

Part Design

Part 2

The Part Design Workbench

What is the Part Design

- **The Part Design is an application makes it possible to design precise 3D mechanical parts with an intuitive and flexible user interface. The Part Design application will enable you to accommodate design requirement for parts of various complexities, from simple to advanced.**
- **This application combines the power of feature-based design with flexibility of a Boolean approach and offers a highly productive and intuitive design environment with multiple design methodologies, such as post-design and local 3D parameterization**

How to enter the Part Design workbench.

1. Select Start -> Mechanical Design -> Part Design from the menu bar (Fig. 1).

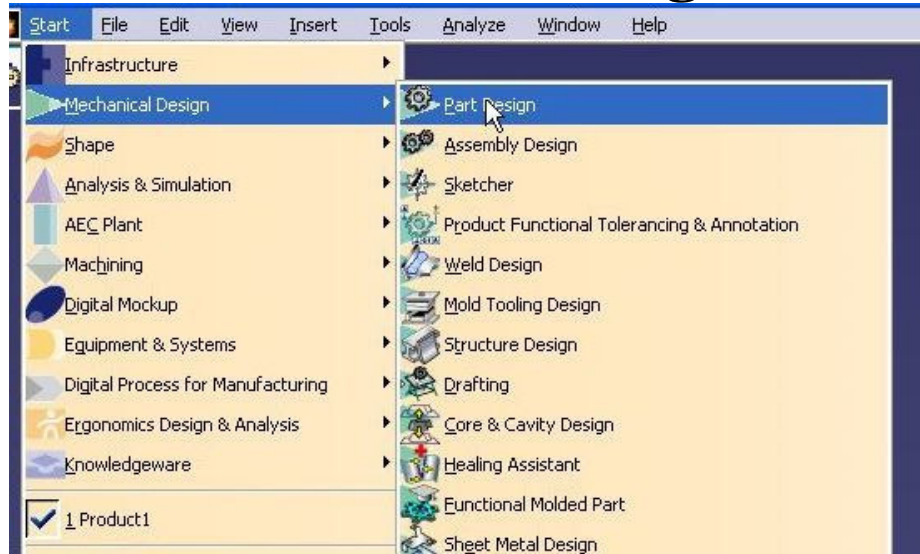


Fig.1. Menu bar

**2. Than you have to select plane.
Click the desired reference plane either in the geometry area or in the specification tree (Fig. 2).**

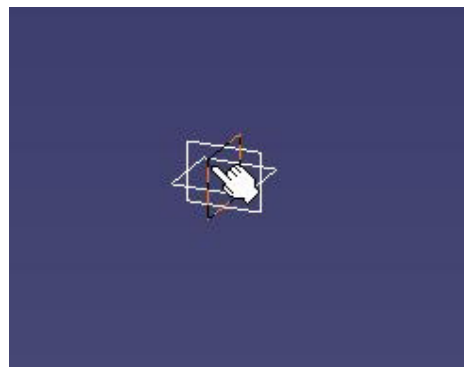
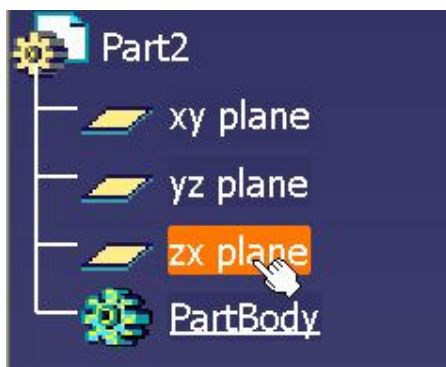


Fig. 2. Specification tree and reference plane

Part Design interface

The Part Design workbench look as follows (Fig. 3):

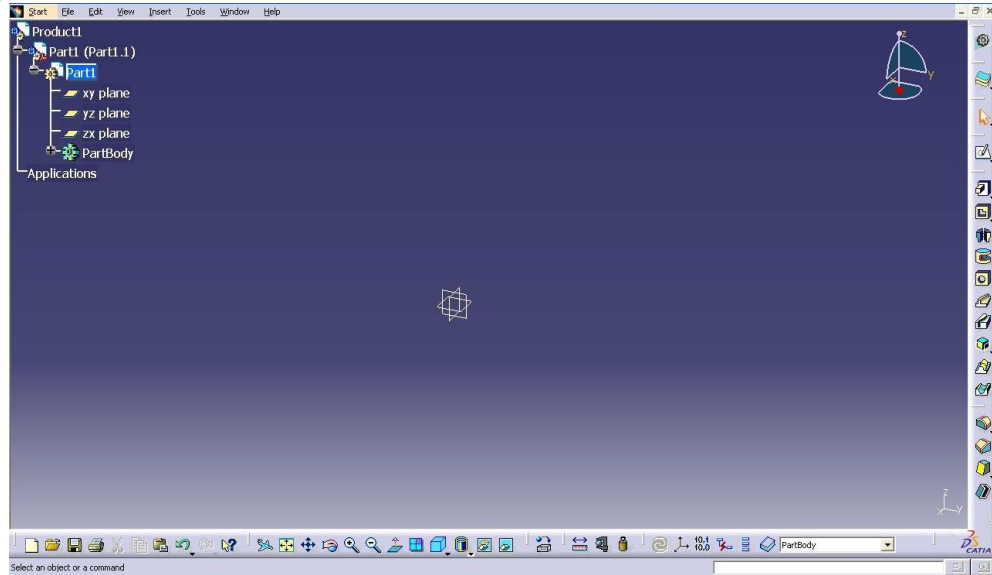
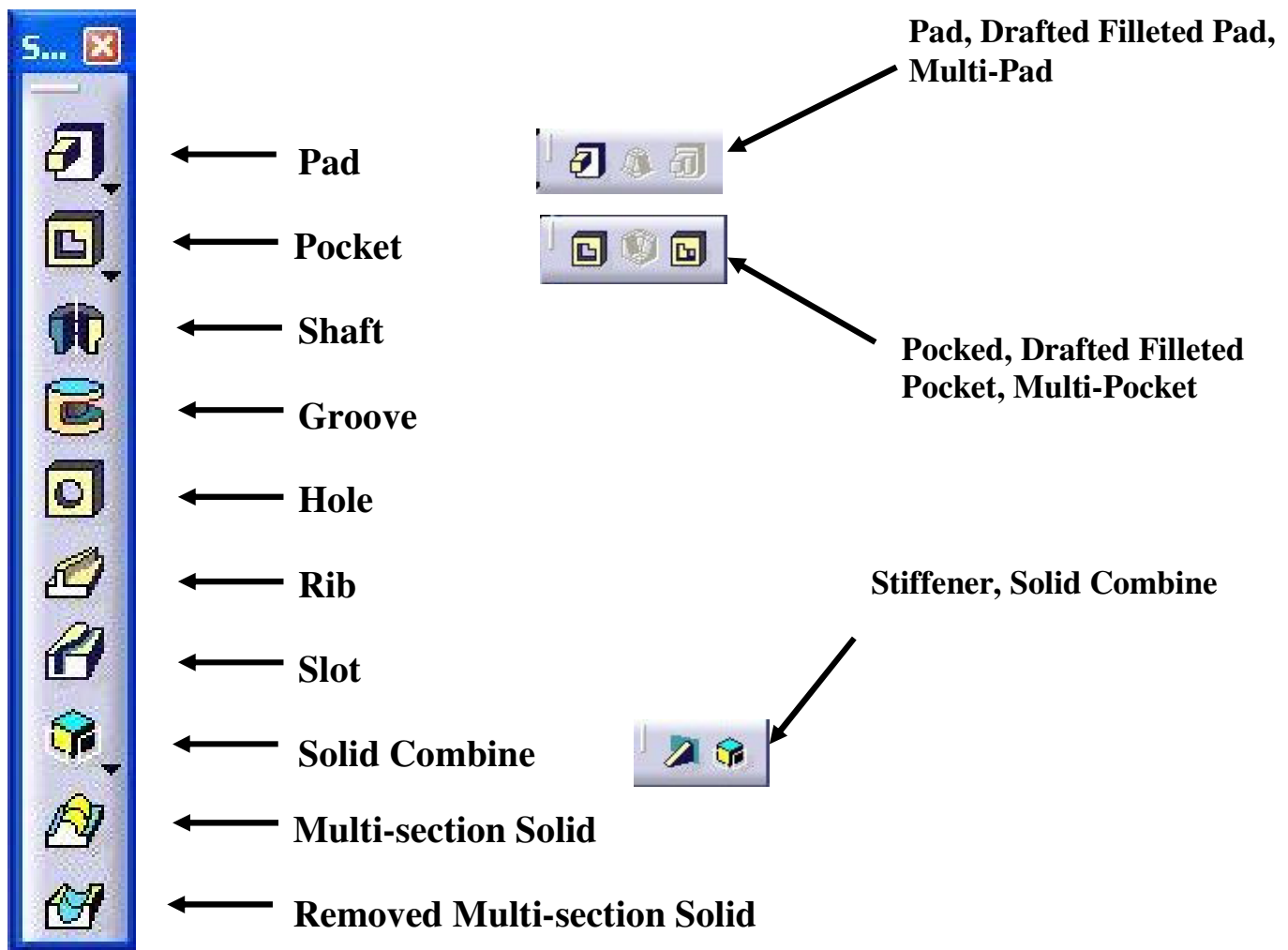


Fig. 3. The Part Design workbench.

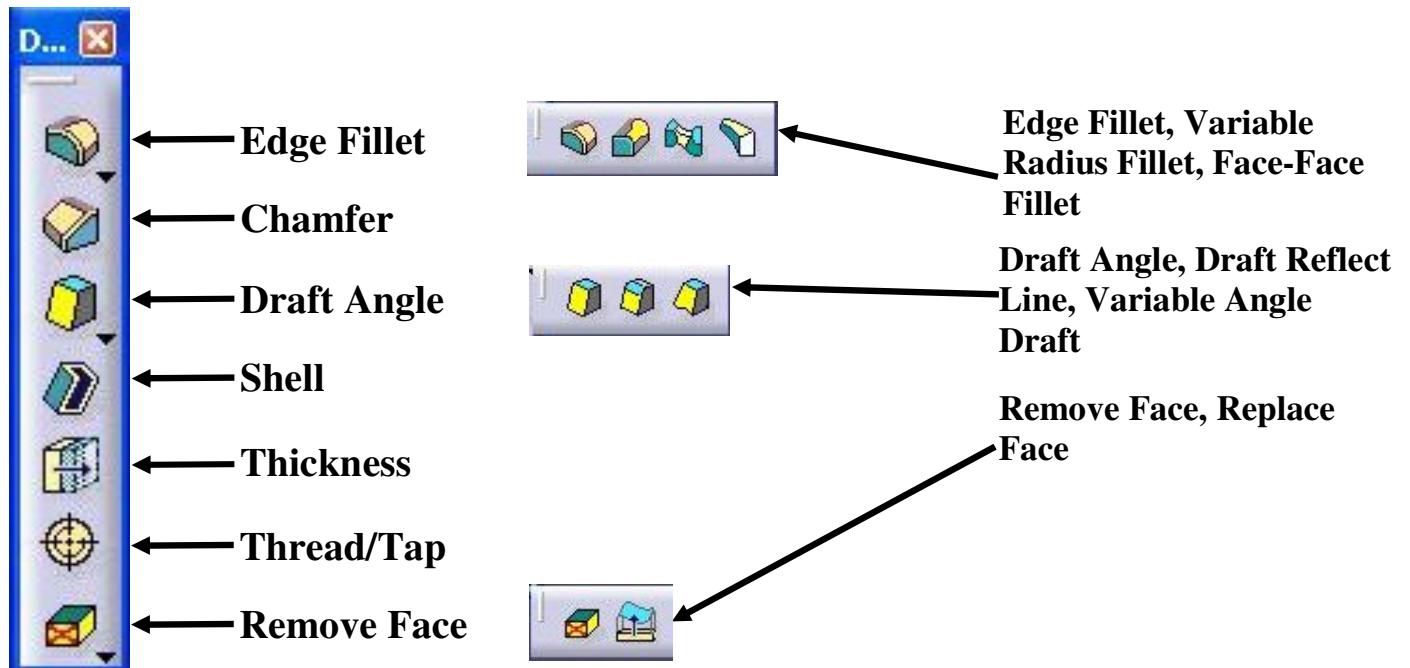
Par Design Tool Bars

There are five standard tool bars found in the Part Design Workbench. The four tool bars are shown below. Some tools have an arrow located at the bottom right of the tool icon. To display the other tool options you must select and hold left mouse button on the arrow.

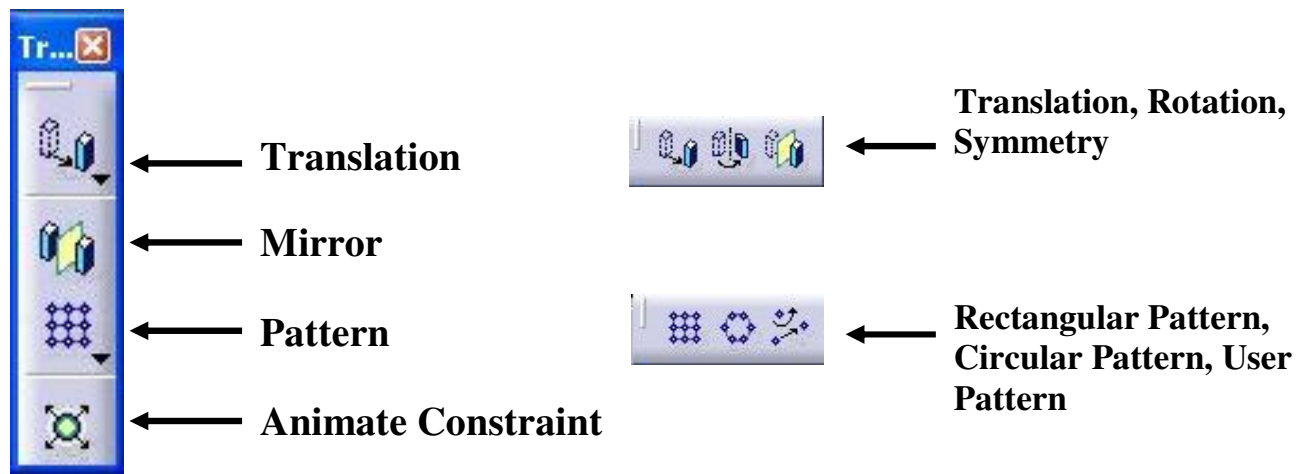
1. The Sketch-Based Features Tool Bar



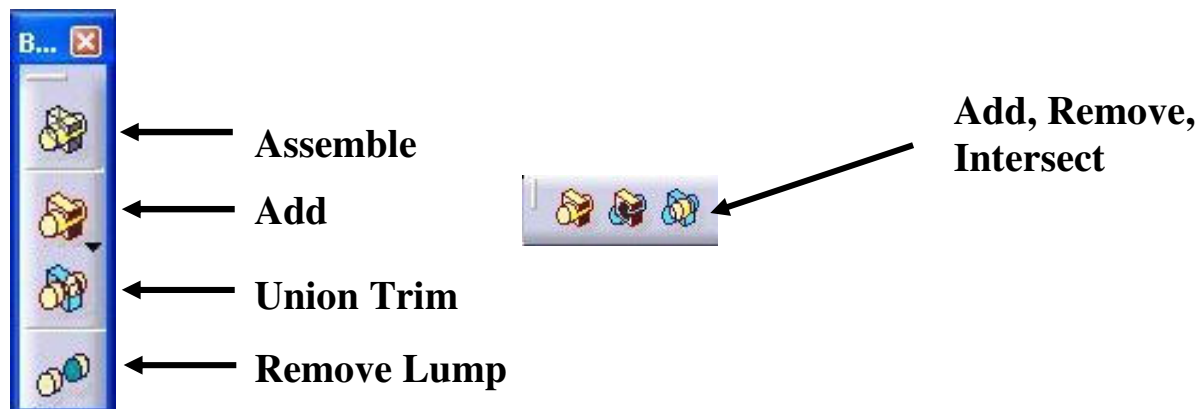
2. The Dress-Up Features



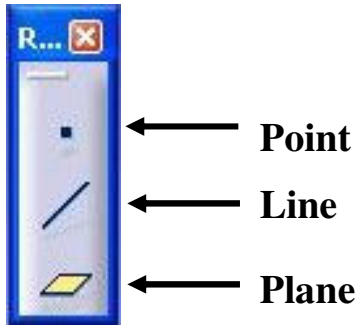
3. The Transformation



4. The Boolean Operations



5. Reference Elements

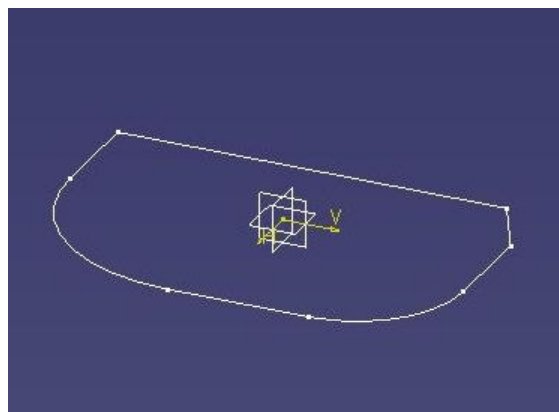
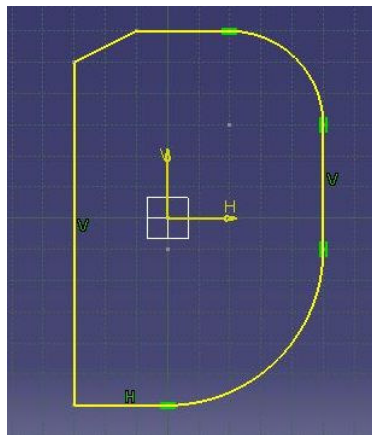



Crating Pads

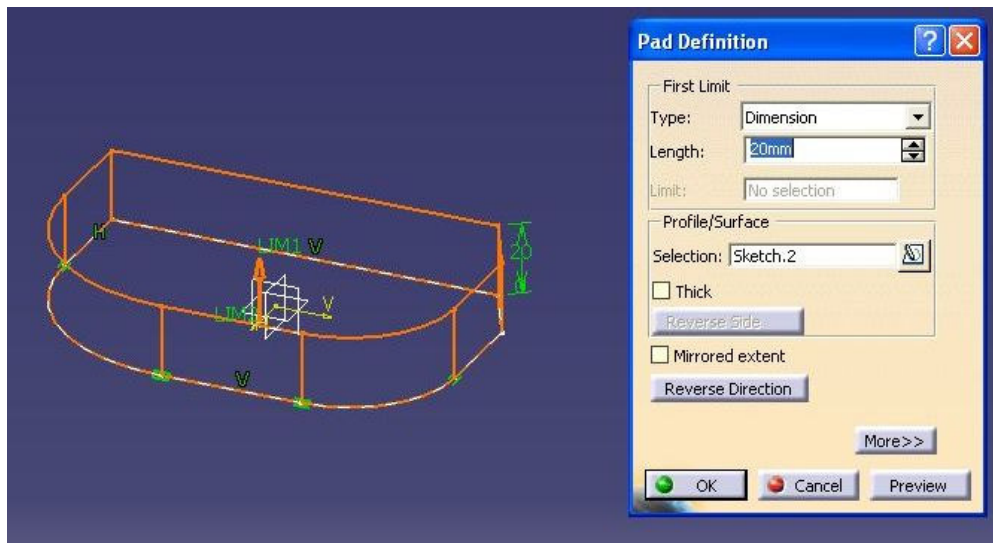
Pad Option

Example No. 1. Creating simple pad

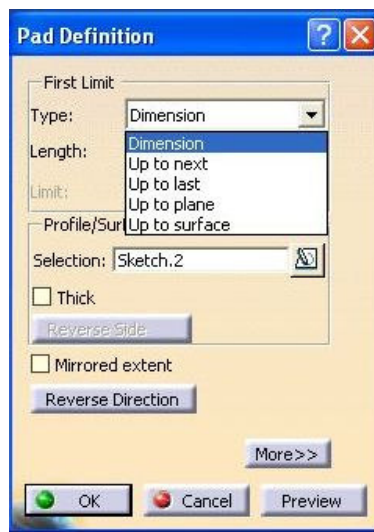
1. Create simple sketch on plane xy in the Sketcher Workbench.



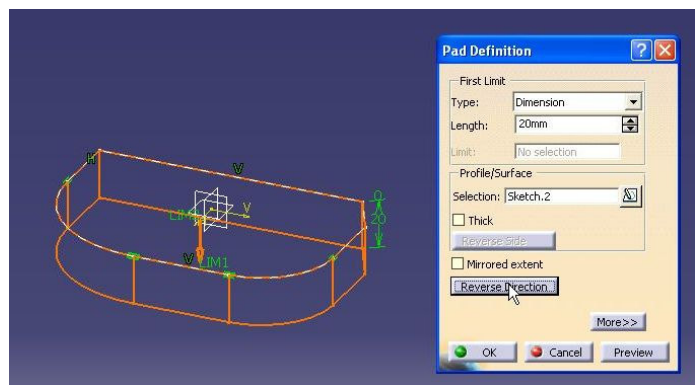
2. Select it and then select the Pad icon . The Pad Definition dialog box appears. Default options allow you to create a basic pad.



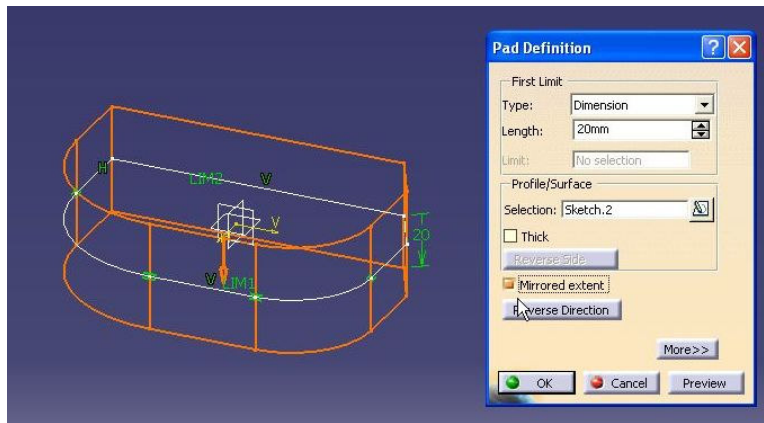
From Type options you can choose another type of the First Limit to crate the pad.



In Length options you can write a dimension of extraction. Reference Direction option changes the direction of extraction.



Mirror extent option creates pad for two directions.




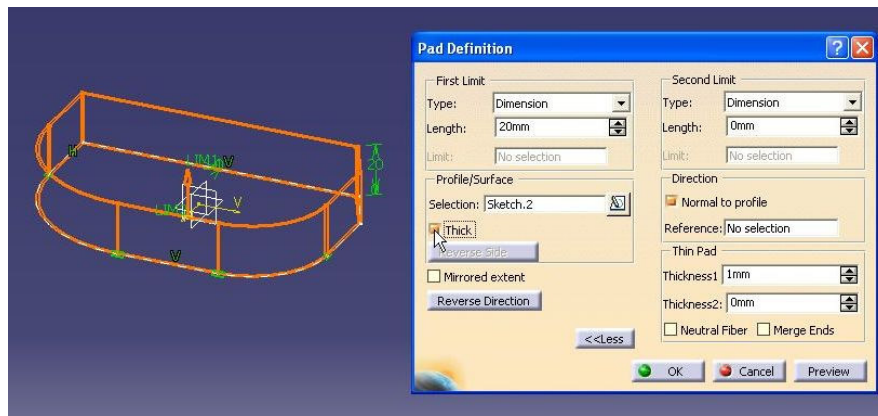
Confirm by clicking Preview button and confirm OK.

Example No. 2. Creating thin solid

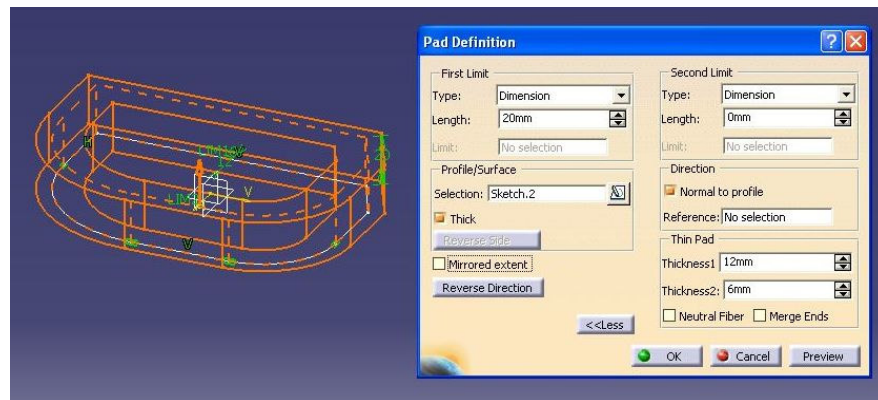
Thin solid is feature resulting from adding Thickness to both sides of a pad's profile.

1. Create simple sketch on plane xy in the Sketcher Workbench or you can use the previous sketch.

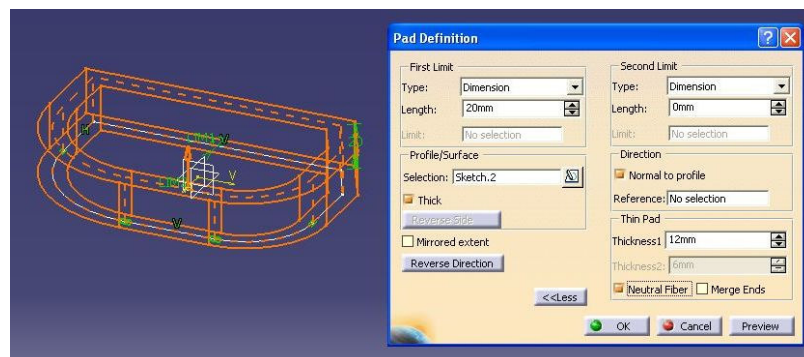
2. Select Pad icon  and select “More.” Then mark Thick options. The Thin Pad Frame appears.



3. Enter either Thickness1 or Thickness2 values or both at the same time. You can see that material has been added to one side or both sides of the profile.



4. If you want to add material equally to both sides, mark Natural Fiber. Thickness1 is then distributed equally (Thickness2 is not available)

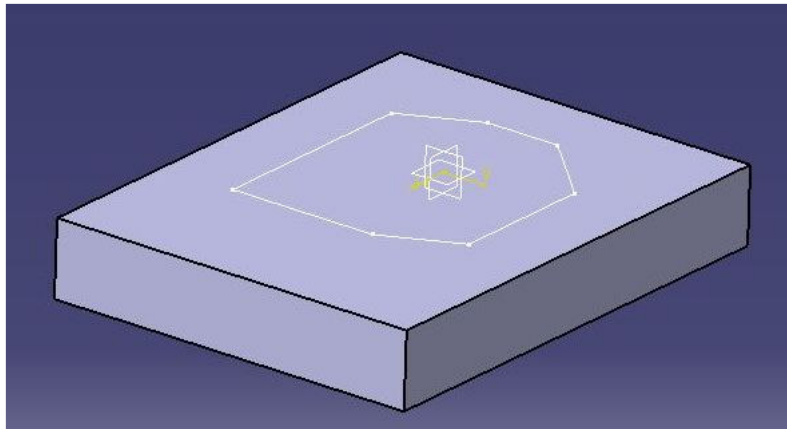



5. Select OK

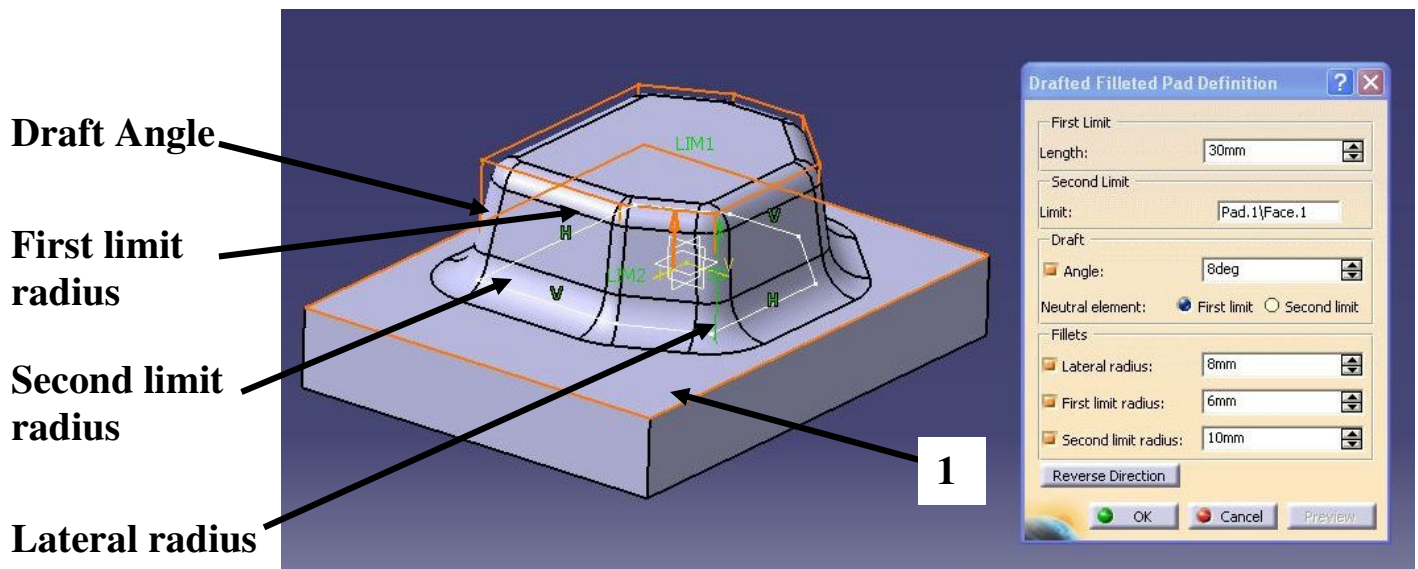
Example No. 3. Drafted Filleted Pad

This option creates a pad with draft and fillets simultaneously. A Drafted Filleted Pad is a solid extruded from 2D profile and its definition includes a draft angle and fillets.

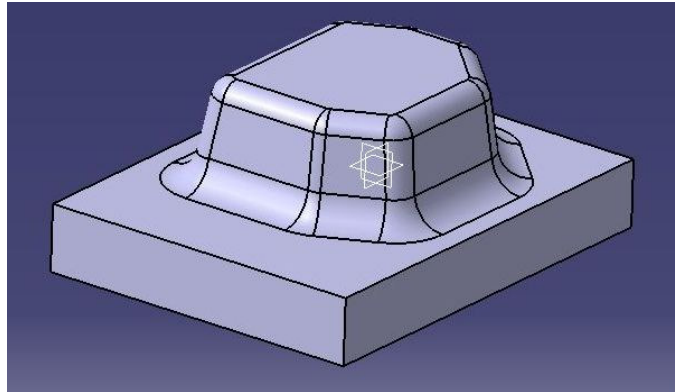
1. Open file Example3.



2. Select a profile on the surface and select Draft Filleted Pad . The Draft Filleted Pad dialog box appears.



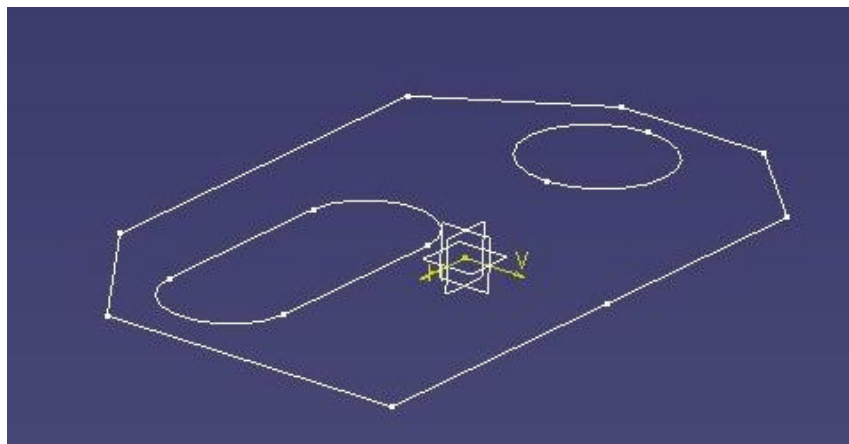
3. From Second Limit option select surface 1. Next, enter the Pad Length, the Draft Angle, the Lateral Radius, the First limit radius and the Second limit radius. Then select Preview button and confirm by clicking OK.



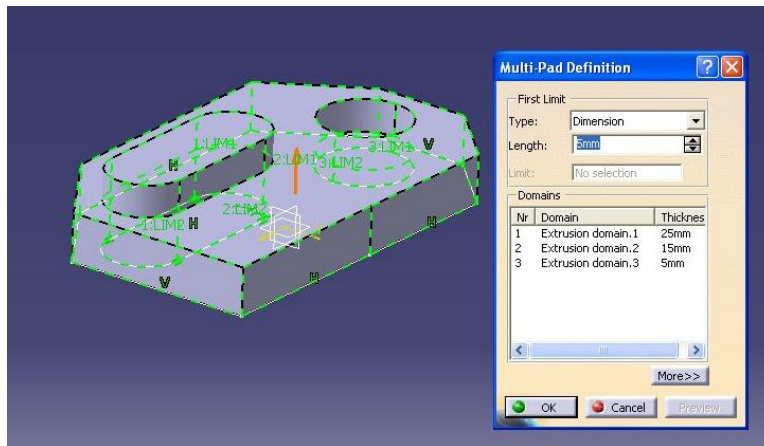
Example No. 4. Multi-Pad

This option extrudes multiple profiles belonging to a sketch using different length values at one time.

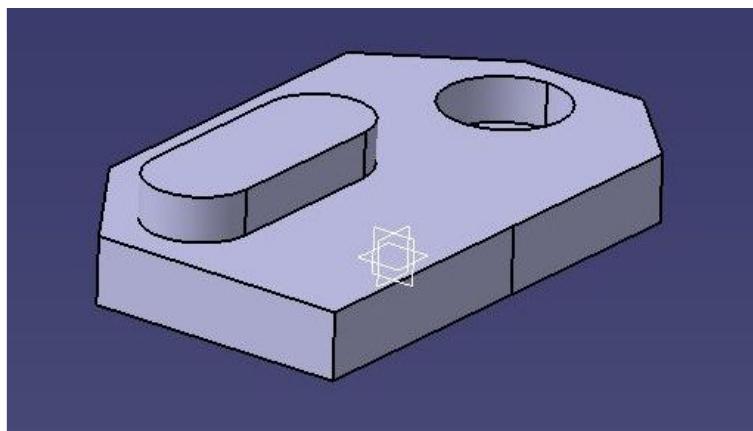
1. Create those profiles.



2. Then select Multi-Pad icon  and select profiles. The Multi-Pad dialog box appears.



3. Select Extrusion Domain1 (the elongated hole) and write in Length option a value of 25. Select Extrusion Domain2 (the biggest profile) and write in Length option a value of 15. Select Extrusion Domain3 (the circle) and write in Length option a value 5. Click Preview and confirm by clicking OK.

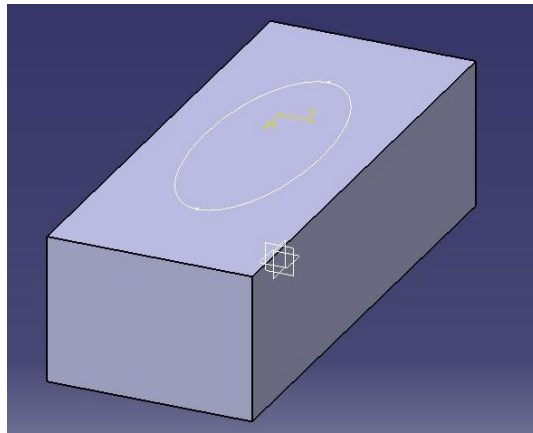



Pocket Option

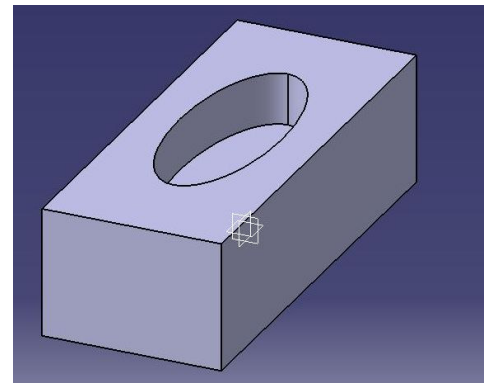
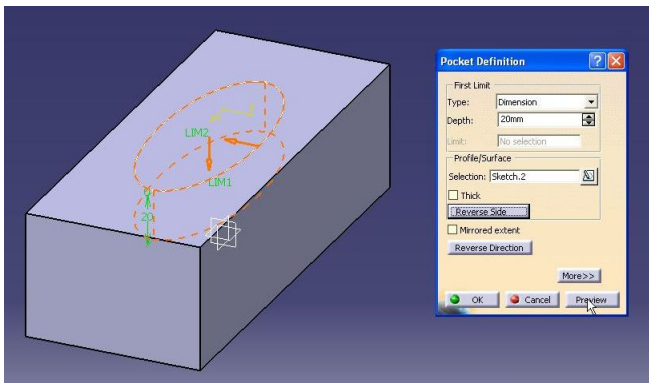
Example No. 5. Creating Pocked

A pocket removes material from an existing feature, by extruding 2D profile.

1. Open the Example5 file

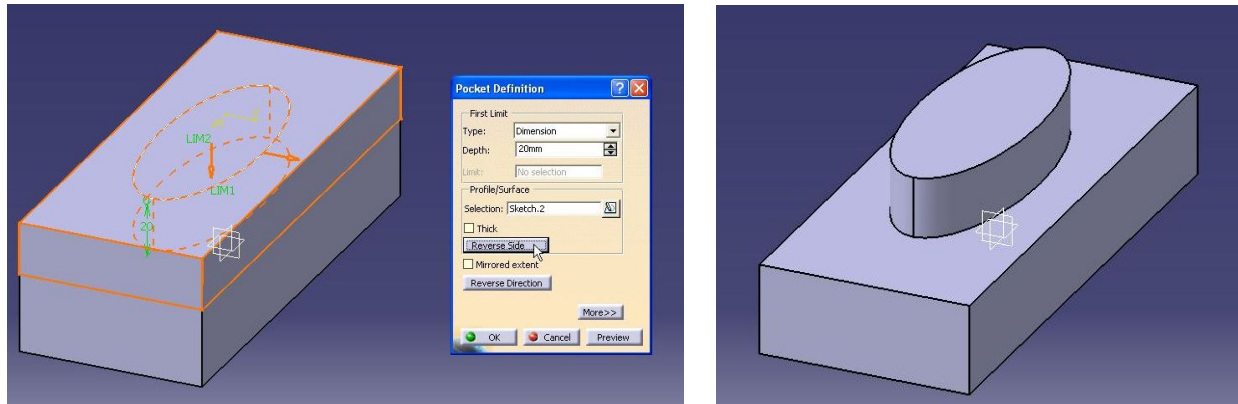


2. Select sketch on the solid and select Pocket icon . In Depth option you change dimension of a removing. Then select the Preview button.



3. If you want to remove material surrounding the profile use the Reverse side option. Click

the Reverse side button and select the Preview button and click OK to confirm.

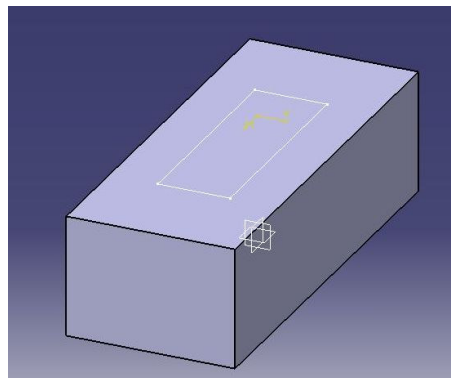


Material around the profile has been removed.

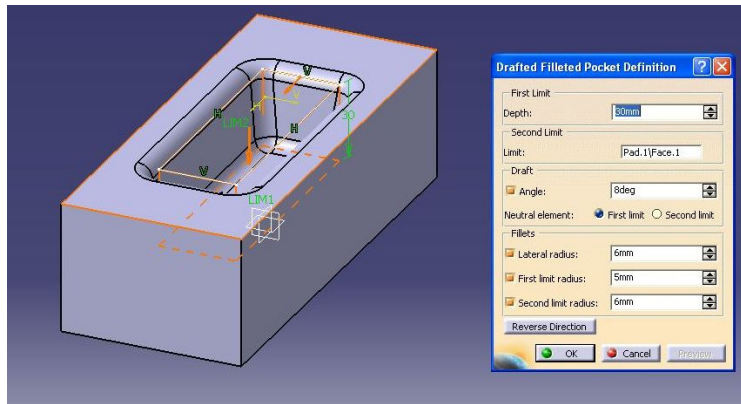
Example No. 6. Drafted Filleted Pocked

This option is similar to Drafted Filleted Pad. Drafted Filleted Pocked creates extraction with draft and fillets simultaneously.

1. Open Example6 file.



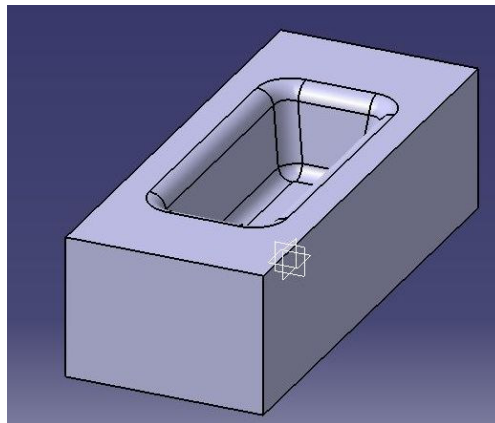
2. Select Drafted Filleted Pocked  and select a profile on the solid.



In First Limit option write in a new Depth dimension value (for example 30).

Selecting a second limit is obligatory.

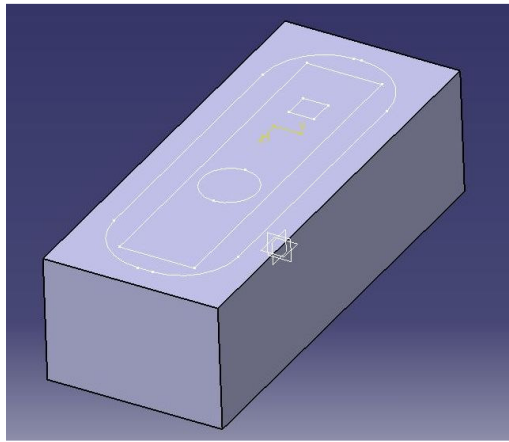
Select top face as the second limit. Then, enter the Draft Angle, the Lateral radius, the First limit radius and the Second limit radius. Then select Preview button and click OK.



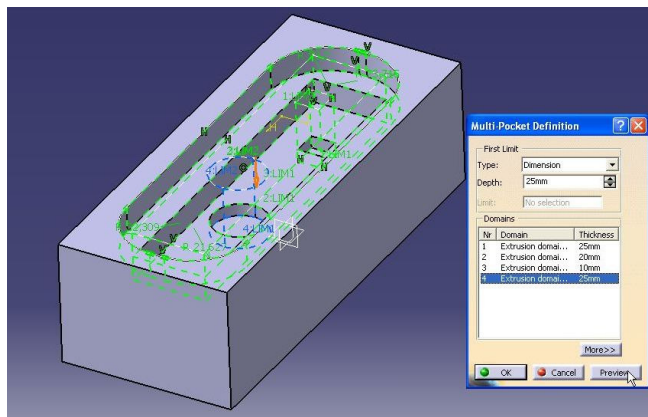
Example No. 7. Multi-Pocket

This option is similar to Multi-Pad. Multi-Pocket creates extraction for multiple profiles belonging to a sketch using different length values simultaneously

1. Open Example7 file

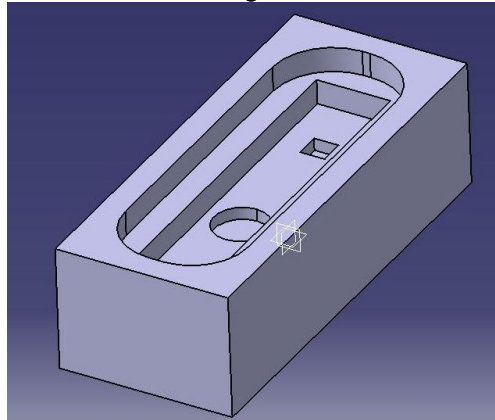


2. Select Multi-Pocket icon  and select profiles. The Multi-Pocket dialog box appears.



3. Select Extrusion Domain1 (the square) and write in Length option value that equals 25.

Select Extrusion Domain2 (the rectangle) and write in Length option value that equals 20. Select Extrusion Domain3 (the biggest profile) and write in Length option value that equals 10. Extrusion Domain2 (the circle) and write in Length option value that equals 30. Click Preview and confirm by clicking OK.



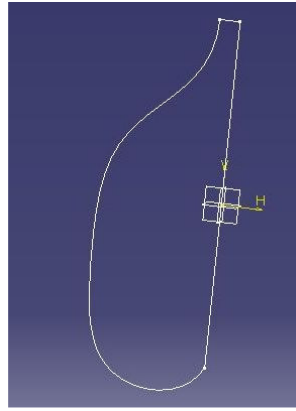
Shaft option


Example No. 8.

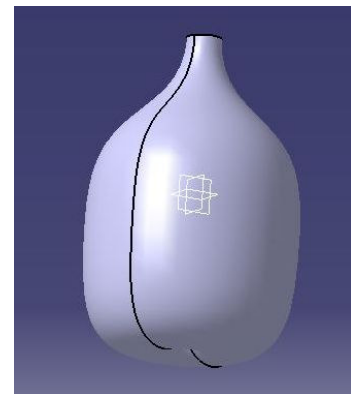
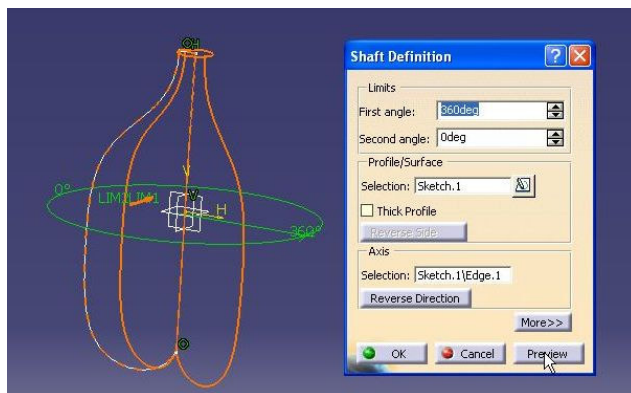
Shaft option has created solids from the revolution of a 2D profile around an axis. The axis and the profile must be created in the same sketch.

Example No. 8. Shaft option

1. Create such profile.



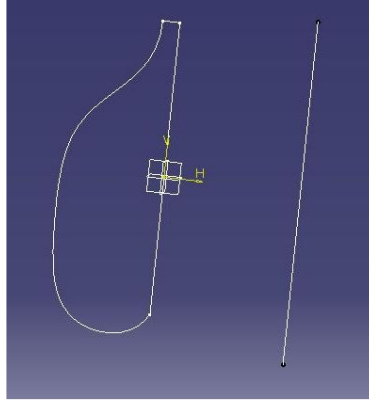
2. Select the profile and select Shaft icon . The Shaft dialog box appears. In the Limits options enter the First angle value that equals 360. This angle defines the revolution angle of the profile around the axis and is orientated in the clockwise direction. In the Limits options enter the Second angle that equals 0. This angle defines the revolution angle of the profile around the axis and is orientated in the counterclockwise direction. Then click OK.



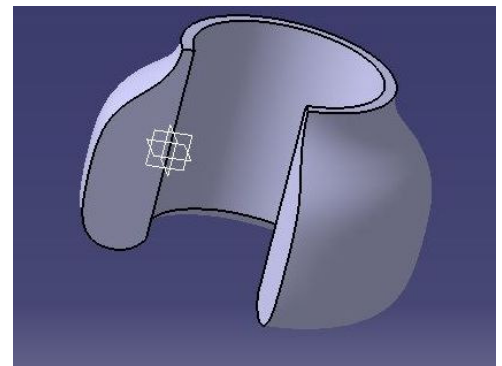
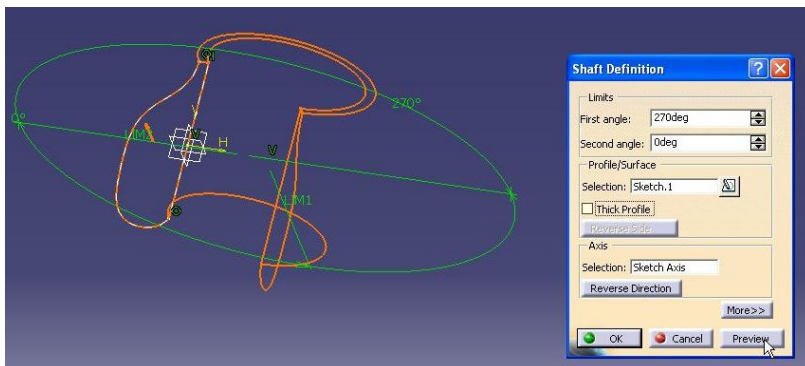
You can create Shaft solid with axis line which is not included in the profile.

Example No. 9.

1. Use as same profile as the previous. Then, create the vertical line on the same plane.



2. Select Shaft icon. In the Profile/Surface option select the profile. Then, from the Axis option select the vertical line. In the Limits options enter the First angle that equals 270. In the Limits options enter the Second angle that has 0 value. Select the Preview button and confirm by clicking OK.

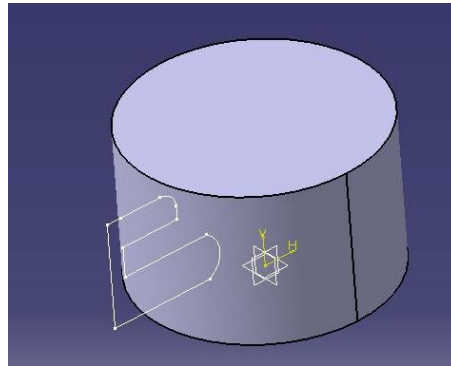



Grove option

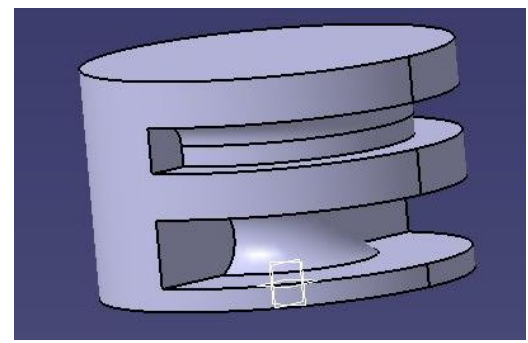
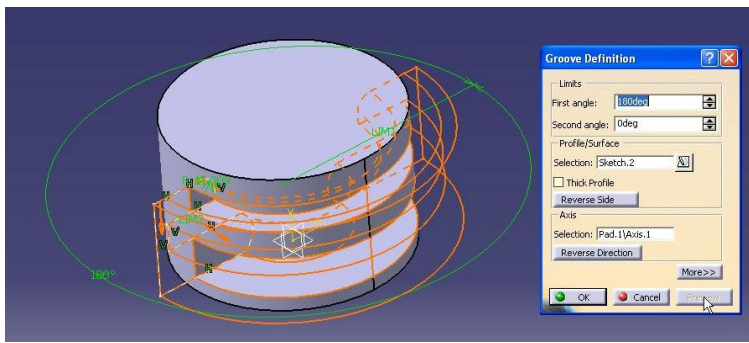
Example No. 10.

Grooves are revolved profiles that remove material from existing solids. A groove is created by selecting the groove profile and the revolution axis.

1. Open Example10 file.




2. Select Grove icon  and select the groove profile. Then in Axis option select the vertical vector (V). In the Limits options enter the First angle value that equals 180. In the Limits options enter the Second angle value that equals 180. Select the Preview button and confirm by clicking OK.

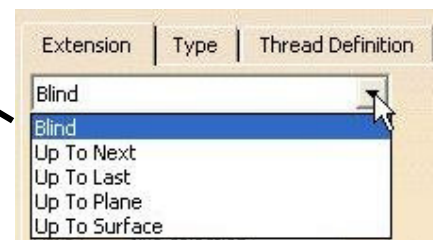
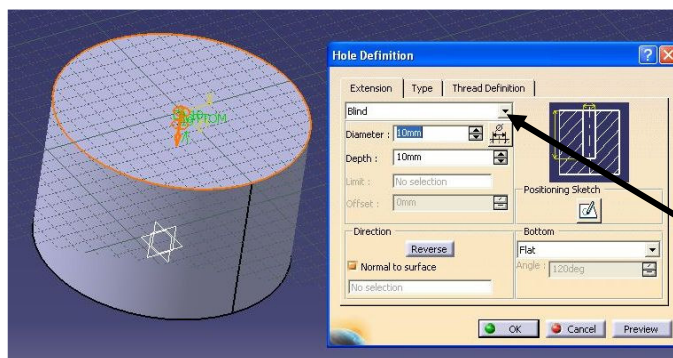


Hole option

Example No. 11.

The Hole removes circular material from an exiting solid at once.

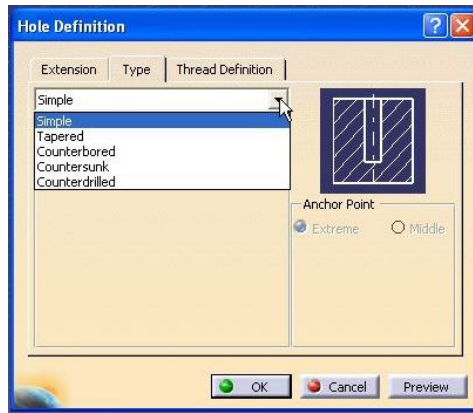
1. Use the same solid as the previous one. Then, select Hole icon  and select the top surface on the solid. The Hole dialog box appears.



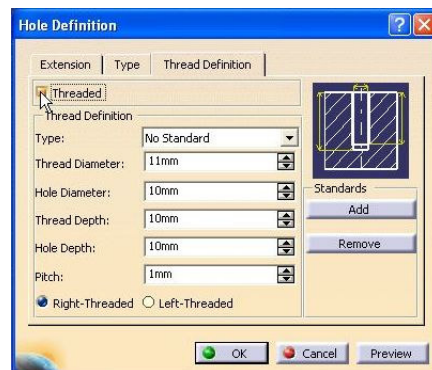
2. Default options allow you to create a standard hole. In Extension options you can change a type of extract hole. If you chose a blind hole, enter the Diameter value that

equals 40 and then enter the Depth value that equals 30 and select Preview.

In Type options you can chose different hole types.

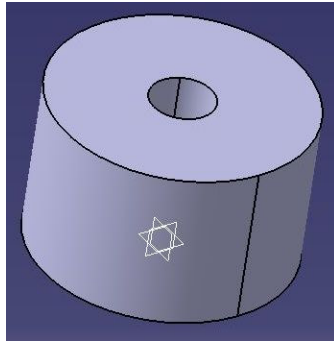


In Thread Definition you can chose type of thread but remember the application displays the hole in the geometry area but not the thread.



3. Change location of the hole. You can change location by clicking the arrow at the hole and moving mouse or selecting Positioning Sketch and changing location in Sketch.

Then click OK.

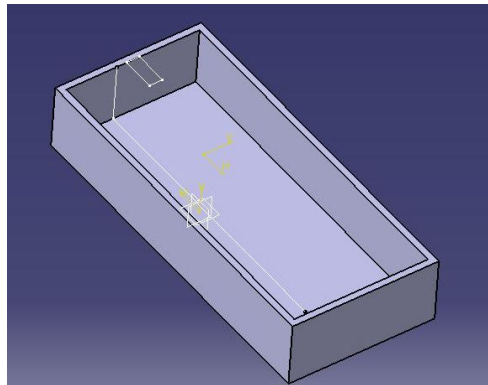


Rib option

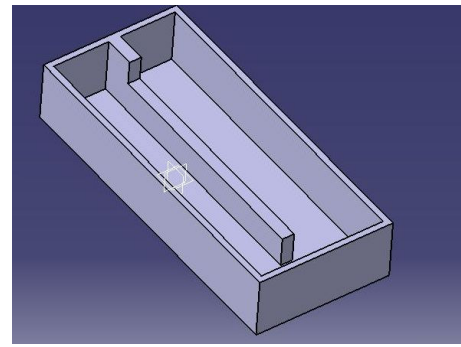
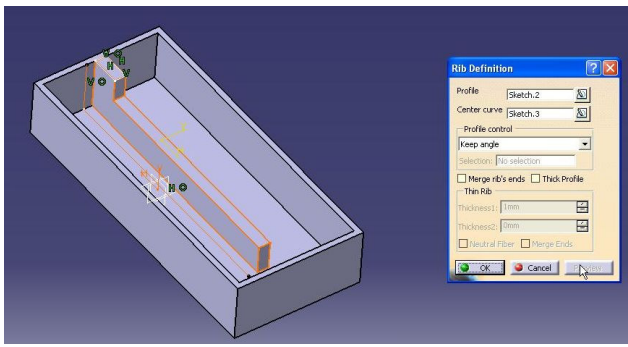
Example No. 12

Rib is a profile swept along an open or closed center curve to create a 3D feature.

1. Open Example11 file.



2. Select Rib icon . In Rib Definition dialog box from Profile option select the rectangle profile. Then, from Center curve option select then line and click OK to create Rib.

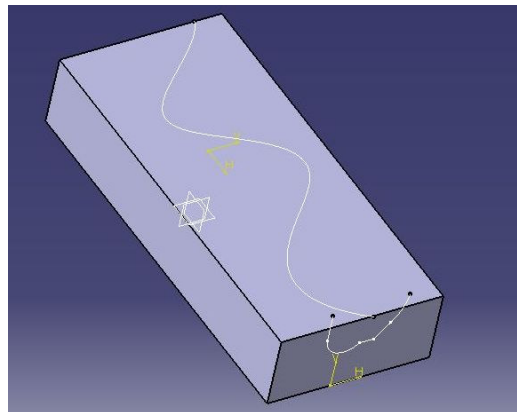



Slot Option

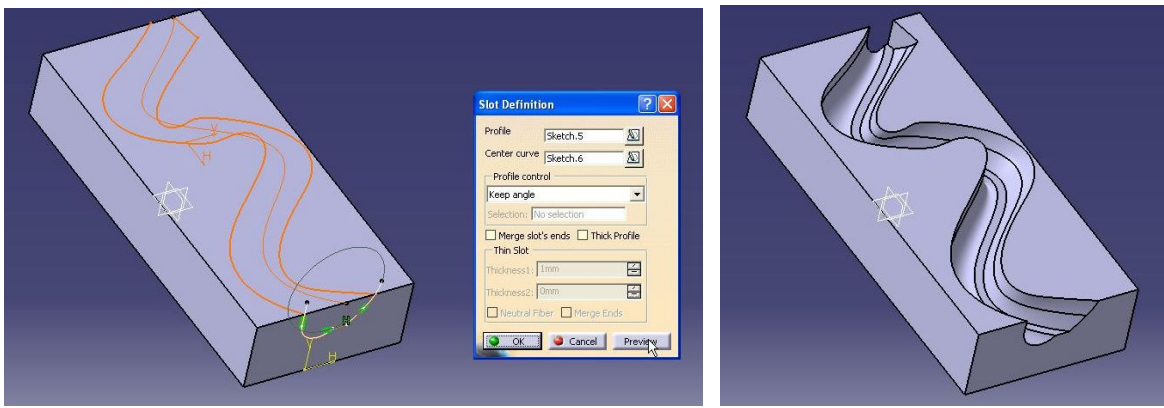
Example No. 13.

This option is similar to Rib but Slot remove material. A profile that will create an extraction can be closed or open profile.

1. Open Example13 file.



2. Select Slot icon . In Slot Definition dialog box in Profile option select the profile. Then, in Center curve option select spline line and confirm by clicking preview and select OK.

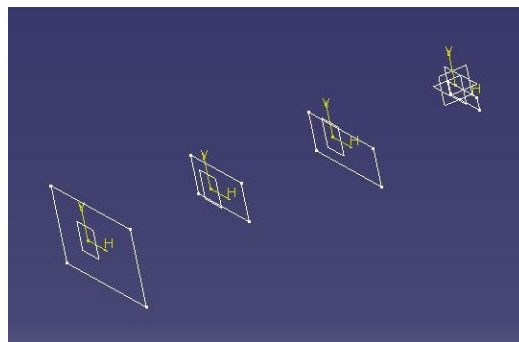


Multi-Sections Solid (Loft)

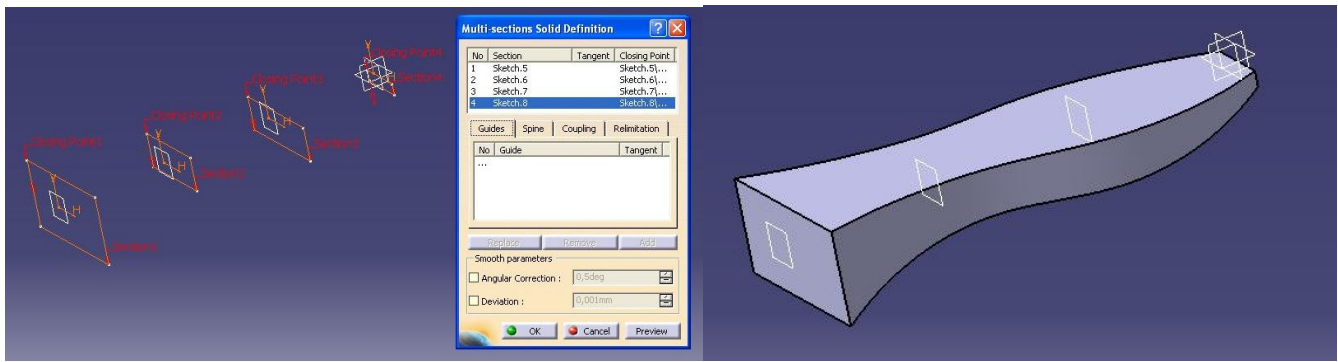
Example No. 14.

Loft creates solid which can be generated by two or more planar sections swept along a spine.

1. Open Example14 file.



2. Select Multi-Sections Solid icon . The Loft dialog box appears. Then, select four profiles in Section options and confirm by clicking OK.

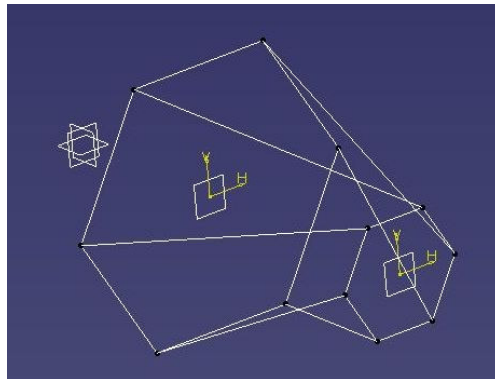


Closing Points on the sketches have to be aligned in order to get the proper orientation of sections. Otherwise Loft would be twisted.

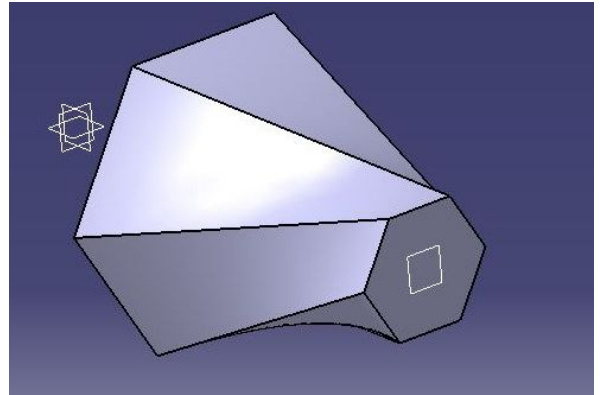
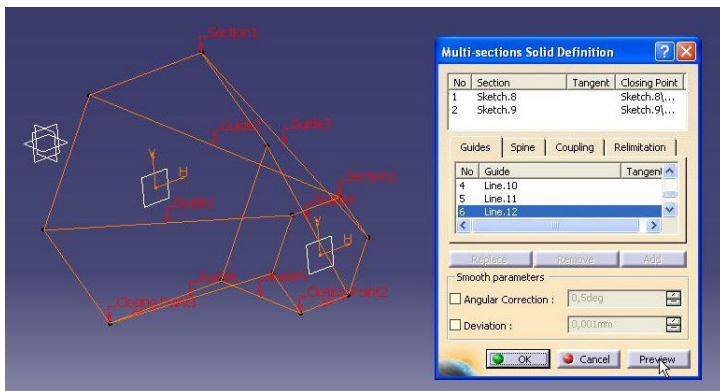
Example No. 15.

Create Loft with Guide line

1. Open Example15 file and create Loft with Guide line.



2. Select Multi-Sections Solid icon. Then in Loft dialog box select two profiles in Section options and six guide lines in Guide option in the clockwise direction. Then, confirm by clicking OK.

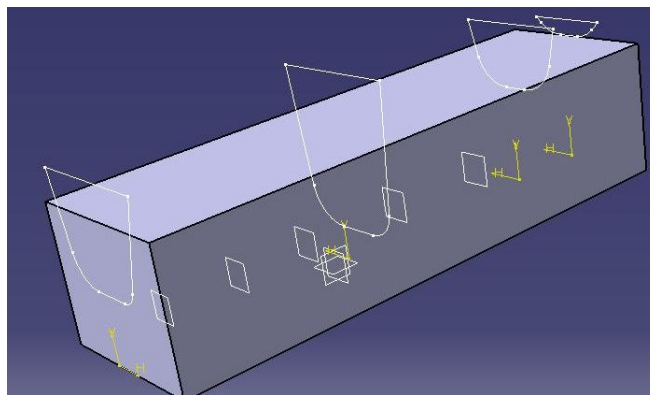


Removed Multi-Sections Solid

Example No. 16.

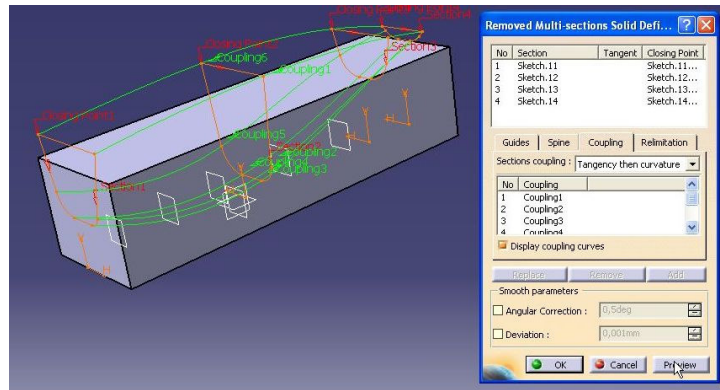
Removed Multi-Sections Solids removes material from an existing feature. Removing solid can be generated in two or more planar sections swept along a spine.

1. Open Example16 file.

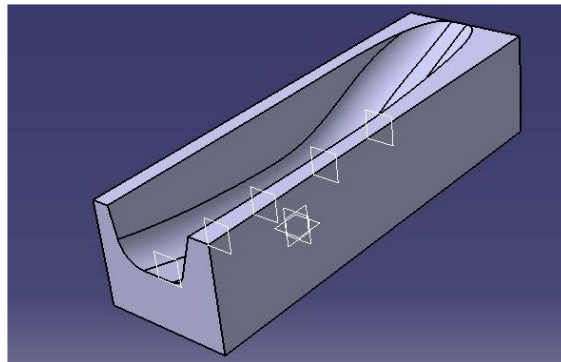


2. Select Removed Multi-Sections Solids icon  and select four profiles. Then, choose

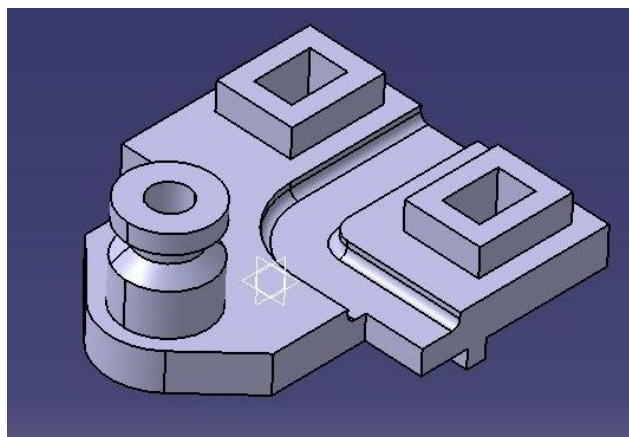
Coupling option and select points on profiles to create couplings. You have to select those points in appropriate order (see on the figure below).



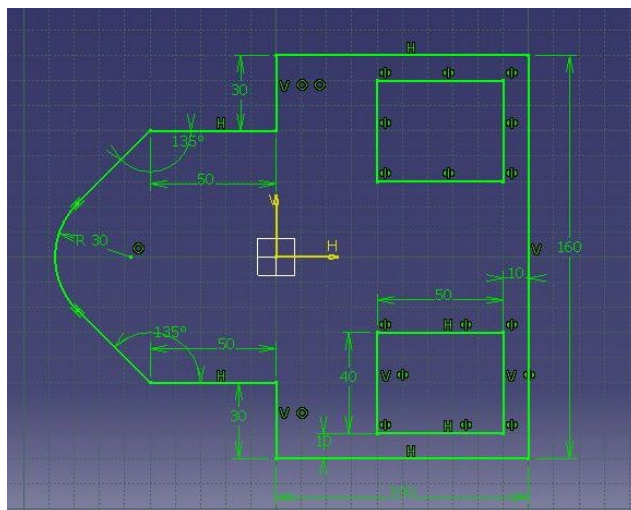
3. After you create six couplings select preview and confirm by clicking OK.



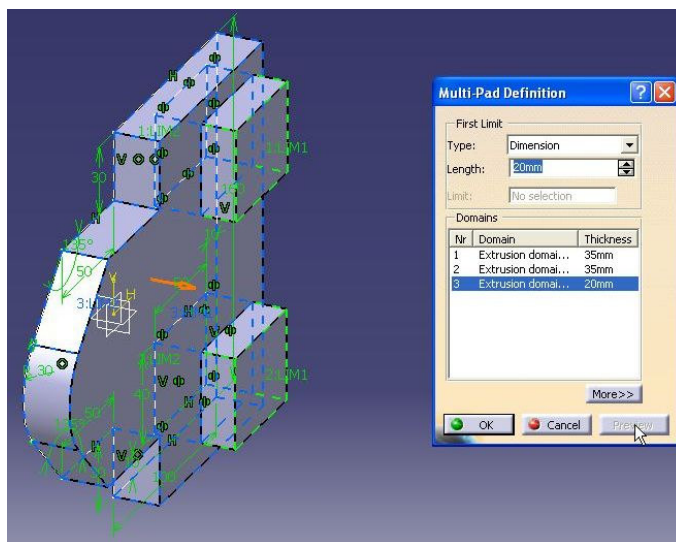
Task No. 1. Create part



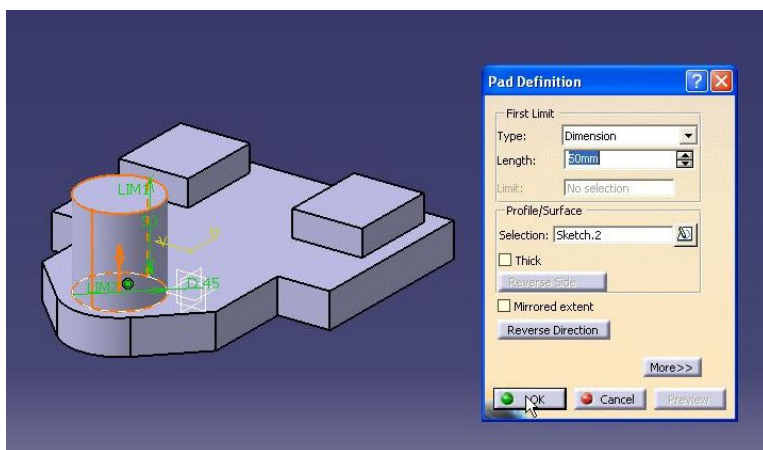
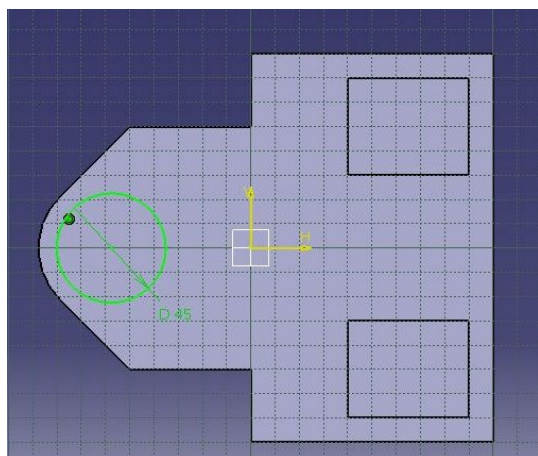
1. Create sketches on yz plane.



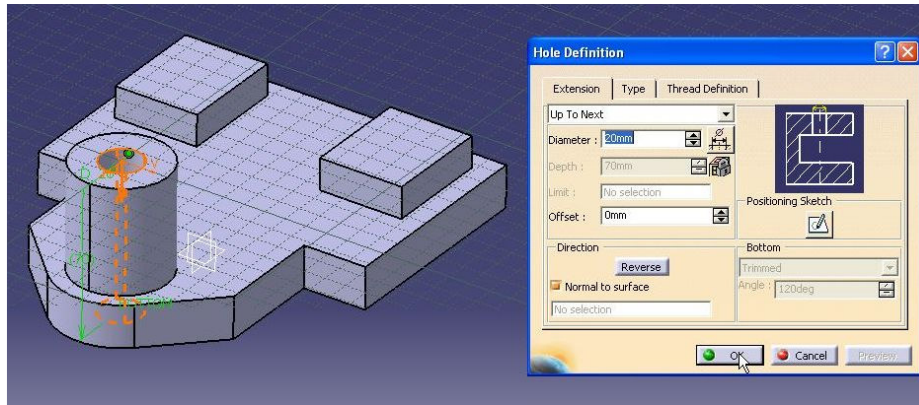
2. Create Multi-Pad.



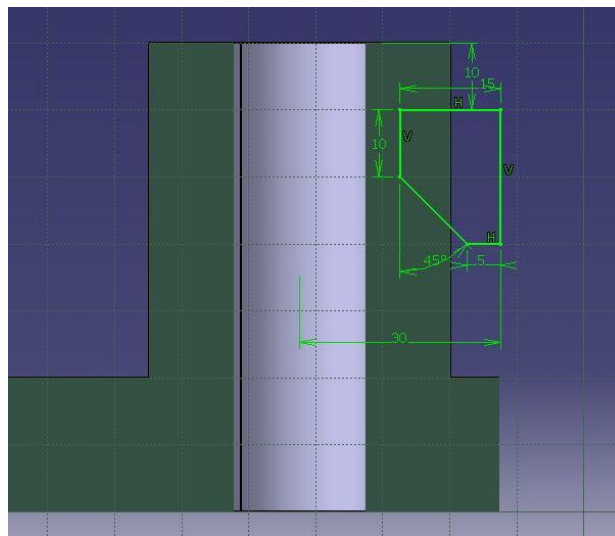
3. Then create circle sketch on top face and create pad.



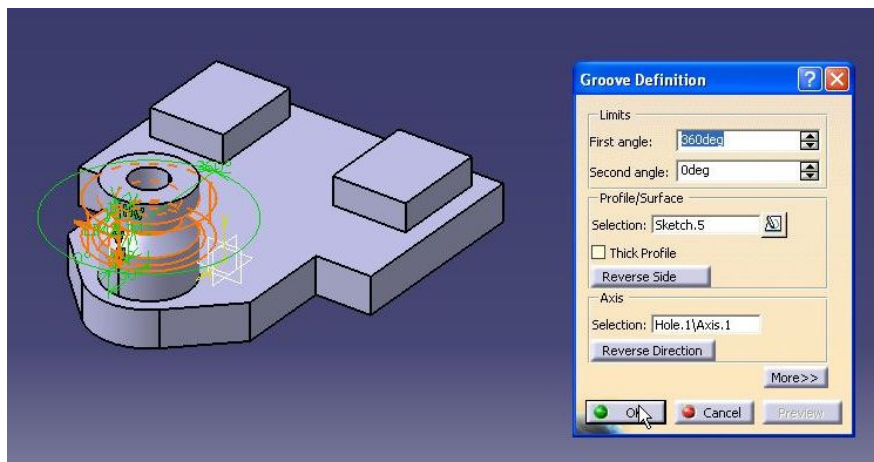
4. Then create hole.



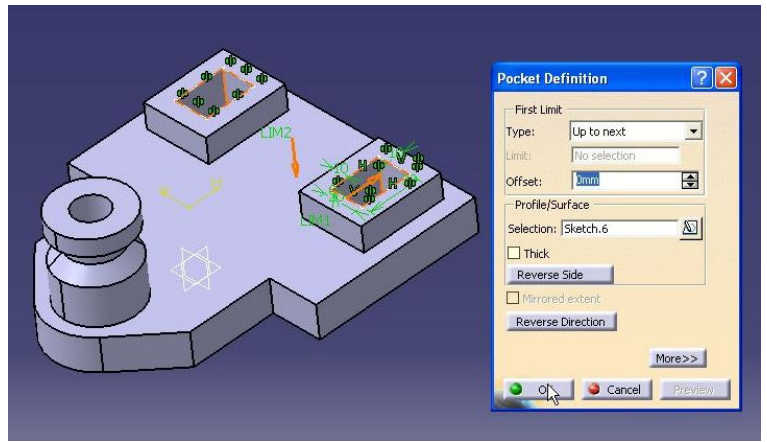
5. Create sketch on zx plane.



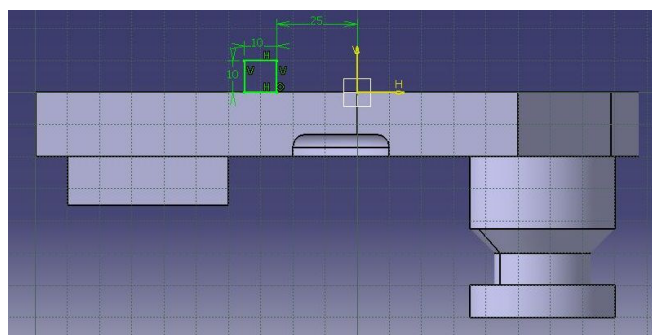
6. Use it to create groove.



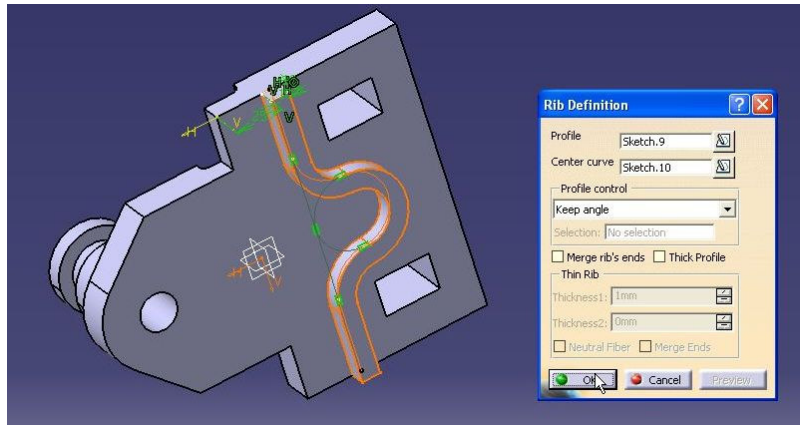
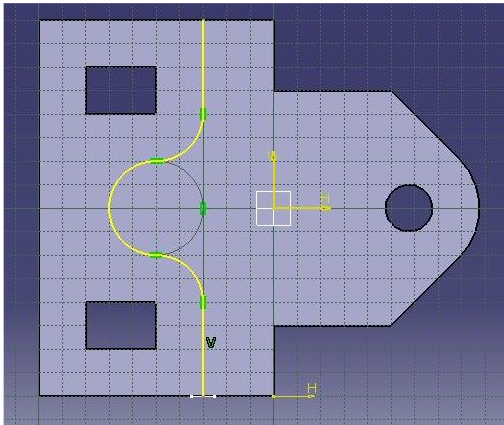
7. Create rectangles sketch and create a pocket.



The technical drawing shows the front view of a mechanical component. The main body is a horizontal plate with a total width of 60 units and a height of 15 units. On the left side, there is a rectangular cutout with a width of 20 units and a height of 7 units. To the right of this cutout, there is a vertical feature consisting of a base with a width of 30 units and a height of 15 units, topped by a smaller section with a width of 10 units and a height of 4 units. A small square hole is located at the top center of the main body, with a width of 10 units and a height of 4 units. The drawing includes dimension lines and arrows indicating the measurements.



11. Then crate profile sketch on the bottom face and create rib



12. Save your file.

