

# Tutorial 2

## Part Creation – Basic Modeling

### Purpose:

To give the student more knowledge about modeling a simple solid part.

### Reference:

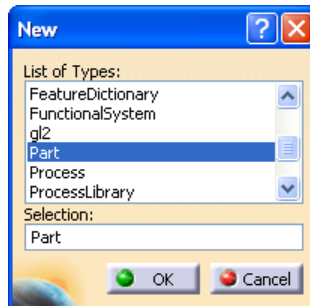
Use the *02-Toy\_Cube.pdf* file.

### 1 – Launch CATIA®

- If a product file is automatically created, close it.

### 2 – Create a new part

- Use *File>New* to launch the *New* dialog box.
- Using the scroll bar, select *Part* in the list.

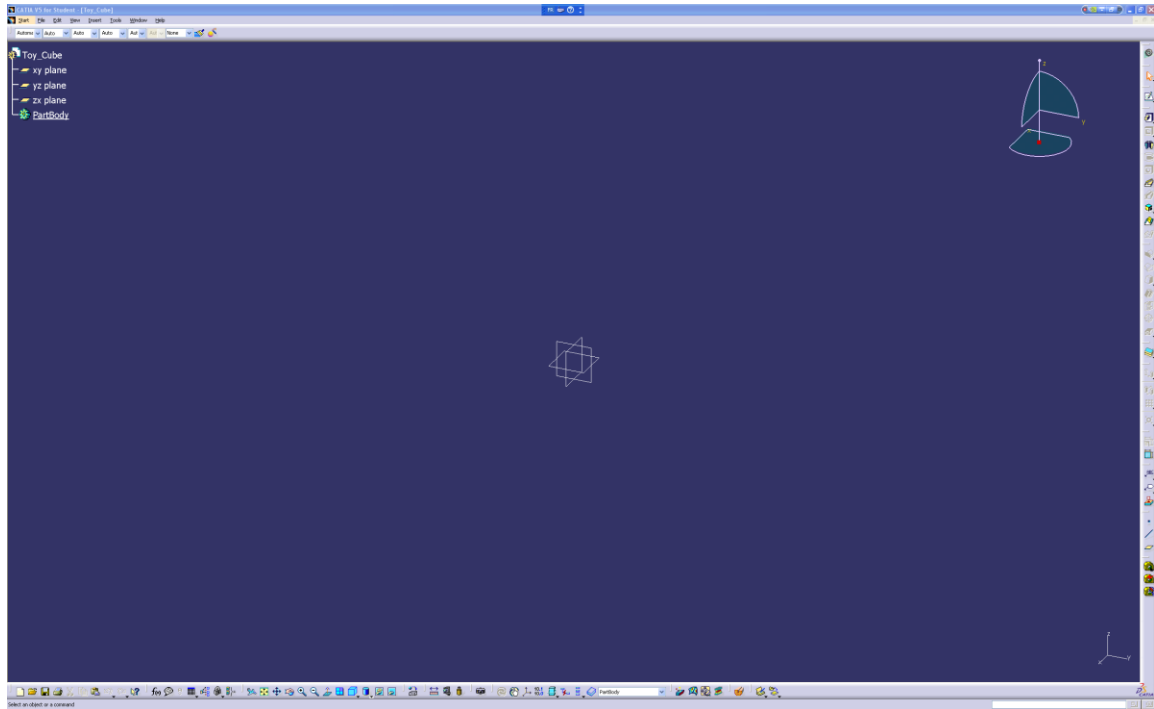


- In the *New Part* dialog box, replace *Part1* by *Toy\_Cube*.



- Make sure the *Enable hybrid design* check box is not selected.


- Click the **OK** button to close the dialog box.
- A new part file is open in CATIA®.

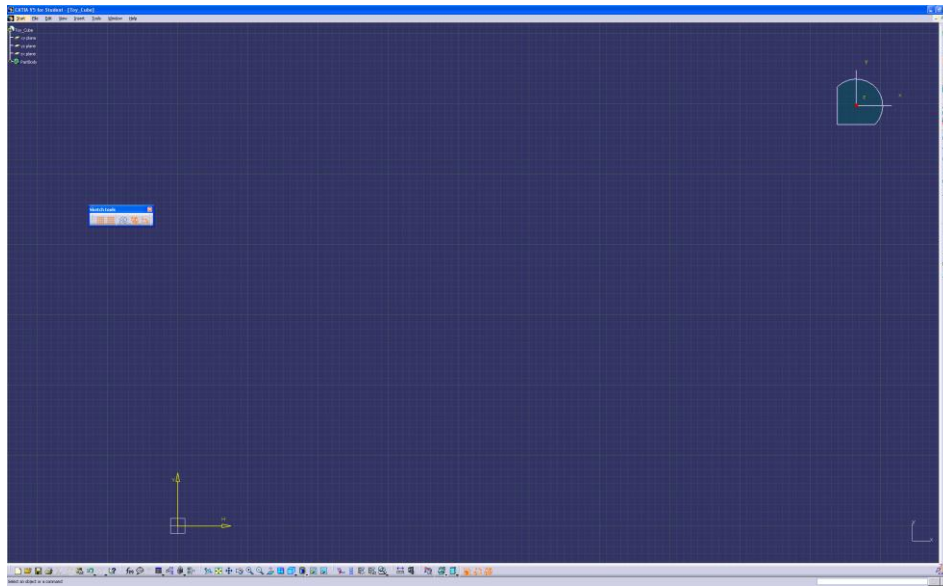


### 3 – If necessary, organize the environment

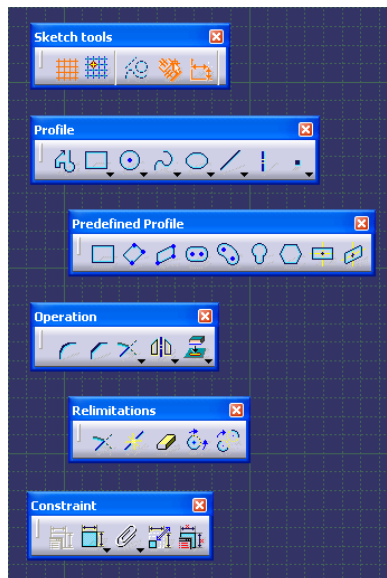
- If more information is necessary about this, review Tutorial 1.


### 4 – Create a sketch

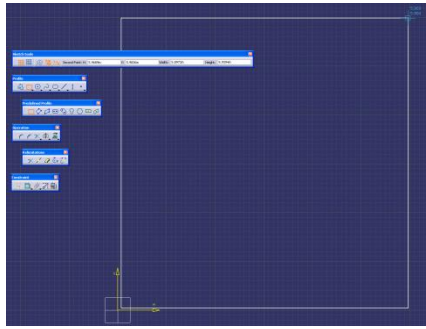
- Click the **Sketcher** tool icon  to create a new sketch on the **XY** plane.
- The **Sketcher's** environment becomes available.
- Use the mouse (or a similar tool) to pan and zoom in order to locate the origin in the bottom left corner and to have a grid about 7 inches high displayed on the screen.




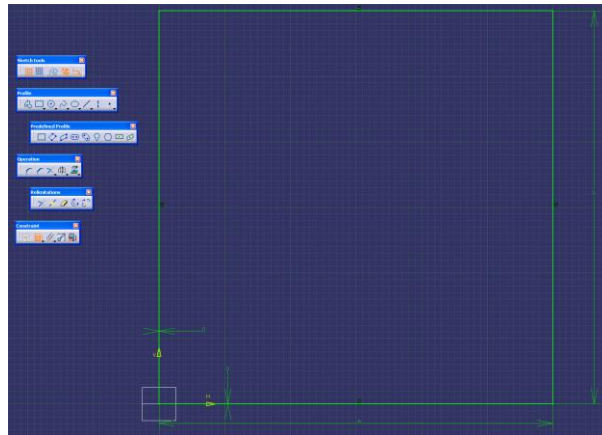
- Bring the **Profile**, **Operation** and **Constraint** toolbars in the working environment. Access the **Predefined Profile** and **Relimitations** toolbars and make them readily available.



- Click the **Rectangle** tool icon  to create the part base.
- Start by clicking the first point close, but not on the **V** and **H** axis origin. Create a square shape having about 6 inches sides.




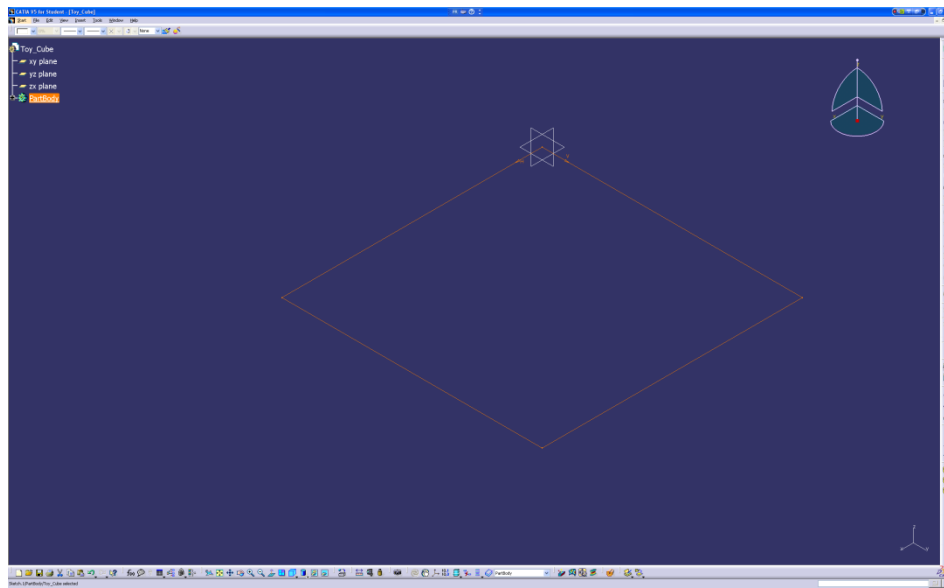
- Click the **Constraint** tool icon  to create dimensional constraints attaching the contour to the appropriate plane elements and setting the size of the square sides to 6.0 inches.




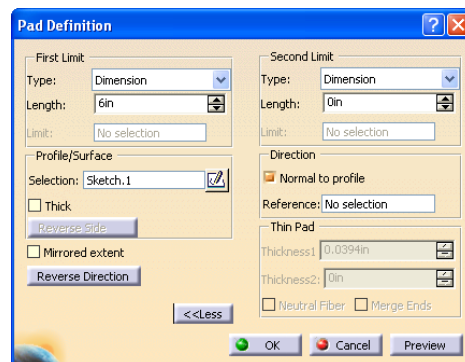
- Click the **Exit Workbench** tool icon  to get back to the 3-D environment.

#### 4 – Create a solid and add a material

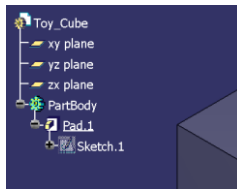
- Click the **Isometric View** tool icon  to better visualize the geometry. Pan and zoom if necessary.




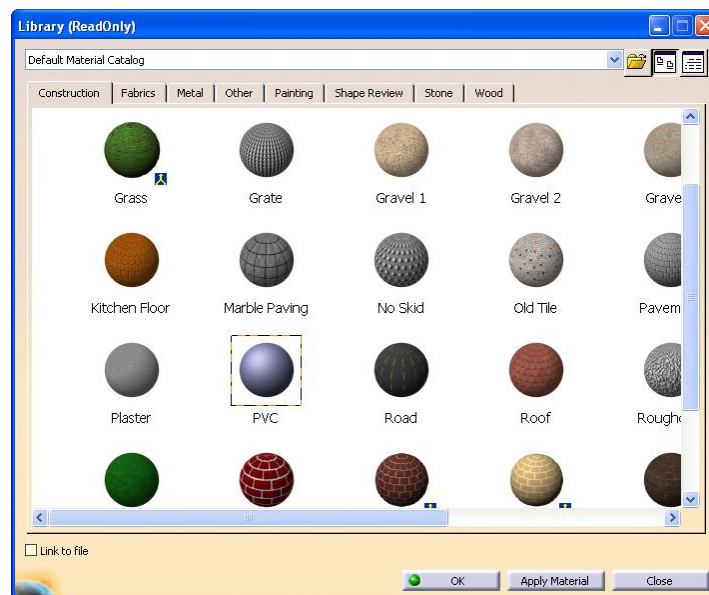
- Click the **Pad** tool icon  to start to create the solid model.
- In the **Pad Definition** dialog box, click on the **More** button to see all settings related to the command.
- Change the value of the **Length** parameter in order to make it 6.0 inches.



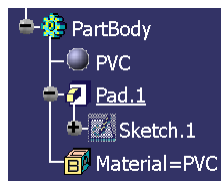
- The solid model is now created.
- Click on the **XYZ** tripod in the screen's lower right corner. The solid object darkens.
- Zoom in in order to increase the size of the **Part Specification Tree** and then click on the **Plus** sign located near the **PartBody** item to expand the tree. The **Plus** sign is now replaced by a **Minus** sign.




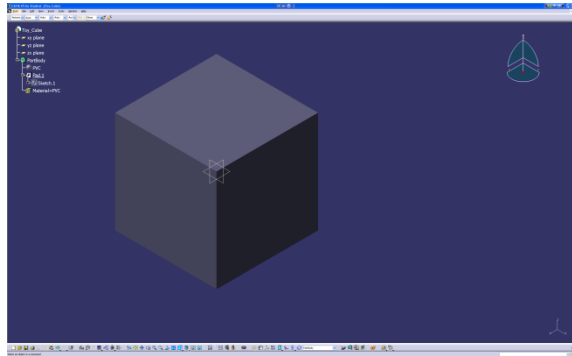
- Click on the **XYZ** tripod again to reactivate the part.
- Click the **Apply Material** tool icon  to assign the part a material.
- Select the **Construction** tab, select **PVC** and select the **PartBody** item in the **Part Specification Tree** prior to clicking the **Apply Material** button.








- The material is now present in the **Part Specification Tree**.

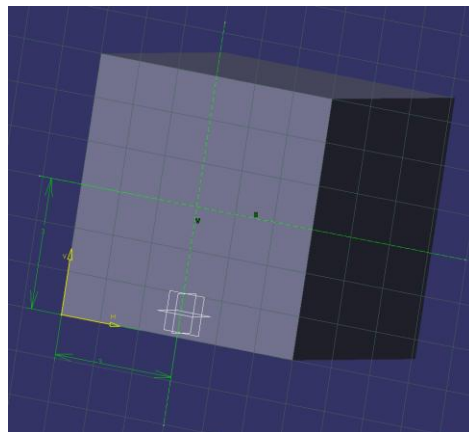



- By clicking the **Shading with Material** tool icon , the part will be displayed with a different texture.




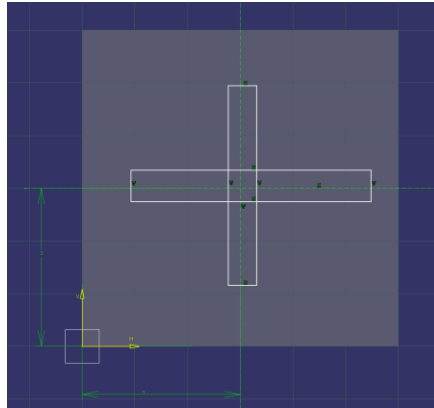
## 5 – Create the top extruded shape



- Click the **Sketcher** tool icon  to create a new sketch. Select the solid cube's top face.
- Click the **Construction/Standard Element** tool icon  prior to clicking the **Line** tool icon . Create two orthogonal lines, one horizontal and one vertical, passing close to the center of the face. Click the **Construction/Standard Element** tool icon  once again to deactivate the mode.
- Click the **Constraint** tool icon  to locate the two lines 3.0 inches from the sides of the cube. Use the mouse to rotate the cube and select the faces instead of selecting the edges of the cube.

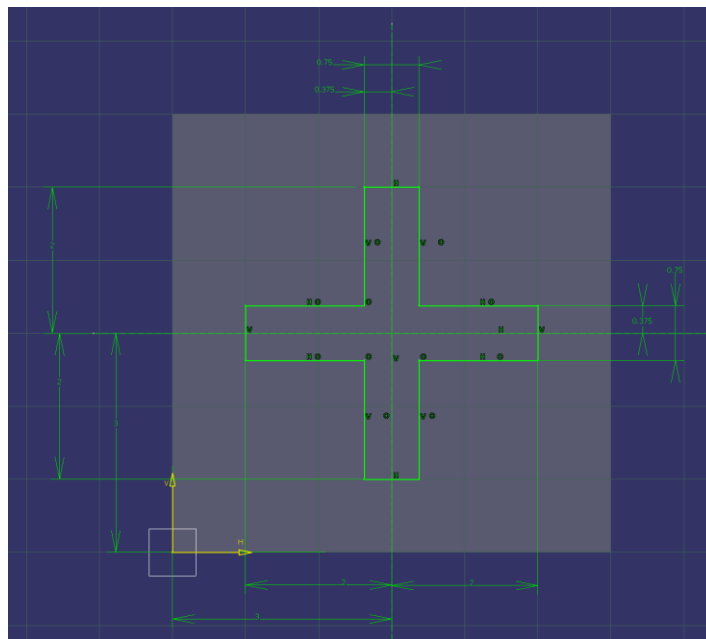


- Click the **Normal View** tool icon  to bring the sketch plane parallel to the screen.

- Double-click the **Rectangle** tool icon  to create two rectangles oriented along the two construction lines. Deactivate the tool when done.





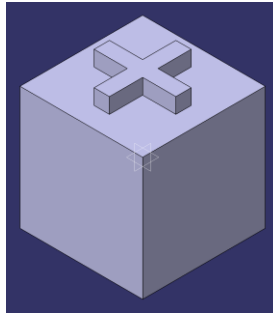
- Double-click the **Quick Trim** tool icon  and clean the geometry to get the desired shape. Deactivate the tool when done.
- Double-click the **Constraint** tool icon  to add all necessary dimensional constraints. Deactivate the tool when done. Note that coincidence constraints make the job much easier to the user.






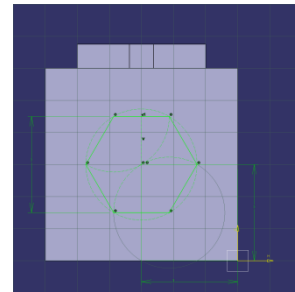
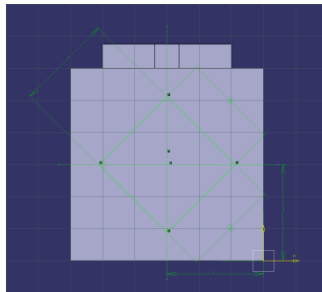
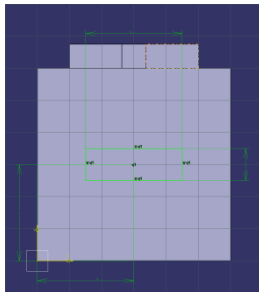
- Click the **Exit Workbench** tool icon  to get back to the 3-D environment.





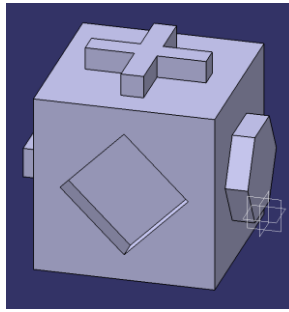
- Click the **Pad** tool icon  to create the 0.75 inch high extrusion.
- Click the **Shading with Edges** tool icon  to get a lighter representation of the solid part.




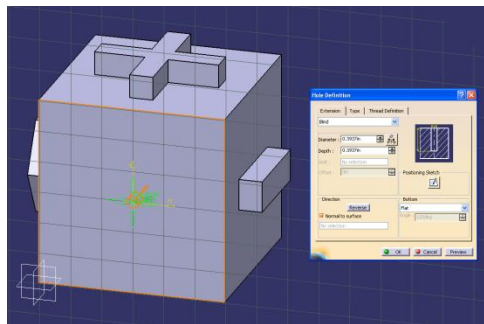
- Repeat the necessary operations in order to create the rectangular shape, the diamond and the hexagon sketches. Note that the **Centered Rectangle** , **Oriented Rectangle**  and **Hexagon**  tools have been used to facilitate geometry creation. Note that completed sketches have been hidden in order to simplify the illustrations.




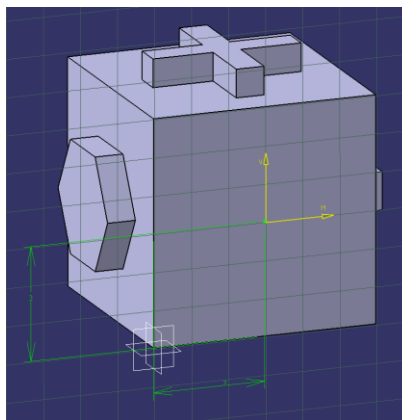
- Click the **Pad** tool icon  to create the 0.75 inch high hexagon and rectangle extrusions. Use the **Pocket** tool  for the 0.75 inch diamond shape.



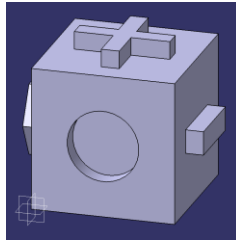
- Click the **Hole** tool icon  to create the cylindrical opening in the last cube's side face. Click anywhere on the face.




- Make sure to have *Flat* as a **Bottom** setting. Set the **Diameter** value to 3.0 inches and the **Blind** hole depth to 0.75 inch.
- Click the **Sketcher** tool icon  present in the dialog box to access the hole center point sketch. The hole center point appears as an asterisk. Create dimensional constraints between this asterisk and the appropriate solid faces.




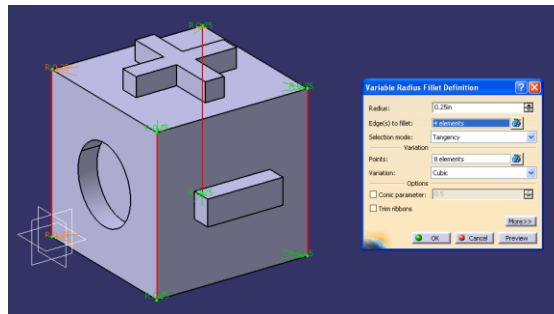
- Exit the sketch and click on the dialog box's **OK** button to complete the hole creation. The rough geometry of the Toy Cube object is now completed.



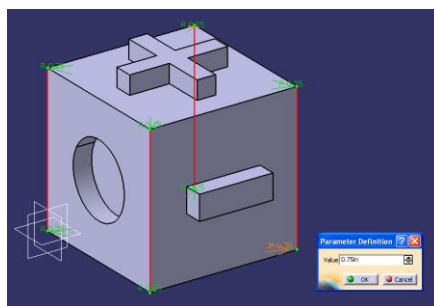
- Click the **black** down pointing arrow present on the **Edge Fillet** tool icon  to access the **Fillets** toolbar. Make it standalone.



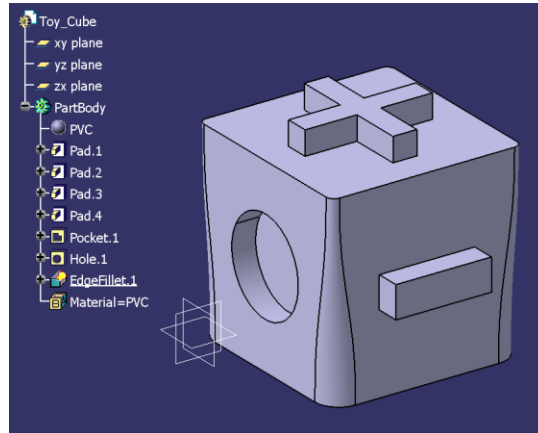
- Click the **Variable Radius fillet** tool icon  to display the command's dialog box. Edit the **Radius** value to make it **0.25** inch. Select all four cube's vertical edges. Note that the **R 0.25** value appears at both ends of the selected edges.




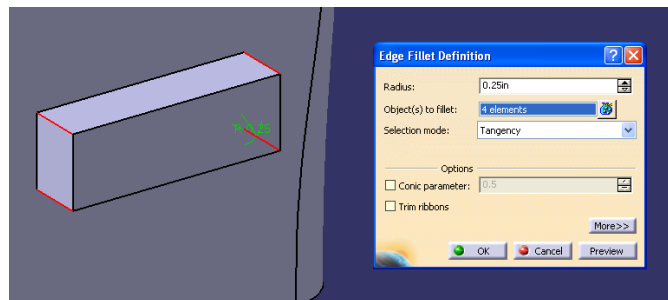
- Double-click on one of the bottom **R 0.25** values. A new dialog box is displayed to the user. Edit the value to make it **0.75** inch.




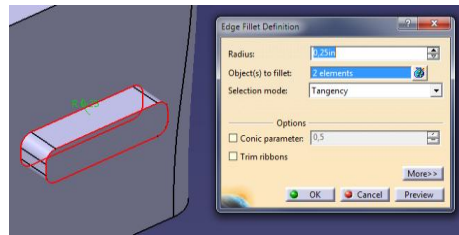
- Click the **OK** button to close the dialog box and repeat the procedure for all three remaining bottom edge values. Close the main dialog box by clicking the **OK** button. All four variable edge fillets are created. Note that a single **Edgefillet** entry is present in the **Part Specification Tree**.




- Click the **Edge fillet** tool icon  to display the command's dialog box. Edit the **Radius** value to make it 0.25 inch. On the rectangular shape, select all four edges normal to the cube face. Note that the radius value appears on the first selected edge only.

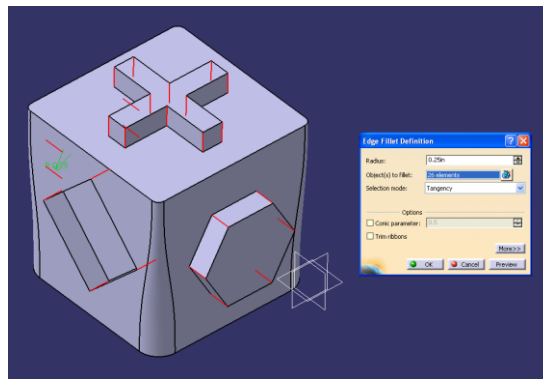


- Click the **OK** button to close the dialog box.
- Click the **Edge fillet** tool icon  to display the command's dialog box once again.
- Select one of the top and one of the bottom edges of the extruded rectangular shape. Note that the complete shape contours are automatically selected. This is due to the **Tangency Selection mode** active in the dialog box.

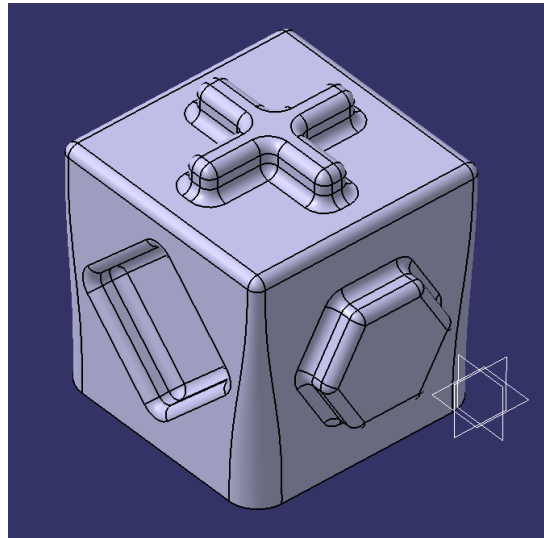



A good filleting strategy will avoid the creation of a great amount of filleting operation. For most simple parts, two or three filleting operations should be enough for a complete part. Especially when fillets are aesthetic, these operations should be located close to the end of the *Part Specification Tree*.

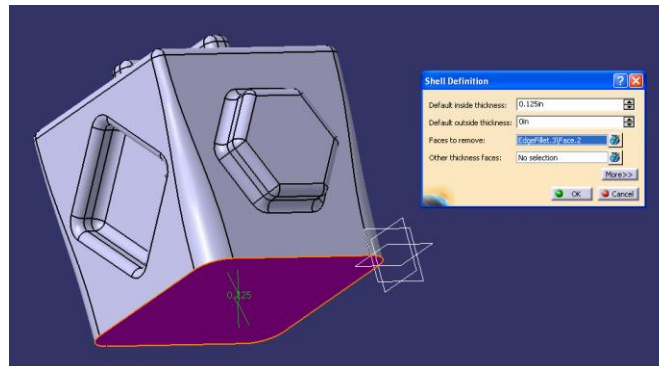
- Double-click the *EdgeFillet* entry  *EdgeFillet.2* in the *Part Specification Tree* to re-access the *Edge Fillet Definition* dialog box.
- Add all shape edges normal to the cube sides to the selection.



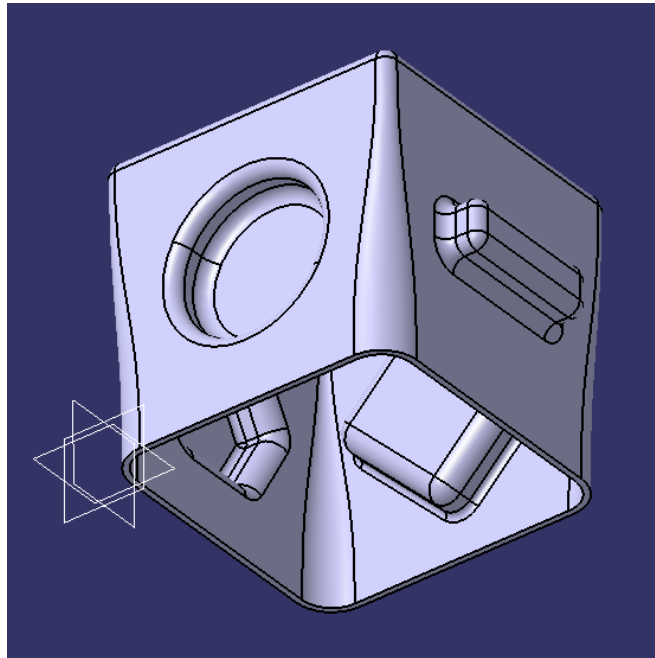
- Click the **OK** button to complete the edition.
- Repeat the procedure with the following *Edgefillet* entry to add missing fillets to all shapes and to the cube top edges. Do not forget to select the hole edges and do not add fillets to the cube bottom edges.



- Click the **Shell** tool icon  to complete the creation of the solid. Set the **Default inside thickness** value to **0.125**. With the dialog box's **Faces to remove** selection box highlighted, select the cube's bottom face. It will turn **purple**.



- Click on the **OK** button to complete the operation. The solid is completed.



- Save the file.

Thanks to Alice Michaud for revising this text.