Getting Started

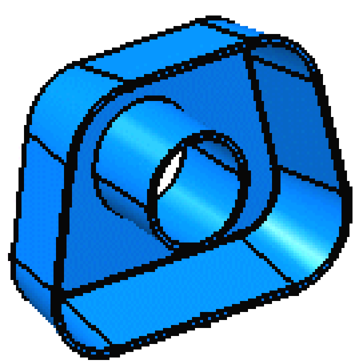
Before getting into the detailed instructions for using CATIA Version 5 parts, the following tutorial aims at giving you a feel as to what you can do with the product. It provides a step-by-step scenario showing you how to use key functionalities.

The main tasks described in this section are:

Entering the Part Design Workbench
Creating a Pad
Drafting a Face
Filleting an Edge
Editing the Pad
Mirroring the Part
Sketching a Circle from a Face
Creating a Pocket
Shelling the Part

All together, the tasks should take about ten minutes to complete.

The final part will look like this:



Now, let's get to sketching the profile!

Entering the Part Design Workbench

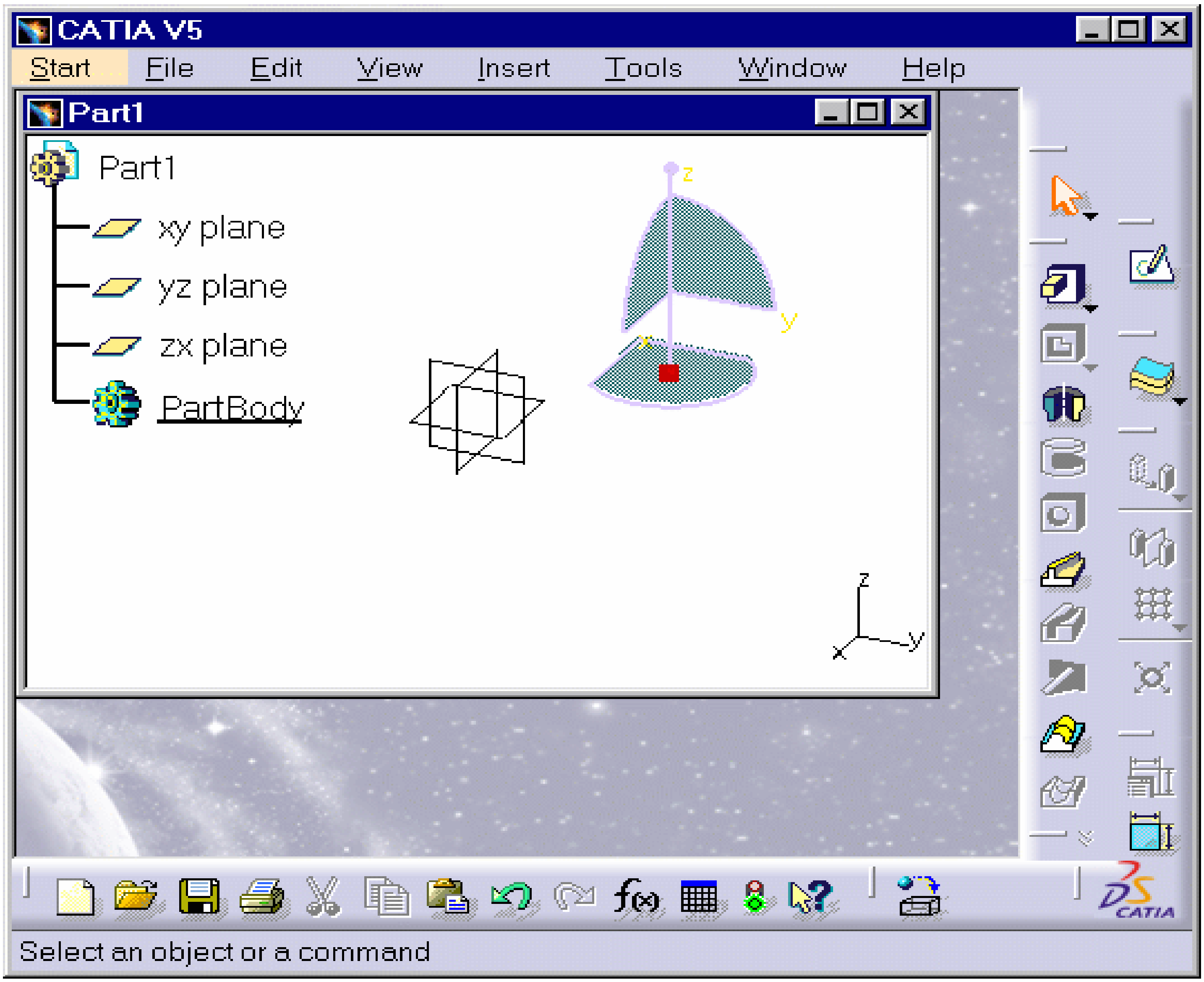
This first task shows you how to enter the Part Design workbench.

1. Select the `File -> New` commands (or click the `New` icon).

   The `New` dialog box is displayed, allowing you to choose the type of document you need.

2. Select `Part` in the `List of Types` field and click `OK`.

   The Part Design workbench is loaded and an empty CATPart document opens.



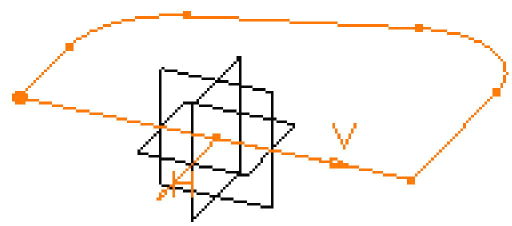
The commands for creating and editing features are available in the workbench toolbar. Now, let's perform the following task Creating a Pad.

Creating a Pad

This task will show you how to create a pad, that is extrude a profile sketched in the Sketcher workbench. For more about this workbench, please refer to *CATIA-Dynamic Sketcher User's Guide Version 5*.

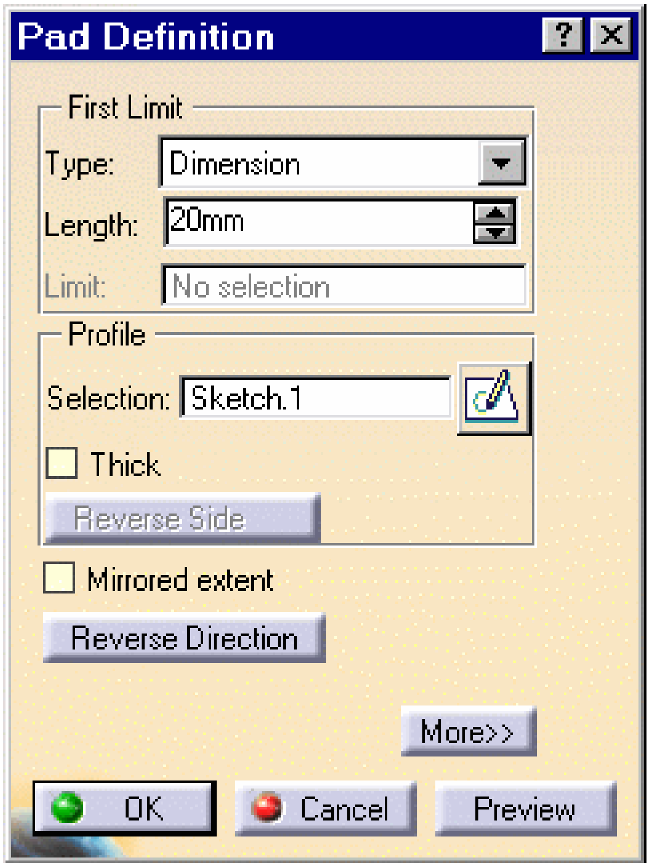
Open the GettingStarted1.CATPart document to open the required profile.

Your profile belongs to Sketch.1 and was created on plane xy. It looks like this:



1. Select the profile if not already selected and click the Pad icon .

The Pad Definition dialog box appears. Default options allow you to create a basic pad.

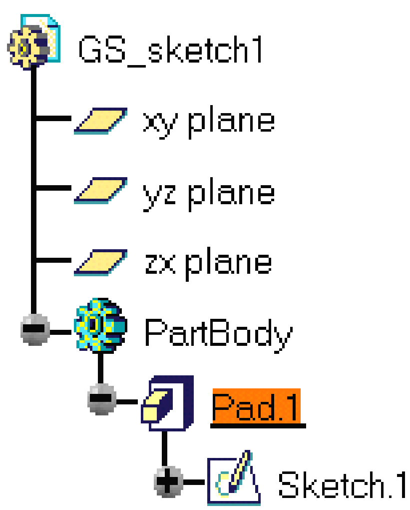


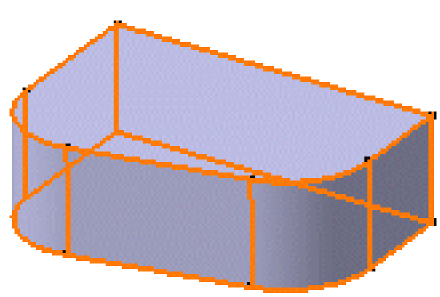
2. As you prefer to create a larger pad, enter 60 mm in the `Length` field.

   The application previews the pad to be created.

3. Click OK.

   The pad is created. The extrusion is performed in a direction which is normal to the sketch plane. The application displays this creation in the specification tree:





CATIA lets you control the display of some of the part components. To know more about the components you can display or hide, refer to Customizing the Tree and Geometry Views.

For more about pads, refer to Pads, 'Up to Next' Pads, 'Up to Last' Pads, 'Up to Plane' Pads, 'Up to Surface' Pads, Pads not Normal to Sketch Plane.

Drafting a Face

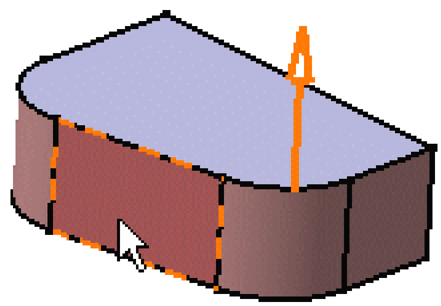
This task will show you how to draft a face.

1. Click the `Draft Angle` icon .

The Draft Definition dialog box appears. The application displays the default pulling direction on the part.

2. Select the face as shown by the arrow as the face to be drafted.

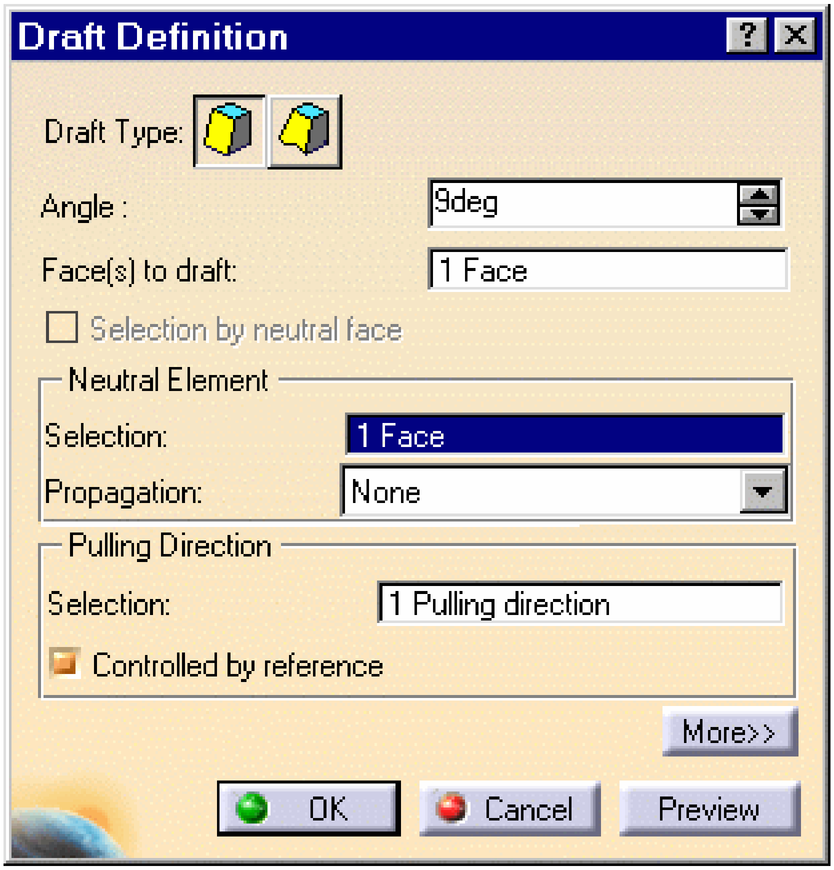
The application detects all the faces to be drafted. The selected face is now in dark red whereas the other faces are in a lighter red.

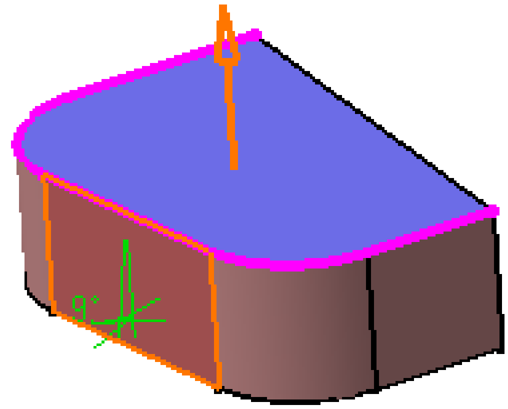


3. Click the Selection field of the Neutral Element frame and select the upper face.

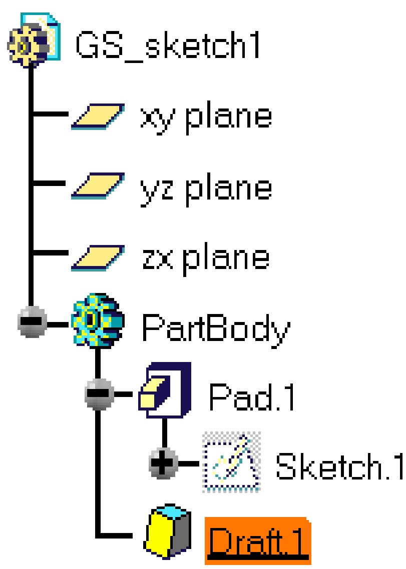
The neutral element is now displayed in blue, the neutral curve in pink.

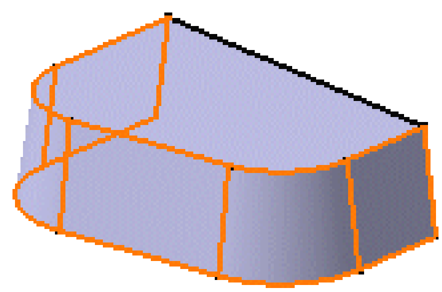
4. Enter 9 degrees in the `Angle` field.





5. Click OK. The part is drafted:





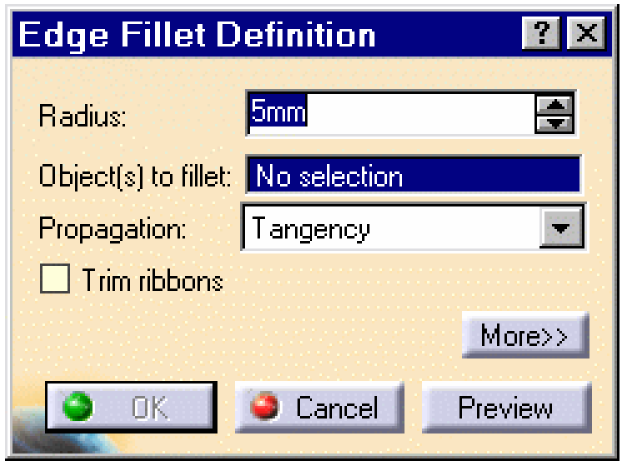
For more about drafts, please refer to Basic Draft, and to Draft with Parting Element.

Filleting an Edge

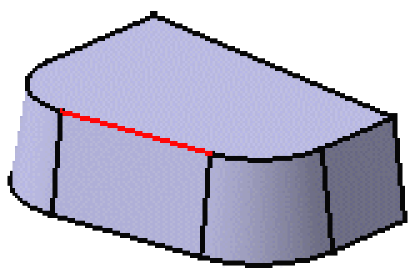
In this task you will learn how to use one of the fillet commands designed to fillet edges.

1. Click the `Edge Fillet` icon .

   The Edge Fillet Definition dialog box appears. It contains default values.



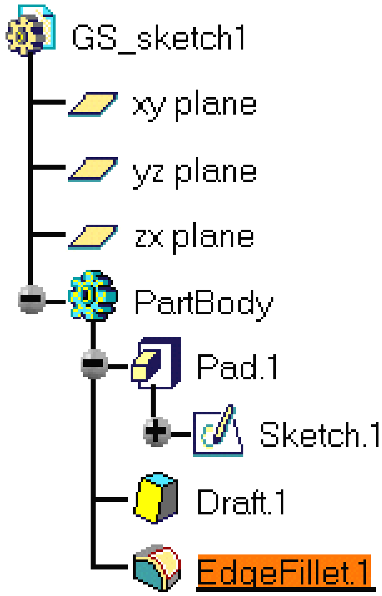
2. Select the edge to be filleted, that is, to be rounded.

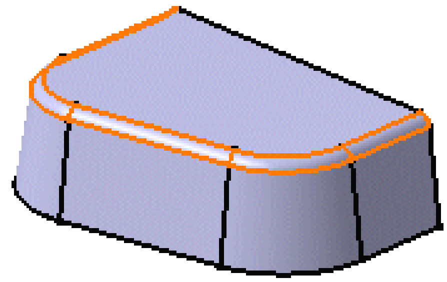


Clicking Preview lets you see what the default fillet would look like.

3. Enter 7 mm as the new radius value and click OK.

Here is your part:





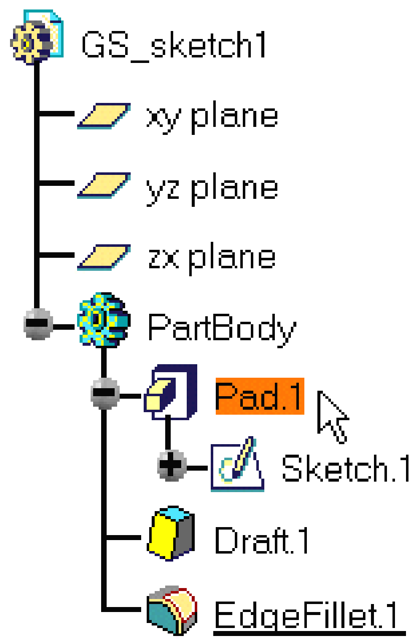
For more about fillets, please refer to Edge Fillet, Face-Face Fillet, Tritangent Fillet , Variable Radius Fillet.

Editing the Pad

Actually, you would like the pad to be thicker. This task shows you how to edit the pad, then how to color the part.

1. Double-click `Pad.1`.

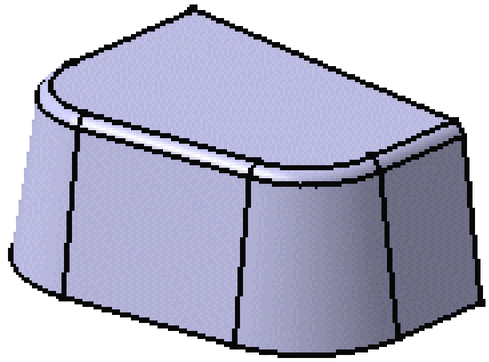
You can do it in the specification tree if you wish.



2. In the Pad Definition dialog box that appears, enter 90 mm as the new length value.

3. Click OK.

The part is modified accordingly.



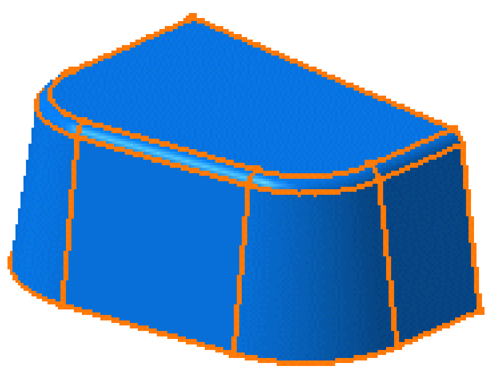
4. Now select `Part Body`.

5. Select the `Edit -> Properties` command and click the `Graphic` tab to change the color of your part.

6. Set the color of your choice in the Color combo box and click `OK`.

   To have details about how to change graphic properties, please refer to *CATIA Infrastructure* User's Guide Version 5.

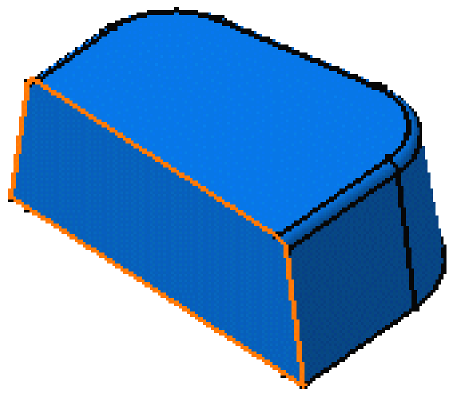
   The part now looks like this:

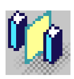


Mirroring the Part

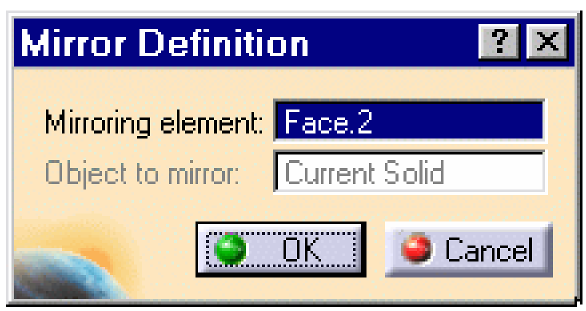
Now, you are going to duplicate the part using the Mirror capability. This task will show you how easy it is.

1. Select the reference face you need to duplicate the part. Select the face as shown:

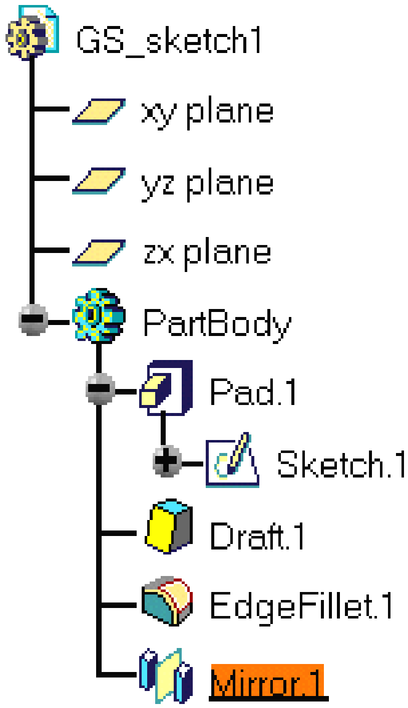


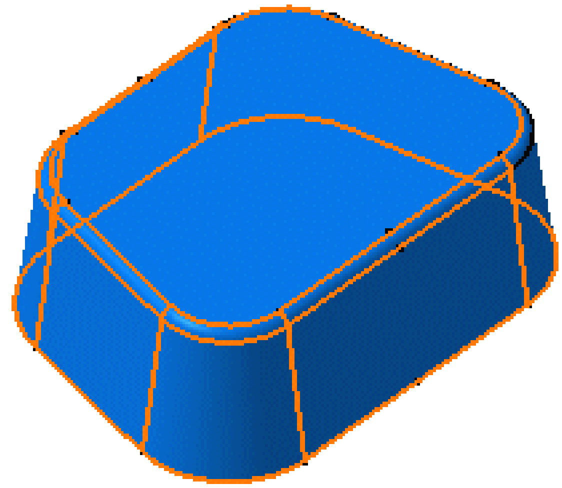
2. Click the `Mirror` icon  .

   The name of this face appears in the Mirroring element field.



3. Click OK.
   The part is mirrored and the specification tree indicates this operation.





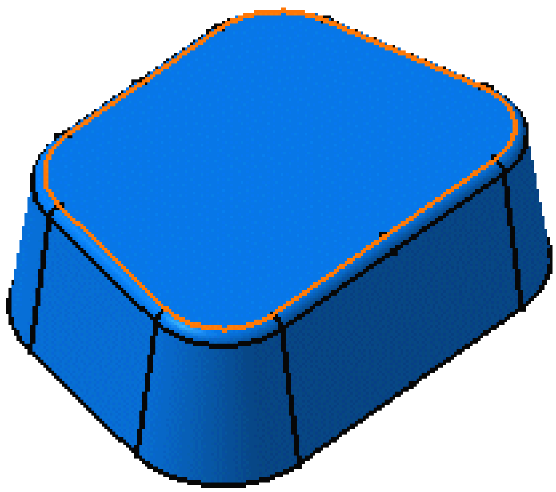
For more about mirror, please refer to Mirror.

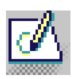
Sketching a Circle from a Face

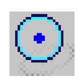
In this task, you will learn how to:

- sketch a circle on an existing face
- use this circle in order to create a pocket

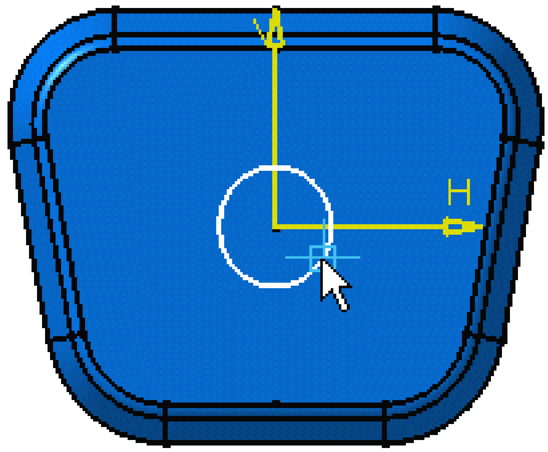
1. Select the upper face to define the working plane.



2. Click the Sketcher icon  to enter the Sketcher workbench.

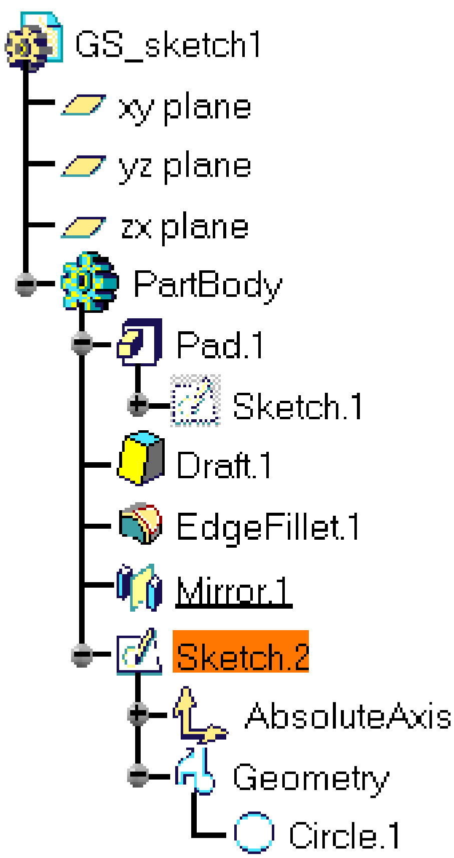
3. Once in the Sketcher workbench, click this Circle icon  to create a basic circle.

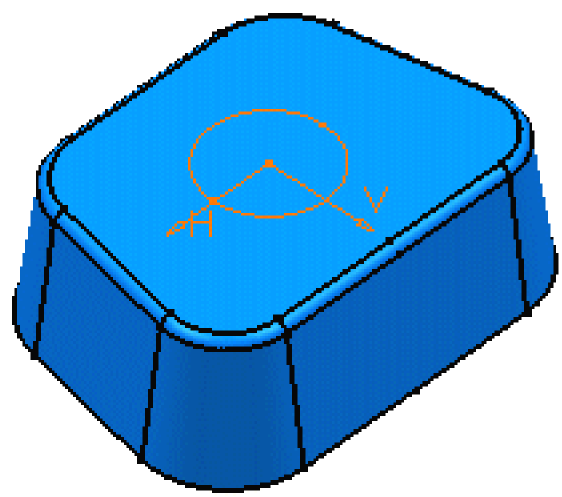
4. Click the circle center in the middle of the face and drag the cursor to sketch the circle.



5. Click once you are satisfied with the size of the circle.

6. Click the `Exit Sketcher` icon  to return to the 3D world. This is your part:





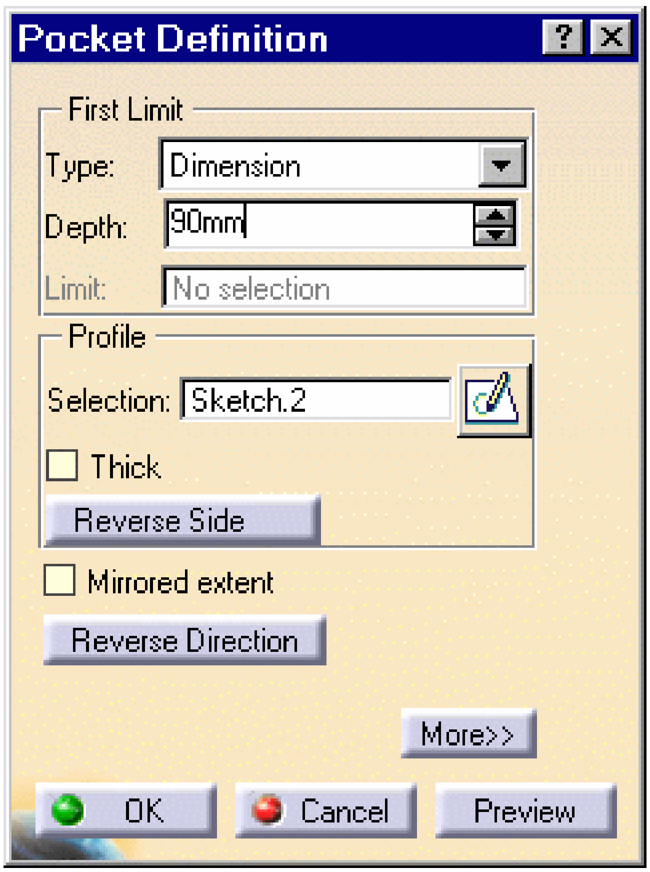
For more about Sketcher elements, please refer to *CATIA-Dynamic Sketcher User's Guide Version 5*.

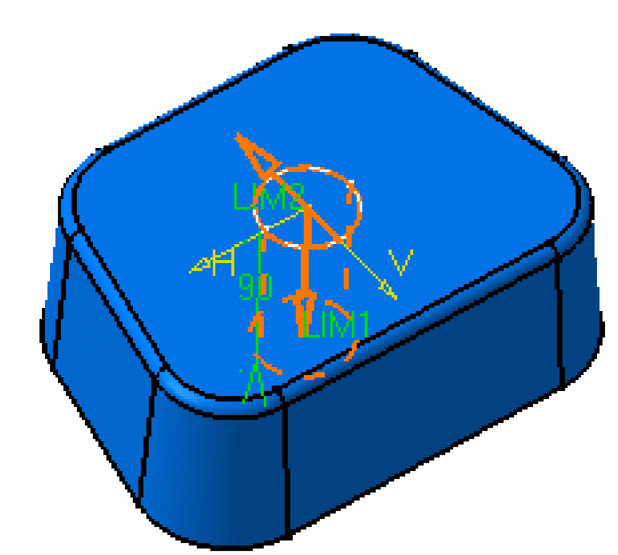
Creating a Pocket

In this task, you will learn a method to create a pocket using the profile you have just created.

1. Select the circle you have just sketched, if it is not already selected.
2. Click the `Pocket` icon .

   The Pocket Definition dialog box is displayed and the application previews a pocket with default parameters.



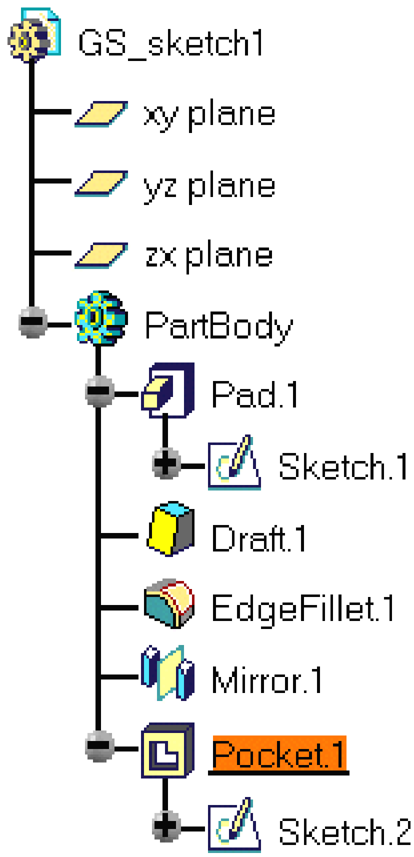


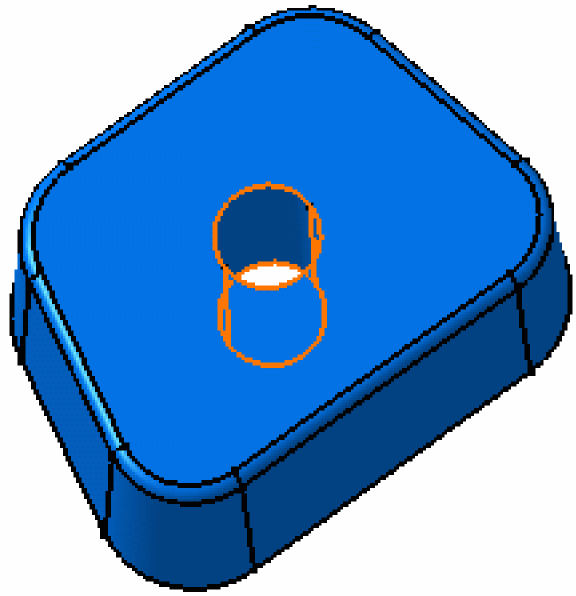
3. Set the Up to last option to define the limit of your pocket.

   The application will limit the pocket onto the last possible face, that is the pad bottom.

4. Click OK.

   This is your pocket:



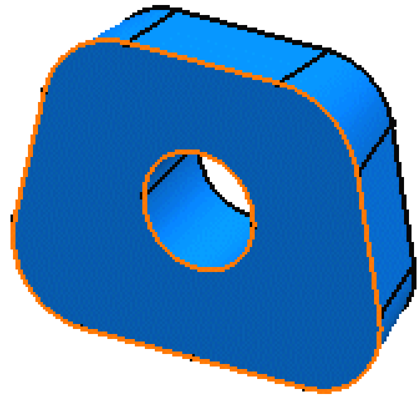


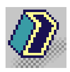
For more about pockets, please refer to Pocket.

Shelling the Part

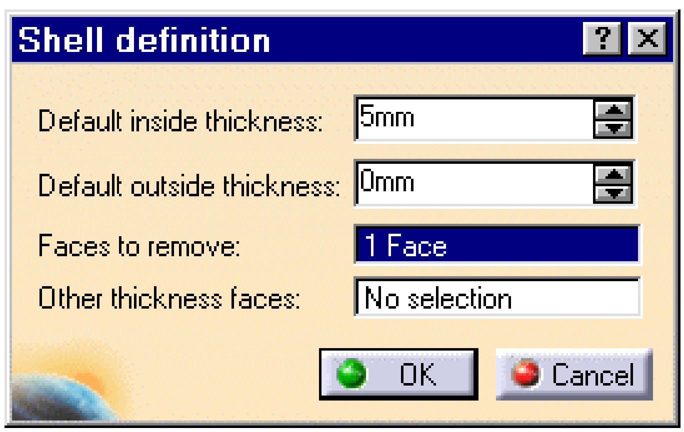
To end the scenario, you will learn how to shell the part.

1. Select the bottom face of the part.



2. Click the `Shell` icon  .

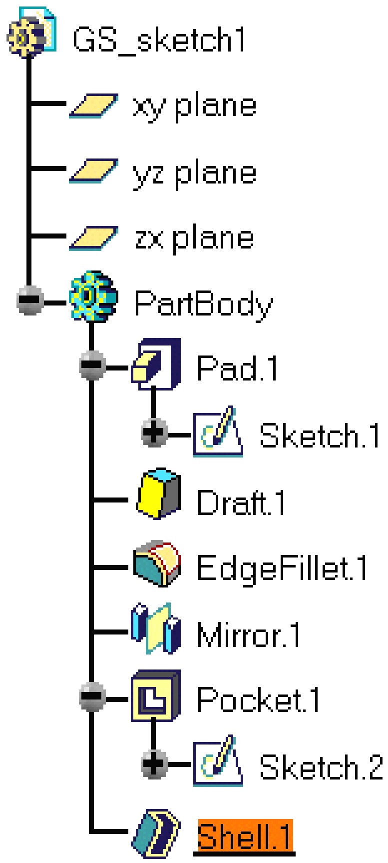
The selected face turns purple and the Shell Definition dialog box appears.

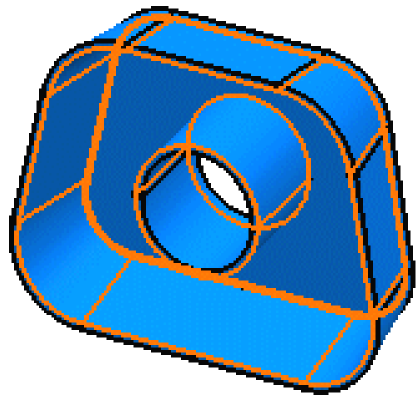


3. Enter 5mm as the inner thickness value.

4. Click OK to shell the part.

You have defined a positive value, which means that the application is going to enter a thin part thickness.





For more about shells, please refer to Shell.

You have finished the scenario. Now, let's take a closer look at the application.