Tutorial 3

Compound angle part

Purpose:

To give the student the basic knowledge related to the creation of compound angle geometry in a part.

Reference:

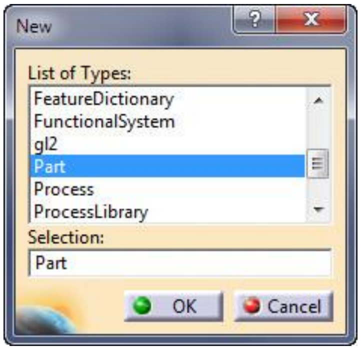
Use the ***03-Compound_angle_part.pdf*** file.

1 – Launch CATIA®

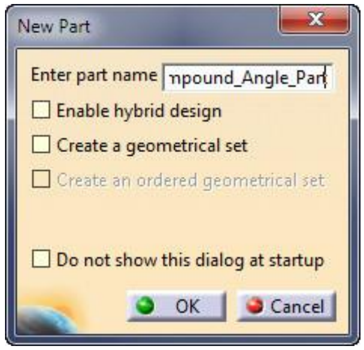
- If a product file is automatically created, close it.

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 *Compound_Angle_Part*



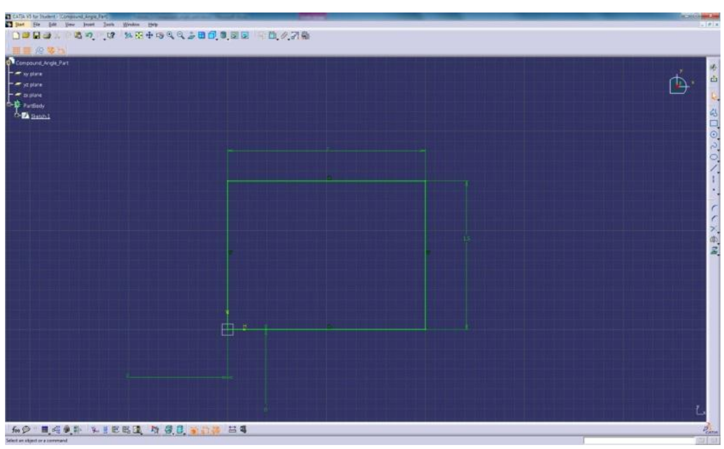
- Make sure the ***Enable hybrid design*** check box is not selected.
- Click ***OK*** to close the dialog box.

3 – If necessary, organize the environment

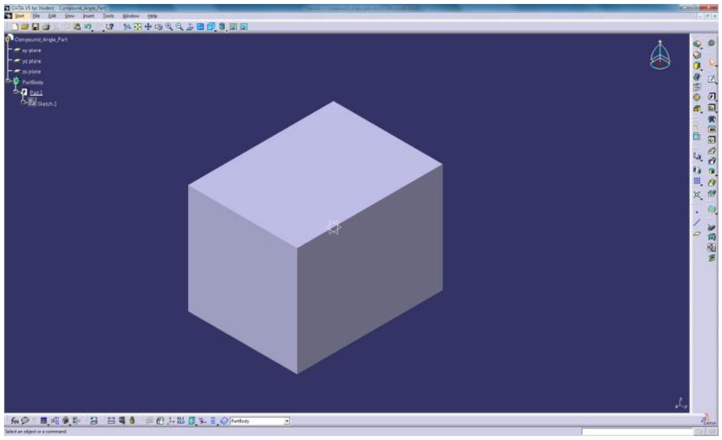
- If more information is necessary about this, review Tutorial 1.

4 – Create the base solid

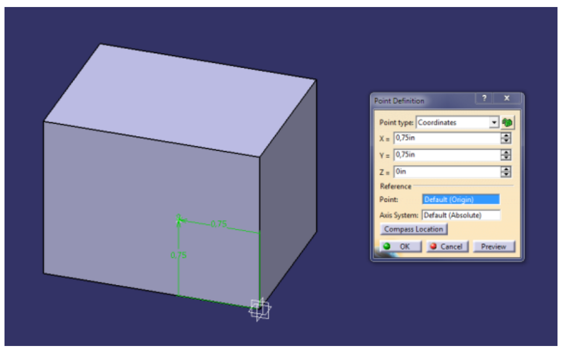
- Click the ***Sketch*** tool icon  to create a sketch using the ***XY*** plane as a reference.
- Click the ***Rectangle*** tool icon  to create a rectangular shape and click the ***Constraint*** tool icon  to make it *2 X 1.5* inches. Start at the sketch's origin and make the longest side oriented along the ***X*** axis (horizontal).



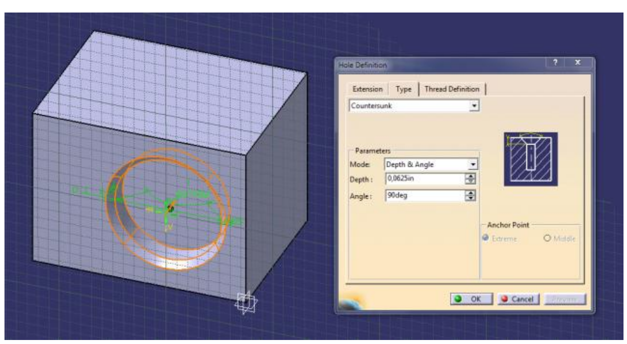
- Click the ***Exit Workbench*** tool icon  to get back to the 3-D environment.
- Click the ***Pad*** tool icon  to create the solid block *1.5* inches thick.



- Rotate the block to see its bottom face and click the ***Point*** tool icon to create a point located at coordinates *x=0.75*, *y=0.75* and *z=0*.

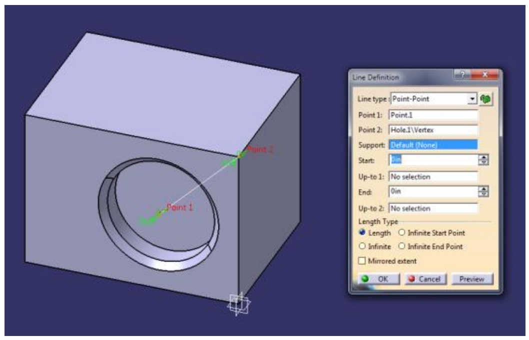


- Click the ***Hole*** tool icon to create a flat bottom *1.0* inch diameter hole having a depth of *0.25* inch in the bottom face, centered on the just created point. Use the ***Countersunk*** option in the ***Type*** tab to add a *45°* chamfer on its edge. Note that a *45°* chamfer will ask for a *90°* countersink setting.

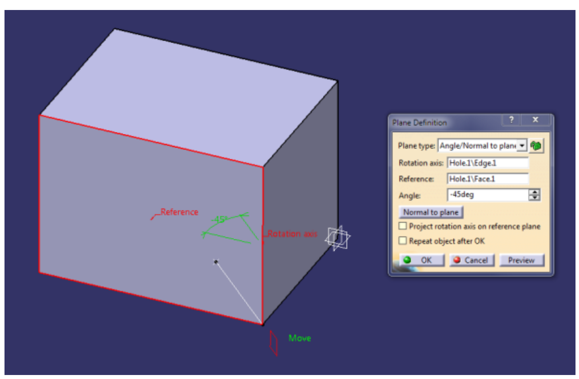


5 – Create the compound angle face

- Click the ***Line*** tool icon  to create a line starting on the point and ending on the solid vertex located at end of the short edge starting at the origin and oriented along the ***Y*** axis.

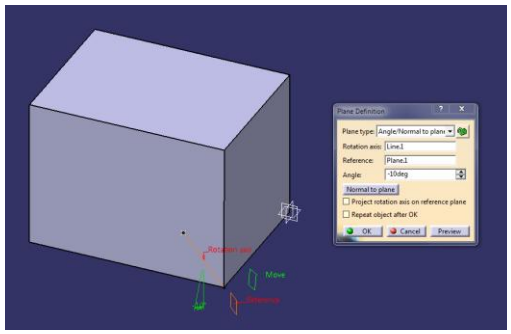


- Rotate the block and click the ***Plane*** tool icon . Select the ***Angle/Normal to plane*** option for ***Plane type***.

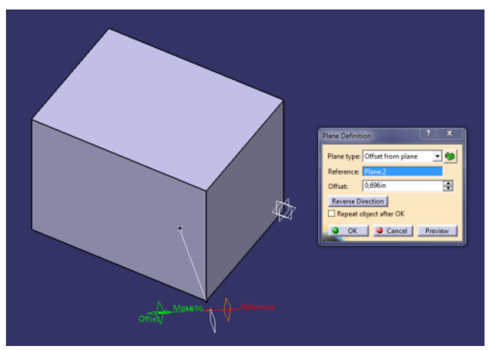


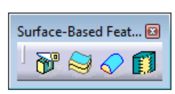
- Select the illustrated vertical edge as ***Rotation Axis*** and any of its adjacent face as ***Reference*** since a *45°* angle will be used. If the newly created plane does not orient in the right direction, use *-45°* as an angle value. While the dialog box is still open, click the ***Move*** grip to relocate the plane out of the solid. To relocate a plane once its dialog box is closed, just double-click the plane.
- Use the line previously created on the bottom face and the plane just created to define a second plane oriented *10°* from the reference one. Again, use the ***Move***

grip to relocate it. The point created at the bottom hole center is coincident to this second plane. This point is also the reference for the dimensions related to the compound angle plane and for the *0.5* inch diameter hole.

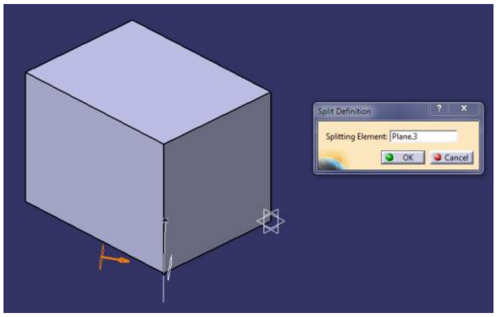


- Click the ***Plane*** tool icon . Select the ***Offset from plane*** option for ***Plane type***. Use the second plane created as ***Reference*** and enter a value of *0.6960* inch as an ***Offset***. Make sure the plane is created on the right side of the block.



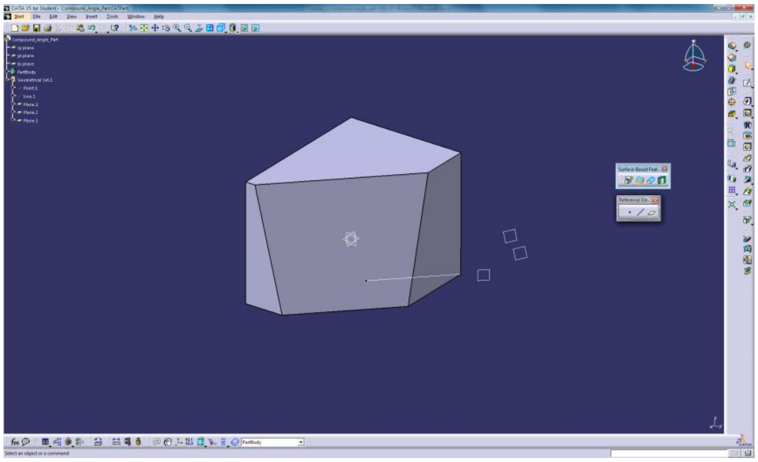
- The ***Surface-Based Features*** toolbar 

is hidden under the ***Thick Surface*** icon . Expand the toolbar and click the ***Split*** tool icon . Select the third plane created and make sure the orientation vector is pointing toward the origin of the part since this vector indicates the portion of the solid that will be kept.

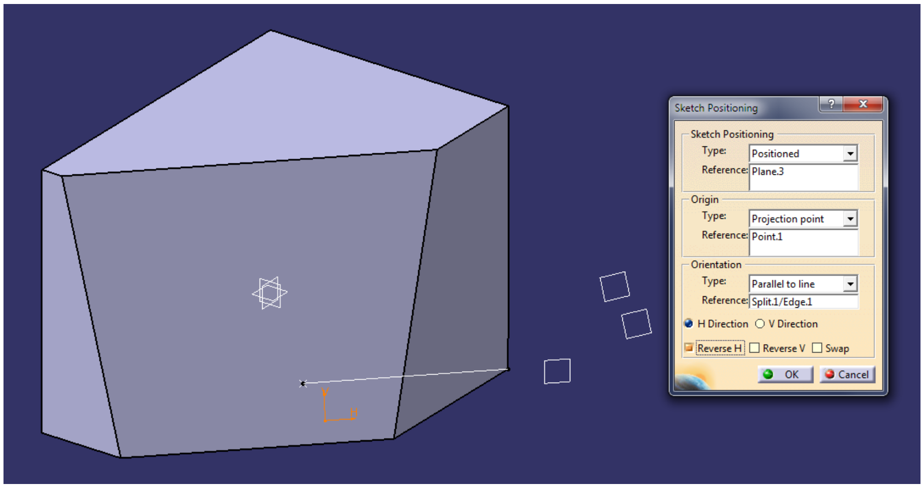


6 – Create the 0.5 inch diameter hole

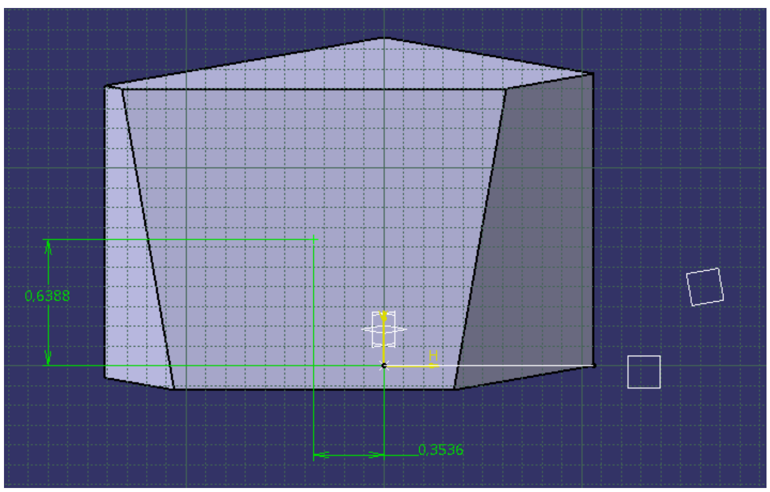
- Rotate about the solid to properly see the compound angle face.



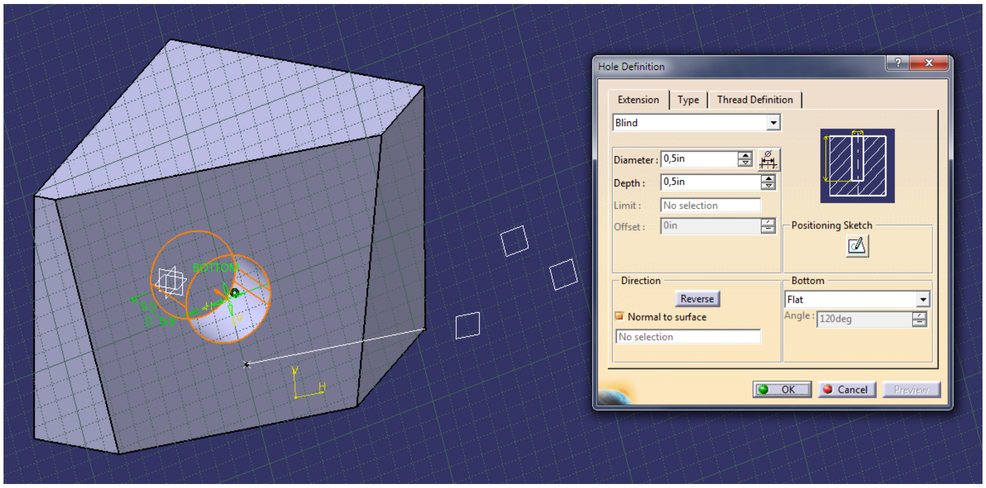
- Click the ***Positioned Sketch*** tool icon , found under the usual ***Sketch*** tool icon .
- Take the third plane created as the sketch ***Reference***, change the ***Origin Type*** as ***Projection point*** and select the point at the center of the hole, make the ***Orientation Type*** as ***Parallel to line*** and select the bottom compound angle face edge. If necessary, swap or reverse the arrow orientations to have the positive vertical axis pointing up and the positive horizontal axis pointing to the right.



- Once in the sketch, create a point and constraint it in order to make it *0.6388* inch over the ***H*** axis and *0.3536* inch to the right of the ***V*** axis.

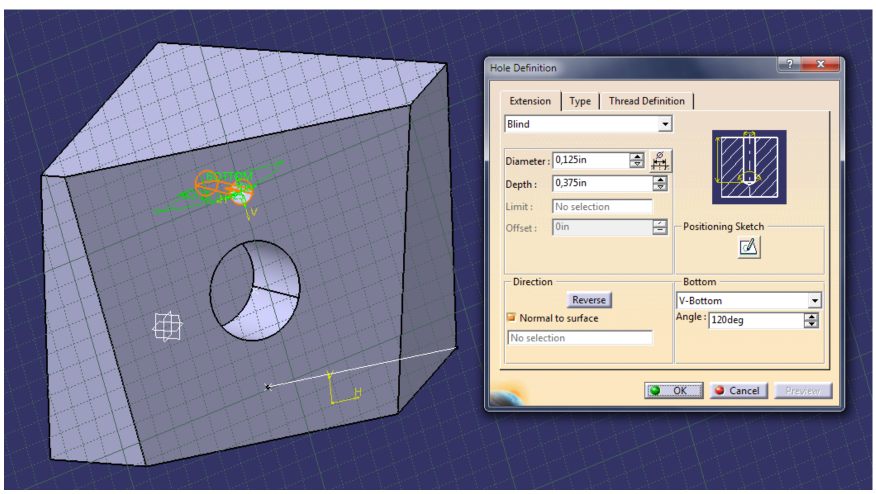


- Click the ***Exit Workbench*** tool icon to exit the ***Sketcher***.
- Click the ***Hole*** tool icon . Create a simple blind flat bottom hole having a diameter of *0.5* inch and a depth of *0.5* inch.

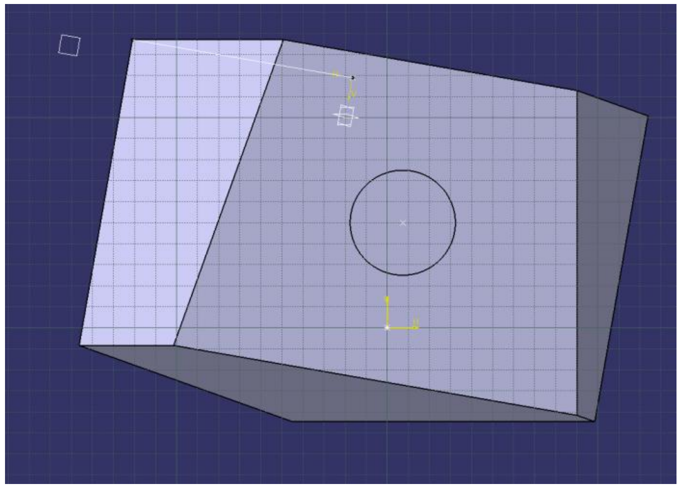


7 – Create the hole pattern

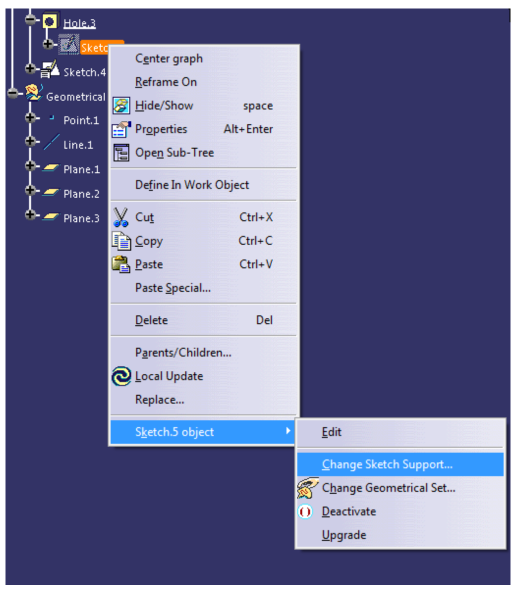
- Click the ***Hole*** tool icon. Create simple blind v-bottom hole having a diameter of *0.125* inch and a depth of *0.375* inch by arbitrarily clicking over the 0.5 inch diameter hole. Complete the hole creation.



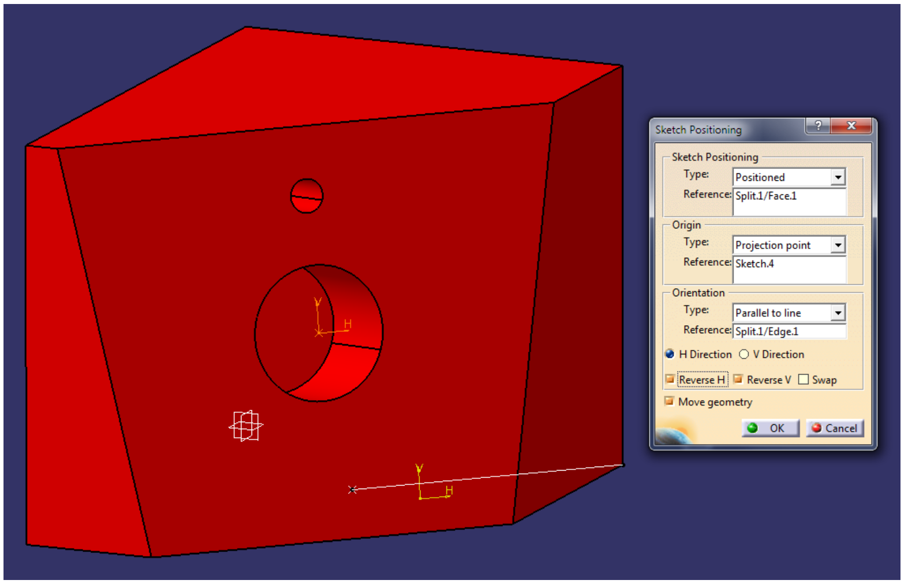
- The new hole is not properly located. Double-click the hole sketch item in the ***Part Specification Tree*** to access the hole reference sketch. The result may be anything…



- Click the ***Exit Workbench*** tool icon  to exit the ***Sketcher***.
- Bring back the mouse cursor over the hole sketch item in the ***Part Specification Tree*** and right-click to access the context menu. Select the ***Change Sketch Support*** option.

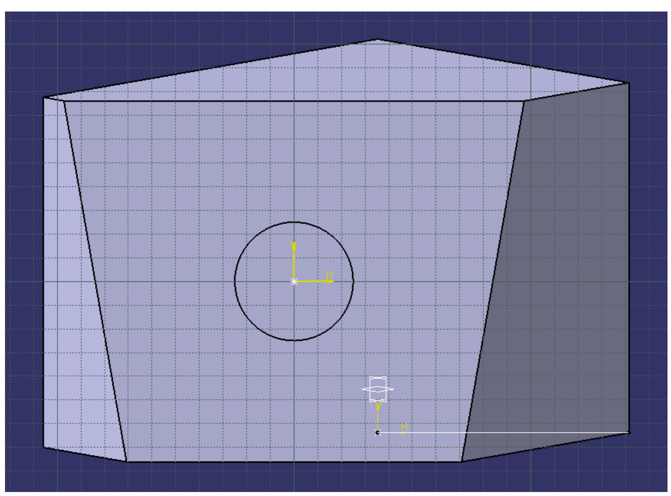


- In the dialog box, first change the ***Sketch Positioning Type*** from ***Sliding*** to ***Positioned***. Then, make the ***Origin Type***: ***Projection Point*** and select the point at the center of the 0.5 inch diameter hole as reference. Finally, make the ***Orientation Type*** : ***Parallel to base*** and select the compound angle face's bottom edge. Click the ***OK*** button to exit the dialog box.

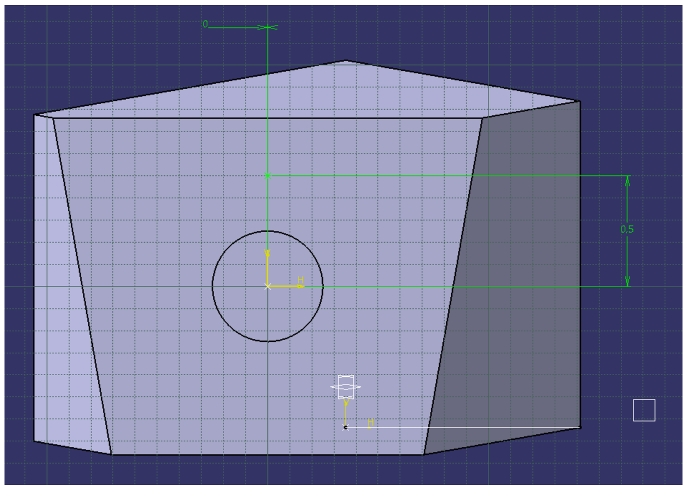


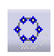
- The sketch will now be redefined in order to be properly oriented according to the base but the hole location may have shifted, making the hole inactive.

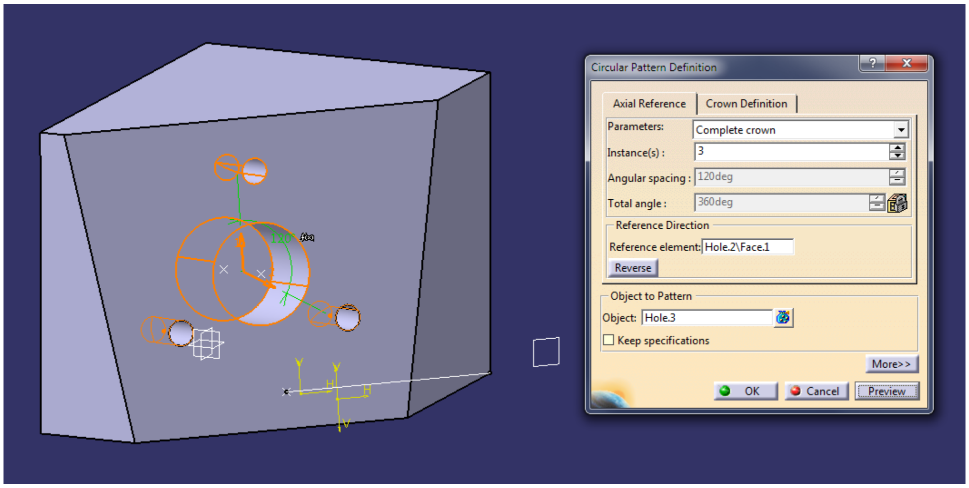
- Double-click the sketch item in the ***Part Specification Tree*** once again to access the hole sketch. The hole center reference asterisk may appear in the middle of the 0.5 inch diameter hole. This means that the small hole has no material to remove due to its size.



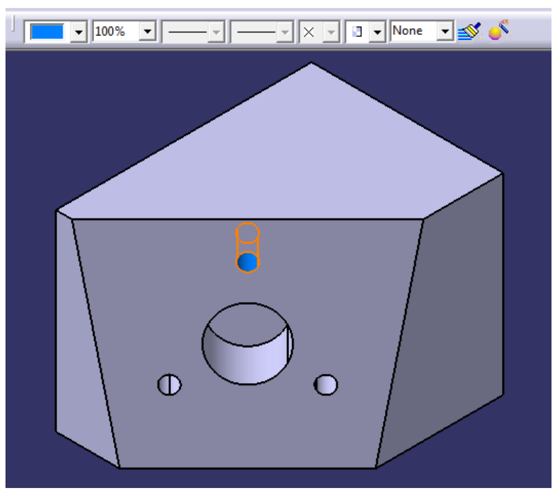
- Relocate the asterisk over the 0.5 inch diameter hole and click the ***Constraint*** tool icon  to locate the asterisk *0.5* inch over the other hole center. Be sure to keep the horizontal alignment. Exit the ***Sketcher***. The first small hole is now properly located.



- Click the ***Circular Pattern*** tool icon  to make the hole repetition. It may be hidden under the ***Rectangular Pattern*** tool icon .
- In the dialog box, click in the ***Object*** edition box and select the small hole in the ***Part Specification Tree***. Then, click in the ***Reference element*** edition box and select the 0.5 inch diameter hole inner cylindrical face as a reference. Finally, make the ***Parameters***: ***Complete crown*** and set the number of ***Instances*** to *3*. Click the ***OK*** button to complete the operation.

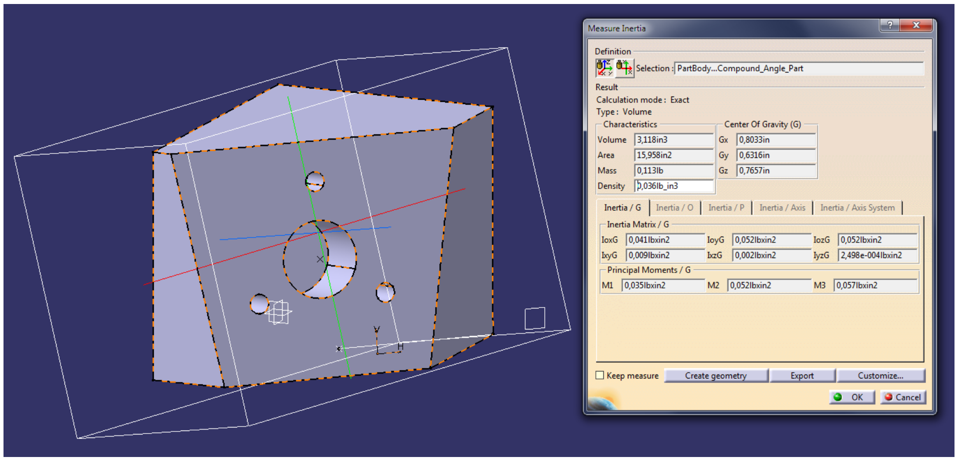


- Change the graphic properties of the first pattern hole inner cylindrical face in order to make it easily recognizable and modifiable when required.



8 – Analyse the part

- Click on the ***Measure Inertia*** tool icon  and select the ***PartBody*** item in the ***Specification Tree*** to check the part. If it is done properly, the coordinates of its center of gravity should be: *x=0.8033*, *y=0.6316* and *z=0.7657*, as noted on the reference drawing.



Thanks to Alice Michaud for revising this text.