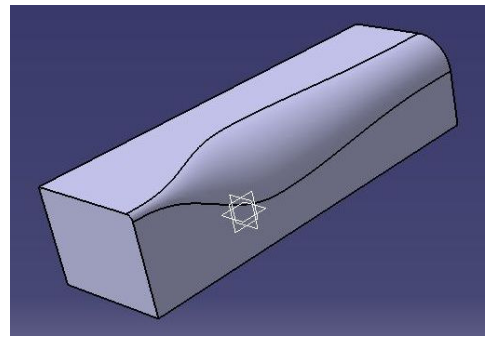
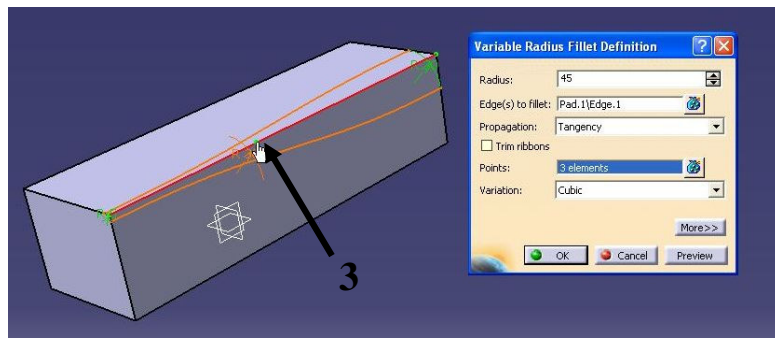
Example No. 2. Variable Radius Fillet.

1. Create simple pad or you can use the previous pad. Select Variable Radius Fillet icon . The Variable Radius Fillet Definition dialog box appears.




2. In Edge(s) to fillet option select edge 1. Then, click on the Points icon (2). In Point Elements dialog box select Point on Edge.1 and enter the Radius value (for example 30) and click two times Preview button.

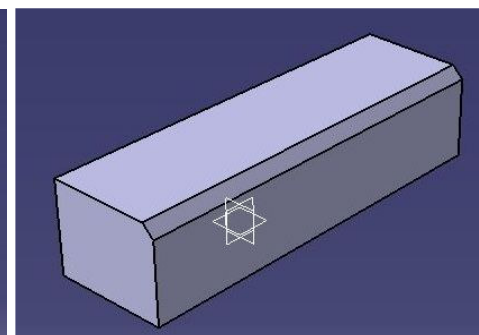
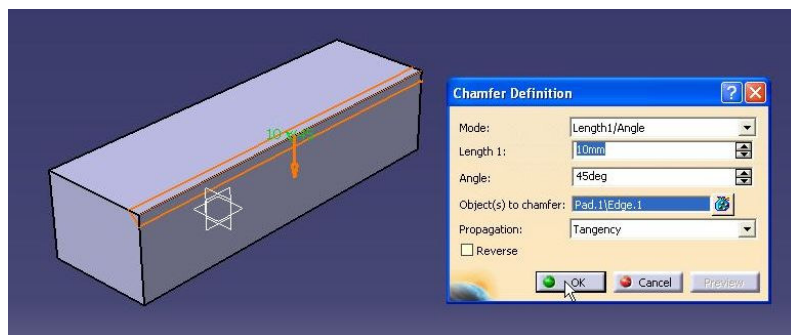
3. If you want to create a new point which will be a beginning of new radius, click on the Points icon and show the point (3) on the edge (1). Next, enter the Radius value (for example 45). Confirm by clicking Preview button two times and select OK.



Chamfer

Example No. 3.

1. Create a simple pad or you can use the previous pad. Select Chamfer icon . Then, chose Mode of chamfer between Length1/Angle and Length1/Length2. If you chose Mode Length1/ Angle enter the Length 1 and the Angle value. If you select Mode Length1/Length2 enter the Length 1 and Length 2 value. Then, select the edge to chamfer and confirm by clicking Preview button two times and select OK.

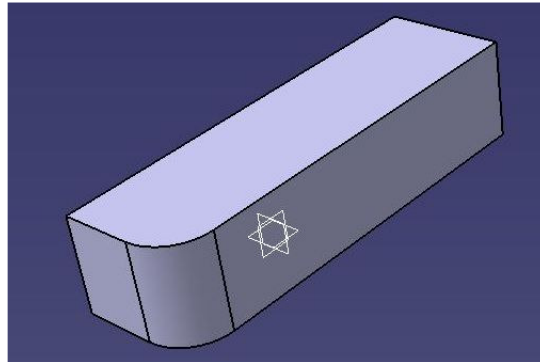



Draft Angle

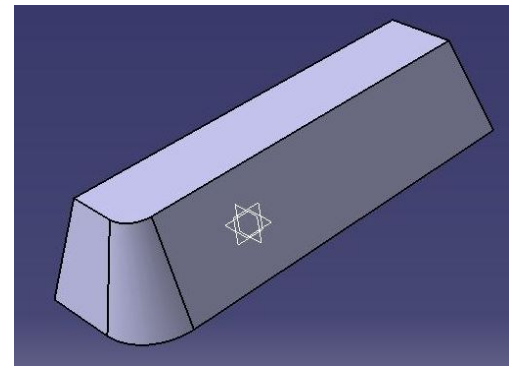
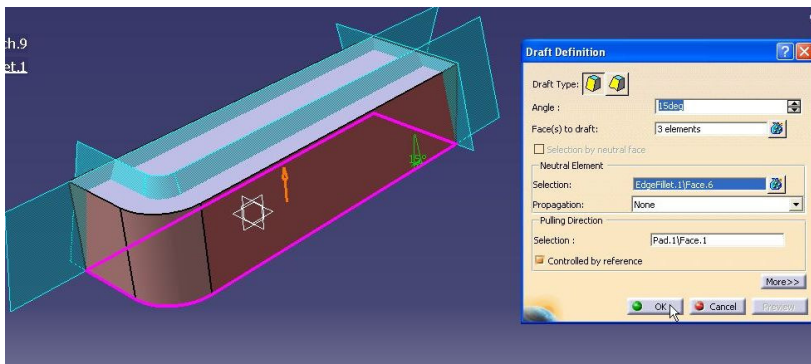
Example No. 4.

Using Draft operations material can be added or removed to the parts during the operations. It depends on pulling directions.

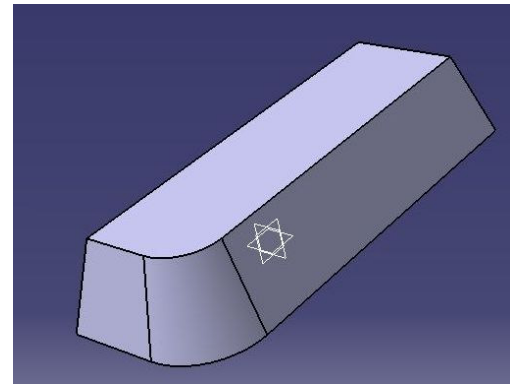
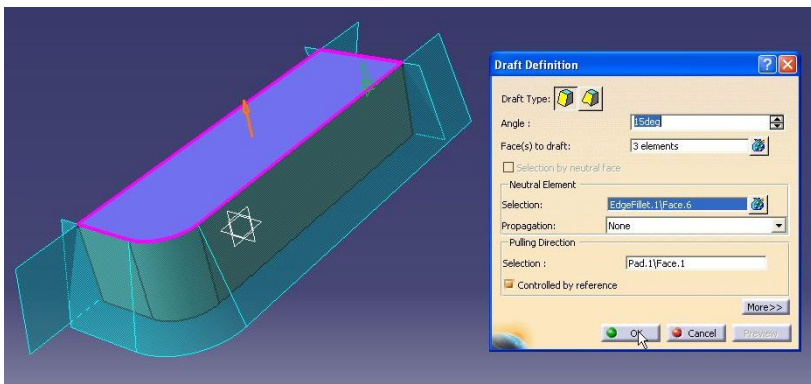
1. Create a simple pad.




2. Select Draft Angle icon . The Draft Angle Definitions dialog box appears. Select faces around the pad to be drafted. Then, define the Draft Angle. Then, select the Natural Element (if you want to remove material, select the bottom surface on the pad). Confirm by clicking Preview button and select OK.

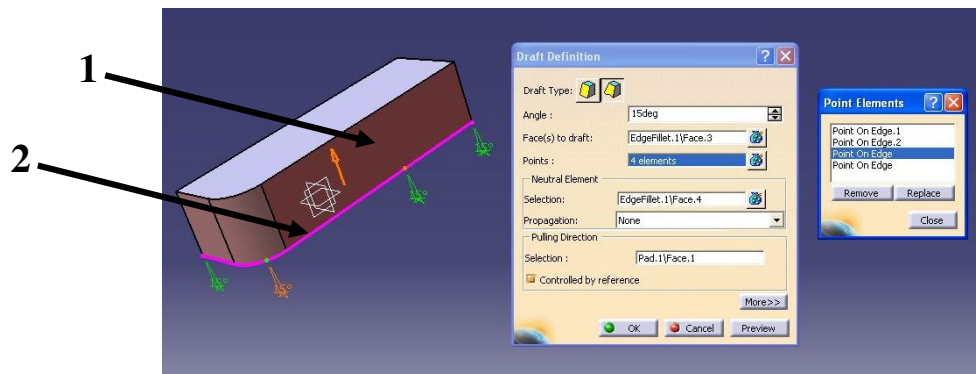


3. If you want to add material, select the top surface on the pad. If you need to change Pulling Directions click on the arrow. Then, confirm by clicking Preview button and select OK.

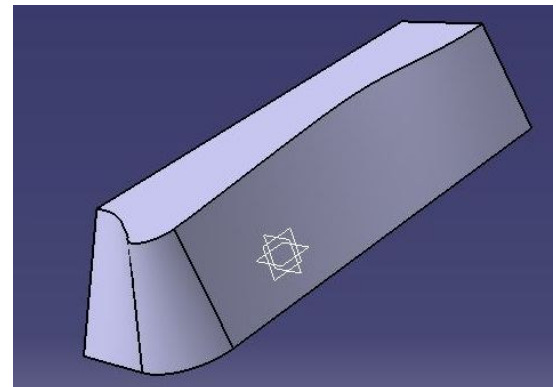
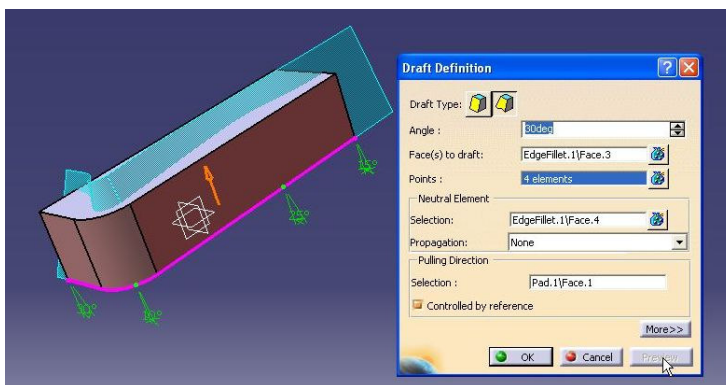


Example No. 5. Variable Angle Draft

1. Create a simple pad or you can use the previous one. Select Variable Angle Draft icon  and select the surface 1 (you cannot create Variable Angle Draft with multiple faces). Then, select the Natural Element (chosed the bottom surface on the pad).




2. Then, create two new vertices on the edge 2 which will begin a new angle draft. Click the Points icon and define vertices on the edge 2. Next, enter the new values of Angle Draft (click on displays values and define new values) Confirm by clicking Preview button two times and select OK.

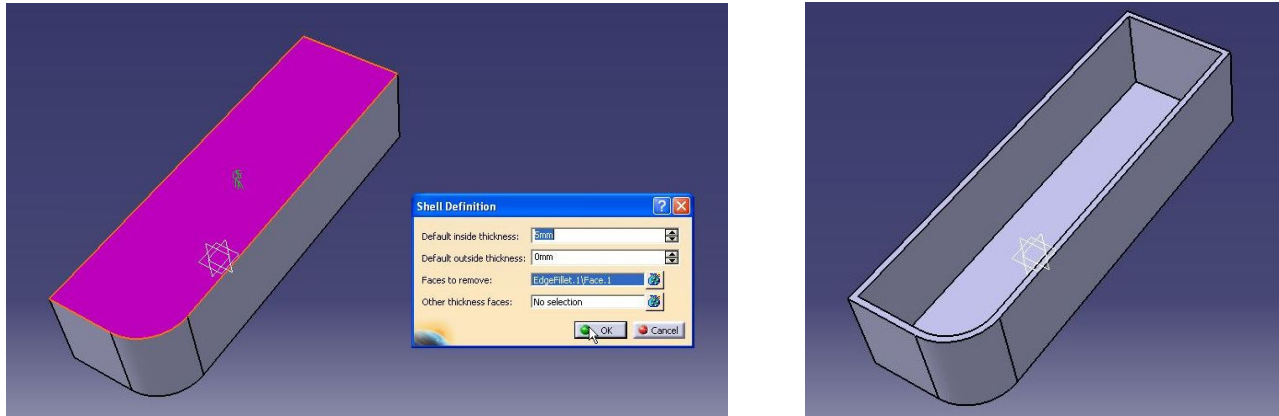


Shell

Example No. 6.

Shelling has a feature of emptying, while keeping a given thickness on its sides.

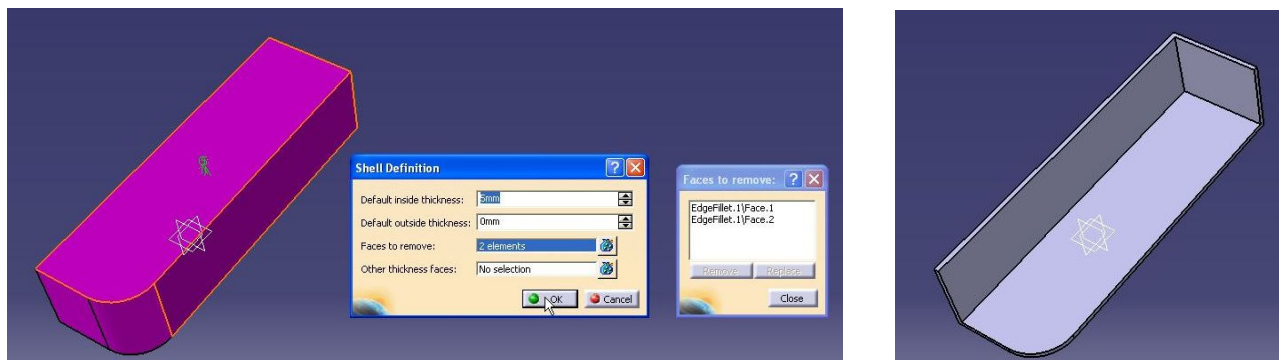
1. Create a simple pad or you can use the previous one. Select Shell icon . Then, select the face to remove and enter the Default inside thickness value (wall thickness will be created to inside of the pad) and select OK.



If you want to create wall thickness to the outside, enter the Default outside thickness value. You can also create wall thickness in booth directions.

You can remove more than one face.

3. Select Shell icon. Then, click the Faces to remove icon and select faces to be removed and confirm by clicking OK.



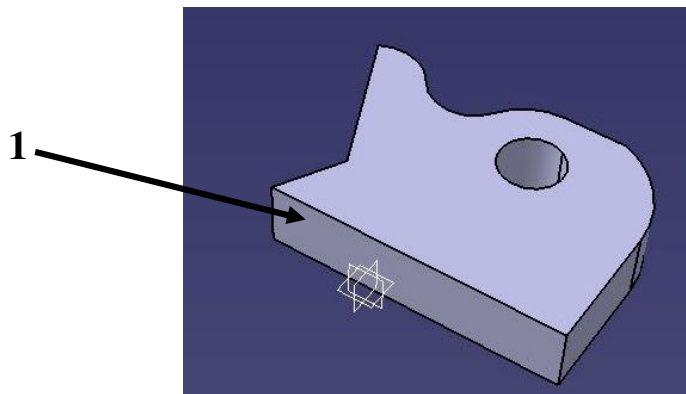
The Transformation Features

Mirror

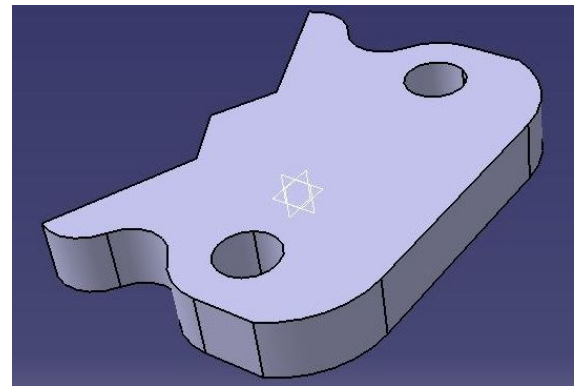
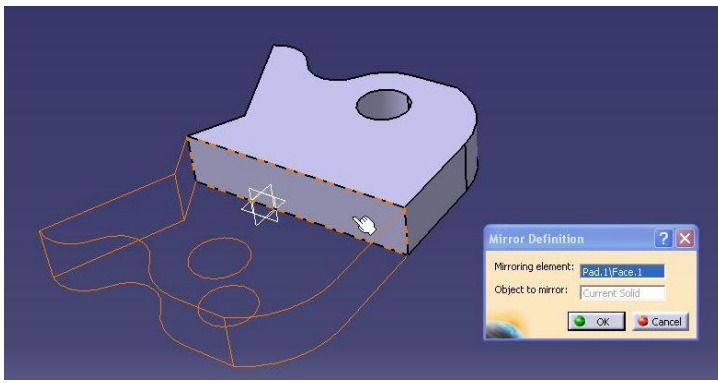
If you build the symmetry part you can search the symmetry line and build only half of the part and use the Mirror option to get whole part.

Example No. 7.

1. Open the Mirror7 file.

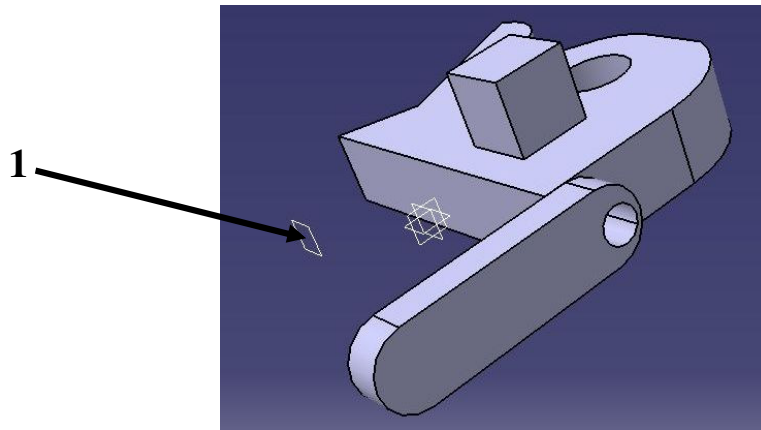


2. Select Mirror icon . Then, select the planar face (1) which will be plane of symmetry. Confirm by clicking OK.

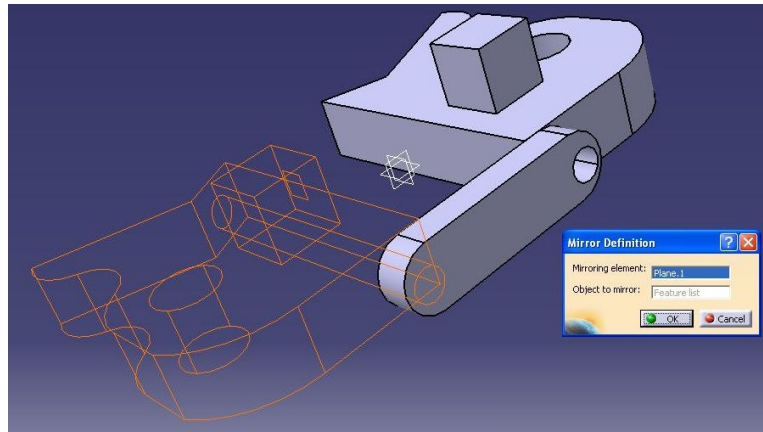


Example No. 8. Mirroring several objects.

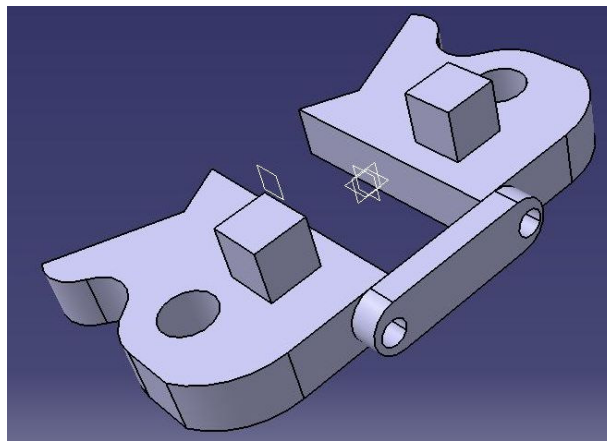
1. Open the Mirror8 file.



2. Select Pad.1, Hole.1, Pad.2 and Hole.2 objects with the Ctrl key. Then, click the Mirror icon and select the plane 1 which will be a Mirroring element.



3. Confirm by clicking OK.

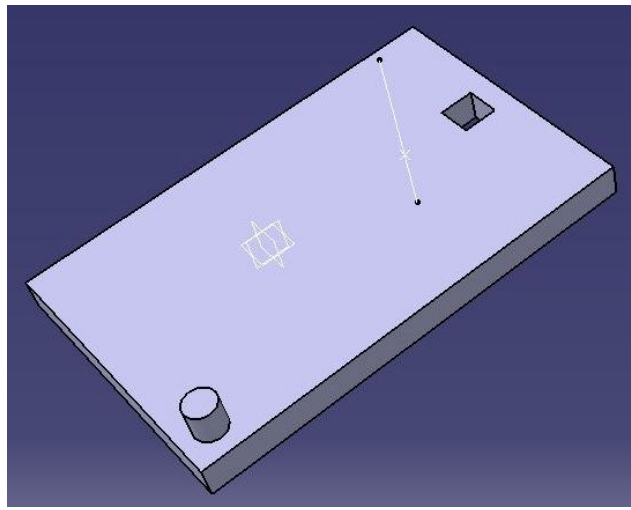



Pattern

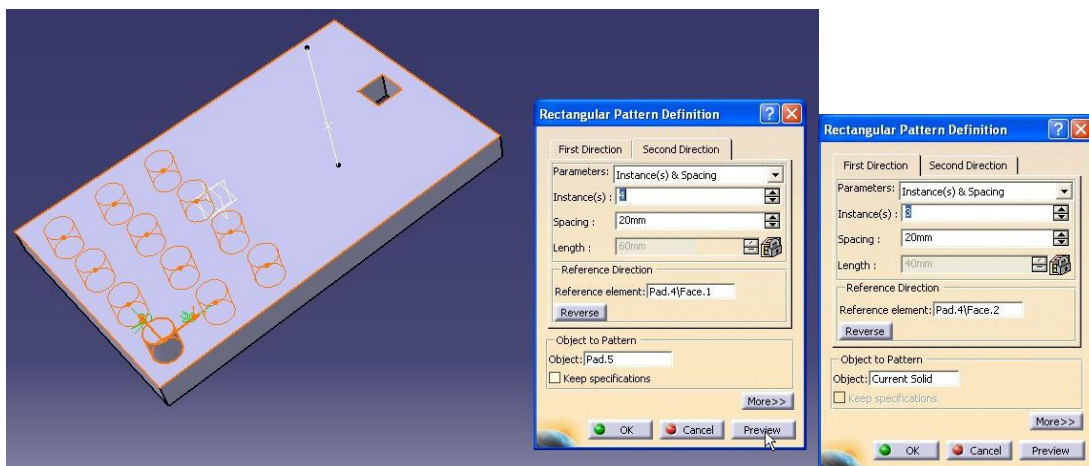
Pattern Options enable to create several identical object from the existing one simultaneously.

Example No. 9. Rectangle Pattern

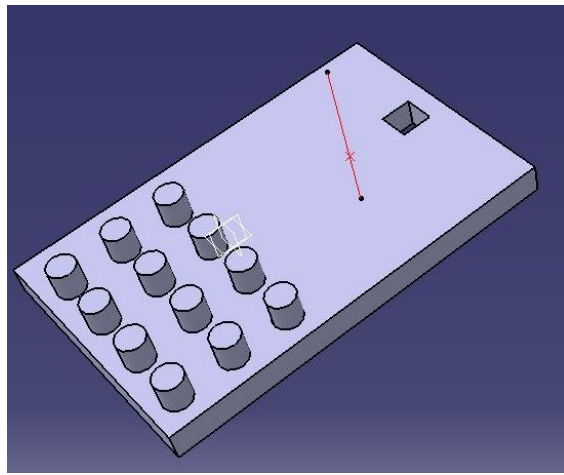
1. Open Pattern9 file.




2. Select the Pad.5 and select Rectangle Pattern icon . The Rectangle Pattern Definitions dialog box appears.

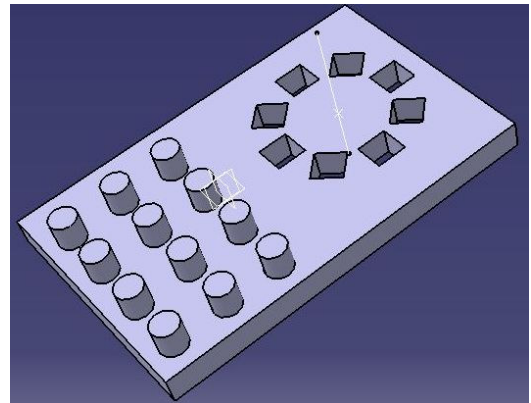
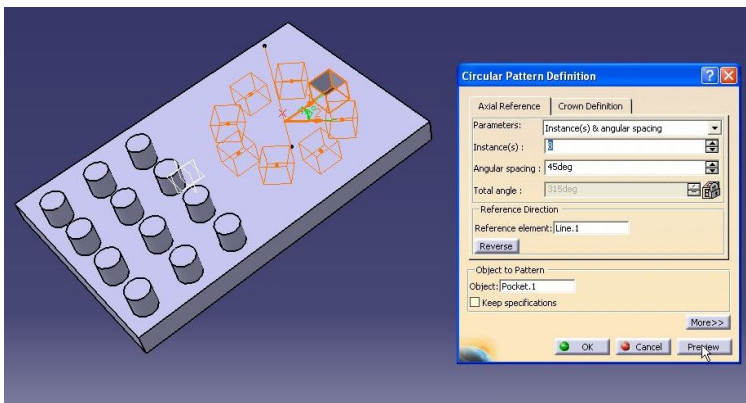


3. In First Direction options enter the Instance(s) value (for example 3). In Reference element select the top face and clicking on Reverse button change direction. Then, in Second Directions options enter the Instance(s) value (from example 4) and clicking on Reverse button change a direction. Select Preview button and click OK.



Example No. 10. Circular Pattern

1. Select the Pocket.1 and select Circular Pattern icon . In Axial Reference options enter the Instance(s) value (for example 8). Next, you can change the Angle spacing value. In Reference element select the vertical line (pocket will be revolving around this line). Select Preview button and click OK.



Reference Elements

Using Reference Elements toolbar you can create points, lines and planes without using the Sketcher.

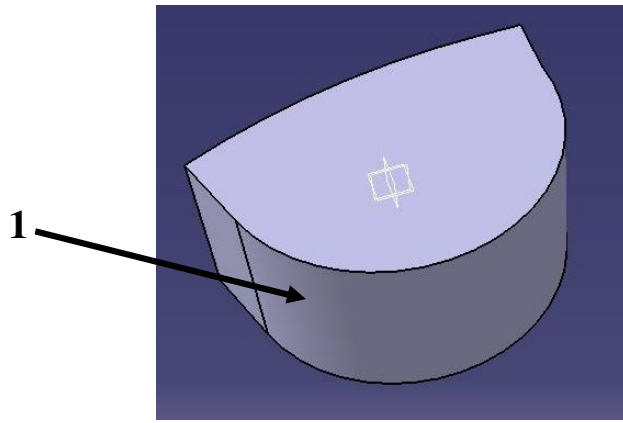
Point


Points can be created with the use of several methods:

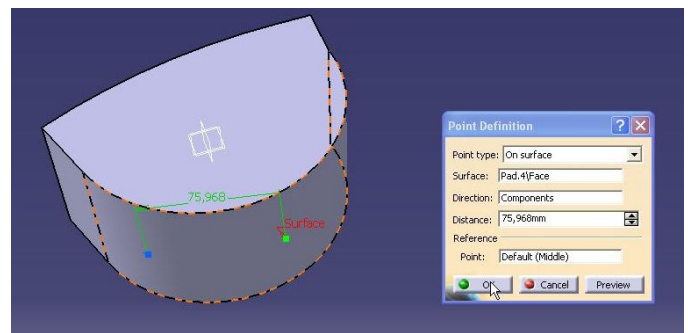
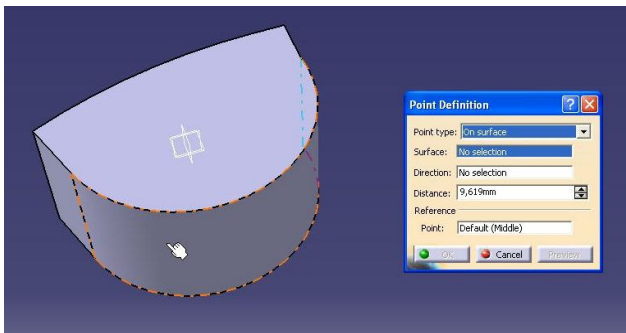
- **by coordinates**
- **on curve**
- **on plane**
- **on surface**
- **circle/sphere center**
- **tangent on curve**
- **between**

Example No. 11. Create points on surface

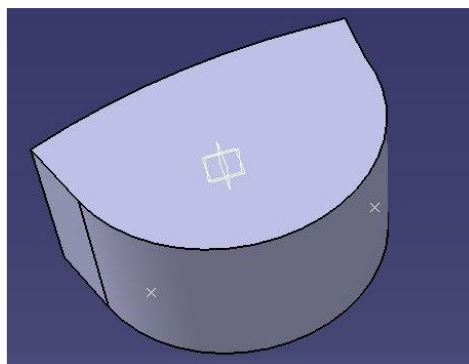
1. Open Point11 file.



2. Select Point icon . The Point Definitions dialog box appears. In Point type options select “On surface.” Then, select surface 1 and choose the position of point on this surface. Confirm by clicking OK.

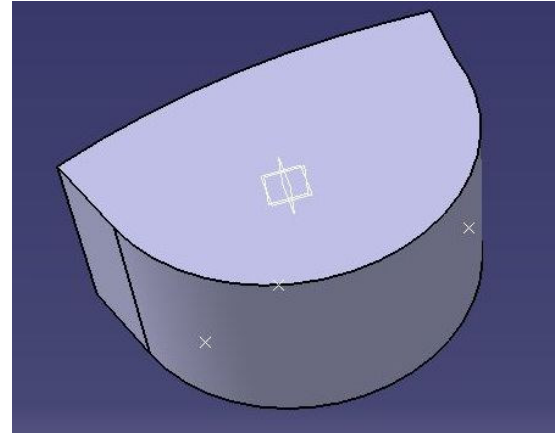
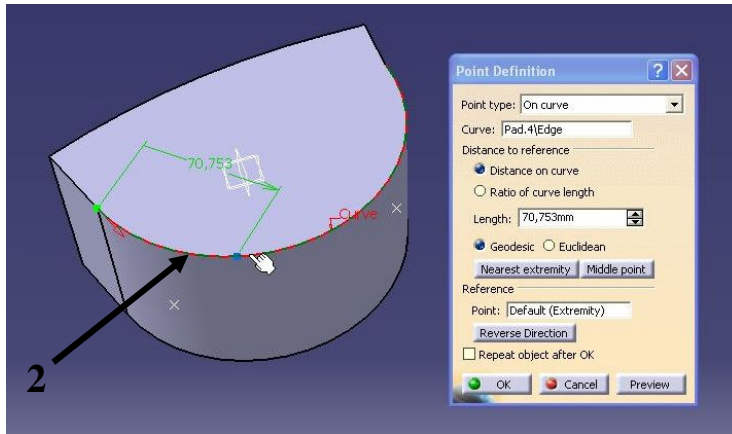


3. Create a second point on the same surface.



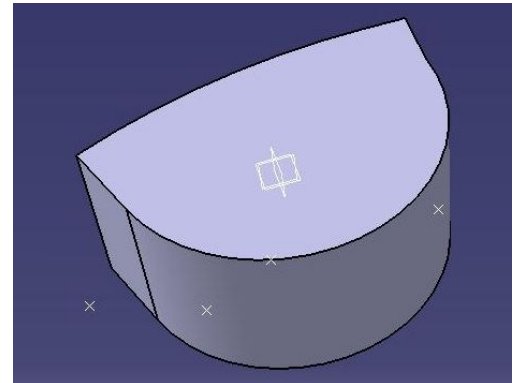
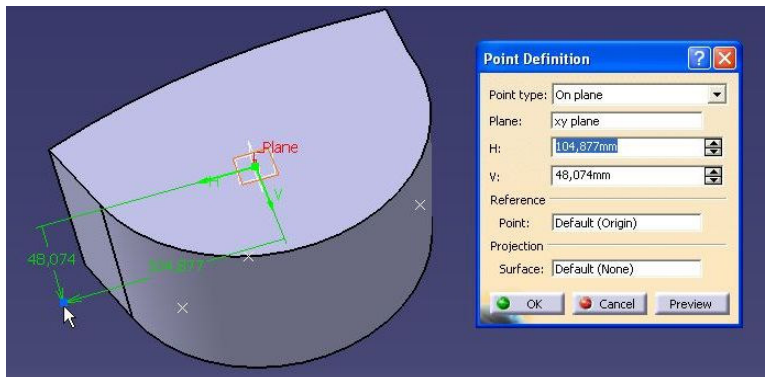
Example No. 12. Create points on curve

1. Use the same pad. Select the Point icon. In Point type options select “On curve.” Then, select curve 2 and choose the position of point on this curve. Confirm by clicking OK.



Example No. 13. Create points on plane

1. Use the same pad. Select the Point icon. In Point type options select “On plane.” Then, select plane xy and choose the position of point on this plane. Confirm by clicking OK.




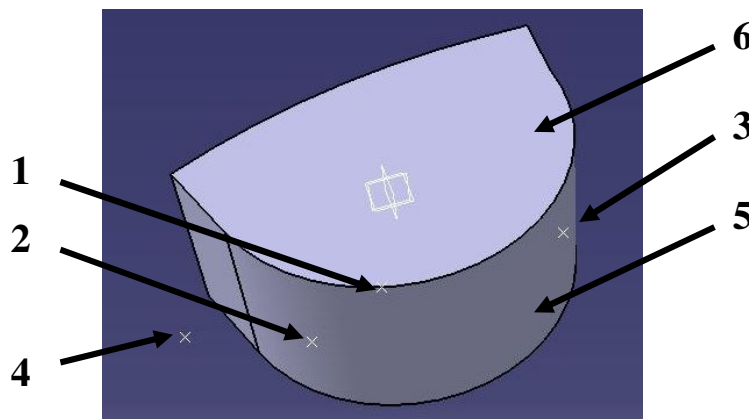
Line

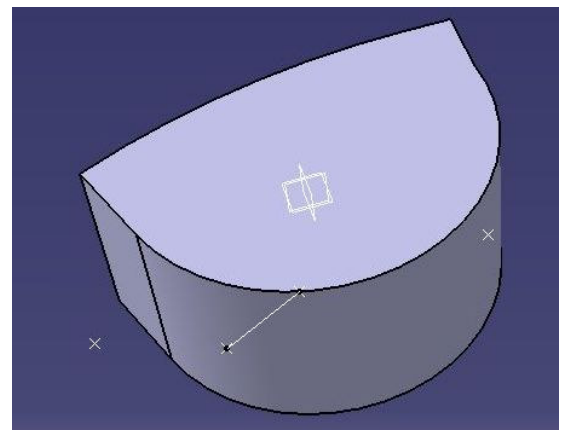
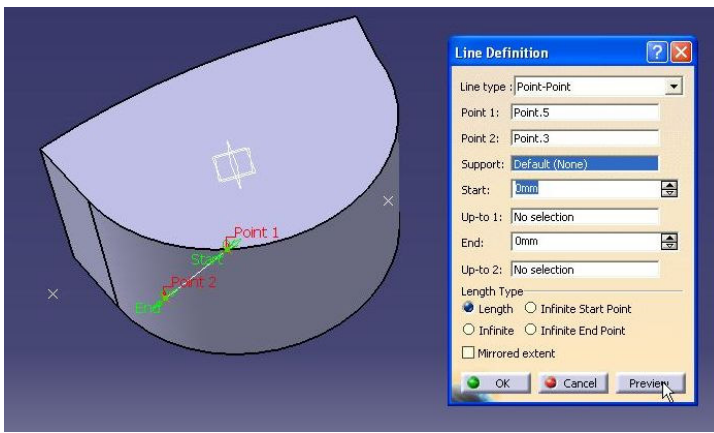
Lines can be created by means of several methods:

- **Point-Point**
- **Point-Direction**
- **Angle/Normal to curve**
- **Tangent to curve**
- **Normal to surface**
- **Bisecting**

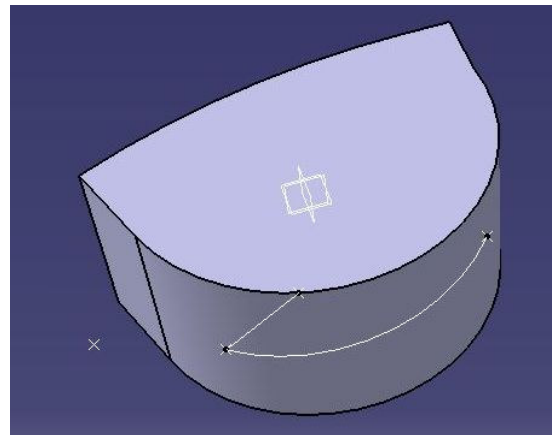
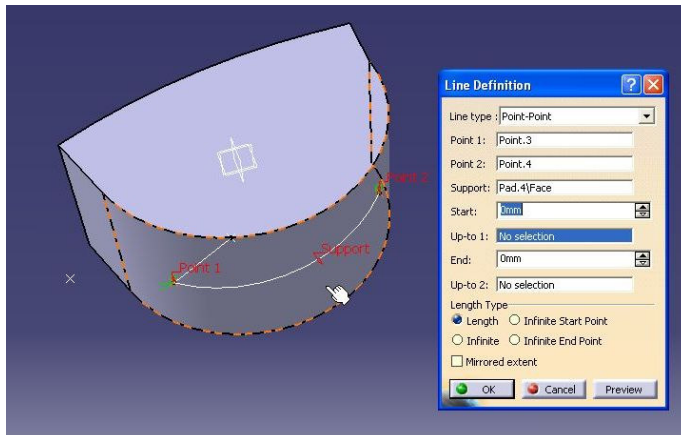
Example No. 13. Create line using Point-Point

1. Use the previous pad with points that were previously created. Select Line icon . The Line Definitions dialog box appears. In Line type options select “Point-Point.” Then, select Point 1 and Point 2. Confirm by clicking OK.





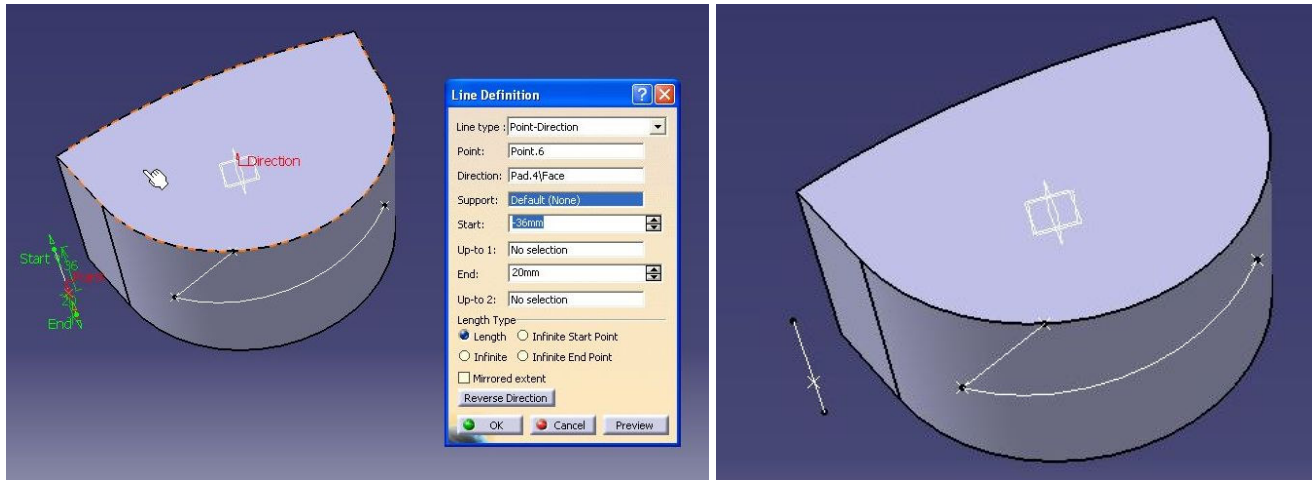
2. Create the next line. Select Line icon. In Line type options select “Point-Point.” Then, select Point 2 and Point 3. In Support option select surface 5 and confirm by clicking Preview button and click OK.



Example No. 14. Create line using Point-Direction

1. Select Line icon. In Line type options select “Point-Direction.” Then, select Point 4. In Direction option select face 6. If you want to change line length click on the arrow and

**move the mouse or enter the Start value.
Confirm by clicking Preview button and click
OK.**




Plane

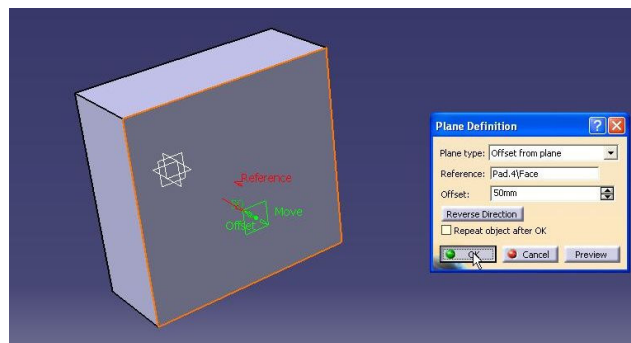
Planes can be created with the use of methods:

- **Offset from plane**
- **Parallel through point**
- **Angle/Normal to plane**
- **Through three points**
- **Through two lines**
- **Through point and line**
- **Through planar curve**
- **Normal to curve**
- **Tangent to surface**
- **Equation**
- **Mean through points**

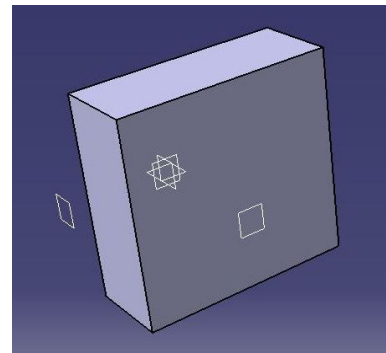
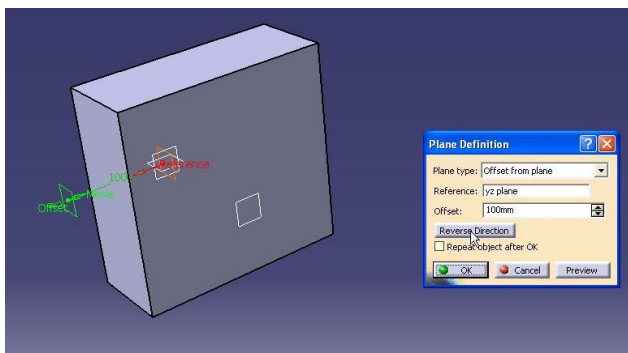
Example No. 15. Create plane using Offset from plane

You can create planes from pad faces or default planes (xy, yz, zx).

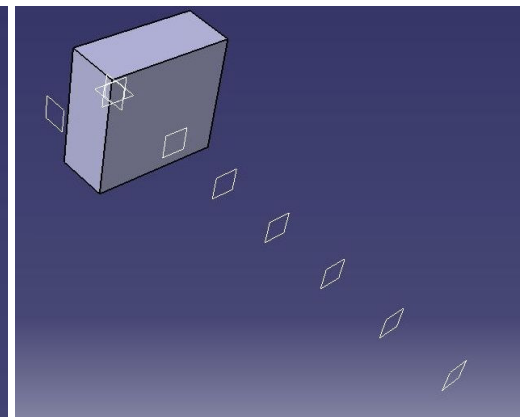
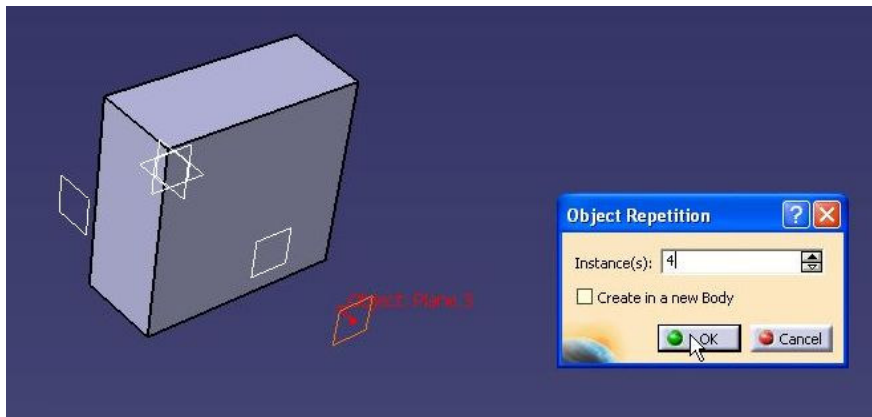
1. Create a simple pad. Select Plane icon . The Plane Definitions dialog box appears. In Plane type options select “Offset from plane.” Then, select Face on the pad end enter the Offset value. You can change direction by clicking Reverse Direction button. Confirm by clicking OK.



2. Select Plane icon. In Plane type options select “Offset from plane.” Then, select “yz” plane and enter the Offset value. Confirm by clicking OK.



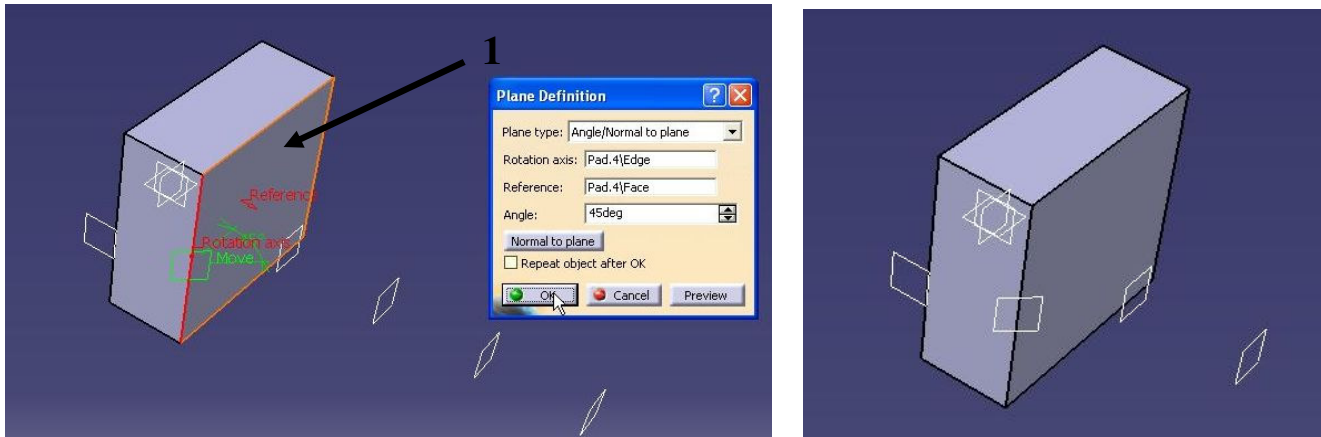
3. If you want to create more planes than one, mark the Repeat Object after OK. Then, select plane to be copied and enter the Offset value and click OK. The Objects Repetition dialog box appears. Enter the Instance(s) value (for example 4). Click OK.



Example No. 16. Create plane using Angle/Normal to plane

1. Select Plane icon. In Plane type options select “Angle/Normal to plane.” Then, in Rotation axis option select one of the edges. Then, select the Face 1 which will be

**Reference element and enter the Angle value.
Confirm by clicking OK.**



Editing Parts

You can edit geometrical parameters parts at any time. You have several ways of editing them. Firstly, you can modify the sketch which was used in the definition of a part, and secondly, you can modify concrete operation which was used to create this part.

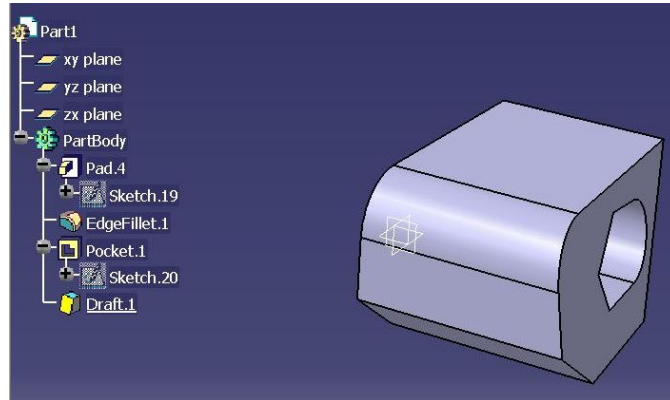
Example No. 17.

You can edit geometrical parameters by double-clicking the one which you want to be

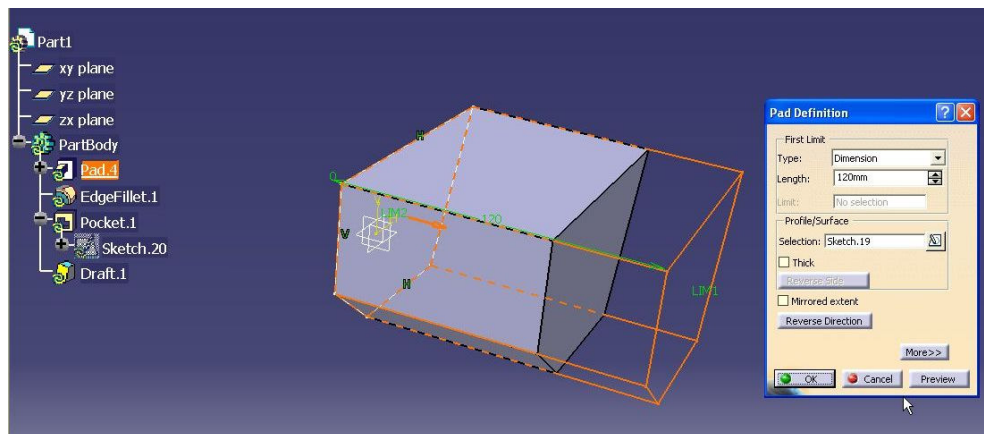
edited. You can use the specification tree or the geometry area.

In this example you will edit Pad, Edge Fillet, Pocket and Draft options.

1. Open Edit17 file.

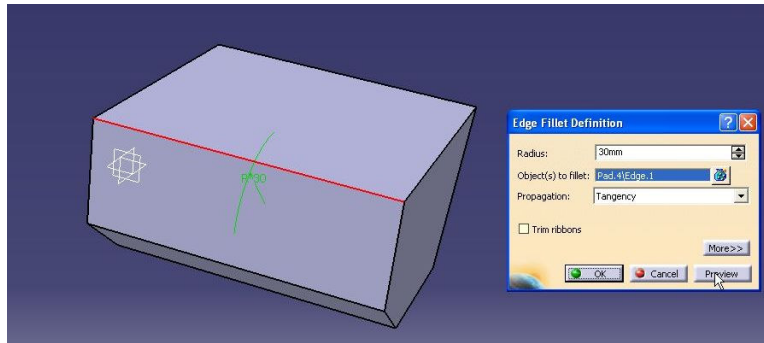


2. Double-click the Pad.4 to be edited. The Pad Definition dialog box appears. Enter a new Length value (120) and click OK. Pad.4 has been edited.

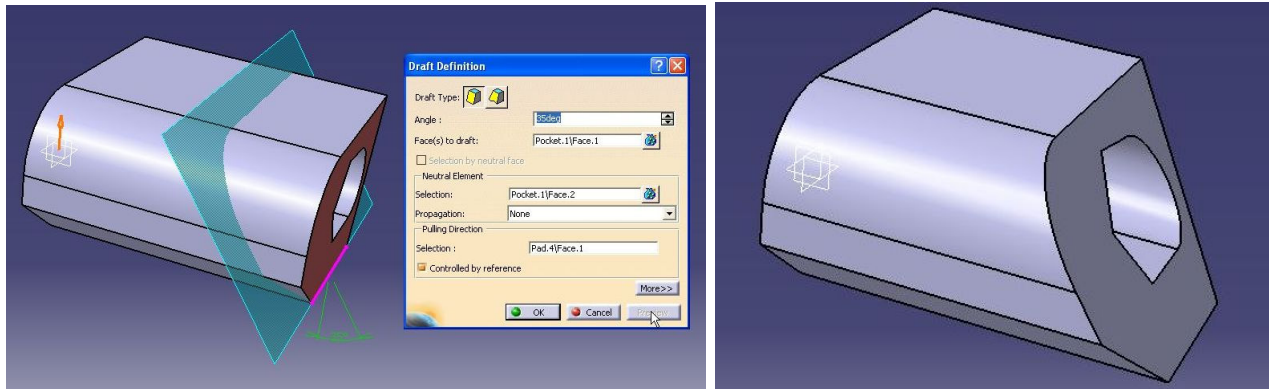


3. Select the EdgeFillet.1 (double-click). The Edge Fillet Definition dialog box appears.

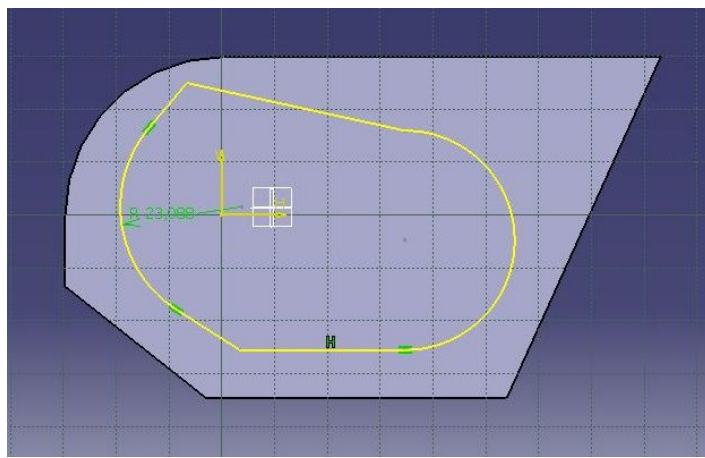
Enter a new Radius value (30) and confirm by clicking Preview button. Click OK



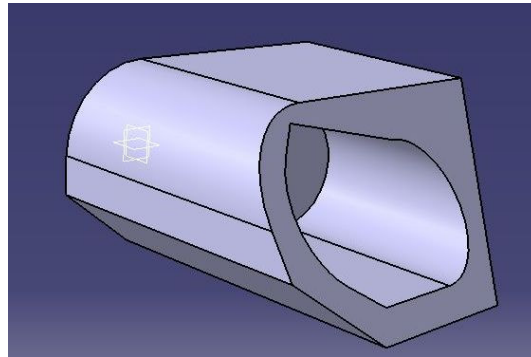
4. Select the Draft.1. The Draft Definition dialog box appears. Change current Draft Angle value on 35deg. Confirm by clicking Preview button two times and then click OK.



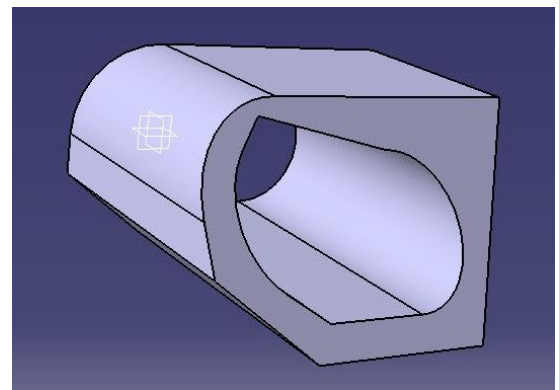
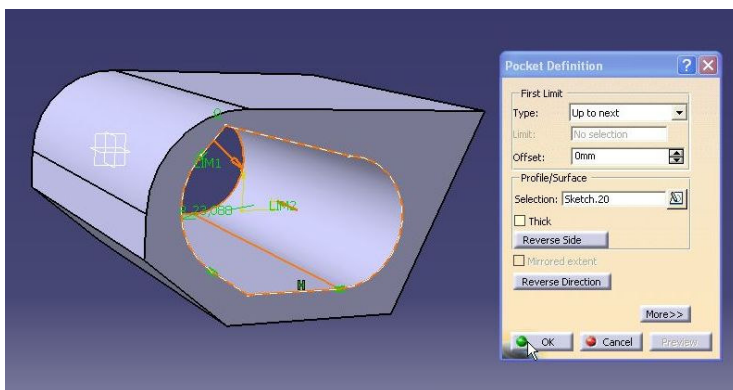
5. You can edit Sketch Pocket geometrical parameters by double-clicking Sketch.20. The Part Design will switch over to Sketcher module automatically.



6. Then, modify the sketch and select Exit Workbench . The Sketcher will switch over to Part Design module automatically.



7. Select the Pocket.1. The Pocket Definition dialog box appears. Change current Type of First Limit on Up to next. Select OK.



Boolean Operations

If you want to use Boolean Operations you have to insert a new body into the part. If you build your part on several bodies you can move and rotate those pads or different elements against one another in any direction without necessity of using the Sketcher or editing elements.

Using several bodies in a part allows you to design different steps of a part without any operations between bodies.

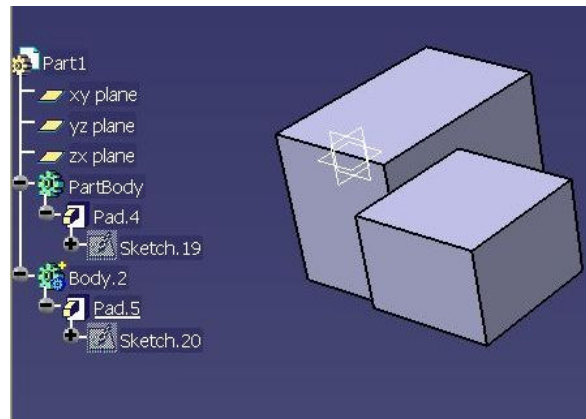
You can create a new body by clicking Body icon  on Insert Toolbar or use Menu bar Insert → Body.

Example No. 18. Create a new body.

1. Create a simple pad on the PartBody. Then, select the Body icon . Application generates a new Body automatically.



2. Select Body.2 (one click) and create a simple pad.



You have created two pads on two different bodies.

Assemble and Add

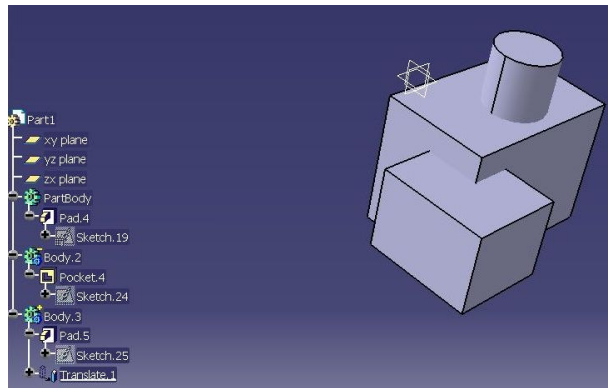
Assemble and Add are very similar operations because they create a union between bodies. There is only one difference. If Body.1 contains a Pocket structure, Assembly will see it as a Pocket and remove material from

PartBody. In this case, if Add is used, the Pocket will be seen by PartBody as a Pad.

Example No. 19. Assembly

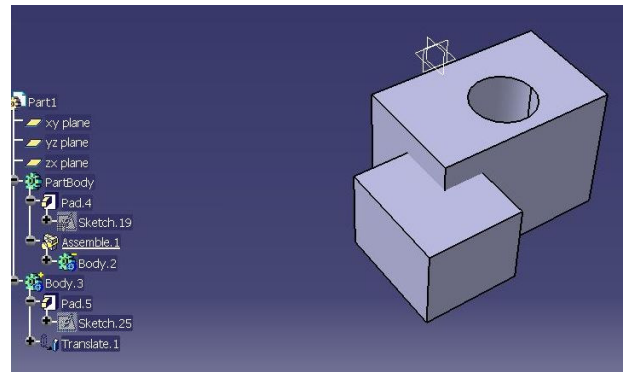
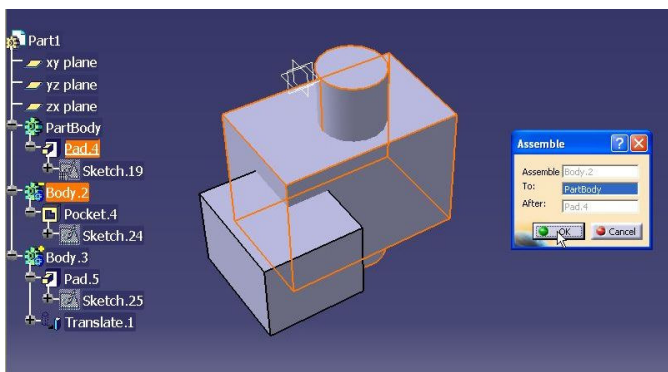
Assembling is an operation integrating your part specifications. It allows you to create a complex geometry.

1. Open Assemble19 file.

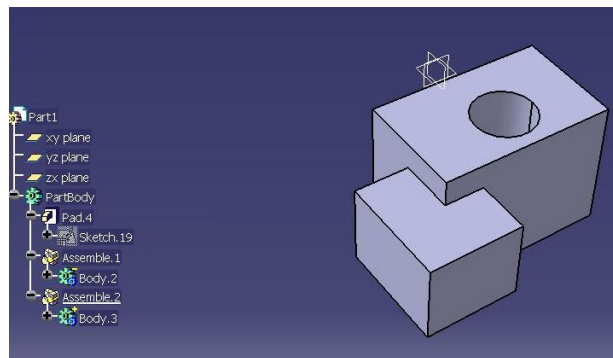


This Part contains three Bodies. PartBody contains the Pad. Body.2 contains the Pocket and Body.3 contains the Pad.


2. Select Assembly icon . From Assembly options select the Pocket in Body.2 and select the Pad in PartBody. Then, click OK.

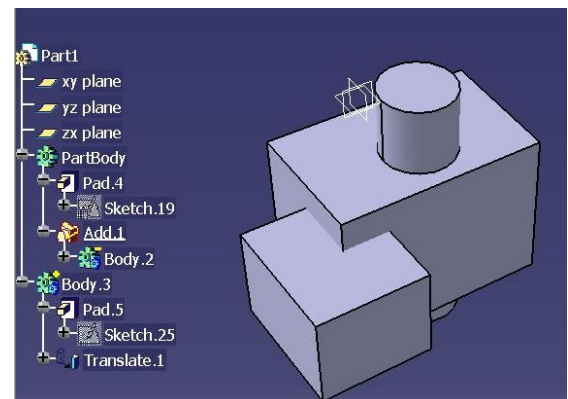
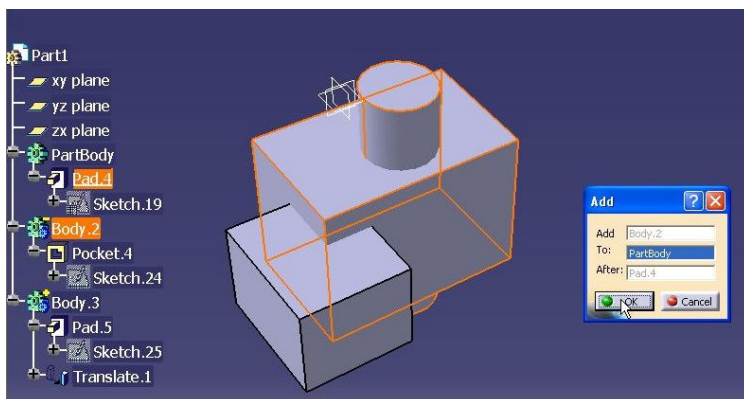


3. Then, create Assembly Body.3 with PartBody.

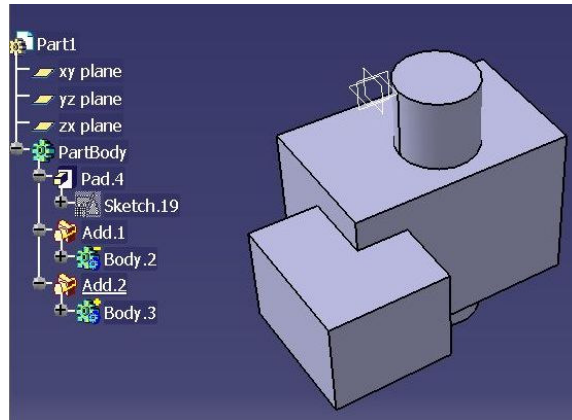


Example No. 20. Add

1. Use the previous file and use the Add option for the same Bodies. Select Add icon . Then, select the Pocket and add it to Pad in PartBody.



2. Then, Add the Pad in Body.3 to PartBody.

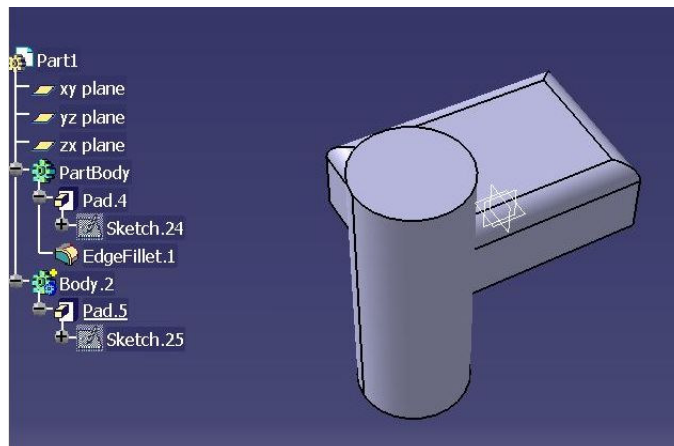


Remove

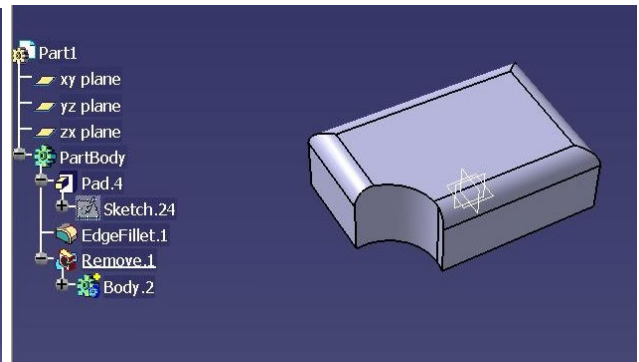
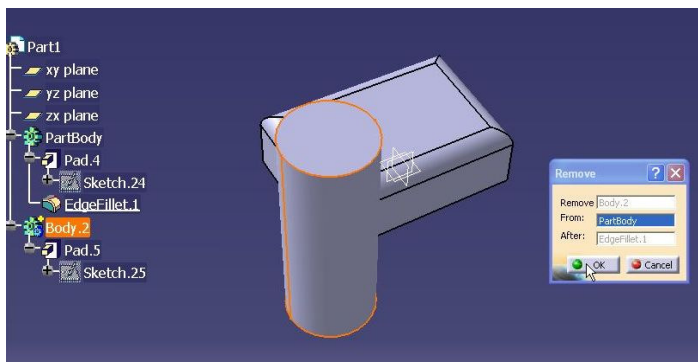
Example No. 21.

If Body.2 is Removed from PartBody, the operation is PartBody minus Body.2.

1. Open Remove21 file.



1. Select Remove icon . Then, from Remove option select the Pad.5 and click OK.



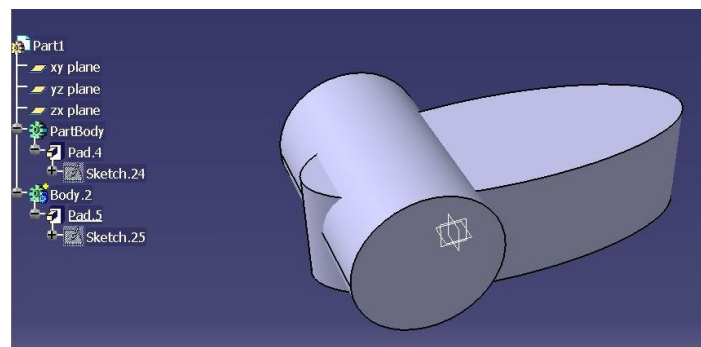
Pad.5 has been removed from Pad.4.

Intersect

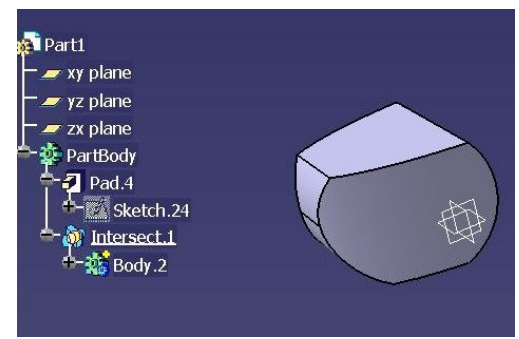
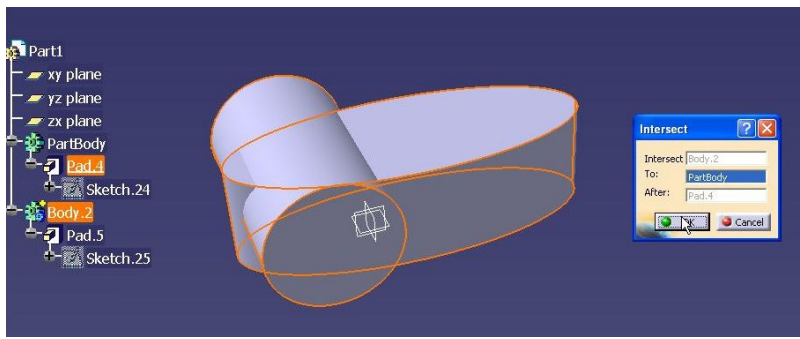
Example No. 22

The resulting material is the intersection between two bodies.

1. Open Intersect22 file



2. Select Intersection icon . Then, select the Part.5 and select the Pad.4. Click OK.

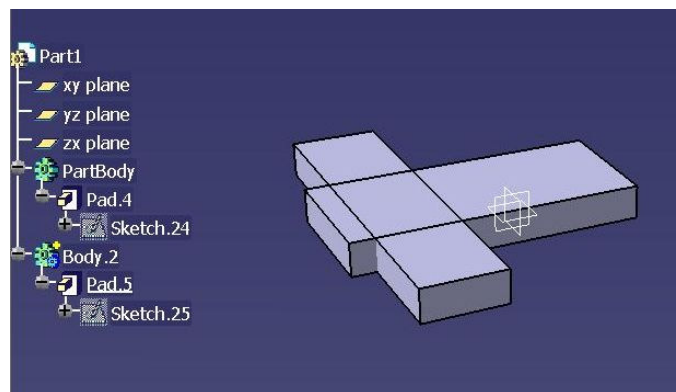


Union Trim

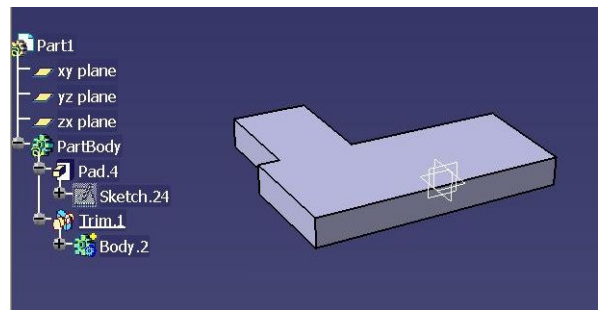
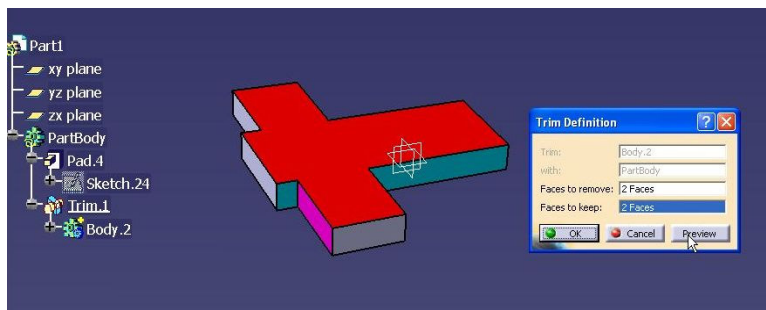
Example No. 23

The Union Trim is basically a union with an option to remove or keep one side or others.

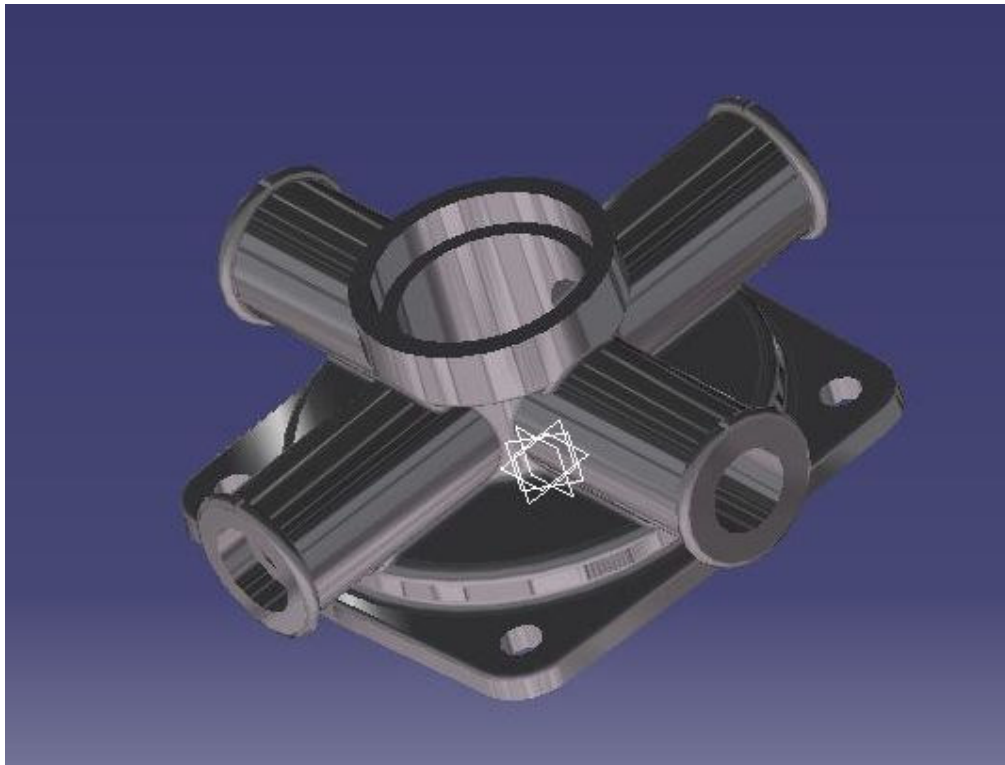
1. Open UnionTrim23.



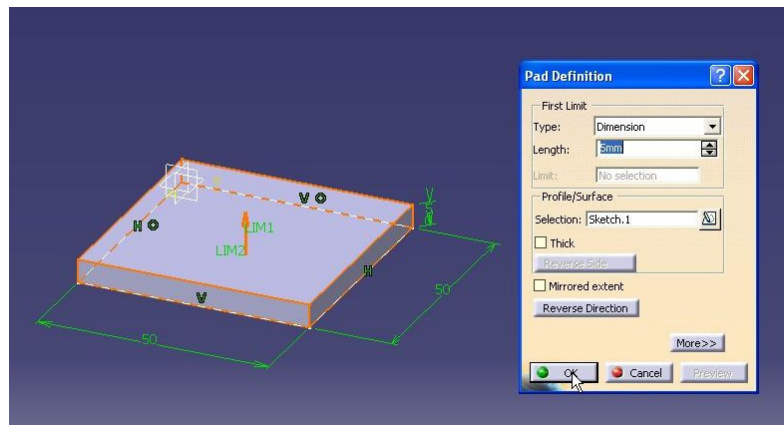
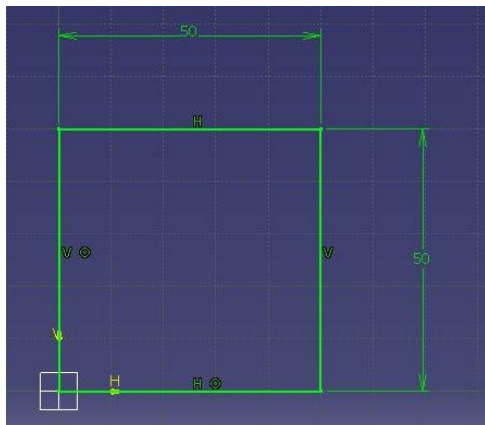
2. Select Union Trim icon . Select Pad.5 to trim. Then, specify face to be removed (select face 1 and 2). Then, specify face to be kept (select face 3 and 4). Confirm by clicking Preview button.



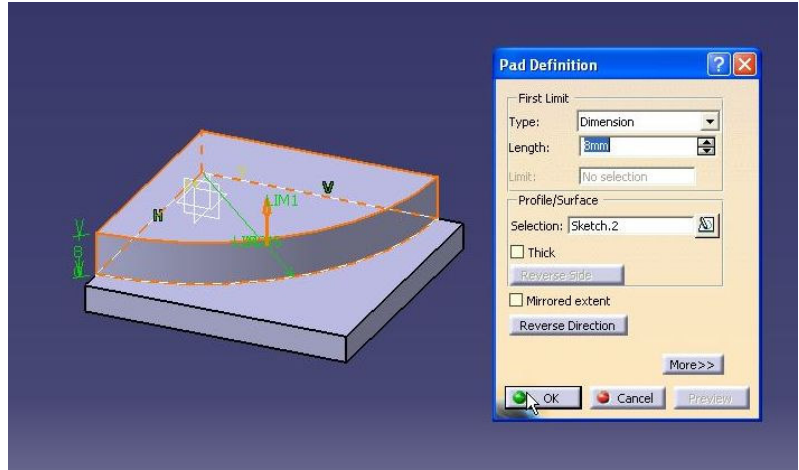
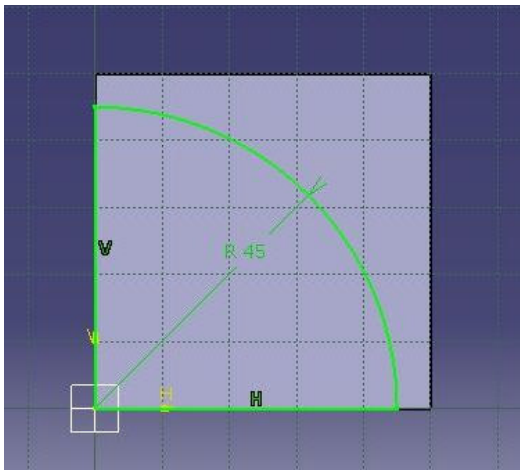
Task No. 1. Create part.



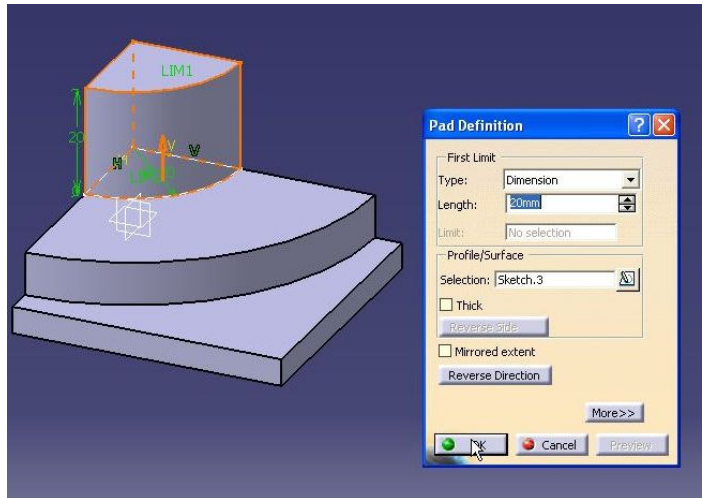
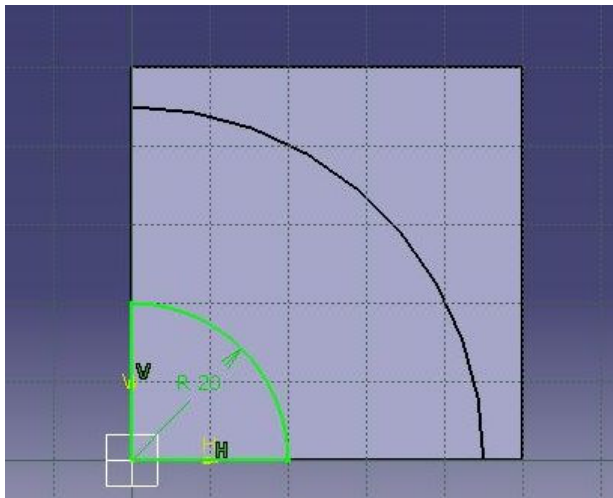
1. In the Sketcher on “xy plane” create quarter sketch part. Then, create a pad.



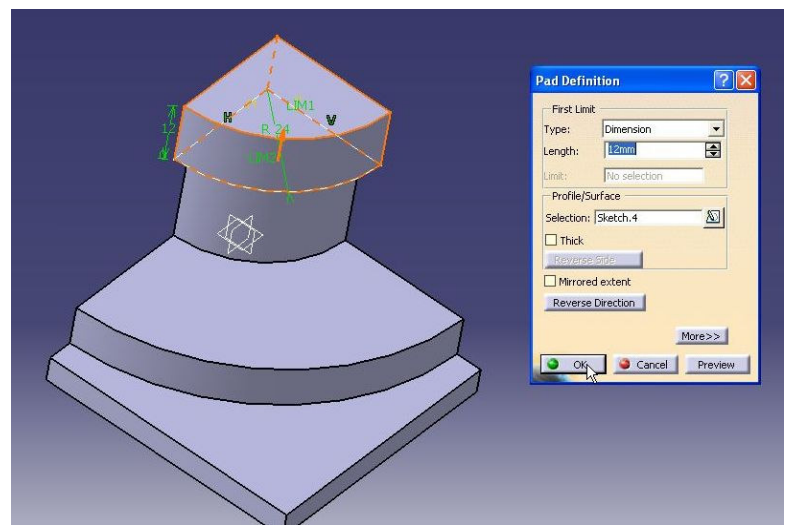
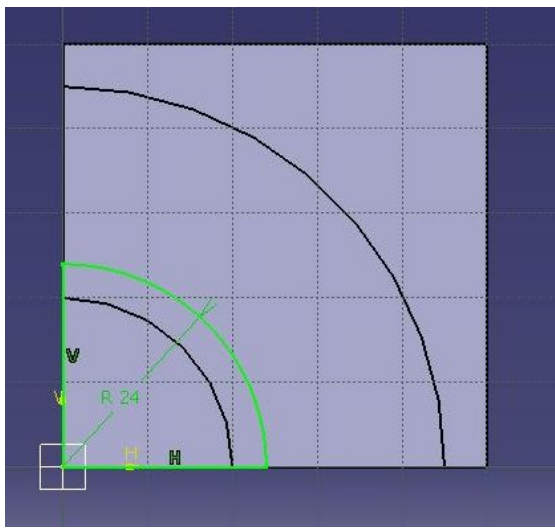
2. On top face create a next profile. Then, create a pad.



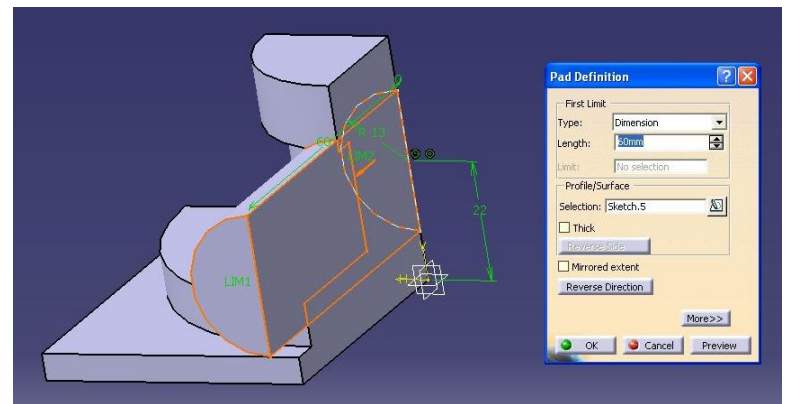
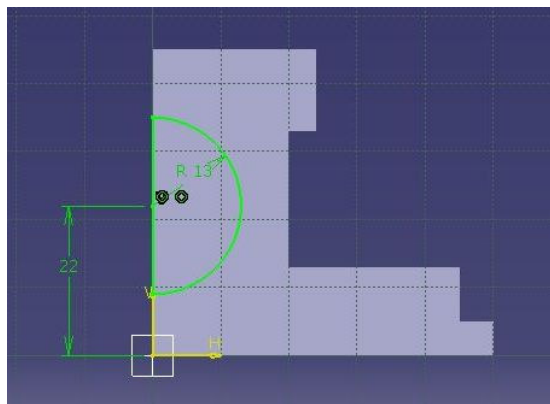
3. On top face of the previous pad create a next profile. Then, create a pad.



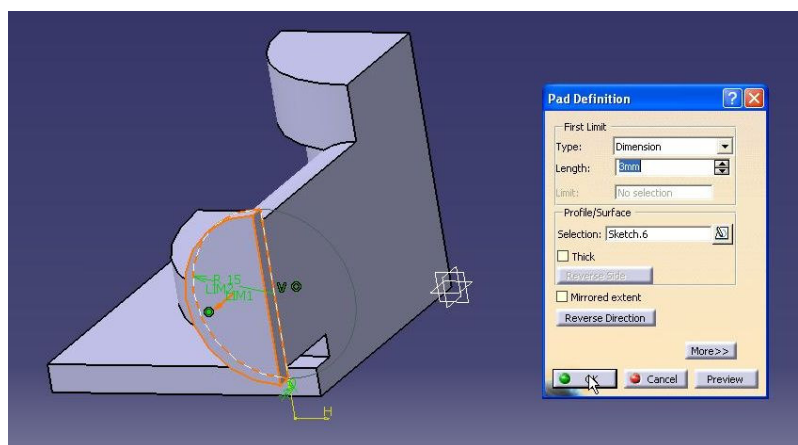
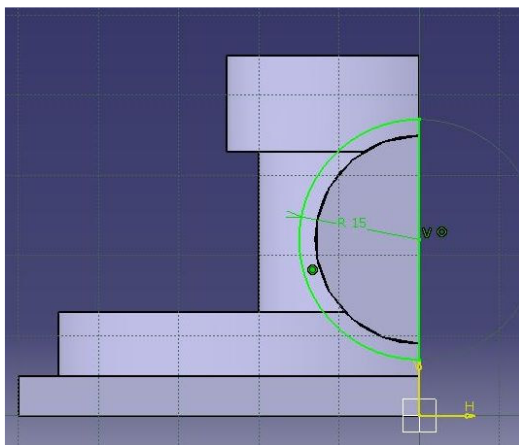
4. On top face of the previous pad create a next profile. Then, create a pad.



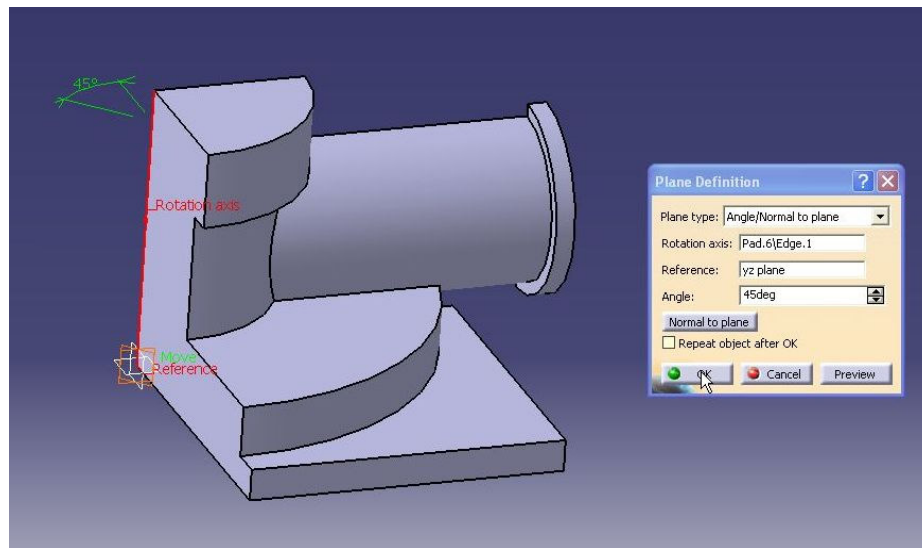
5. On “zx plane” create sketch. Then create a pad.



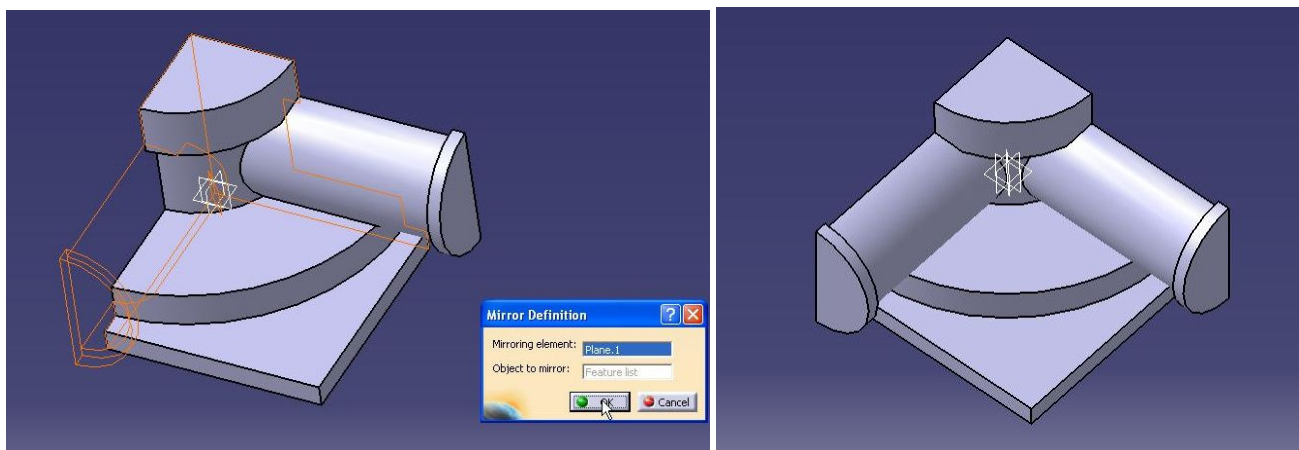
6. On top face of the previous pad create a next profile. Then, create a pad.



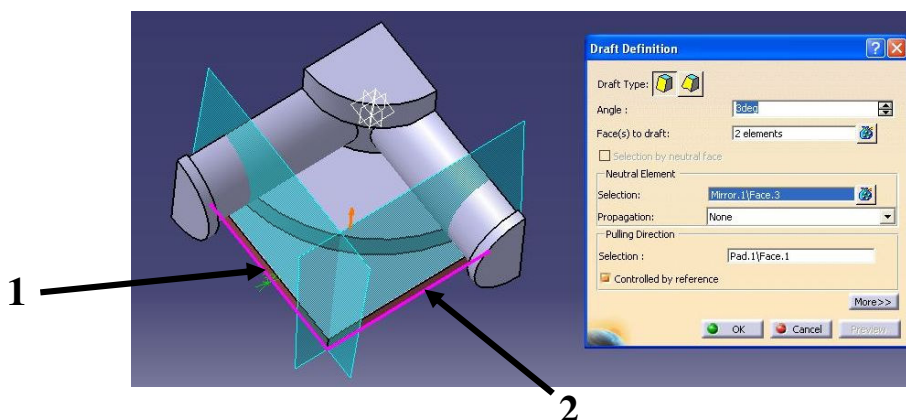
7. Create a new plane which will be rotated about 45deg against “yz plane.”



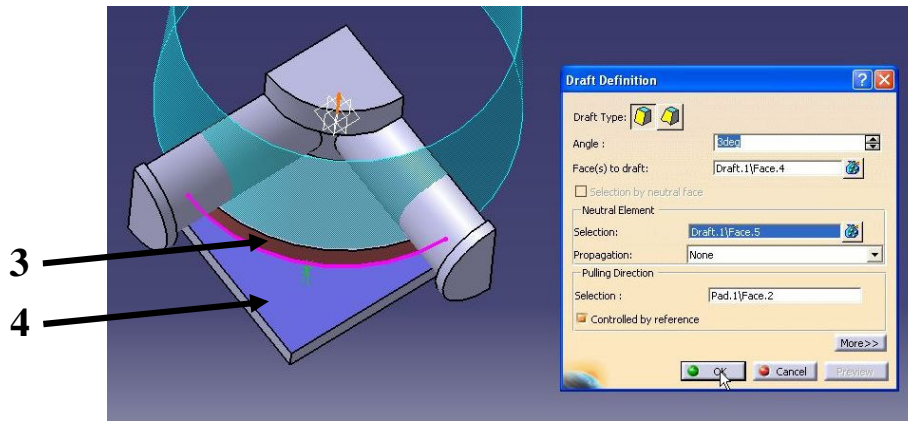
8. Use Mirror option to copy pad.5 and pad.6.



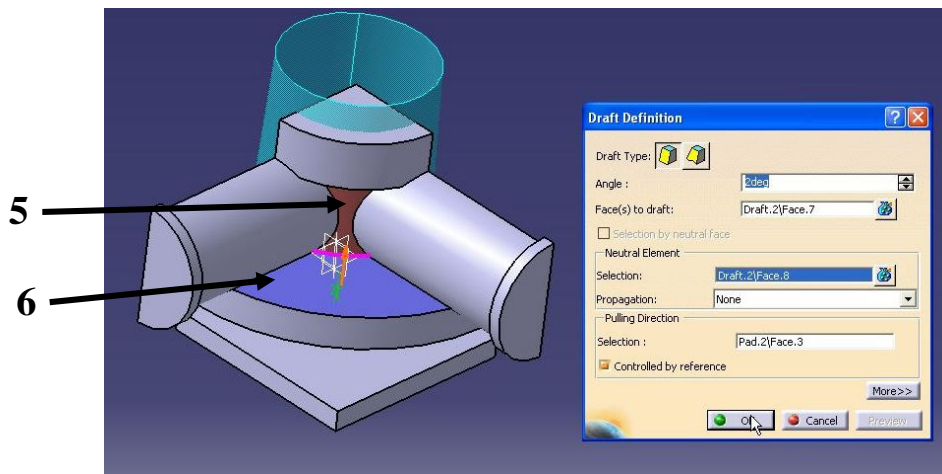
9. Use Draft Angle option to draft surfaces.
a) select face 1 and 2 to be drafted. In Pulling Direction select top face. In Neutral Element select bottom face.



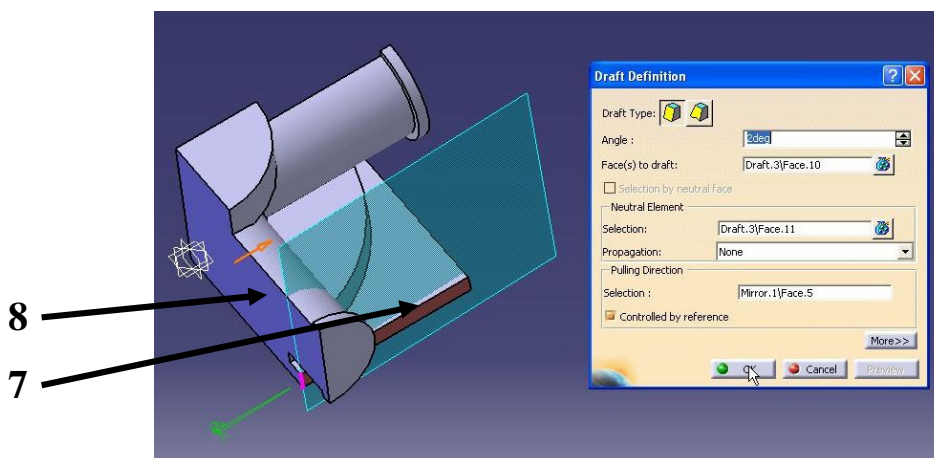
b) select surface 3 to be drafted. In Neutral Element select face 4.



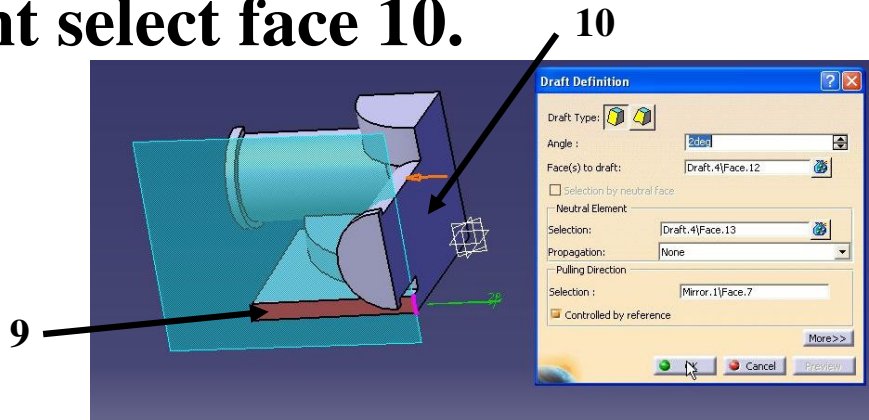
c) select surface 5 to be drafted. In Neutral Element select face 6.



d) select face 7 to be drafted. In Neutral Element select face 8.

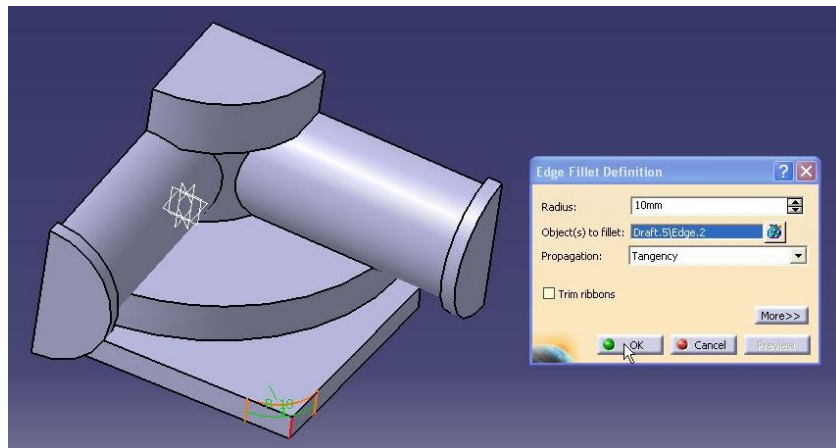


e) select face 9 to be drafted. In Neutral Element select face 10.

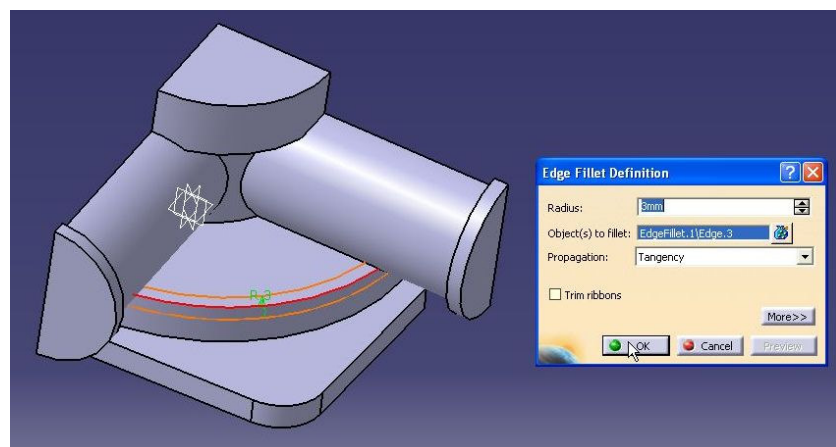


10. Use Edge Fillet option to fillet.

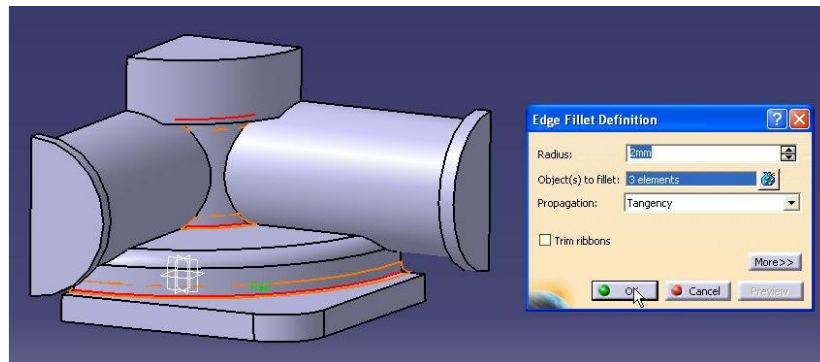
a)



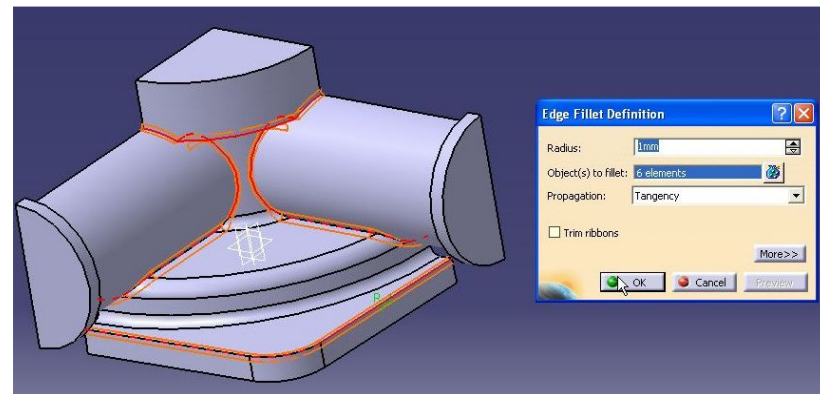
b)



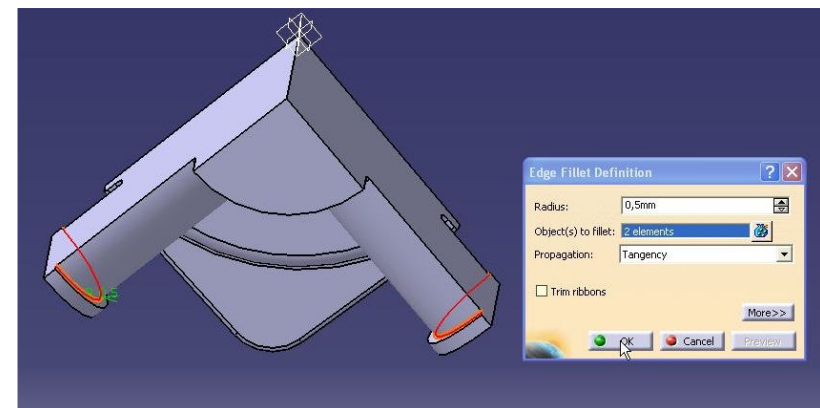
c)



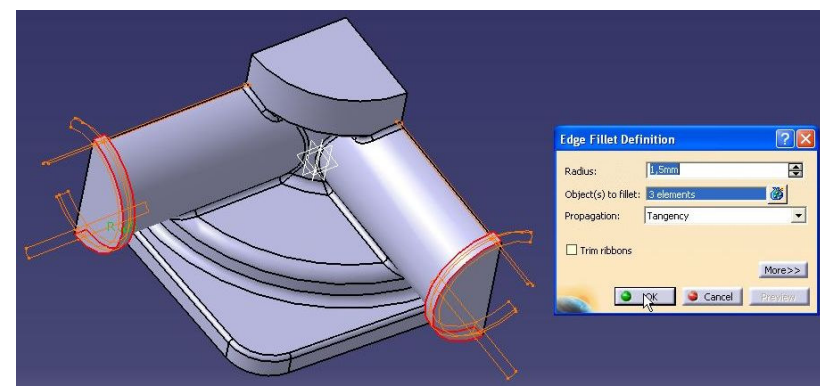
d)



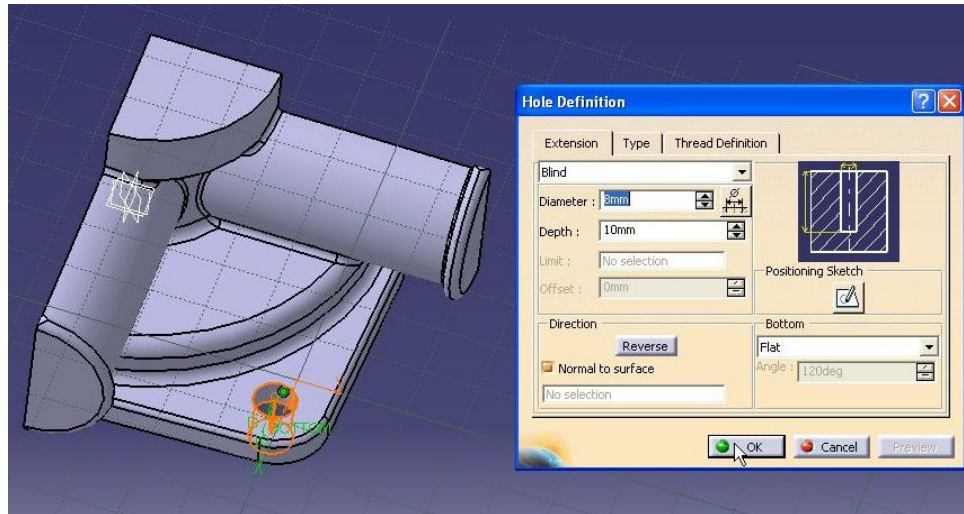
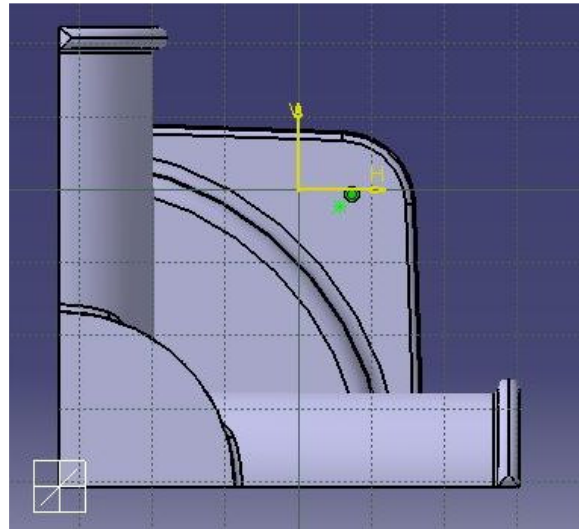
f)



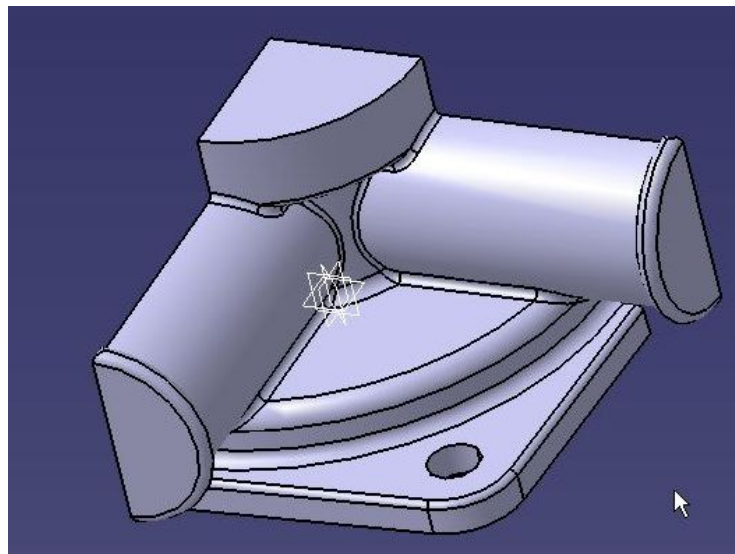
g)



11. Use Hole option to create a hole (use Positioning Sketch)

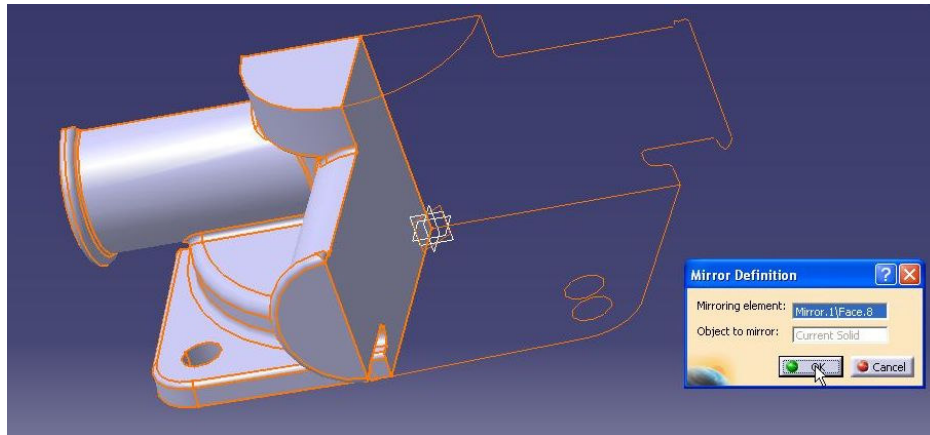


12. Save your file.

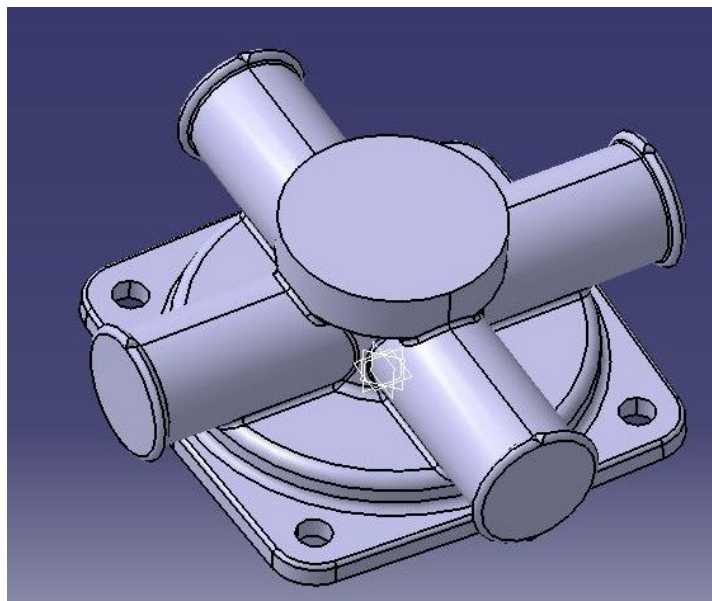
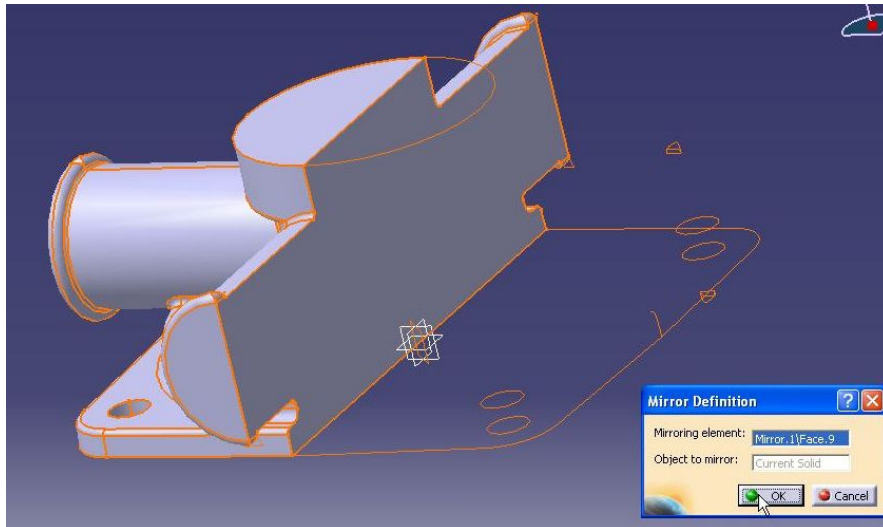


13. Use Mirror options to copy the pad.

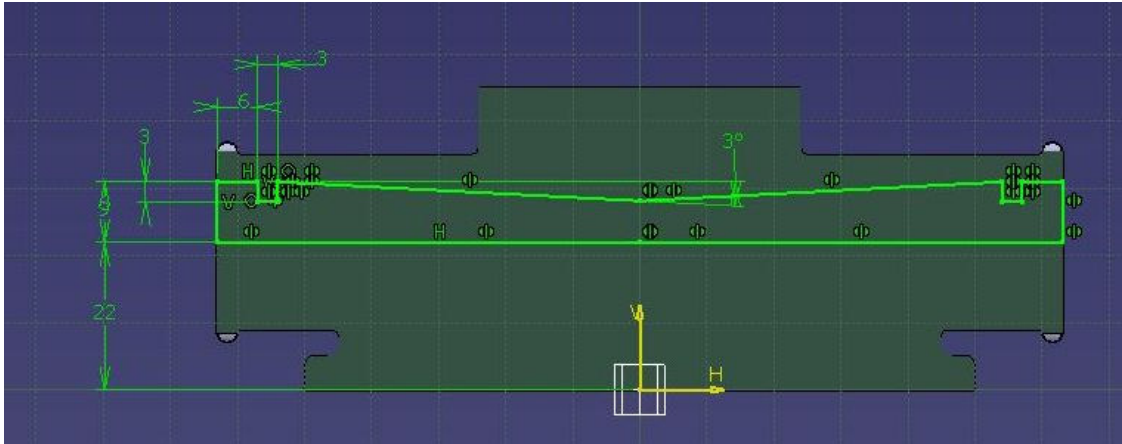
a)



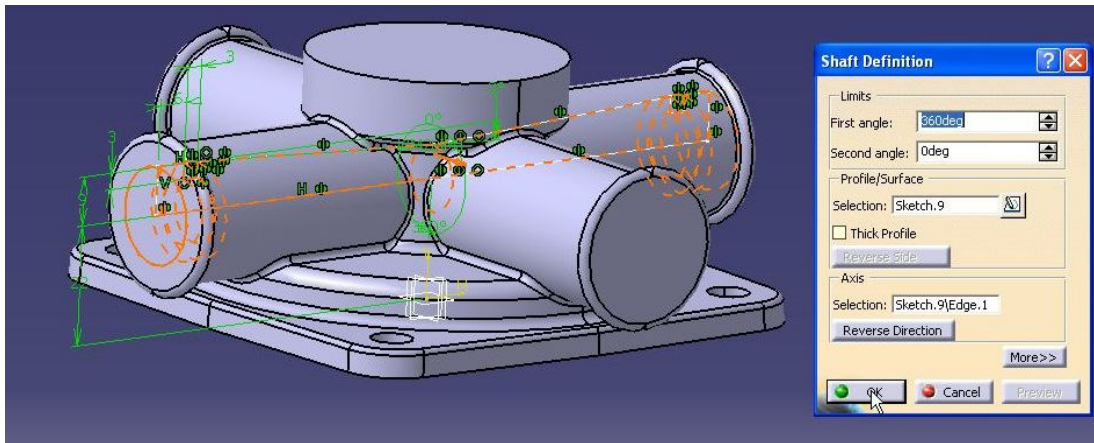
b)



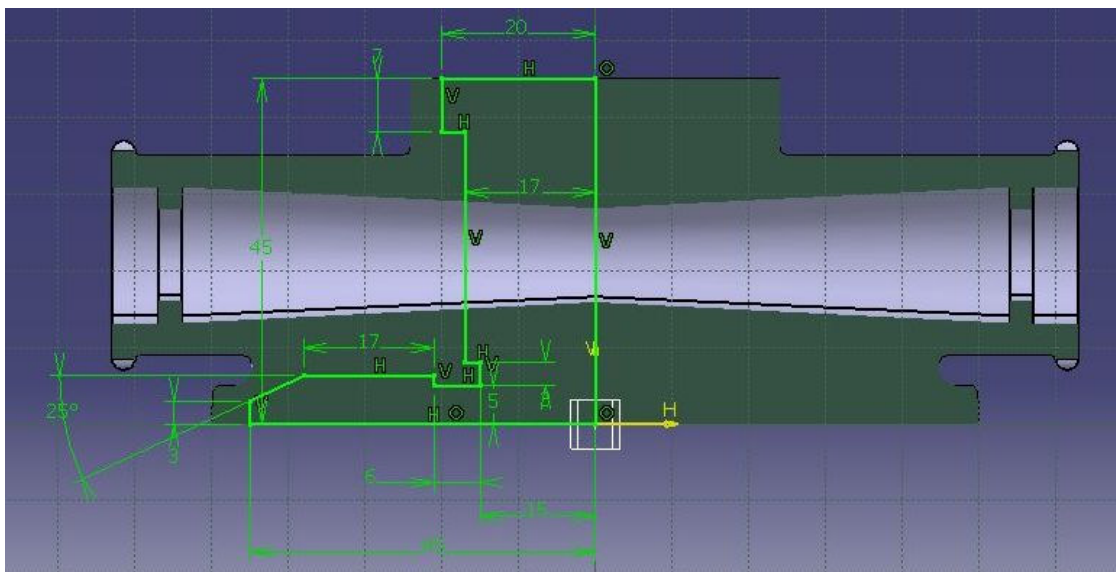
14. Insert a new Body. Then, create a sketch on “yz plane.”



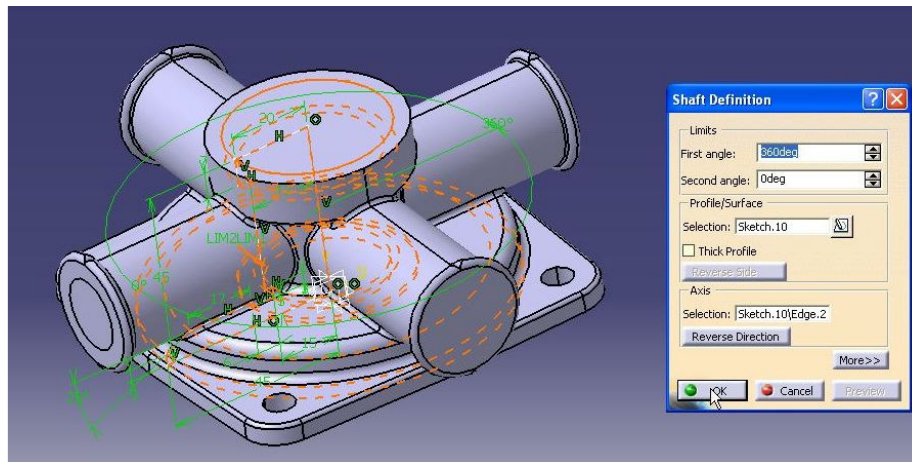
15. Use Shaft option to create a pad.



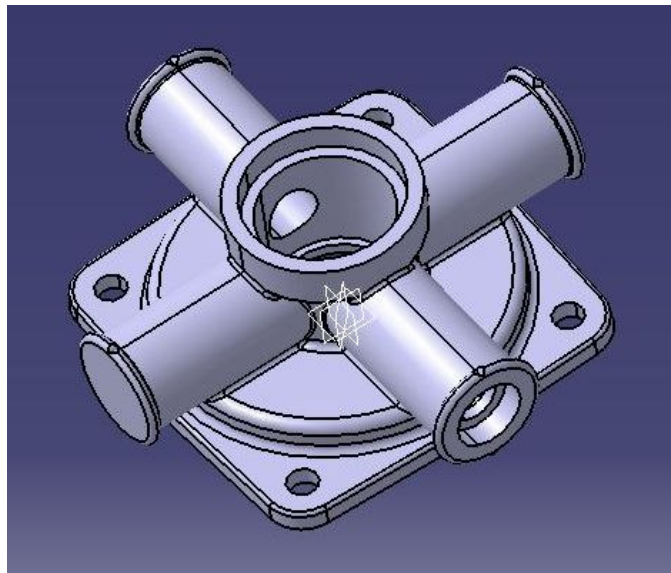
16. Create a sketch on “yz plane.”



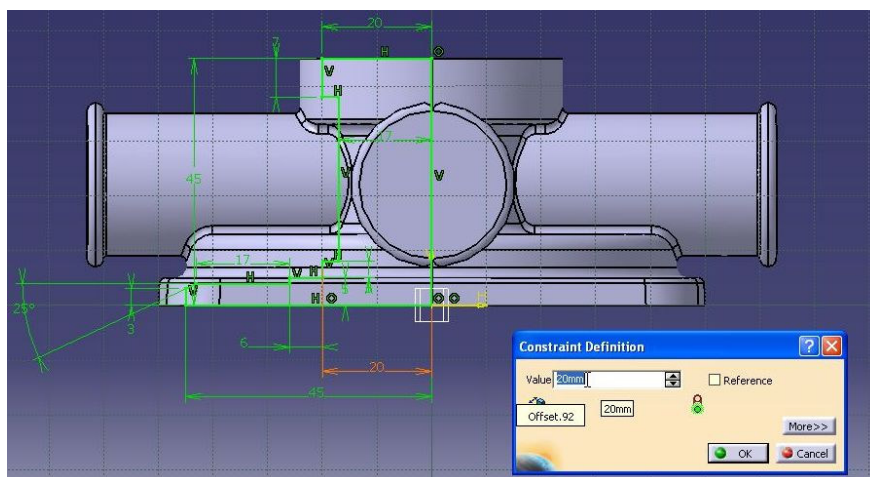
17. Use Shaft option to create a pad.



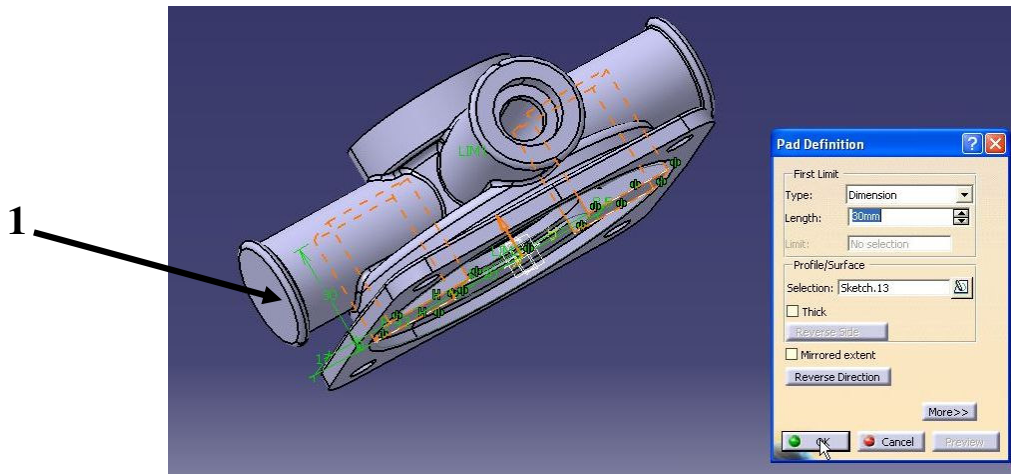
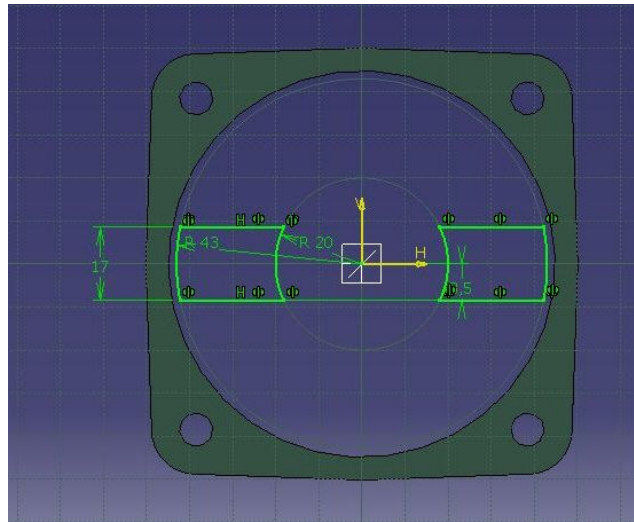
18. Use Boolean operation to remove Body2 from PartBody.



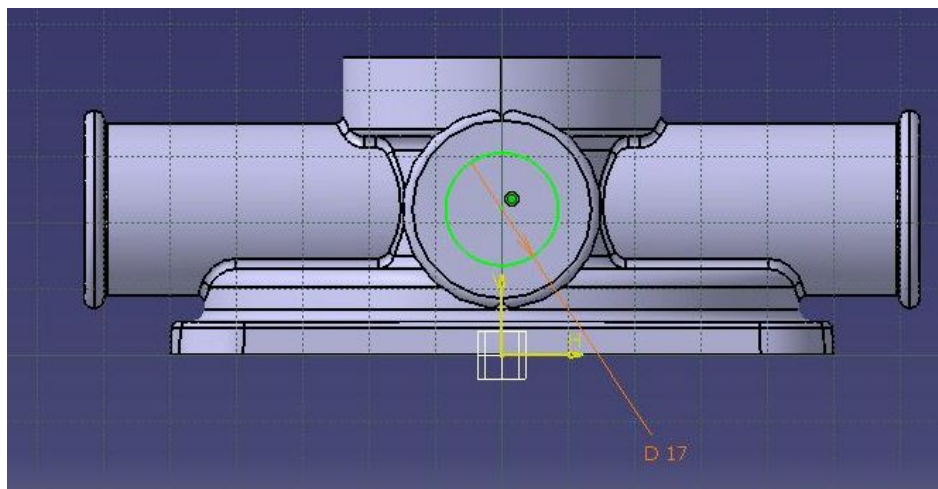
19. Modify the Sketch.10.

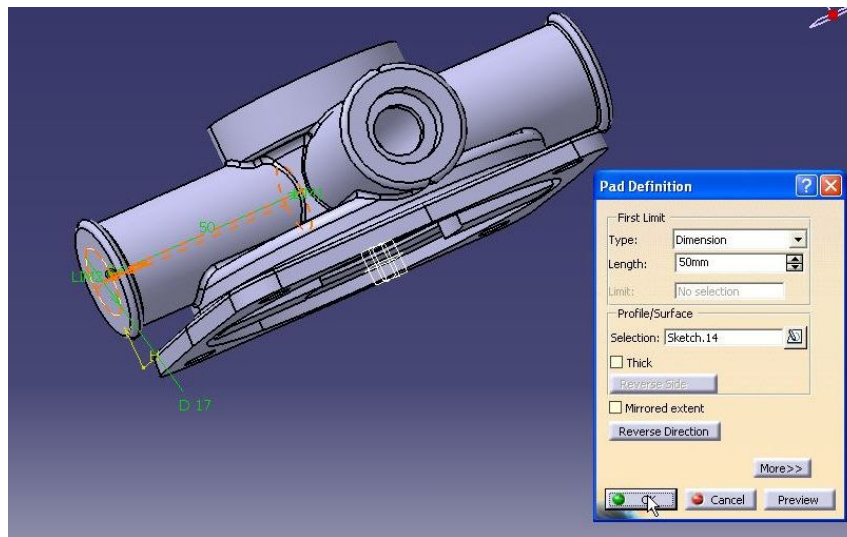


20. Insert a new Body. Then, create a sketch on “xy plane” and crate a pad.

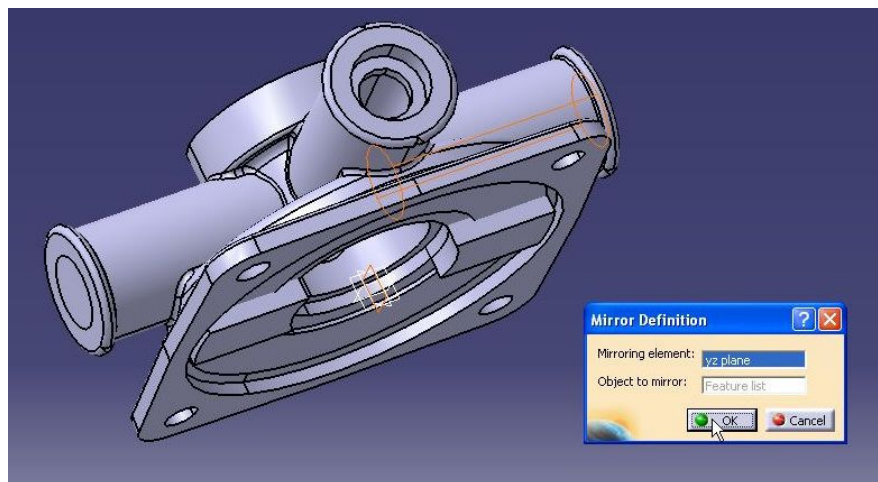


22. Insert a new Body. Then, create a sketch on face 1. Then, create a pad.

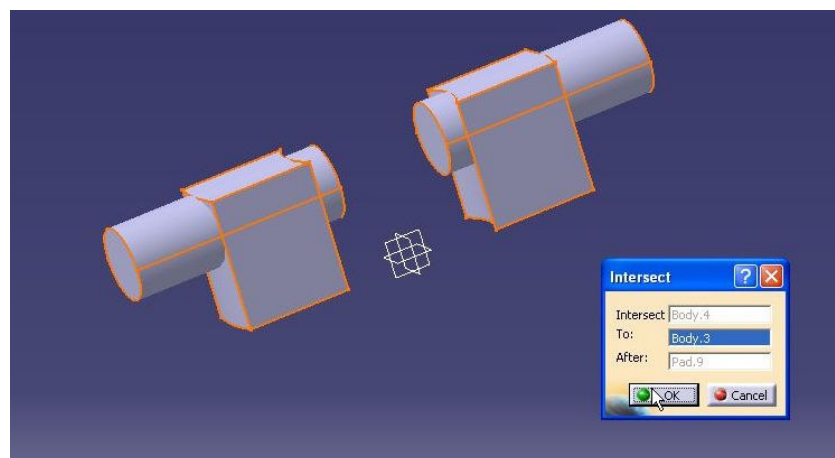




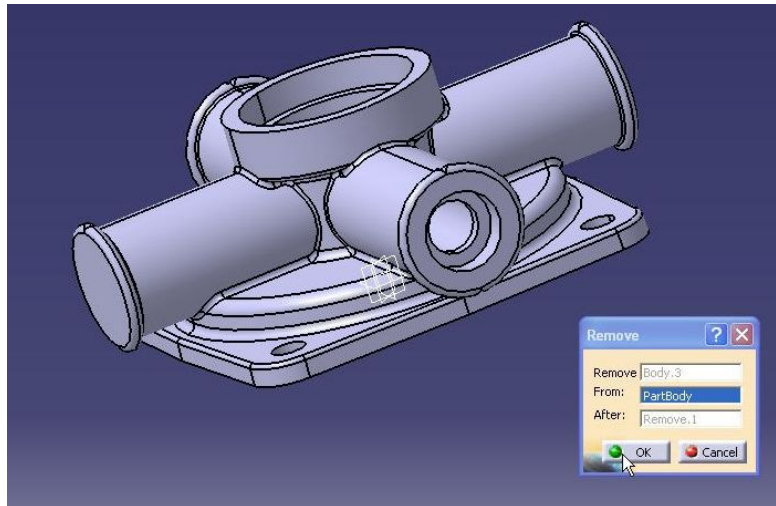
23. Use Mirror options to copy the pad.



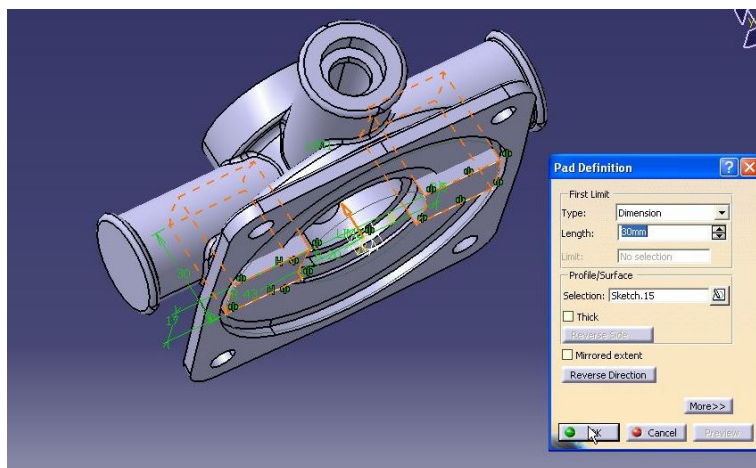
24. Use Hide/Show options to hide the PartBody. Then, use Boolean operation to intersect Body.3 and Body.4.



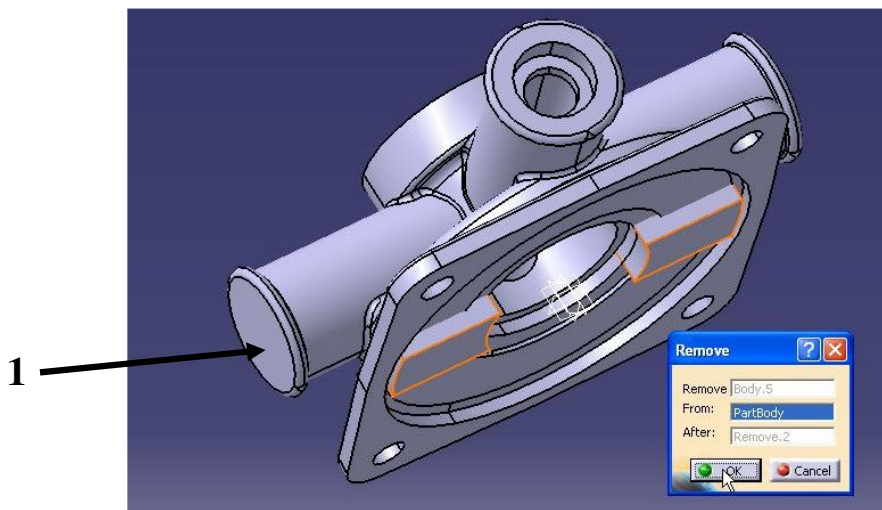
25. Remove Body.3 from PartBody.



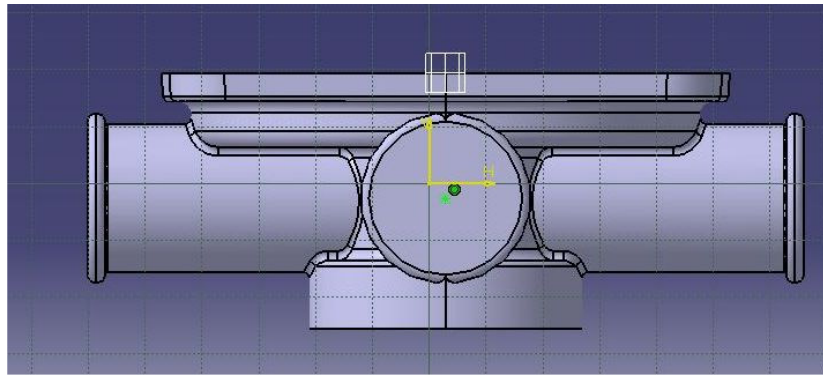
26. Insert a new Body. Then, copy the sketch.11 to Body.5 and create a pad.



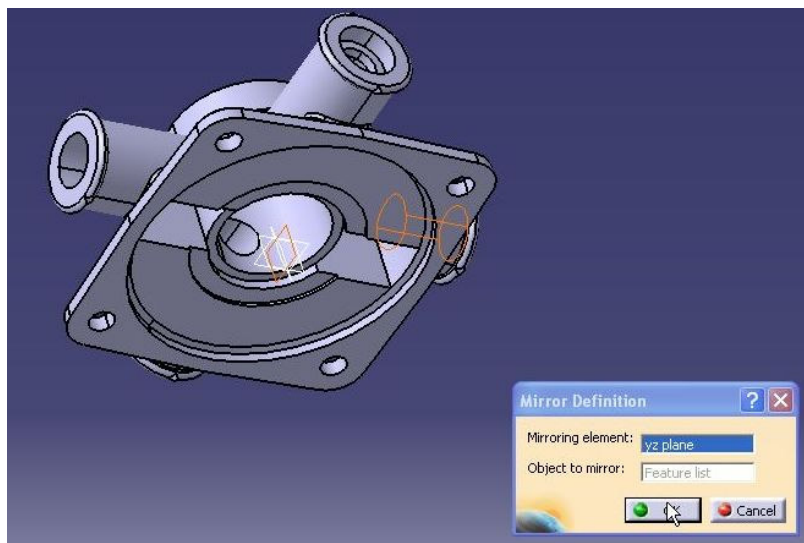
27. Remove Body.5 from PartBody.



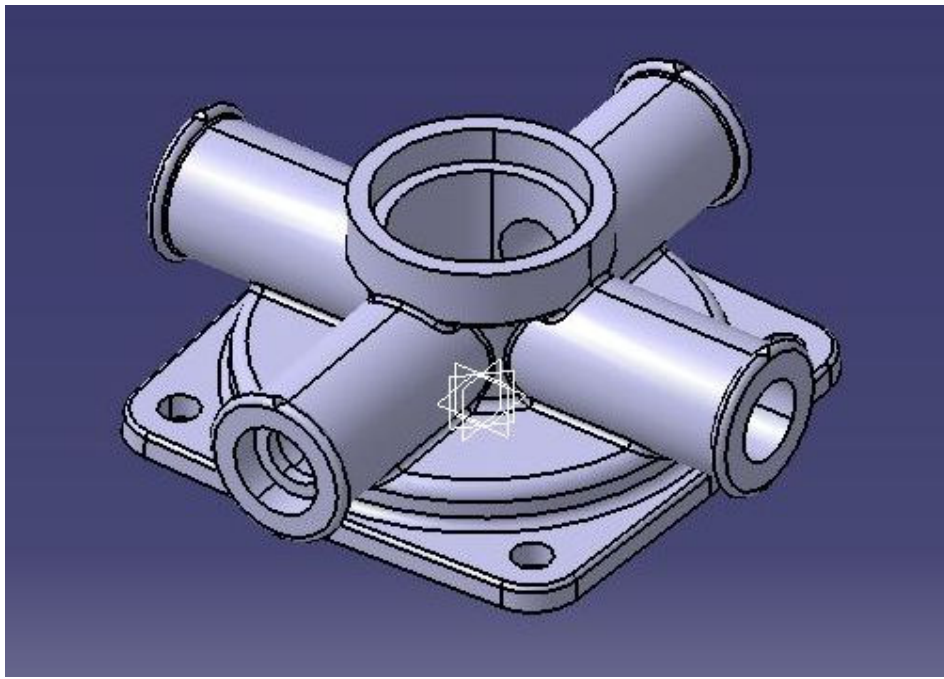
28. Create a hole on face 1.

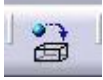


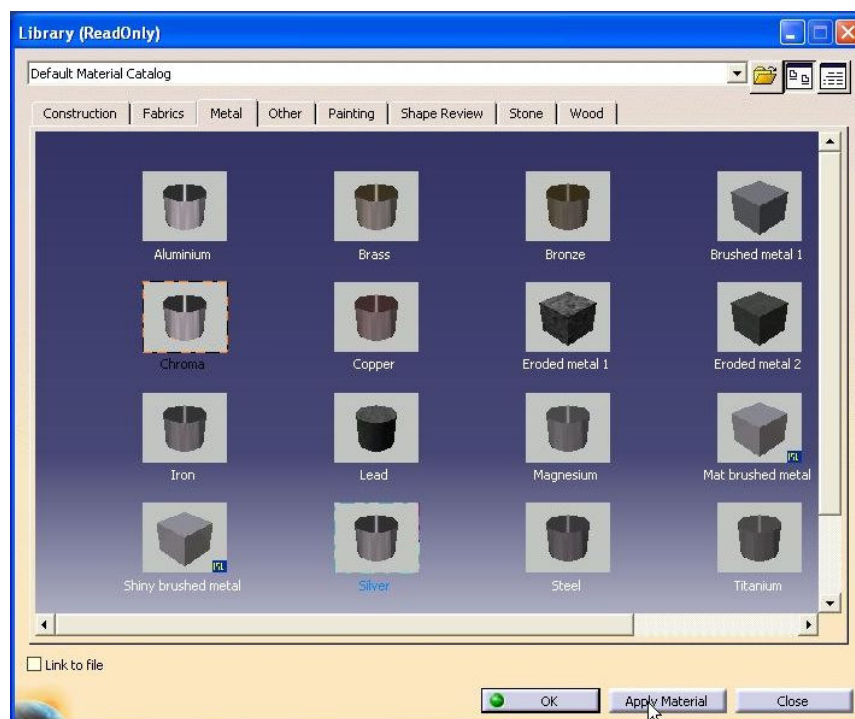
29. Use Mirror options to copy the hole.



30. Save your file.



31. Select PartBody and select Apply Material icon . The Library Material dialog box appears. Select Metal Material and chose material (for example Chroma). Confirm by clicking Apply Material button.



32. Save your file.

