PDHonline Course G350 (8 PDH)

CATIA-5 PART-A: 3D CAD, Parts, Assemblies and Drawings

Instructor: John R. Andrew, P.E.

2020

PDH Online | PDH Center

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CATIA-5 PART-A

3D CAD, Parts, Assemblies and Drawings

John Andrew, P.E.

SOME OF THE 20,000+ COMPANIES USING CATIA

Boeing, Air Bus, KelseyHayes, Lear Jet, Northrop Grumman

BMW, Daimler Chrysler, Volvo, Toyota, Ford, Honda Hyundai

Ferrari, Lockheed Martin, Porsche, Fiat Peugeot, Mercedes-Benz

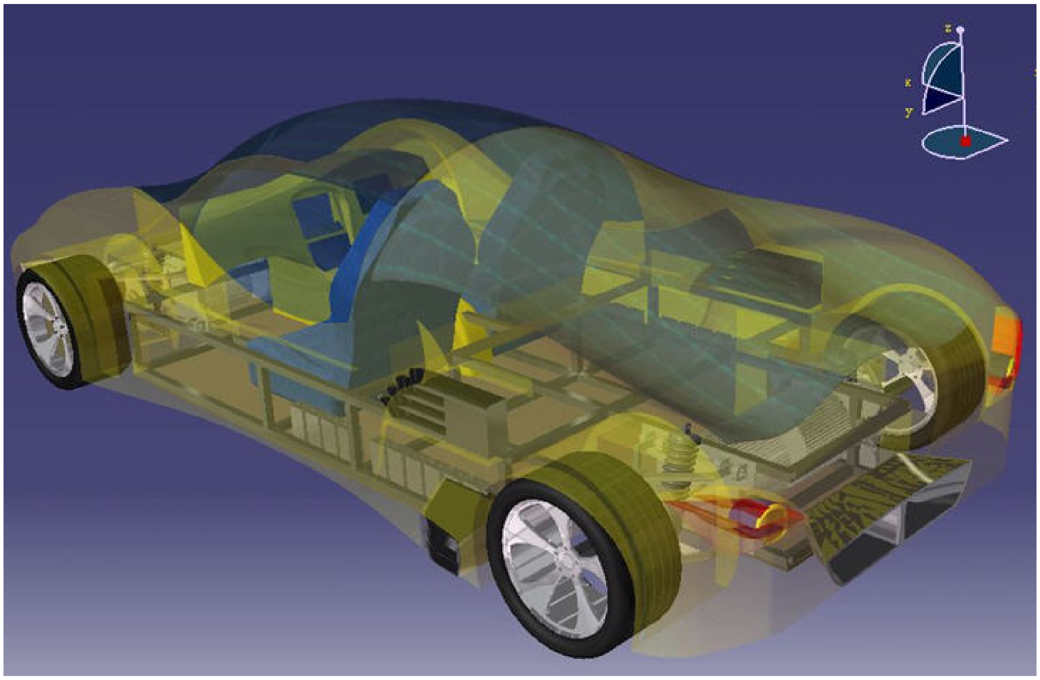
Freightliner, Allied Signal, Volkswagen, Pratt Whitney, United Airlines

Black and Decker, Goodyear

- **52% of all the cars built in 2000 were designed with CATIA. More than all other CAD design engineering products combined together.**
- **14 out of the Top 20 automotive manufacturers use CATIA as their core design system.**
- **CATIA 3D product development is now used by 22 of the top 30 global automotive manufacturers and is the de facto global standard in automotive manufacturing.**
- **87% of civilian/ commercial airplane designers use CATIA.**
- **79% of helicopter designers use CATIA, and 50% of military airplane designers use CATIA.**
- **76% of the world's aircraft designers use CATIA - clearly CATIA is the world standard for aircraft design.**

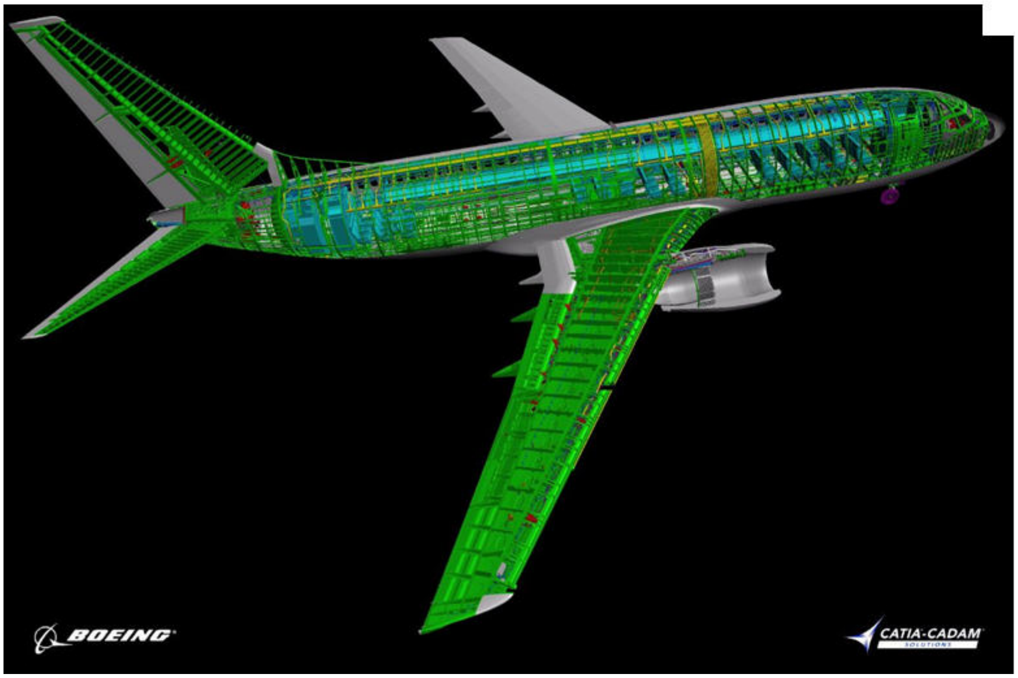
SUMMARY

AUTOMOBILES



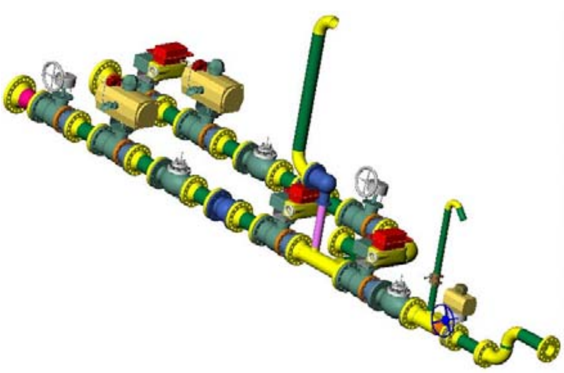
Catia CAD software dominates in automobile and truck design worldwide.

AIRCRAFT



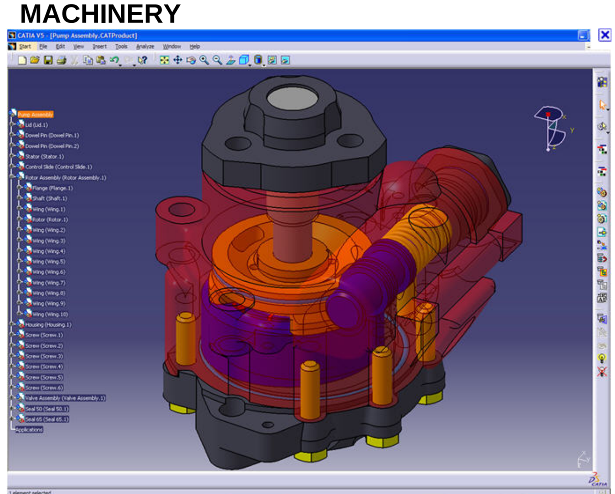
Boeing and most of the world's aircraft are Catia designed. Image at: www.nextcraft.com

PROCESS PIPING



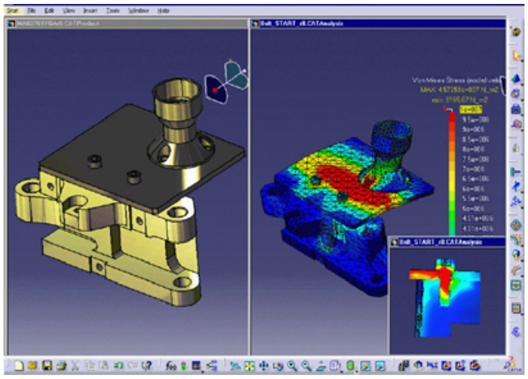
Website for above image: elysiuminc.com

MACHINERY

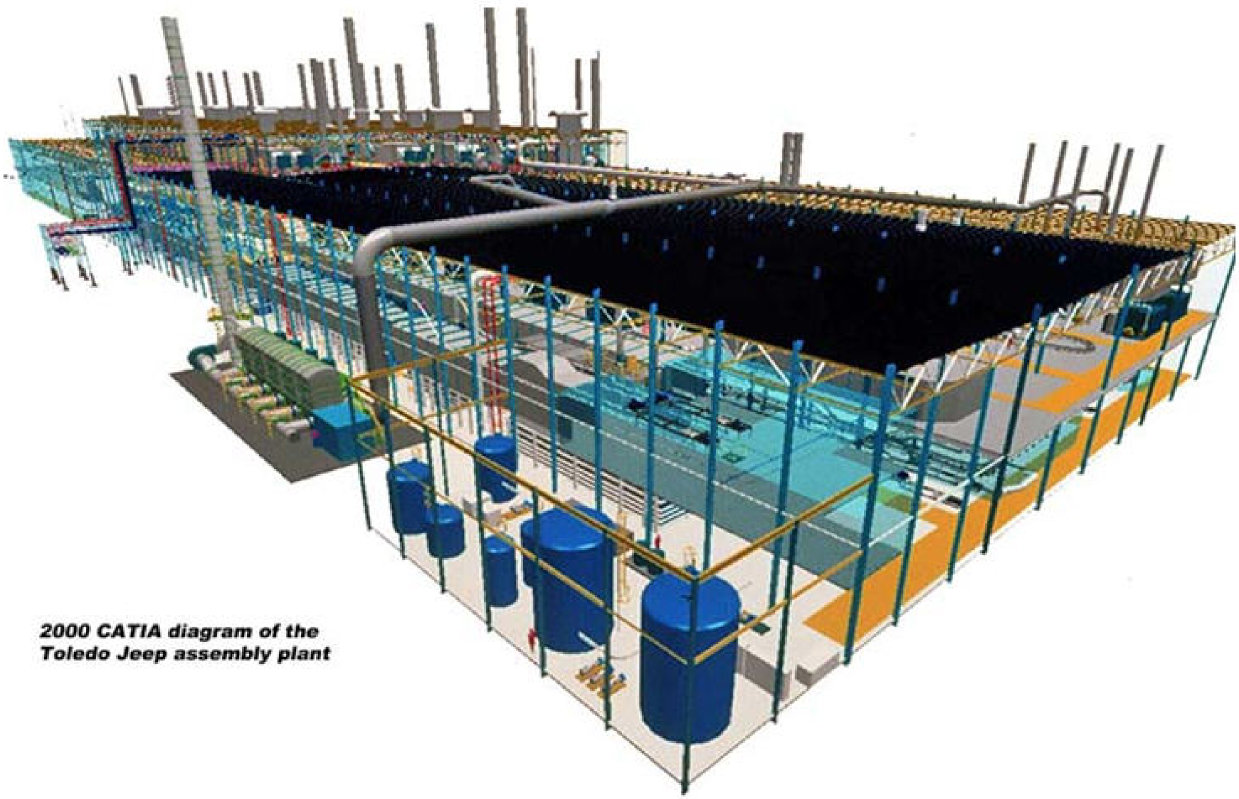


Website for above image: dezignstuff.com.

MOLDS AND FINITE ELEMENT ANALYSIS



FACTORIES

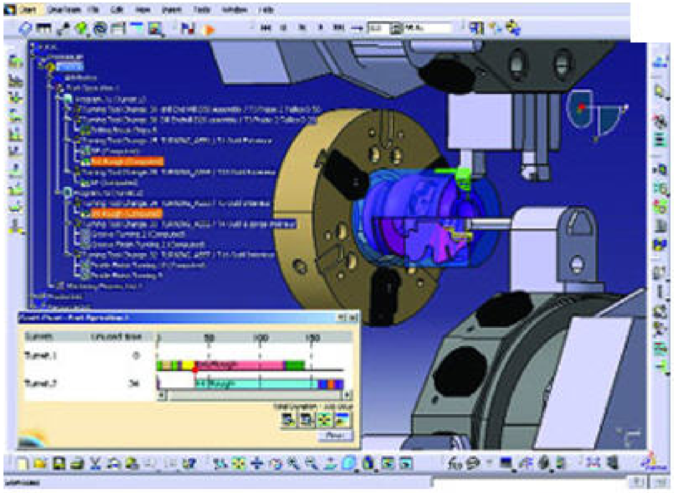


Website for above image: allpar.com

PRODUCT DESIGN MANAGEMENT

- **CATIA is the world's leading solution for product design management excellence.**
- **It addresses all manufacturing organizations, from OEMs through their supply chains, to small independent producers.**
- **Delivering: process, automation techniques and industry-specific Product Lifecycle Management PLM services.**
-

CNC MACHINING



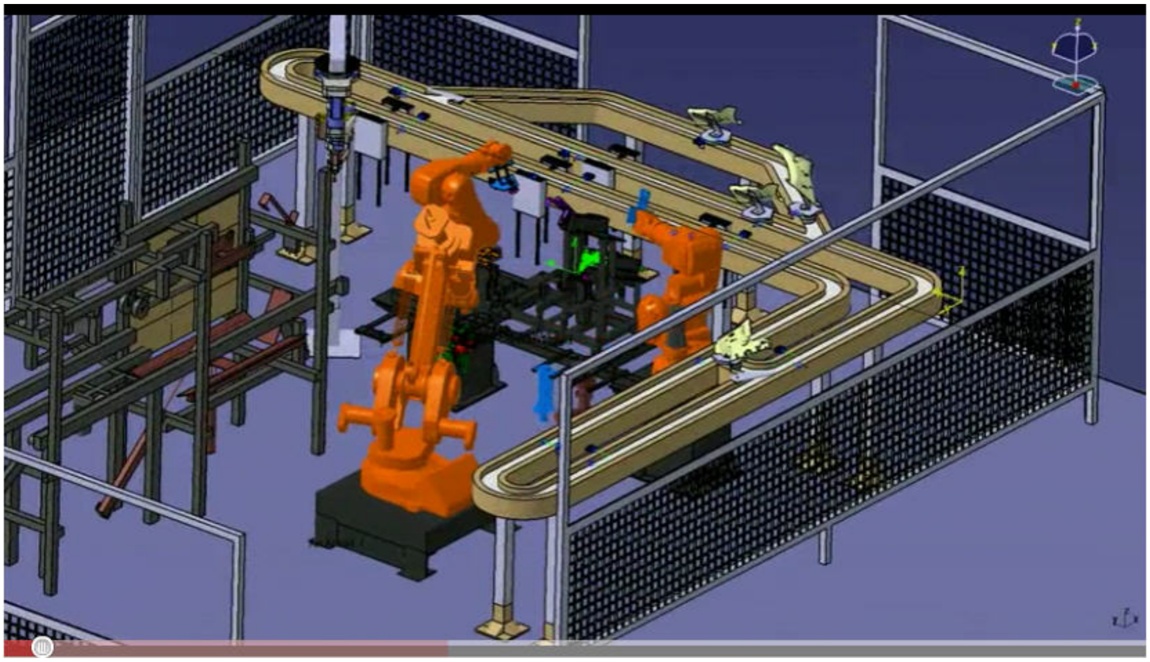
Website for above image: www.deskeng.com

ERGONOMICS

http://www.3ds.com/products/delmia/solutions/human-modeling/#vid3

“As a leading **Digital Human Modeling** solution provider for over 20 years, Dassault Systèmes **Virtual Ergonomic Solutions team** has developed and implemented dedicated solutions that are based on the industries’ best practices and standards. Built on a **lifelike human manikin model**, our solutions empower our customers with the capability of **evaluating Ergonomics and Human Factors** at all levels of Product Lifecycle Management (PLM): Virtual Design, Manufacturing & Maintainability.”

CATIA ROBOT SIMULATION



http://www.youtube.com/watch?v=qdHdZHSsS_0

A GROUP OF CATIA ROBOT VIDEOS

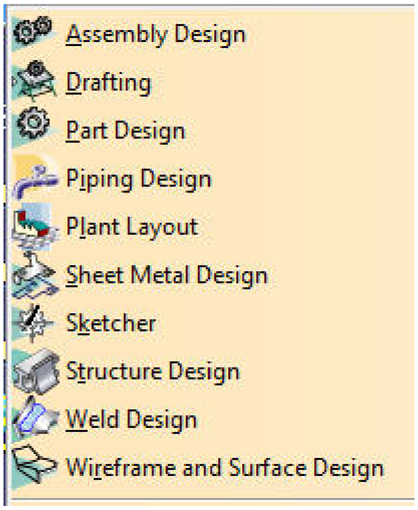
http://www.youtube.com/watch?v=UkaNe6r15q8&NR=1

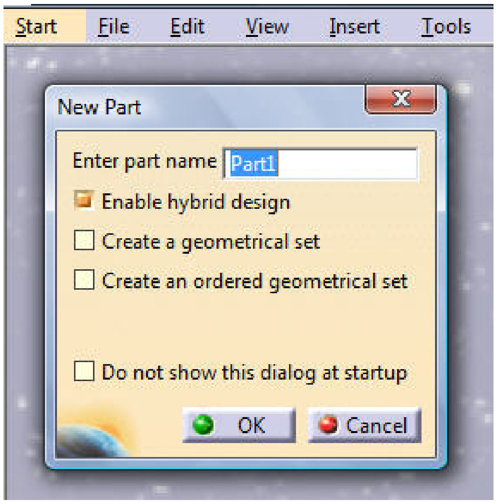
Free Catia Robotics literature: http://www.pdftop.com/ebook/catia+robotic+arm/

1- CATIA SKETCH WORKBENCH

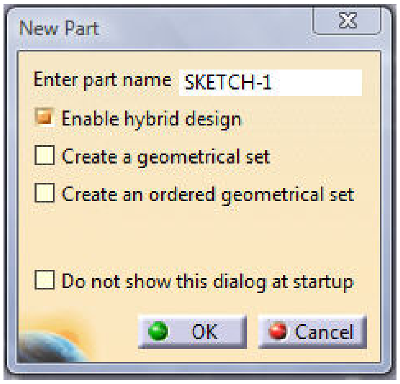
All 3D solid parts begin with a sketch in the "Sketcher Workbench".

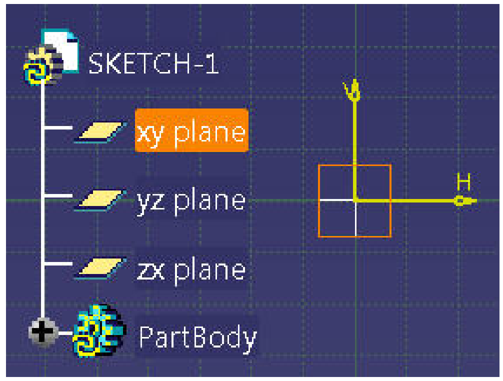
START A CATIA SKETCH





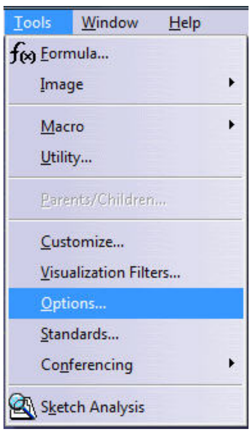
Open Catia and pick: Start >> Part Design >> Edit "Part1" >> SKETCH-1.



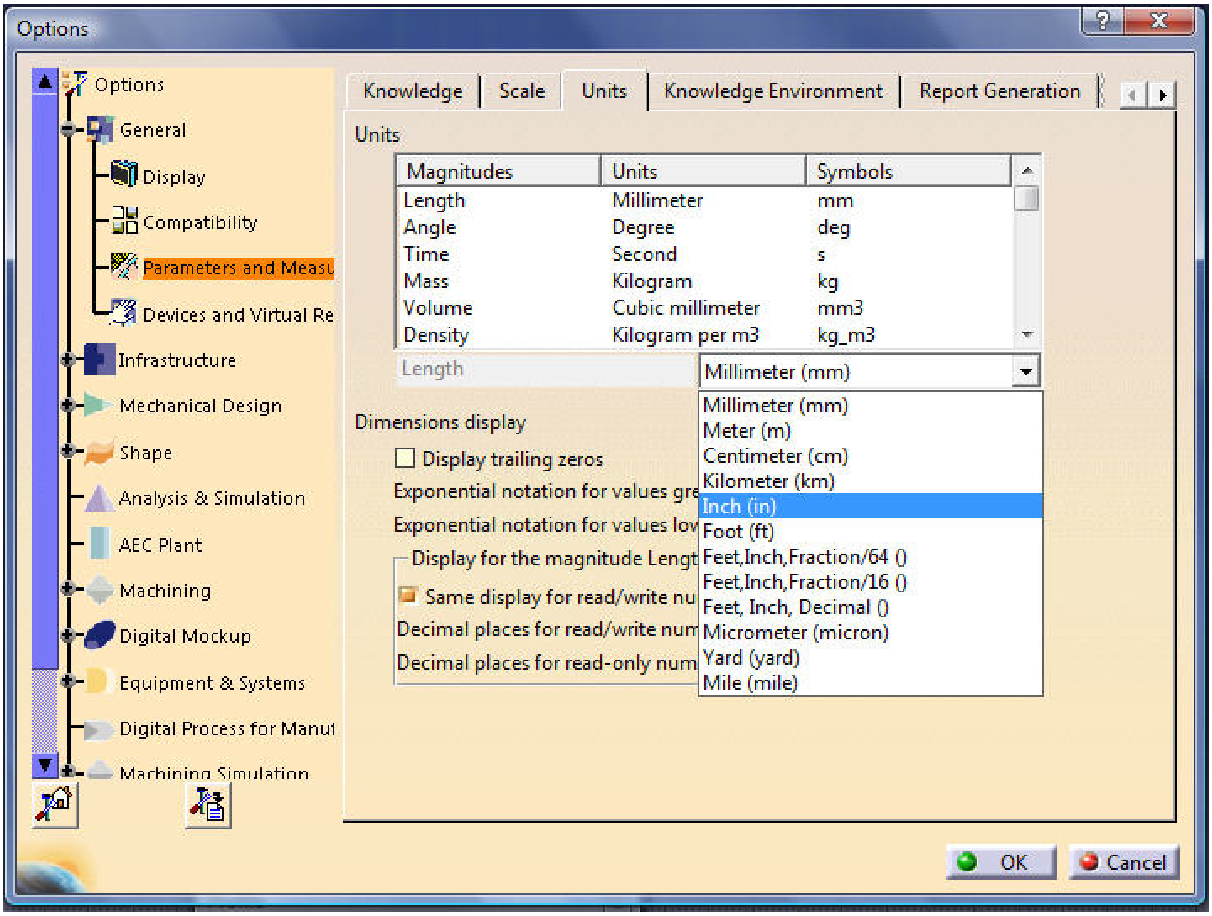


Pick the "xy plane" in the Tree with the left mouse pointer. Pick "Sketch" tool.

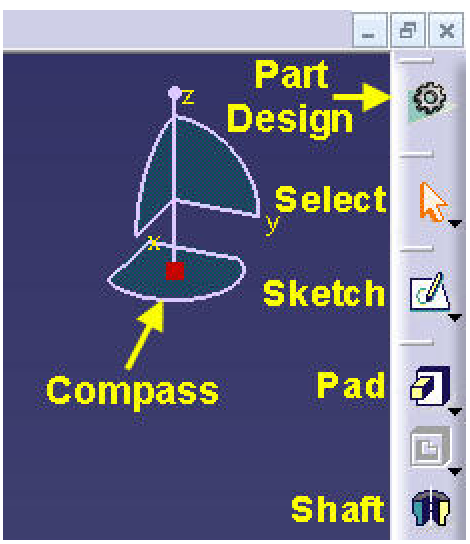
<u>Always pick or drag with the left mouse pointer unless stated otherwise.</u>

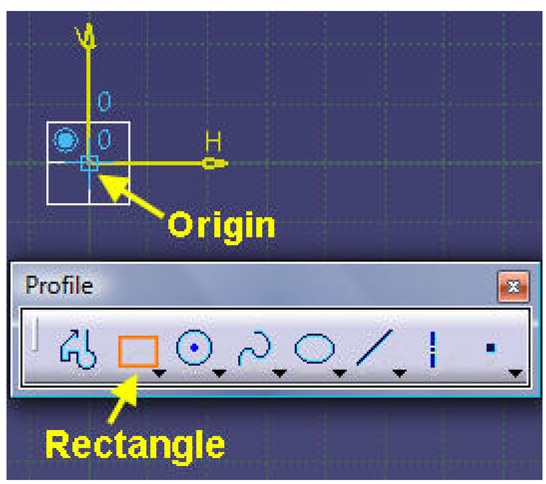


Pick drop down menu: Tools >> Options.



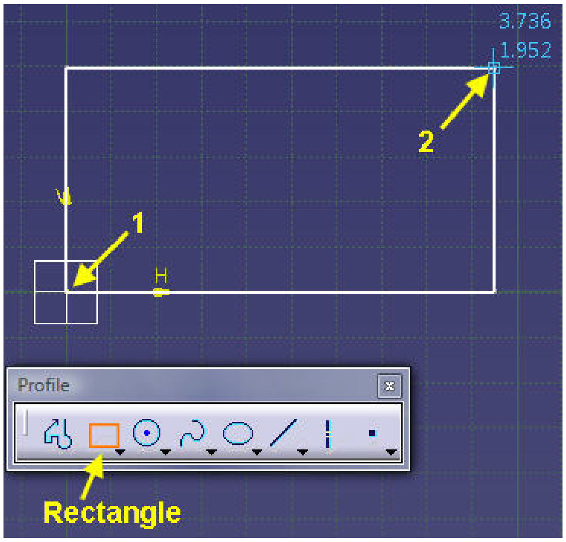
Pick: Parameters and Measure >> Inch (in) >> OK.

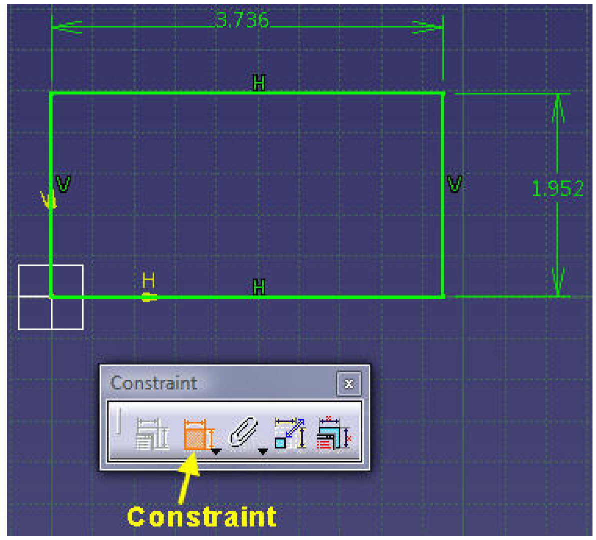




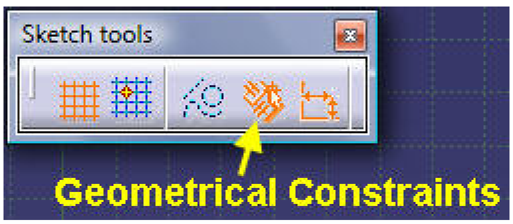
Pick the “Sketch” tool.

Pick the “Rectangle” tool >> Pick the Origin.





Drag the mouse pointer from 1 to 2 >> pick point 2.



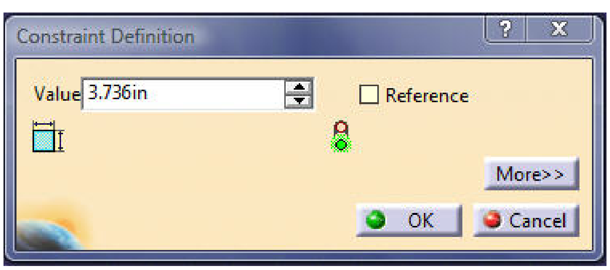
Note that if the "Geometrical Constraints" tool is orange the: Horizontal (H) and Vertical (V) are added by Catia. If this tool is blue it is not active.

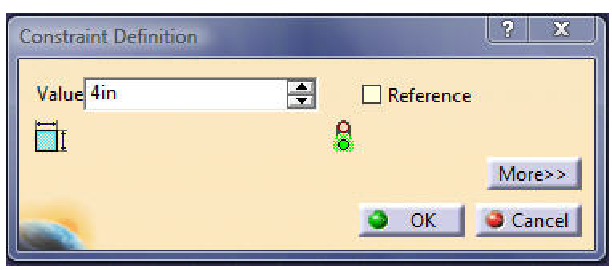
Double-click on the "Constraint" tool >> Pick the top of the rectangle > Place the 3.736 inch dimension.

Pick the right side of the rectangle >> Place the 1.952 inch dimension.

Your dimensions may be different.

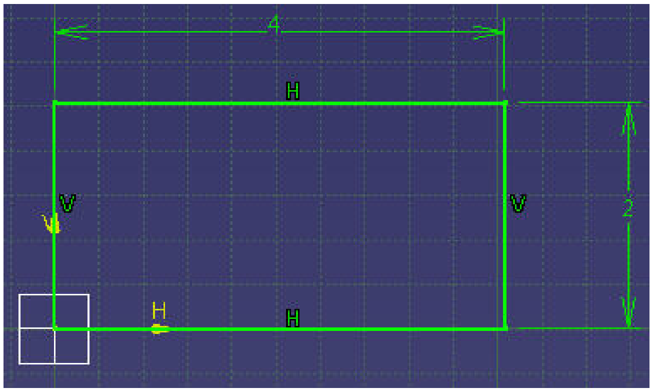
The next step is to edit the two dimensional constraints.

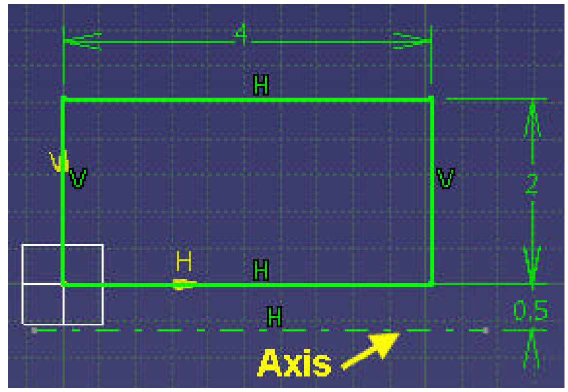




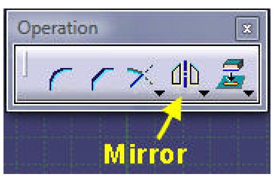
Double-click on the 3.736 dimension and the "Constraint Definition" box will open.

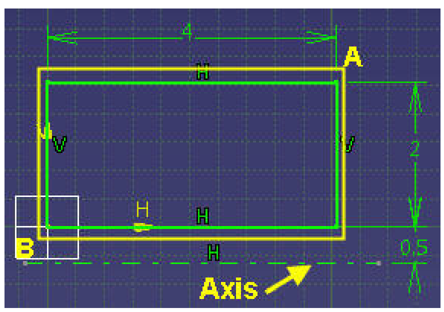
Edit the 3.736 dimension to 4 >> OK.

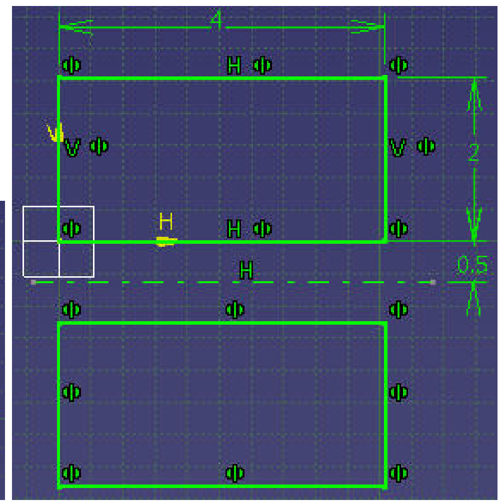




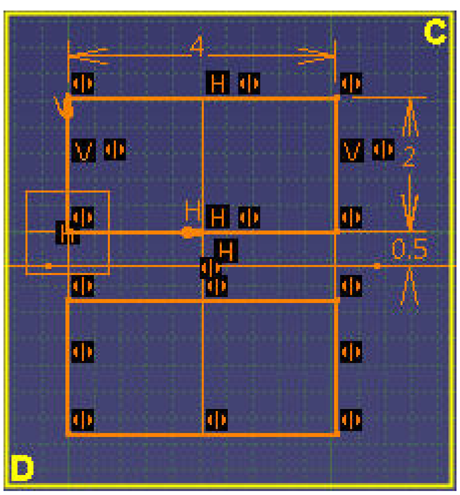
Edit the 1.952 to 2 by the same method. **On the "Profile" toolbar pick the Axis tool.**







Pick the "Mirror" tool. Pick point A and drag to B >> Pick the Axis >> Mirror is complete.



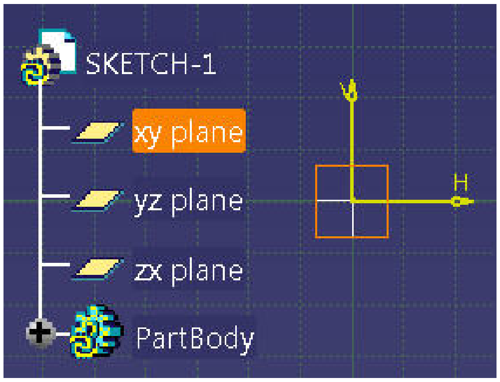
DELETE SKETCH

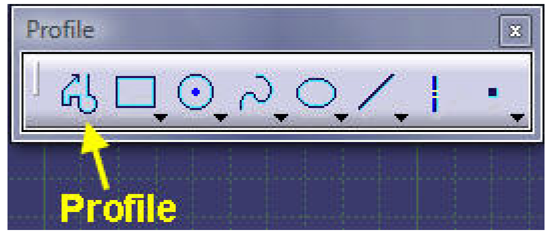
Pick point C and drag to D >>

Delete Key >> Sketch left is erased.

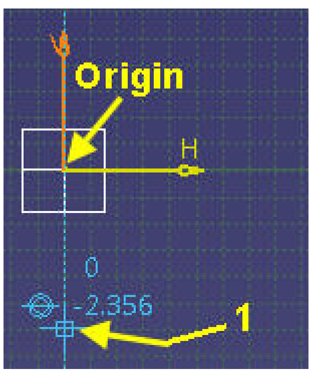
"Profile" tool below

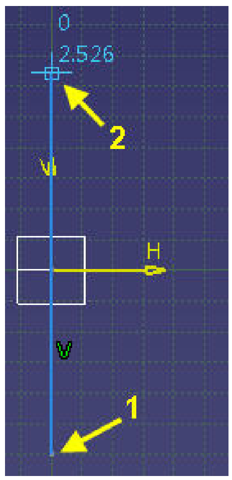
SKETCH WITH PROFILE TOOL

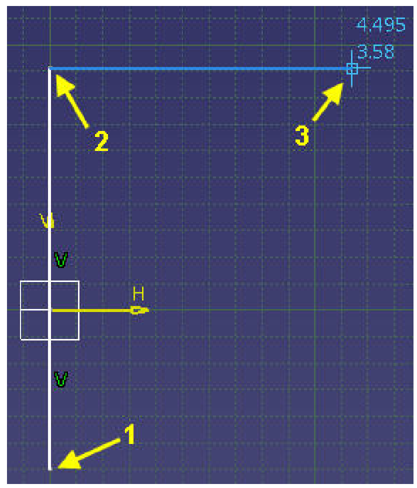




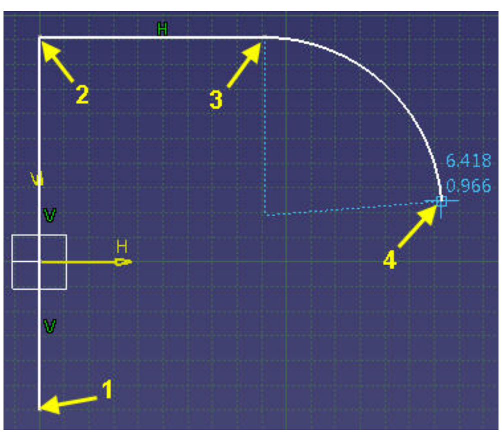
Pick the "xy plane" in the Tree with the left mouse pointer. Pick "Sketch" tool >> Pick the "Profile" tool above >>

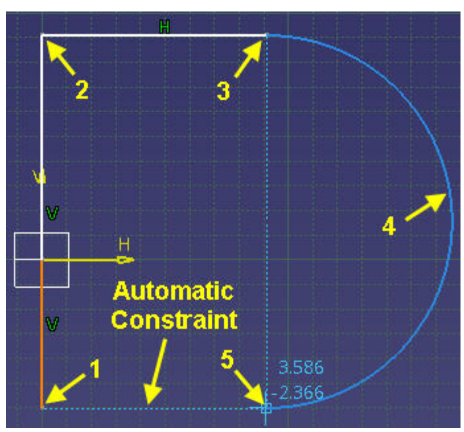




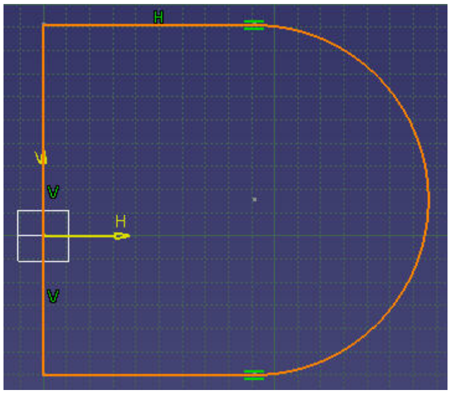


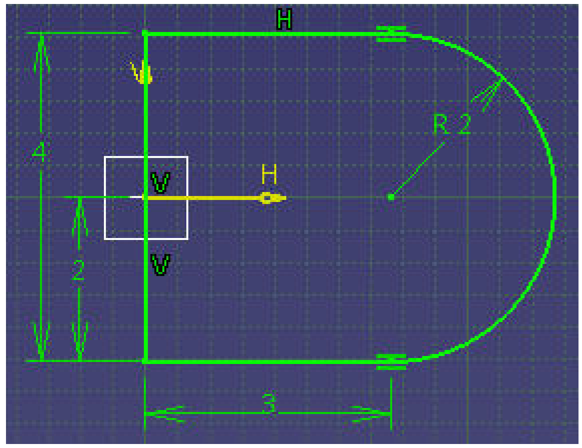
Pick-1 when on Origin line >> Release mouse button >> Pick-2 >> Release mouse button >> Pick-3 >> Hold mouse button >> Move mouse pointer to 4 >>





Continue moving mouse pointer to 5 >> Obtain the "Automatic Constraint" line Release mouse button >> Pick-1.





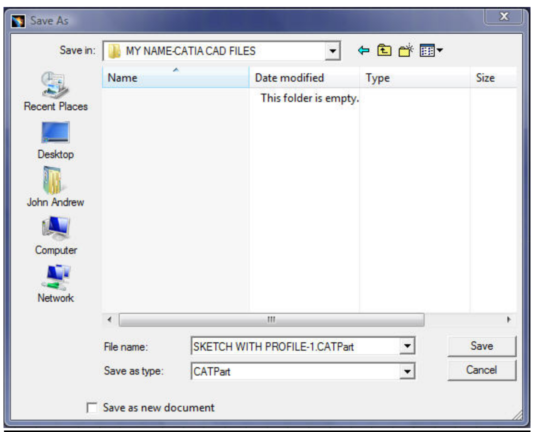
Un-constrained profile is red. **Fully constrained profile is green.**

The 2 inch dimension is necessary. **The Sketch must be constrained to the Origin.**

SAVING THE SKETCH

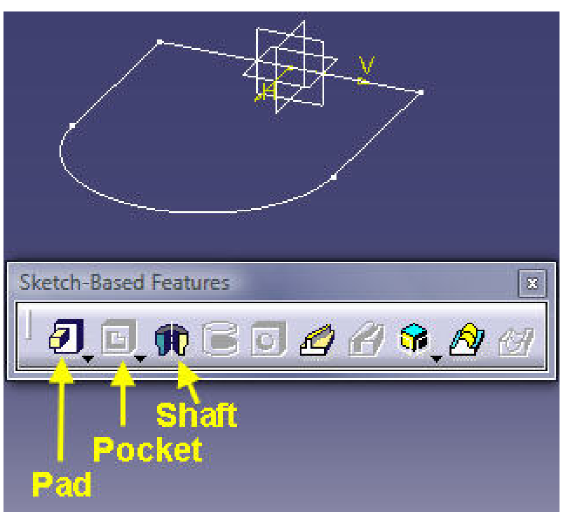
ALL CATIA CAD FILES SHOULD BE IN ONE FOLDER

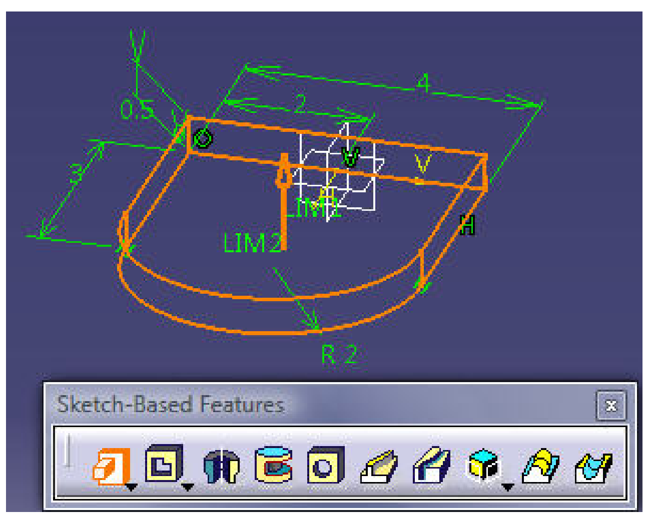
EACH CATIA PROJECT CAD FILES CAN BE IN A SUB-FOLDER



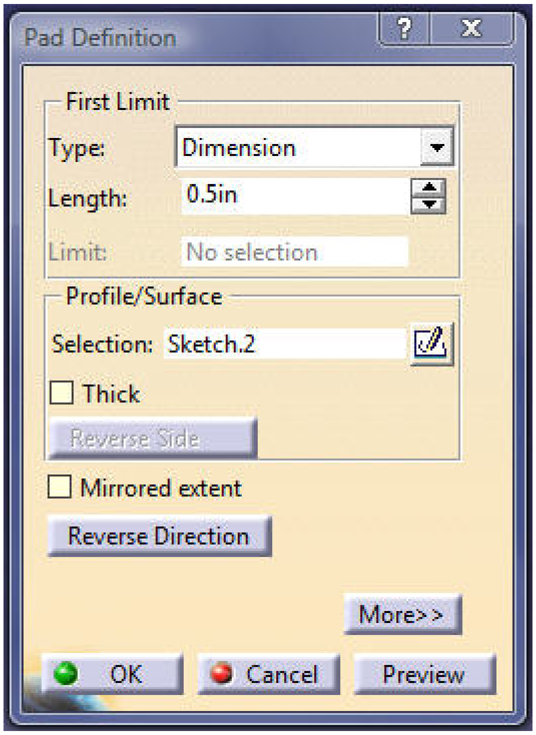
1. Create a folder in the: X drive >> Y folder >> Z sub-folder >> TITLE.CATPart

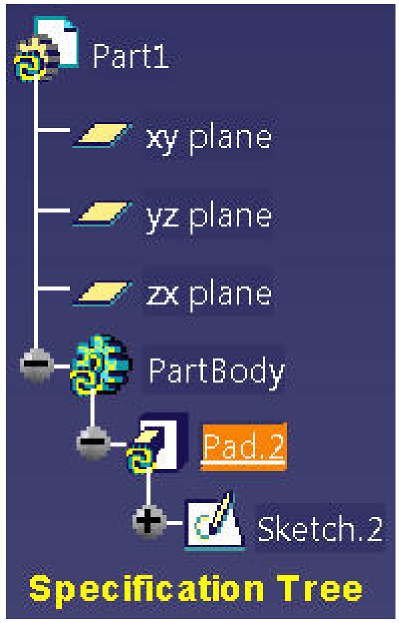
PADDING THE SKETCH





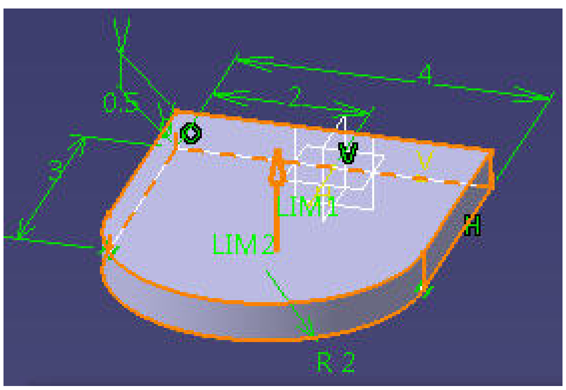
Exit the "Sketch Workbench" >> Pick the above sketch >> Pick the "Pad" tool.





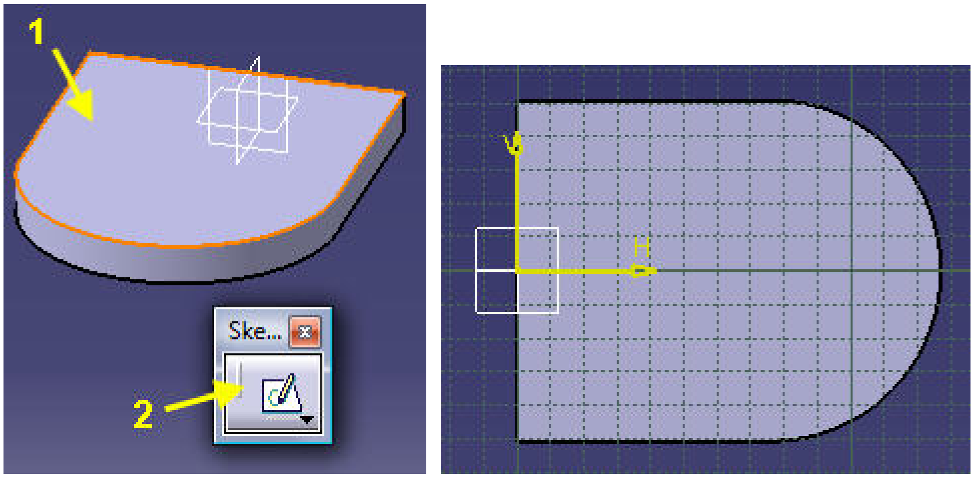
In the "Pad Definition" box >> Length >> 0.5in >> Preview.

Pad.2 has been added to the Specification Tree.

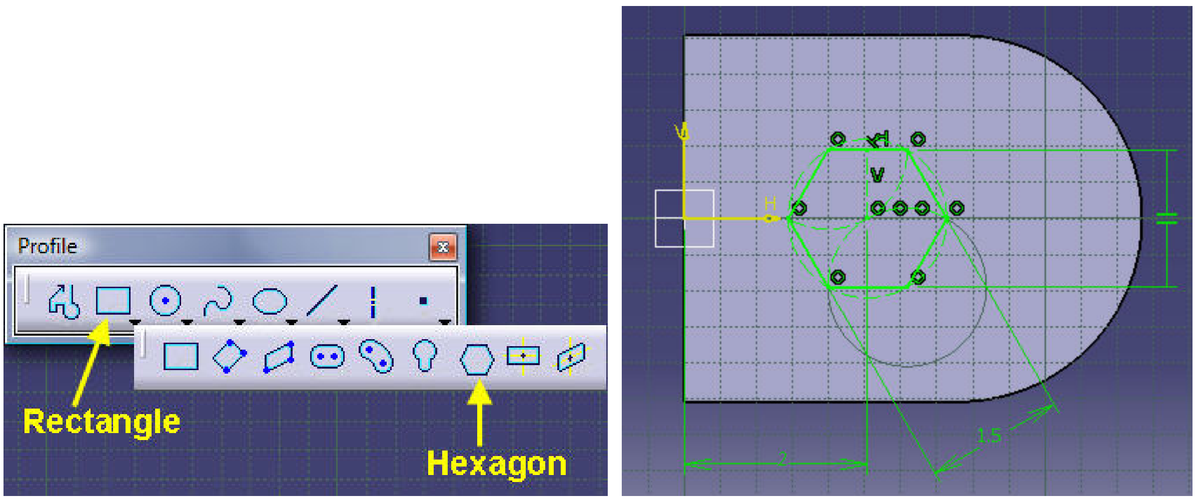


The Pad preview is shown above.

File >> Save as >> Browse >> My CAT Folder >> My Initials-PAD-1.

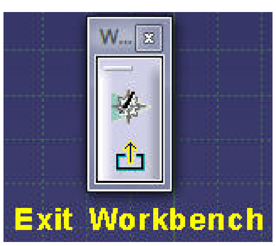


Pick-1 pad top surface >> Pick-2 Sketch tool.

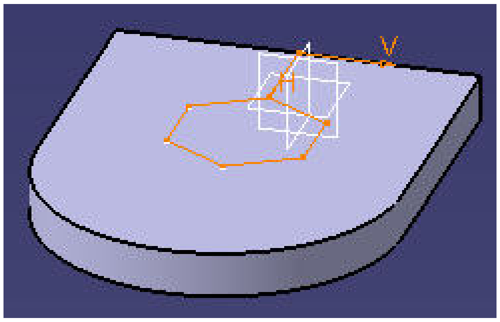


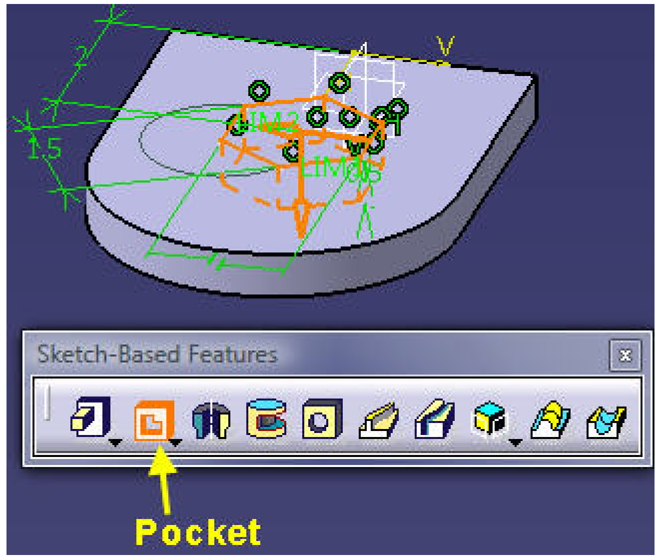
Pick the Drop-down menu under the Rectangle tool >> Pick Hexagon.

Place the hexagon shown above at 2in from the origin and 1.5in across flats.

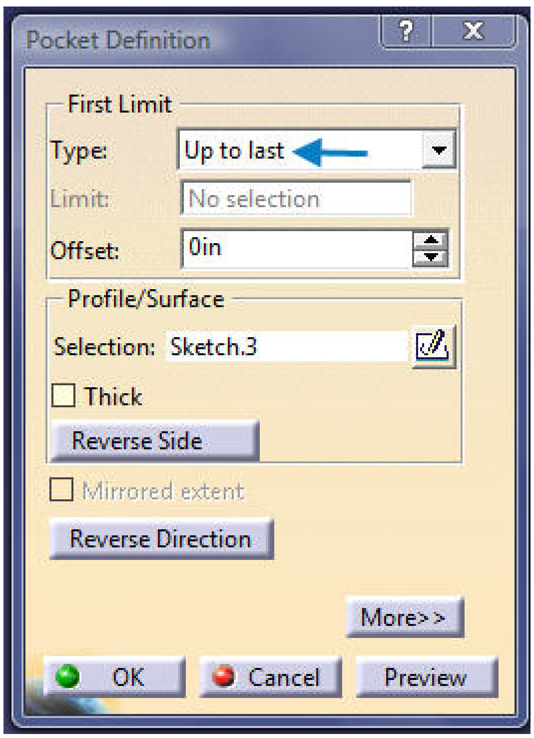


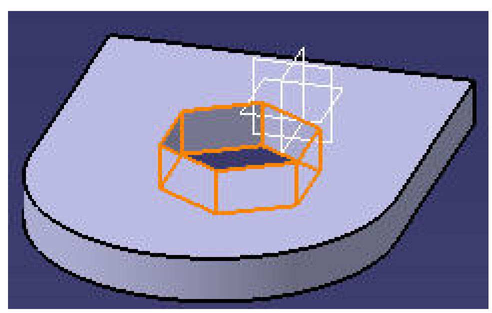
Exit the Sketch Workbench.



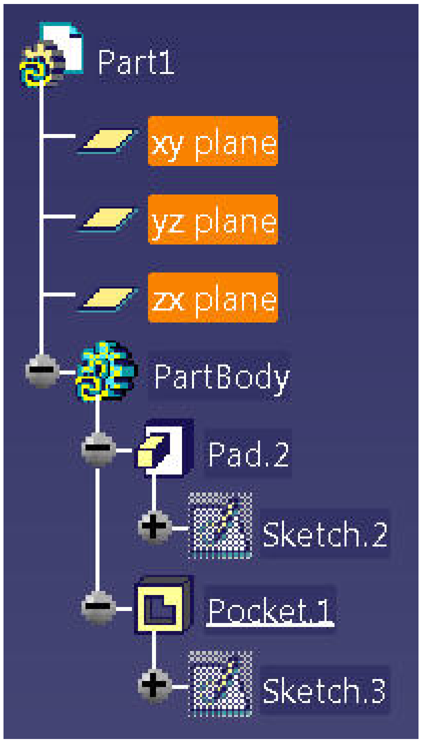


Pick the hexagon sketch >> Pick the "Pocket" tool >>





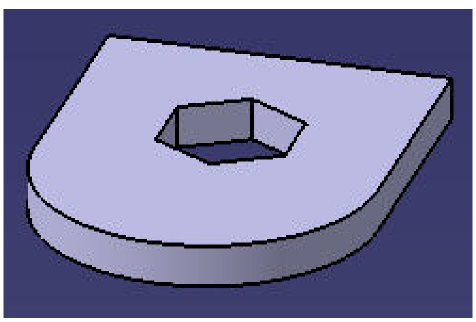
First Limit >> Type: >> Up to last >> OK. **The hexagon pocket is shown above.**



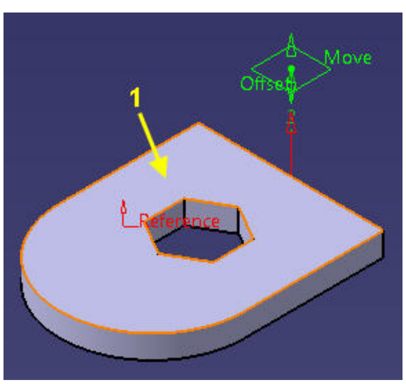
Pocket.1 has been added to the Specification Tree.

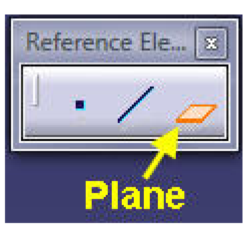
Double click on the xy plane >> hold the Shift key >> pick the zx plane.

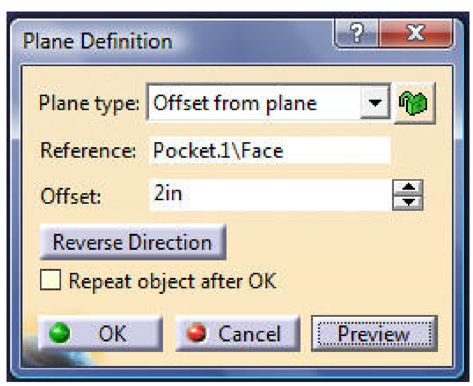
All three planes are selected as above >> Pick the "Hide / Show" tool.



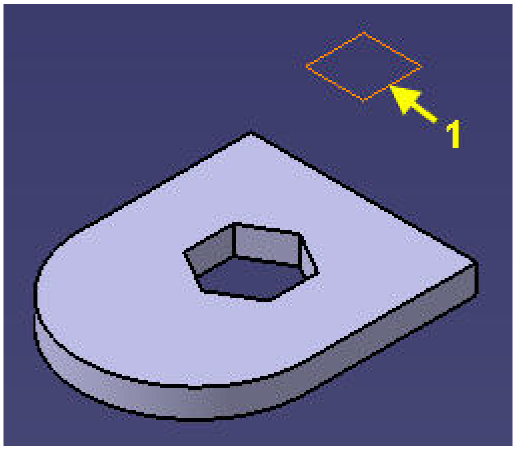
The reference planes in the pad are hidden as shown above.

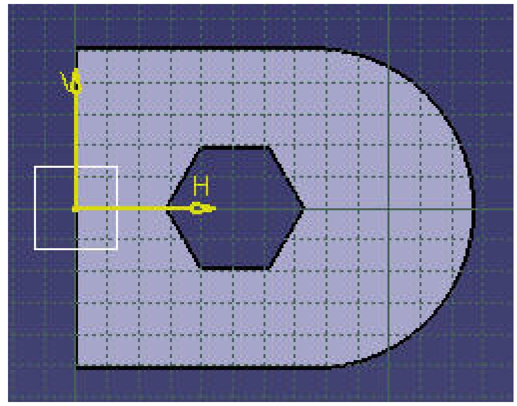




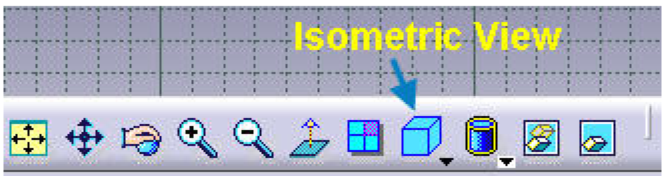


Pick-1 >> "Plane" tool >> Offset: >> 2in >> OK.

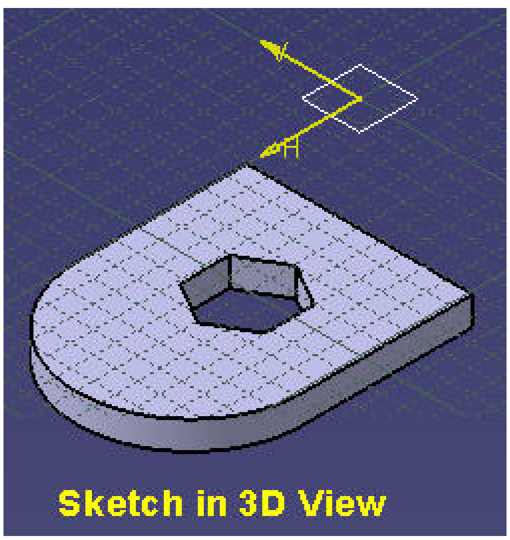


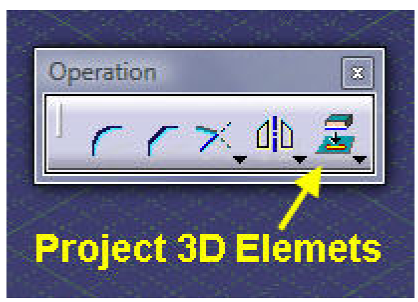


Pick-1 >> Sketch >>

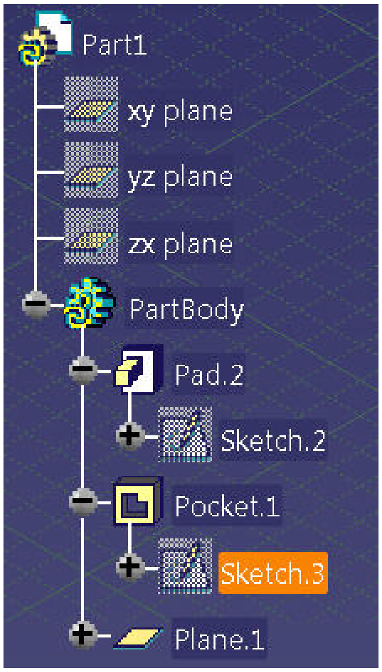


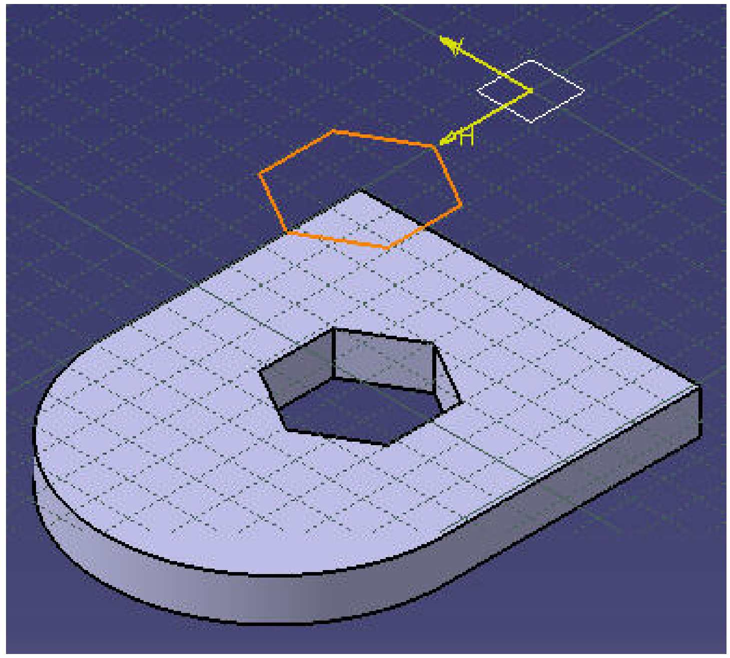
Isometric View >>





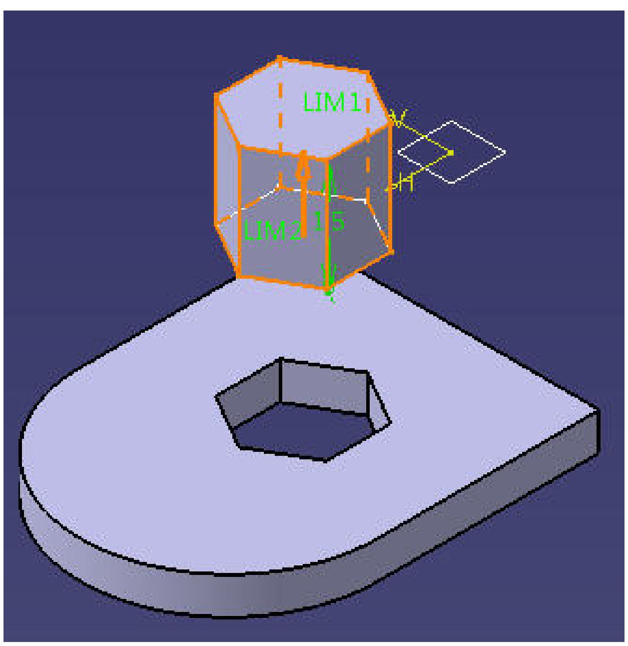
Sketch plane in 3D view >> Project 3D Elements >>

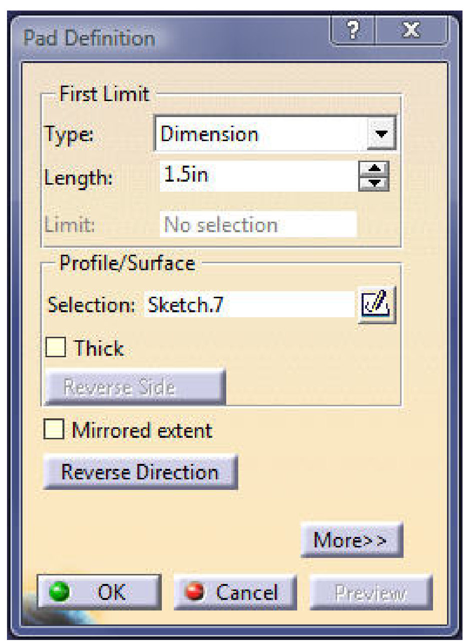




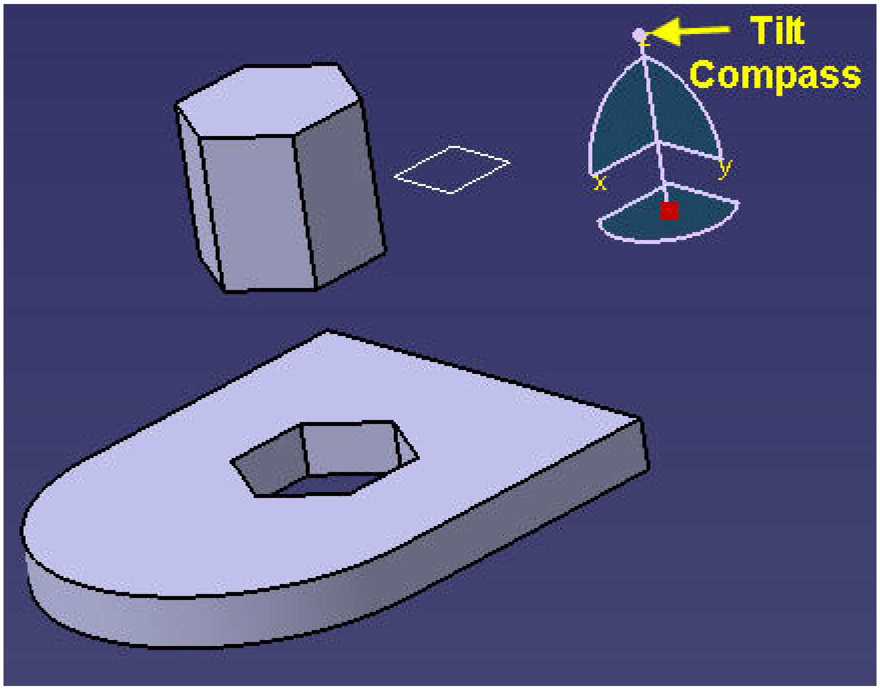
Pick the "Hexagon Pocket Sketch.3" >> Exit Workbench.

Pad the sketch >>



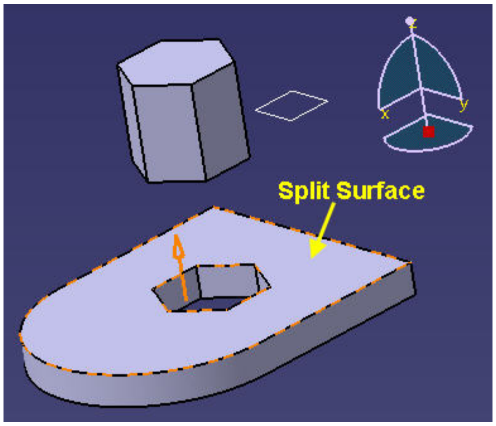


PLUG length >> 3in >> OK.

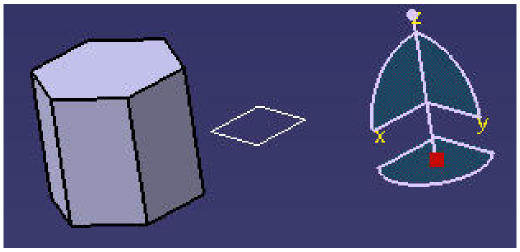


File >> Save As >> PAD & PLUG, one part needing to be split into two parts.

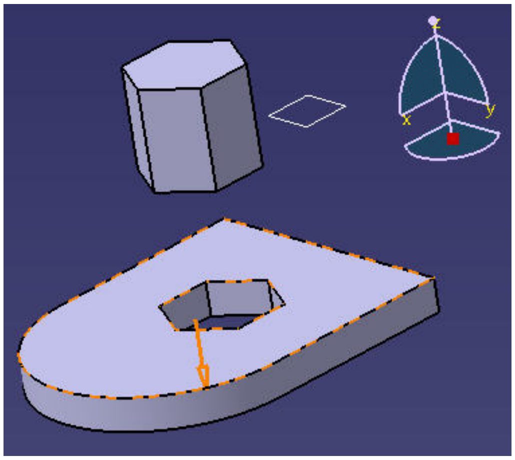
Pick top of compass >> tilt compass.

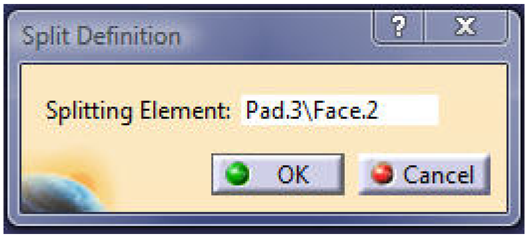


Split >> Pick "Split Surface" >> Pick Arrow to change included area >> OK.



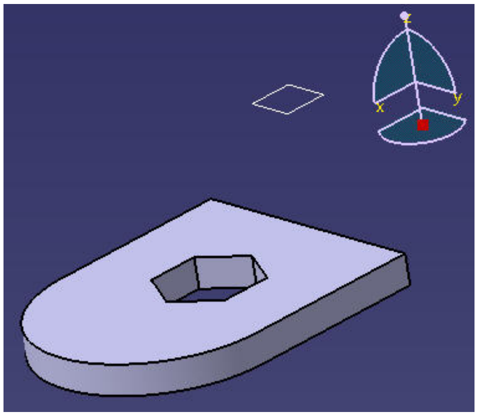
File >> Save As >> PLUG.





File >> Open >> PAD & PLUG

Split >> Pick "Split Surface" >> Pick Arrow to change included area >> OK.

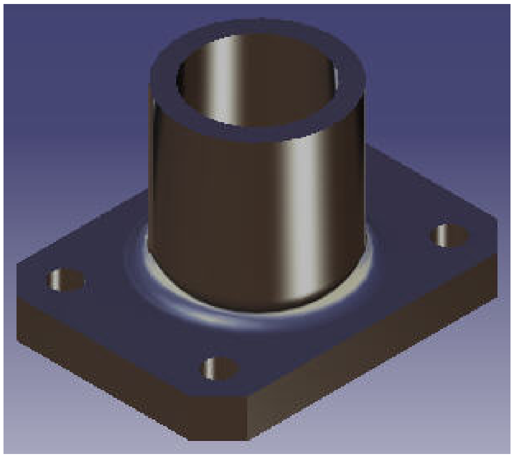


File >> Save As… >> PAD

2. PART DESIGN WORKBENCH

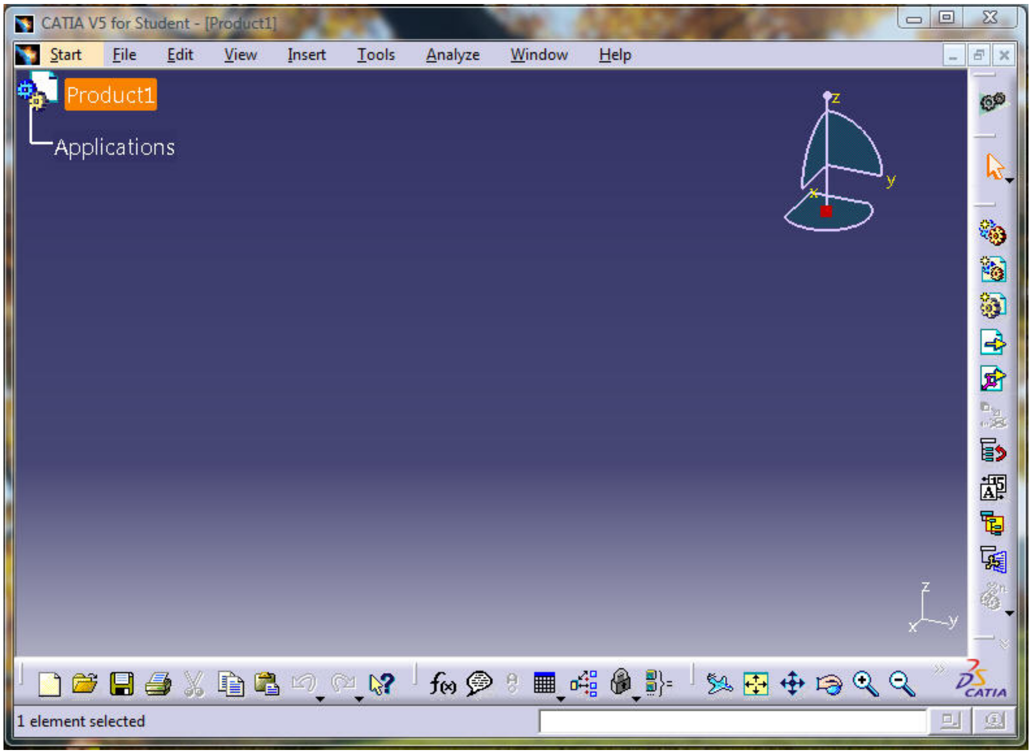
All 3D solid "Parts" begin with a sketch in the "Sketcher Workbench".

BASE BRACKET EXAMPLE



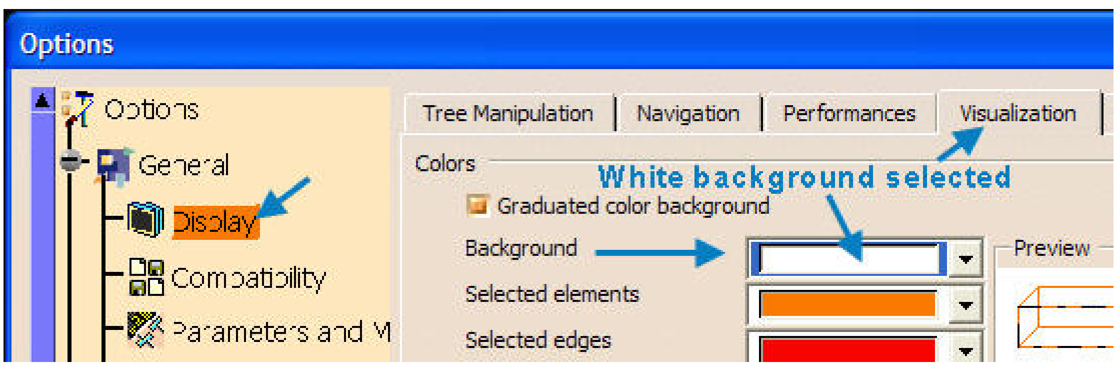
Objective: Create the part above named "BASE-M101".

OPEN CATIA



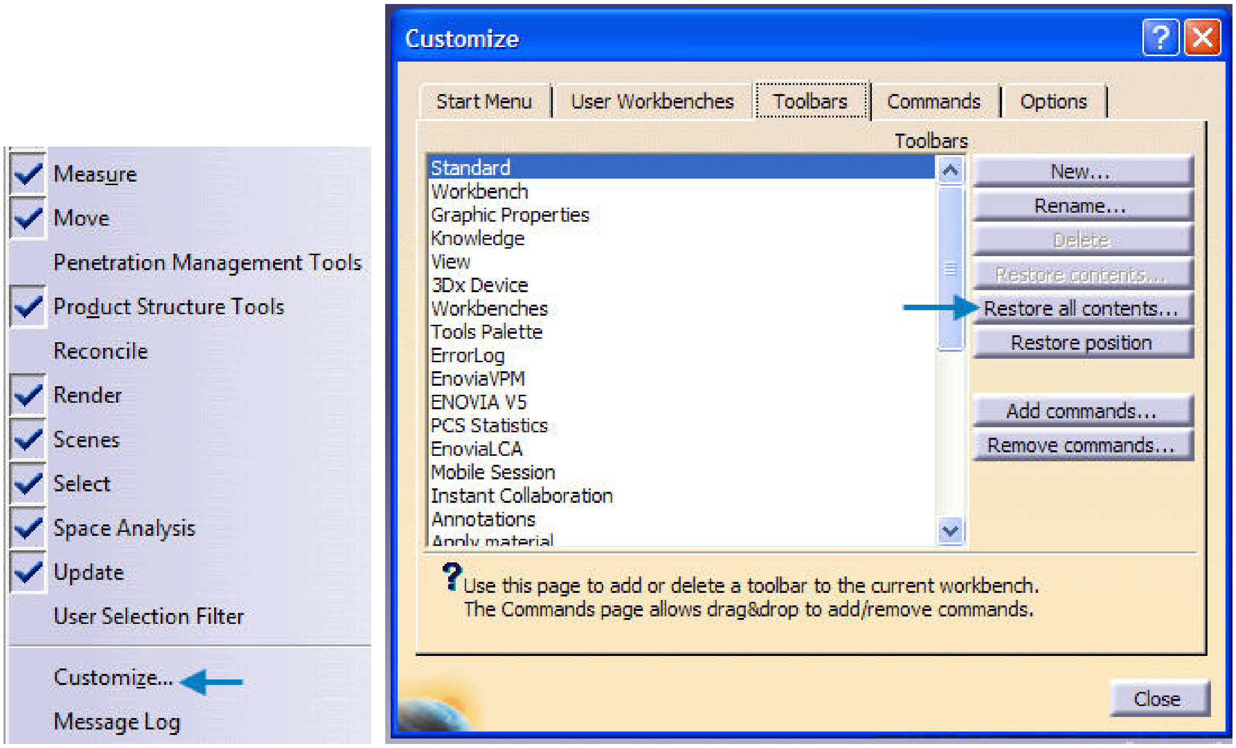
TO CUSTOMIZE THE CATIA ASSEMBLY DESIGN AREA

A. CHANGE BACKGROUND COLOR



Tools >> Options >> Display >> Visualization >> Background >> White or other color.

B. RESTORE ALL TOOLBARS IN A WORKBENCH



1. Right click on the toolbar on right side of the Catia display.

2. List of toolbars opens above left.

3. Click on toolbars one by one to restore or

4. Click on "Customize" at the bottom of the list and select:

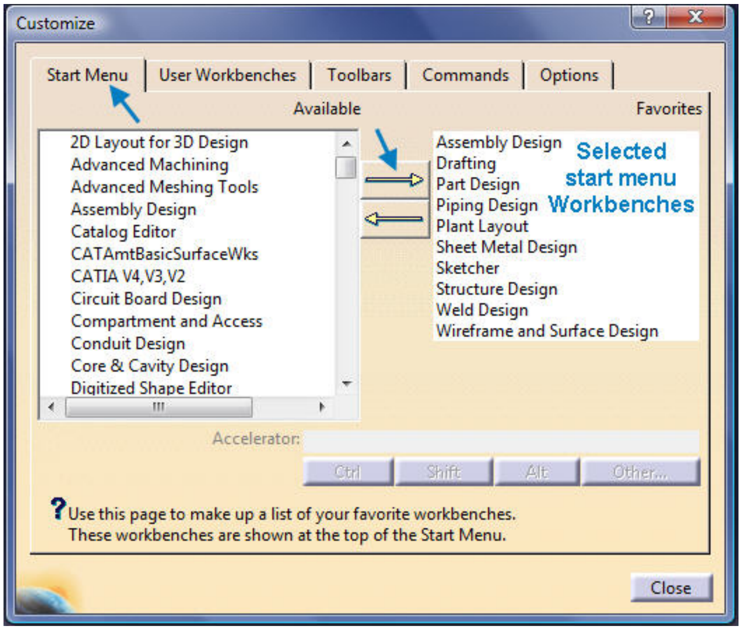
Restore all contents… >> Restore position

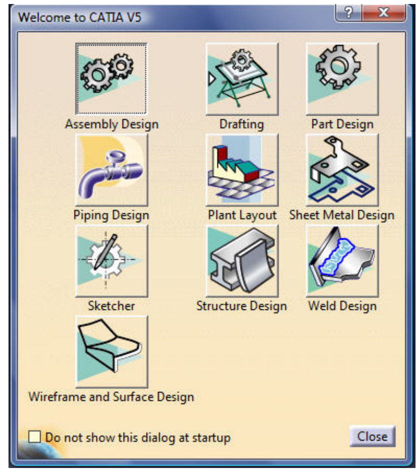
under the Toolbars tab in the Customize box above right.

5. If shafts or holes are not smooth round click: Tools >> Options >> Display >> Performance >> 3D Accuracy >> Fixed >> 0.01 >> 2D Accuracy >> Fixed >> 0.01 >> OK

Parts may be created and assembled in the "Product Workbench" shown above.

C. CUSTOMIZE THE START MENU

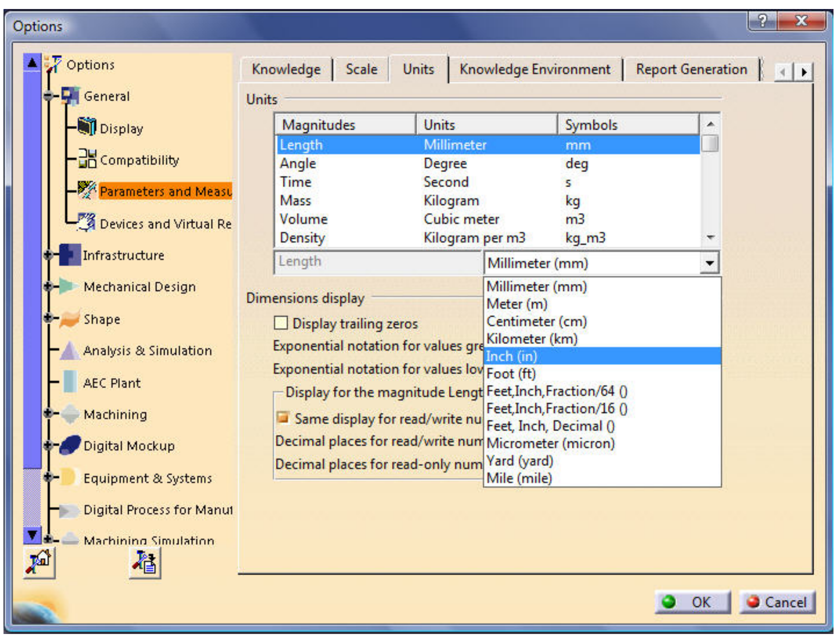




1. Right click on the toolbar on right side of the Catia display.
2. Click on "Customize" at the bottom of the list and select:

   Start Menu > Click on an item in the left side >> Right arrow to move the item to the right side.
3. Ten Catia "Workbenches" have been added to the "Welcome to CATIA V5" box above right.

D. CHANGE UNITS OF LENGTH



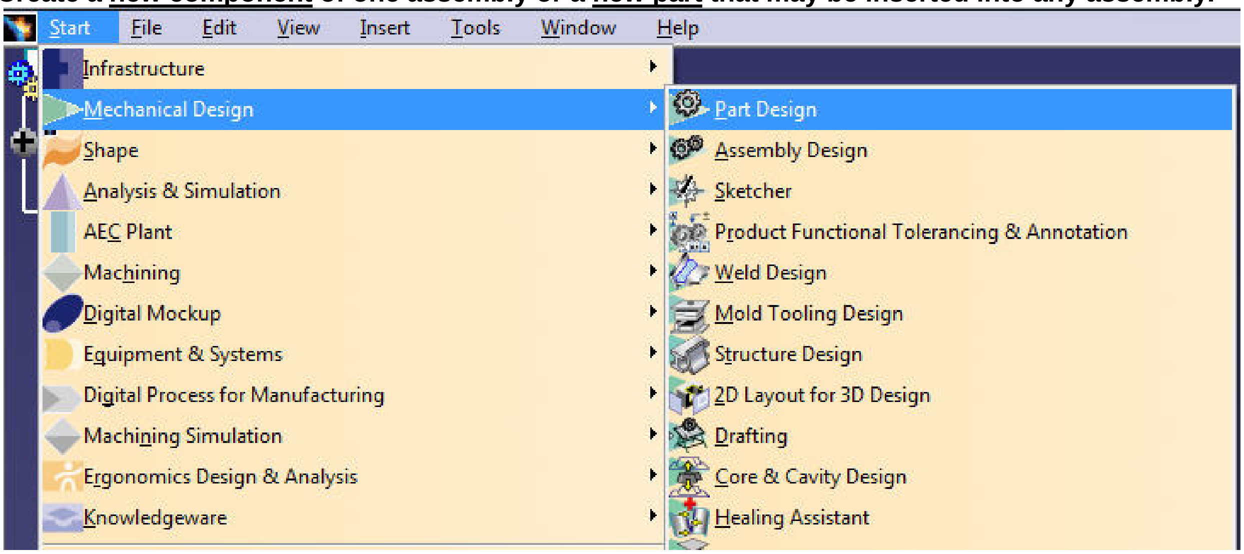
Select drop-down menu:
Tools >> Options >> Parameters & Measure >> Units >>
Select Inches or Millimeters, as shown above.

E. CHANGE AUTOMATIC BACKUP INTERVALS

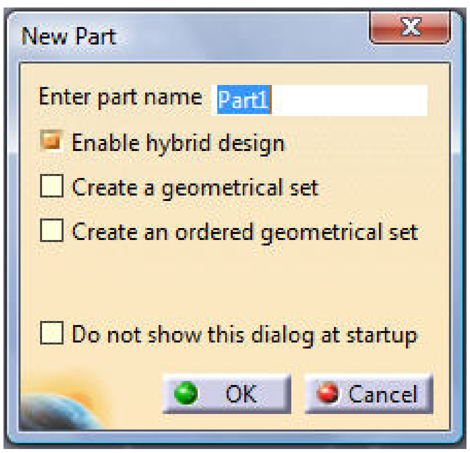
Automatic save is obtained at: Tools >> Options >> Automatic backup every >> 10 minutes.

START A NEW PART

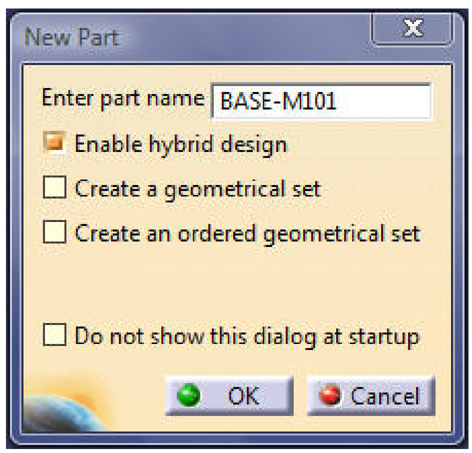
Create a new component of one assembly or a new part that may be inserted into any assembly.



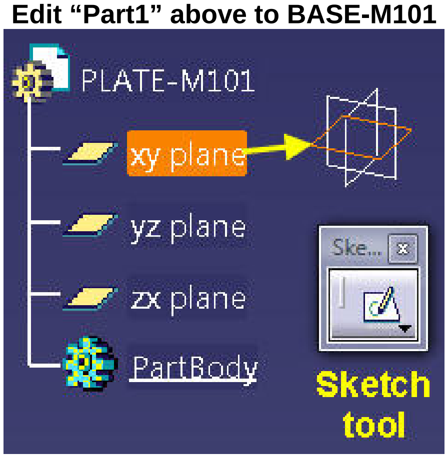
Select: Start >> Mechanical Design >> Part Design



Edit "Part1" above to BASE-M101

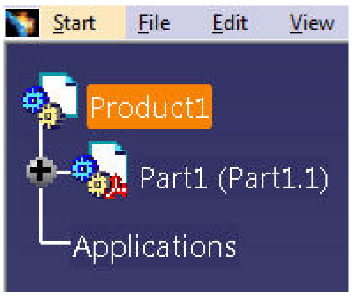


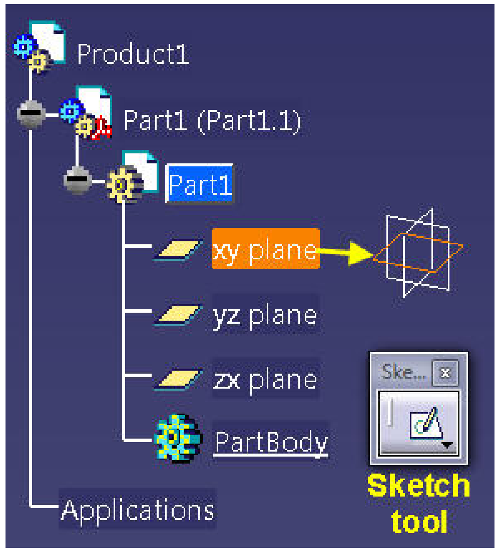
"BASE-M101" has been entered



If the part name is not changed Catia will provide the name "Part1".

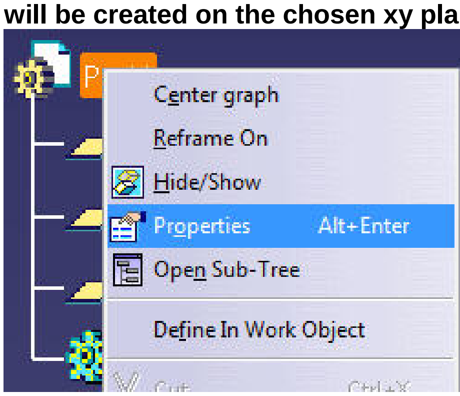
The part name can be changed later as shown below.



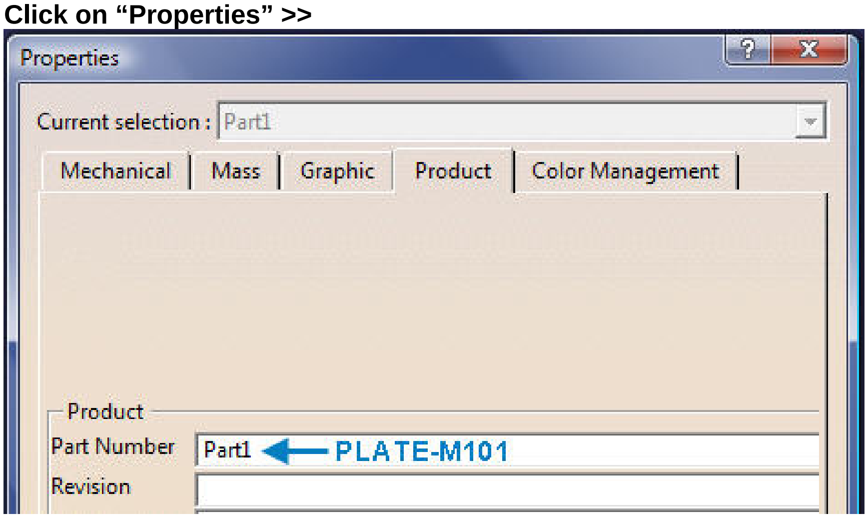


Pick the "+" sign to expand the "Specification Tree"
Pick one of the 3 planes to make a sketch on.
Pick the "Sketch" tool icon >> "xy" plane in this example.

A two dimensional sketch of one face of the proposed part will be created on the chosen xy plane above.

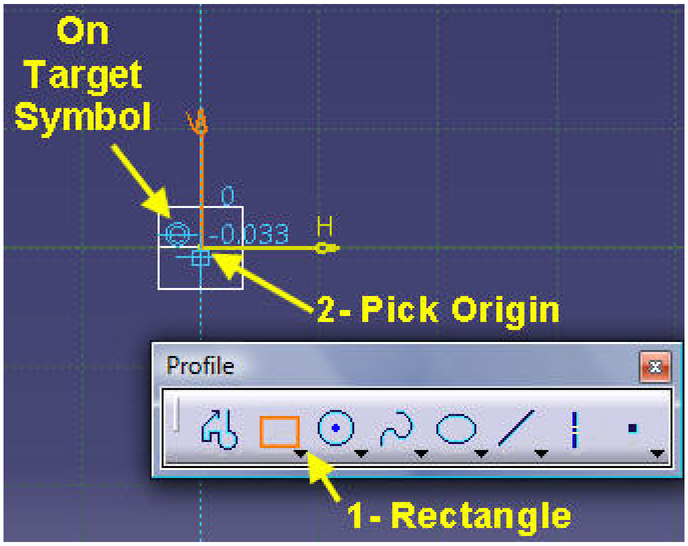


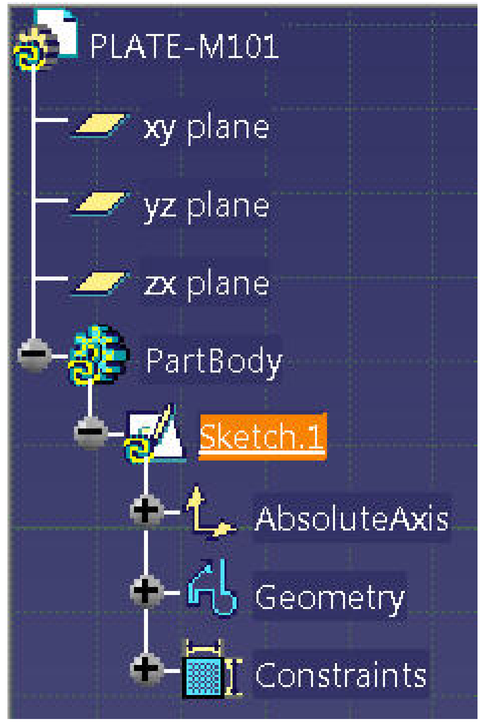
To change the Part name at any time, right click on the title at the top of the specification tree >> Click on "Properties" >>



>> Select top tab "Product" >> edit Part Number >> PLATE-M101.

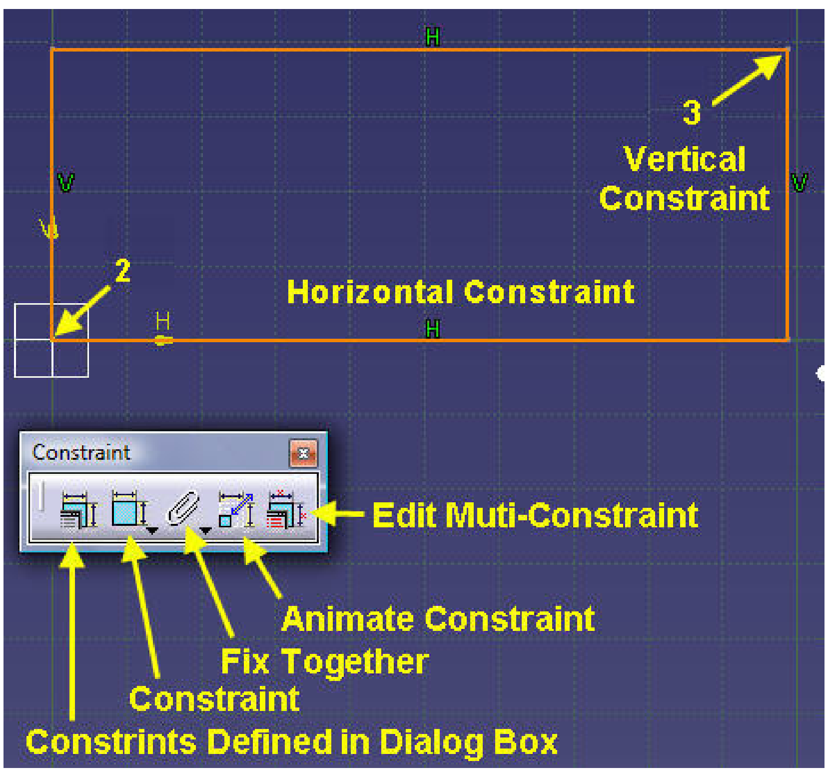
SKETCH A RECTANGLE





Select 1-Rectangle on the "Profile" toolbar. Move the mouse pointer until the "On Target" symbol appears.

The "Specification Tree" above lists all part construction elements.

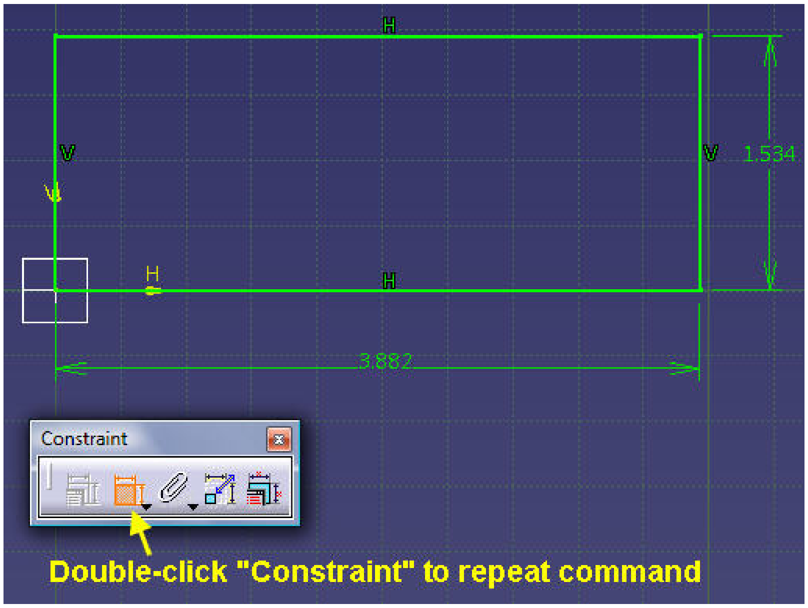


Create the above rectangle by dragging the mouse pointer from point 2 to 3.

AUTOMATIC CONSTRAINTS:

H (horizontal) and V (vertical) are created by Catia.

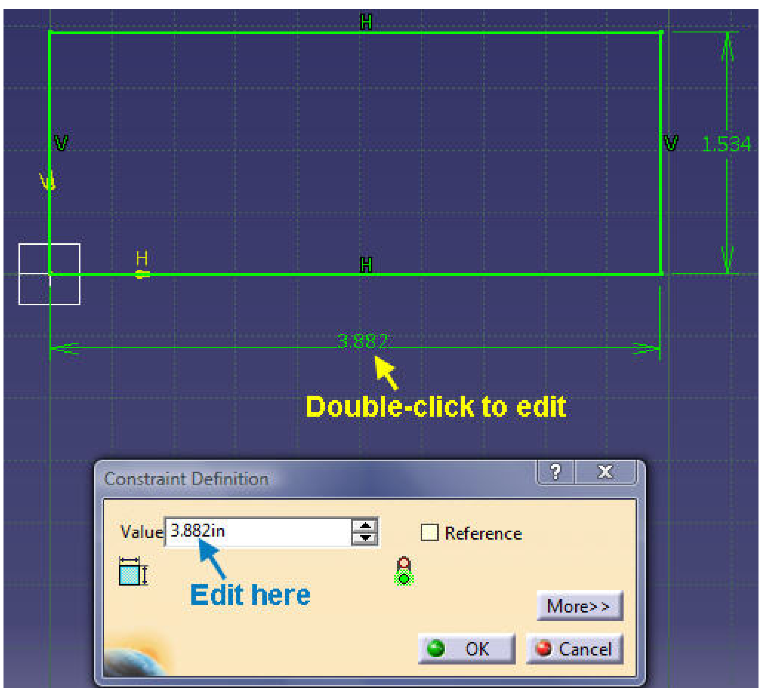
ADD DIMENSIONS



Double-click the "Constraint" icon to dimension objects.

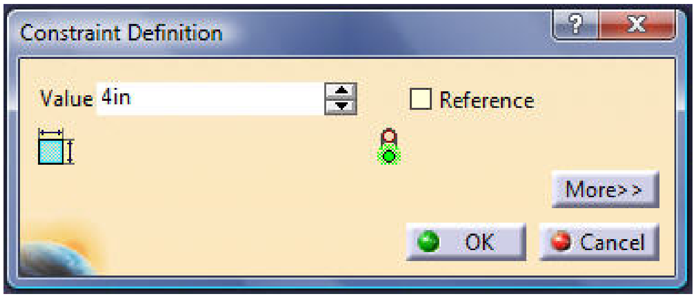
Pick the bottom horizontal line and place its dimension below.

Pick the right vertical line and place its dimension to the right.

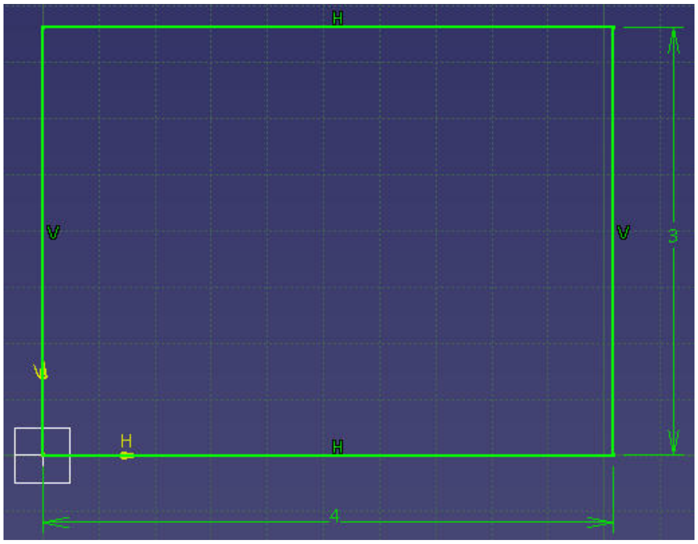


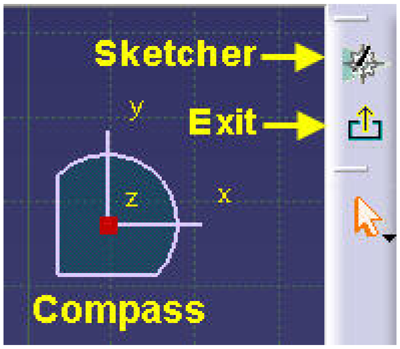
EDIT DIMENSIONS: Double-click the horizontal dimension (3.882 inches left).

Type 4 in the Constraint Definition "Value" box as shown left.



The 3.882 inch dimension has been changed by typing, "4" >> OK.

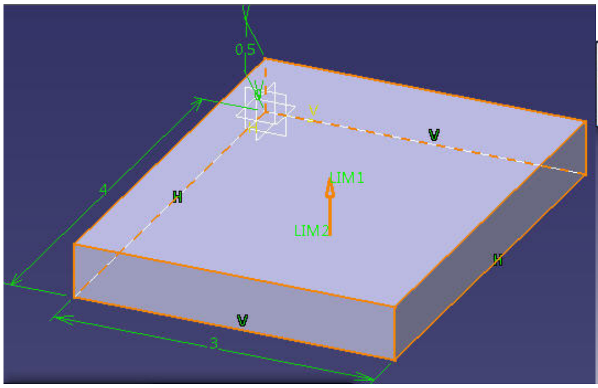


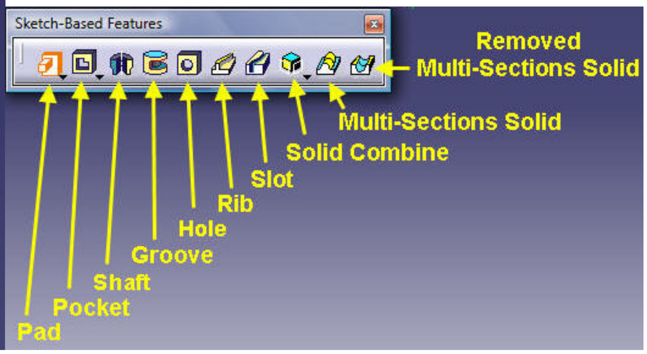


Double-click the vertical 1.534 dimension and change it to 3 inches.

Pick the "Exit" icon to leave the "Sketcher Workbench".

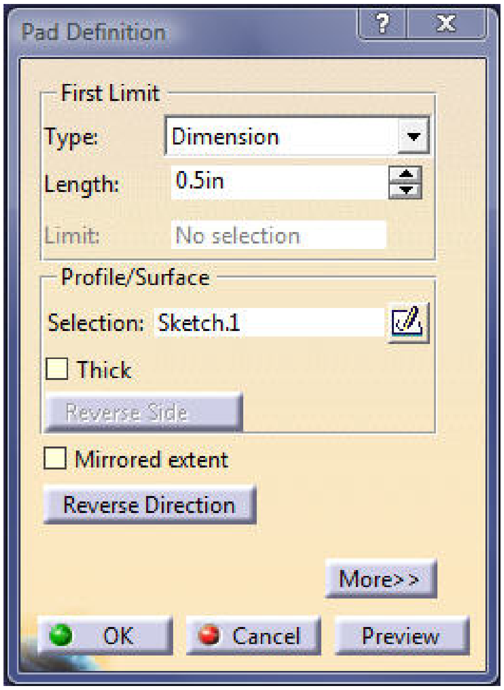
ENTER THE 3 DIMENSIONAL ZONE

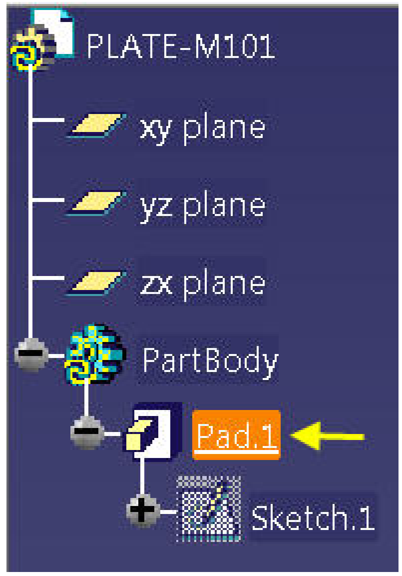




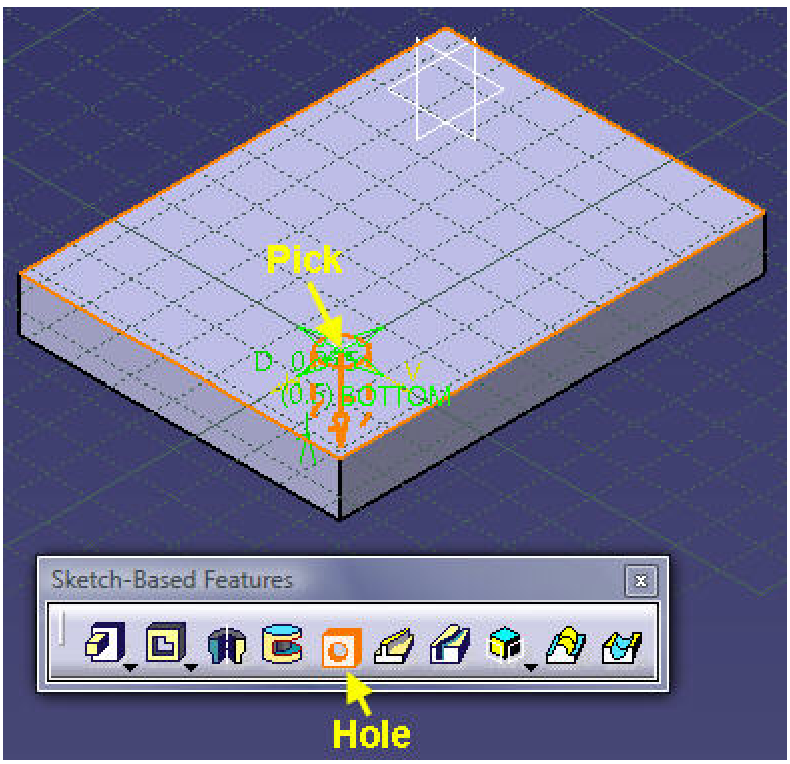
With the mouse pointer select "Pad" on the "Sketch-Based Features" toolbar above.

Each icon on the "Sketch-Based Features" toolbar represents its function.

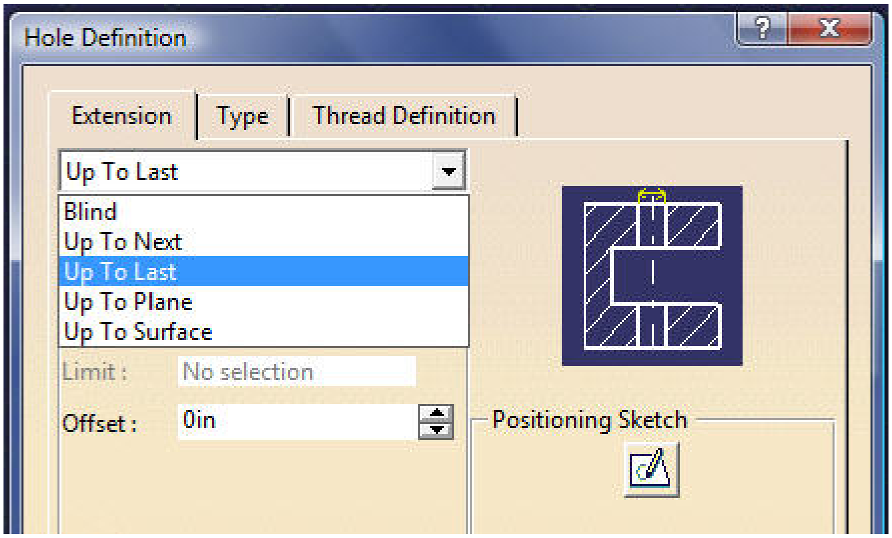




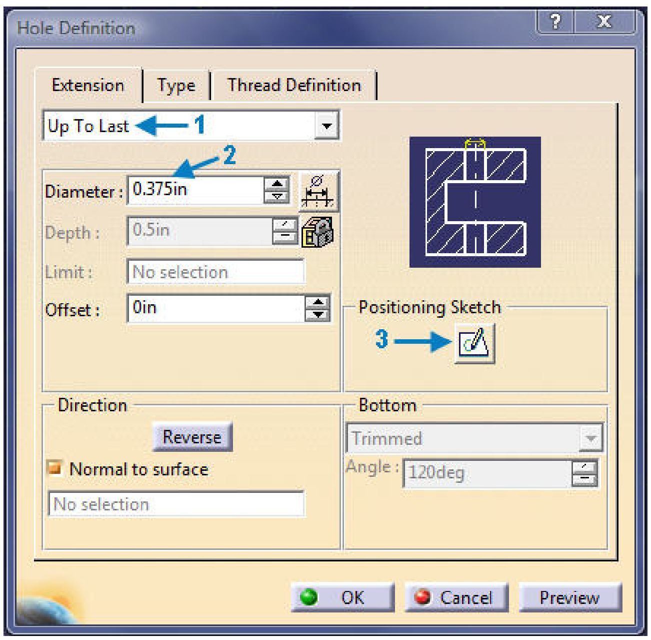
Set the PAD Length to 0.5in above.

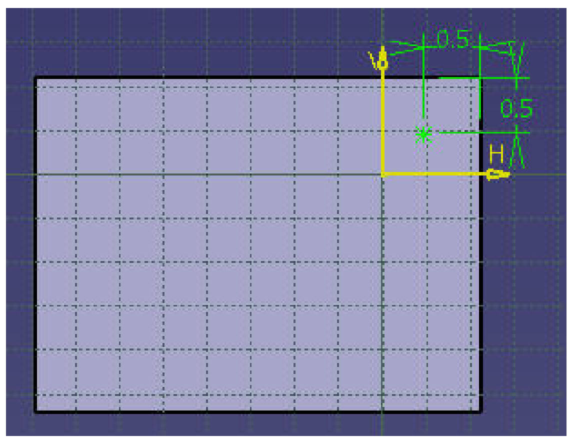


Pick Pad.1 in specification tree. Select: Hole >> Pick hole location

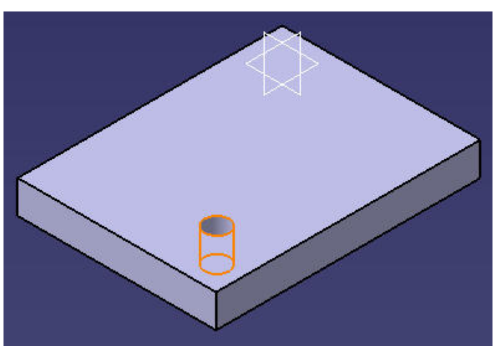


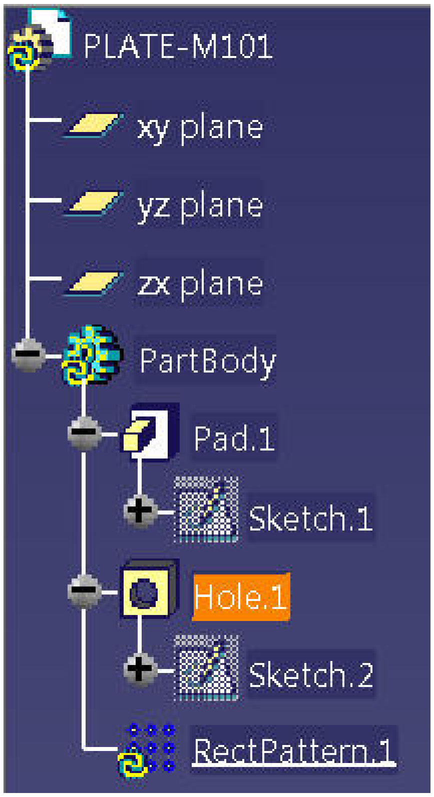
1. **Hole Definition box opens >> Drop down "Extension" >> select Up To Last**

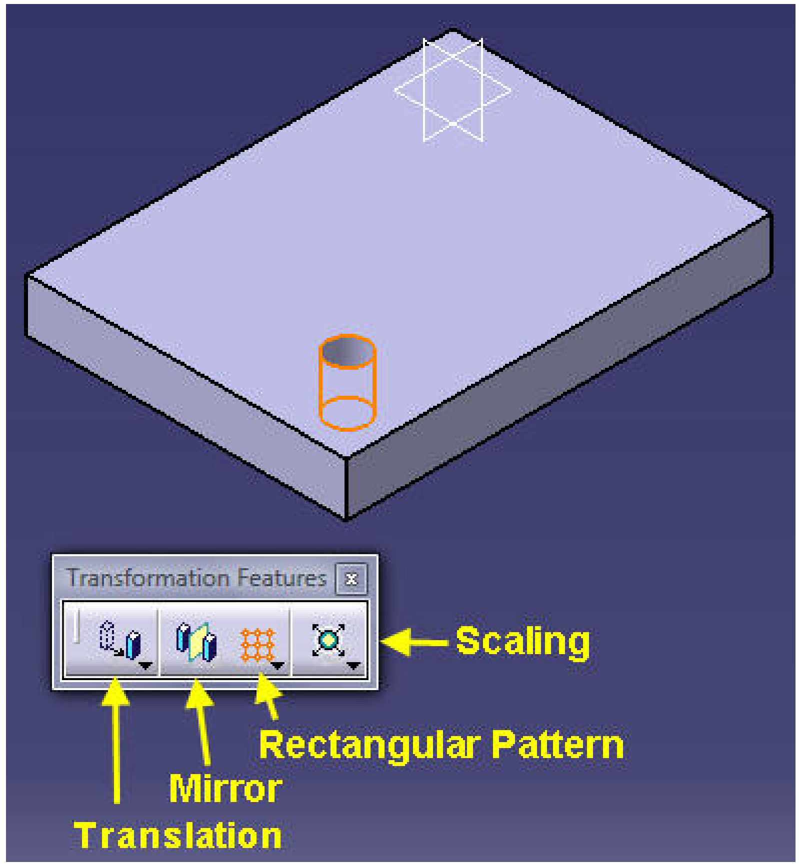




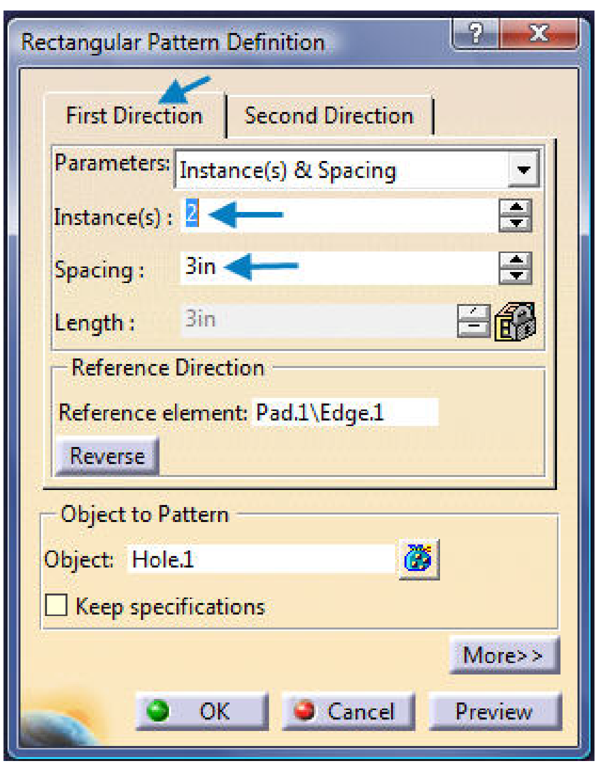
2. **Edit diameter to: 0.375in as above.**
3. **Pick: "Positioning Sketch" and add hole location dimensions 0.5 and 0.5 as above right.**

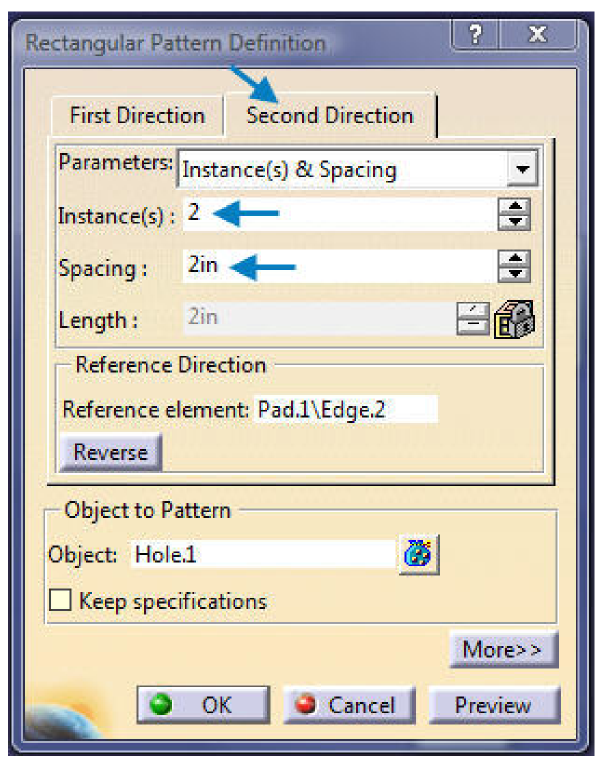


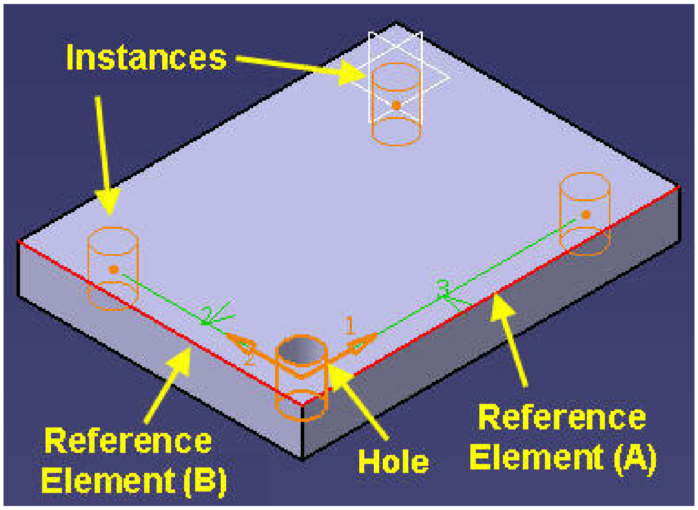




Hole has been added to the Tree. Pick "Rectangular Pattern" on Transformation Features bar.



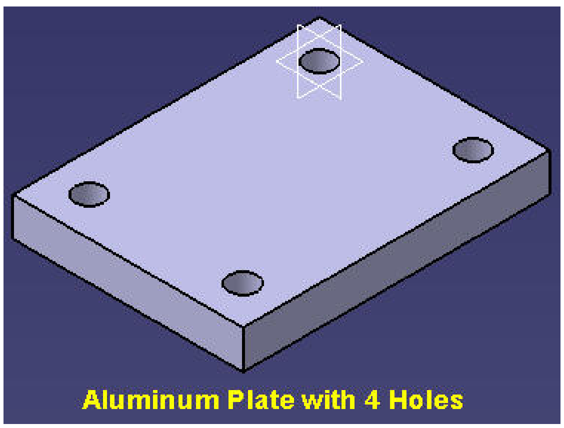


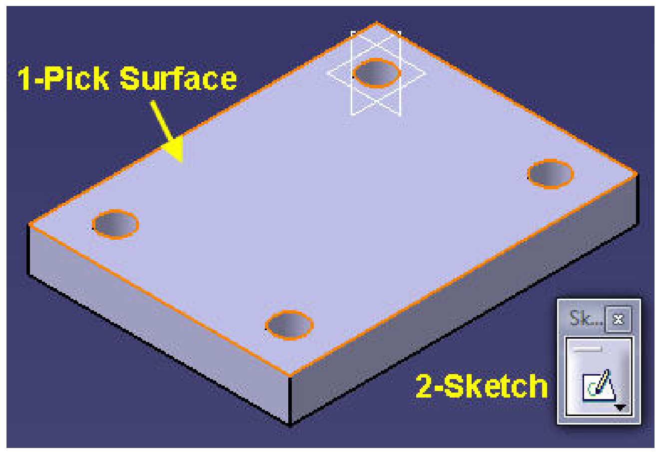


Pick: First Direction >> Instances >> 2 >> Spacing >> 3in >> Reference Element (A)

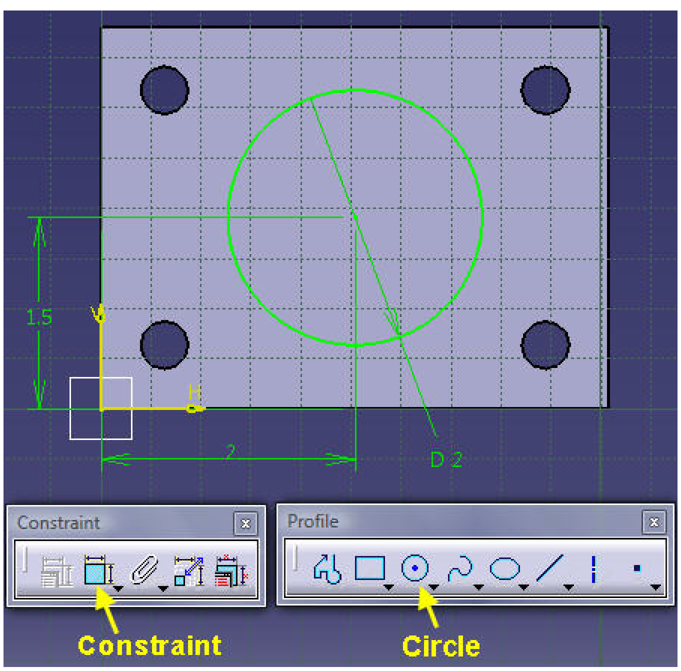
Pick: Second Direction >> Instances >> 2 >> Spacing >> 2in >> Reference Element (B)

Preview >> OK



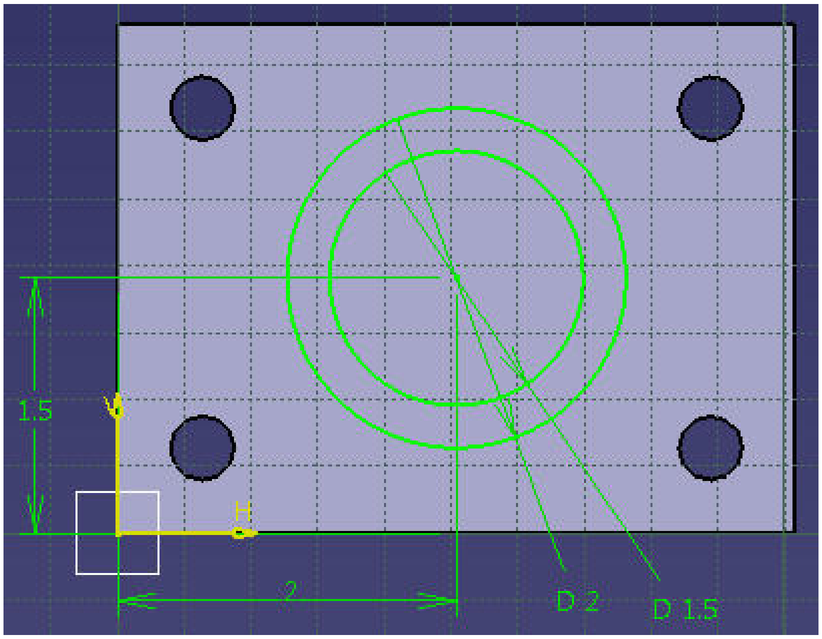


Pick any point on the plate top surface >> Pick the "Sketch" icon.



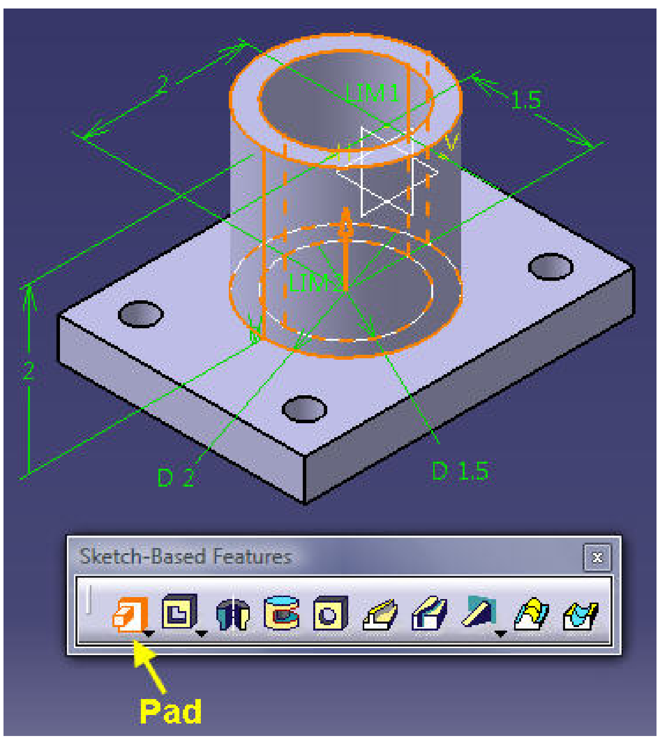
Pick the "Circle" icon on the Profile toolbar.

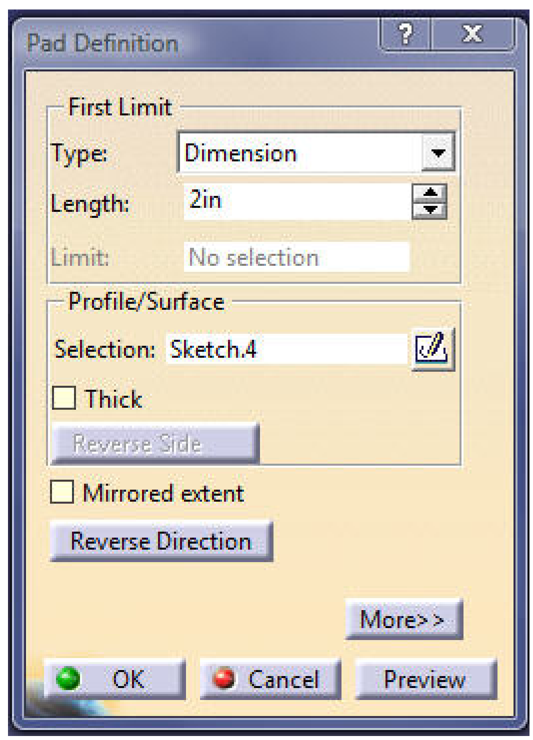
Place the circle center in the approximate plate center with the mouse pointer.



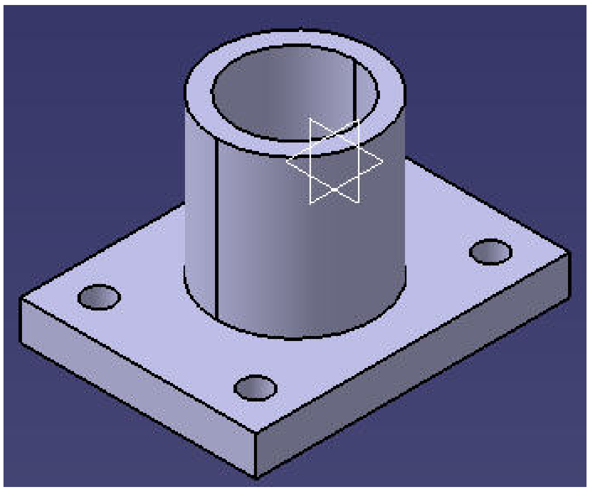
Double click the "Constraint" icon >> Pick the circle center >> Pick the plate bottom edge >> Position the vertical dimension. Create the hole horizontal dimension.

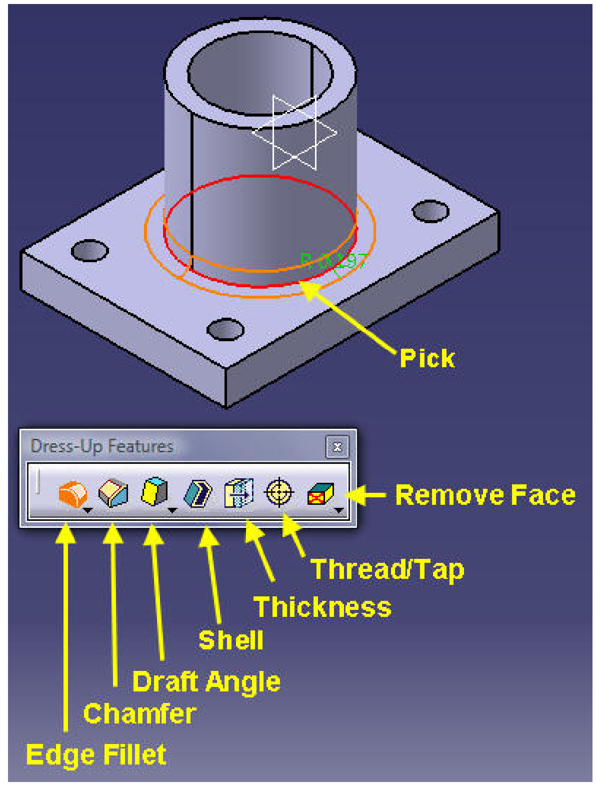
Double click on one dimension and edit as shown above. Edit the other dimension.



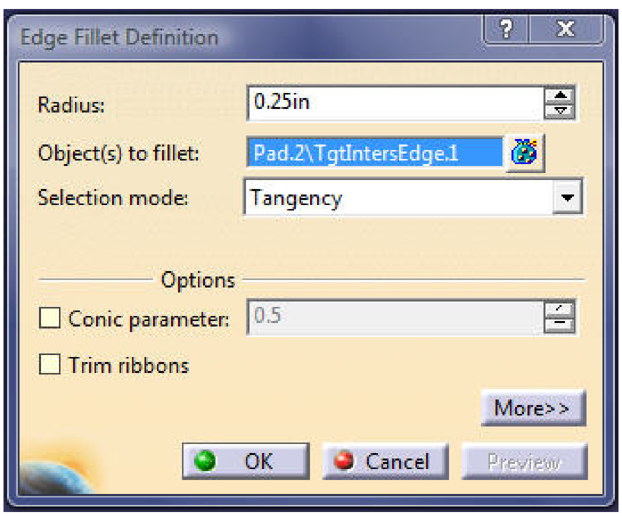


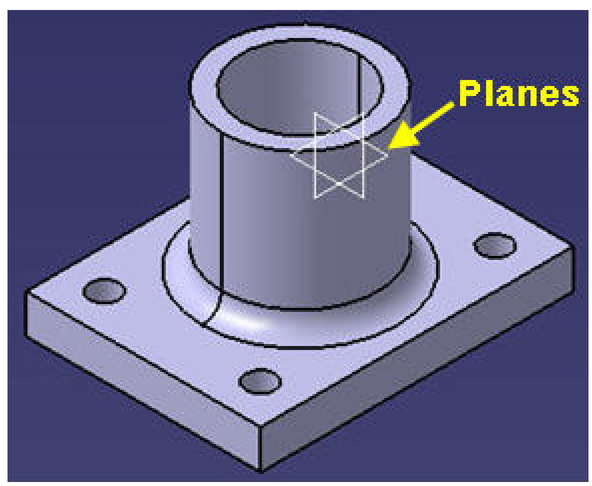
Pick "Pad" >> Edit Length to 2in as above right >> OK.





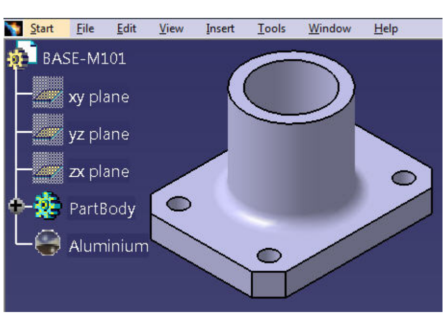
On the "Dress-Up Features" toolbar select: Edge Fillet >> Pick the circle shown above.

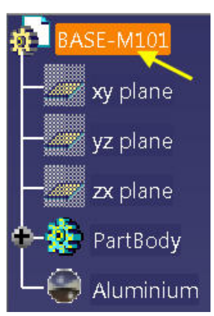




Edit the fillet radius to 0.25in.

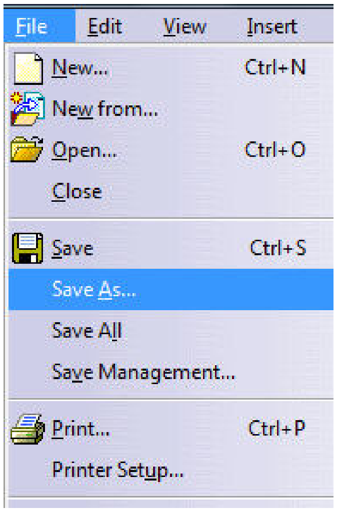
Finished fillet.

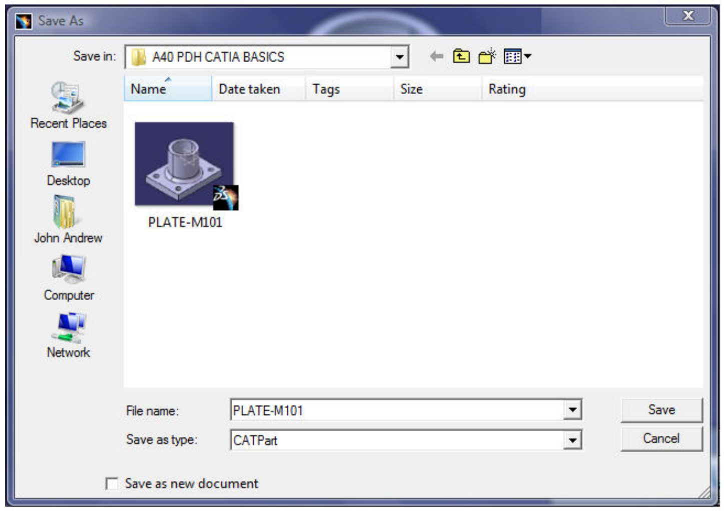




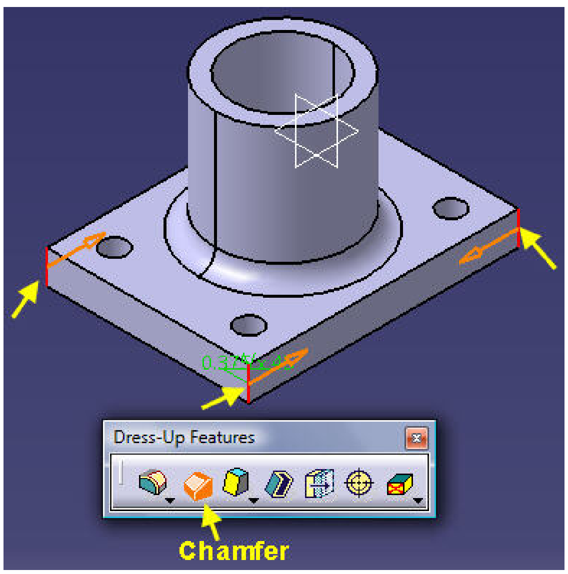
ERASE Planes above.

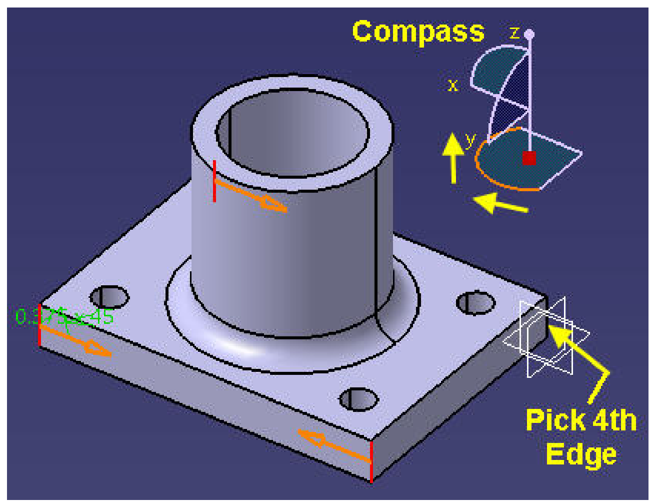
Hold Ctrl key >> Pick each plane >> Hide / Show.



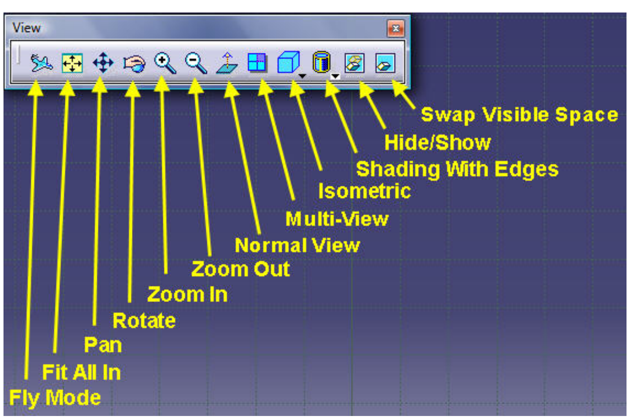


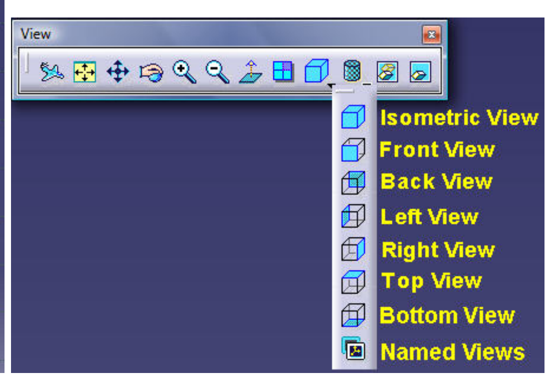
File >> Save As… >> Browse >> Folder >> BASE-M101





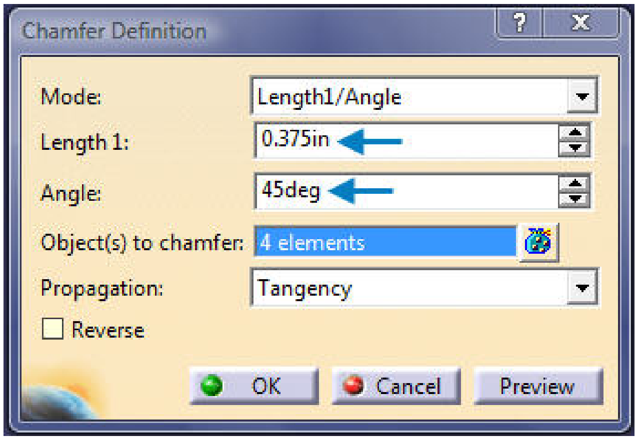
VIEW TOOLBAR

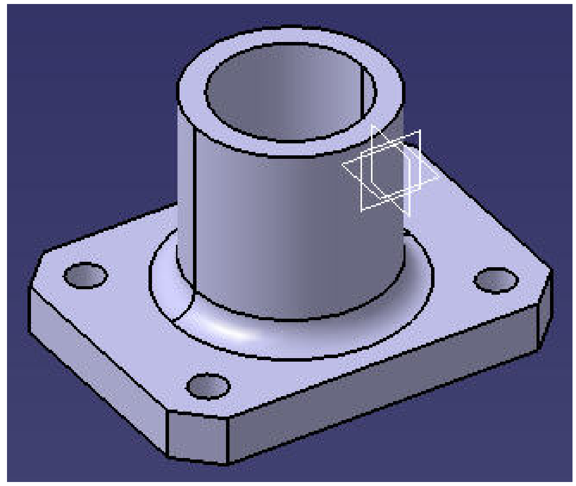




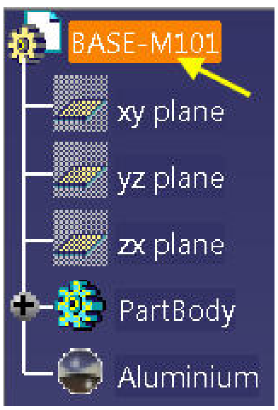
The "View" toolbar icons are listed above. **The "Isometric View" drop-down menu.**

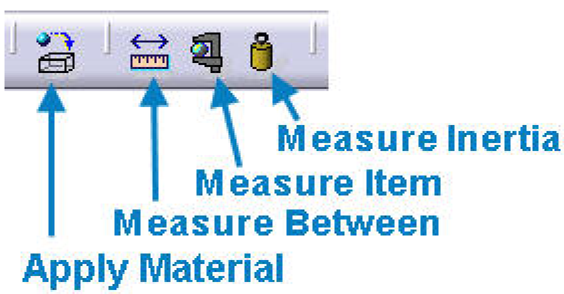
Icons with a "Down Arrow" have additional icons.



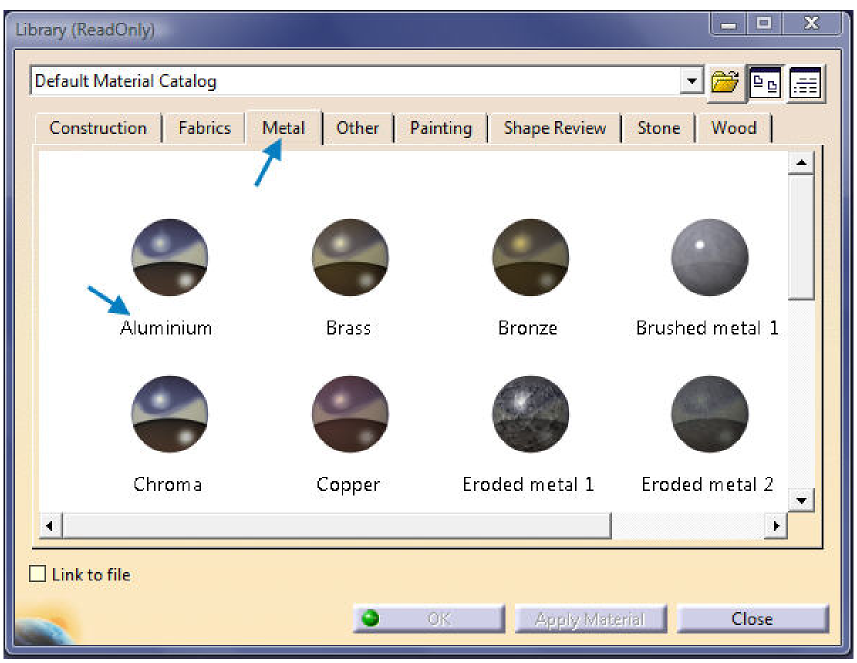


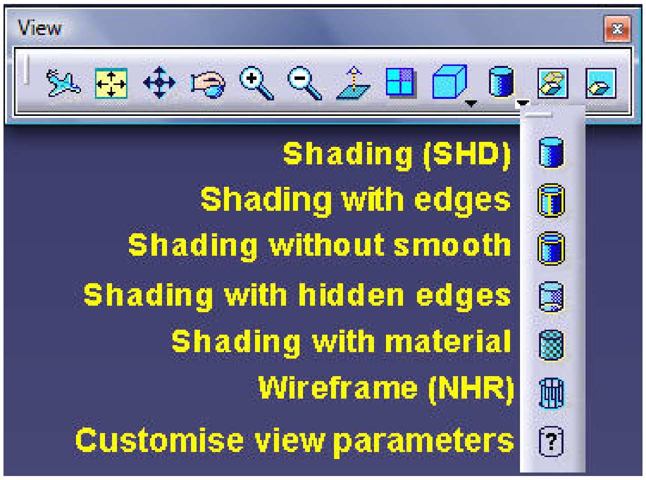
On the "Dress-Up Features" toolbar select: Chamfer >> Length 1: >> 0.375in >> Angle >> 45deg

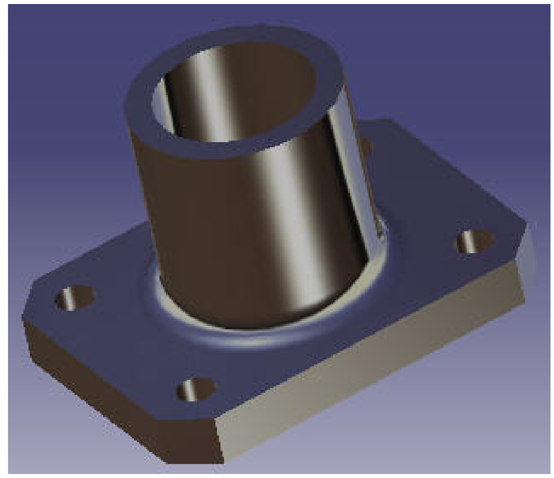




Select tree heading "BASE-M101" >> Apply Material >> Library >> Metal >> Aluminum >> Apply.



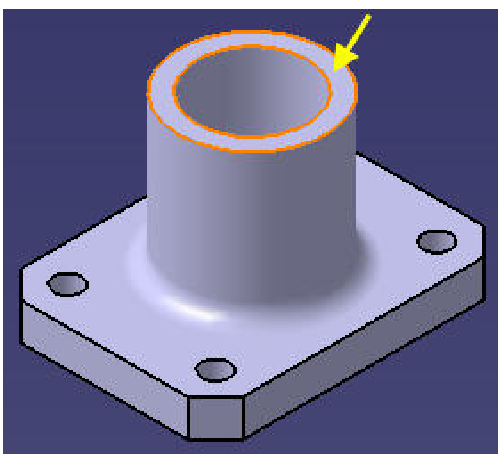




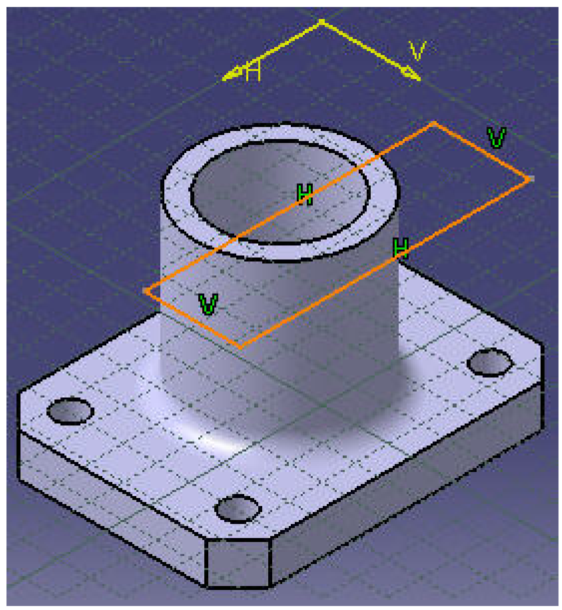
Select “Shading with material” on the “View” toolbar. Result is above right.

POCKET

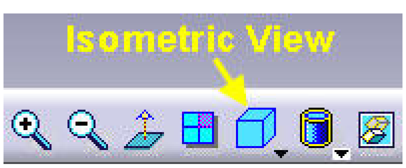
SKETCHING in Isometric View can improve design visualization.



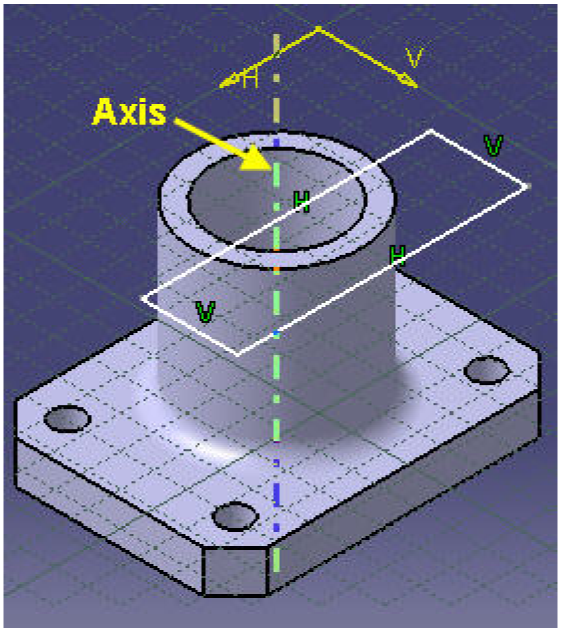
Pick the top surface of the tube.

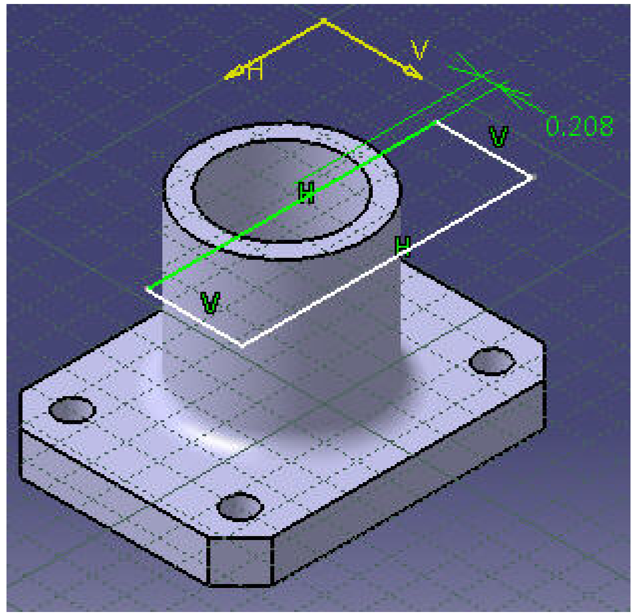


Pick: Sketch tool >> Isomeric View tool >>Rectangle

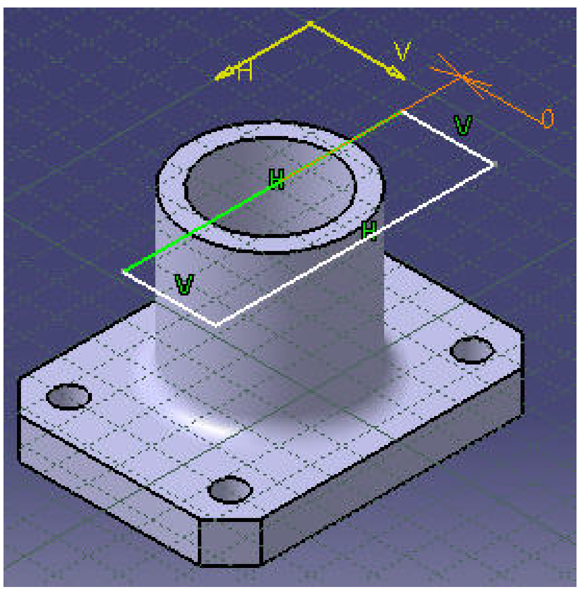


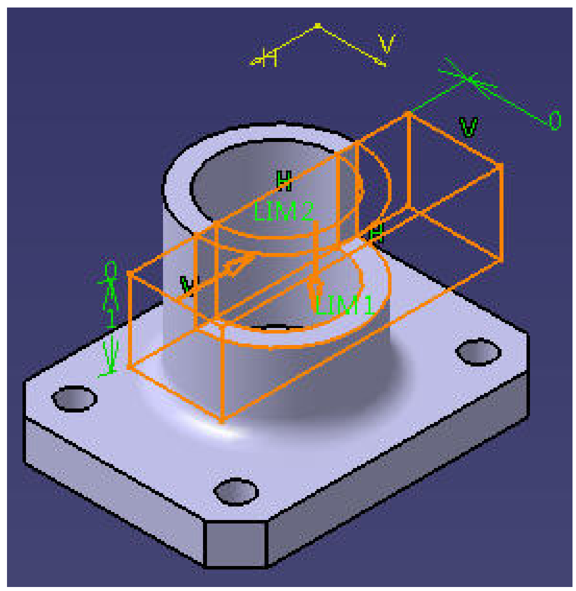
Sketch the rectangle shown above.



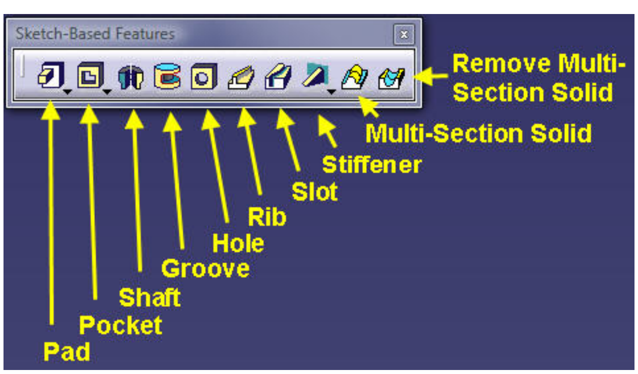


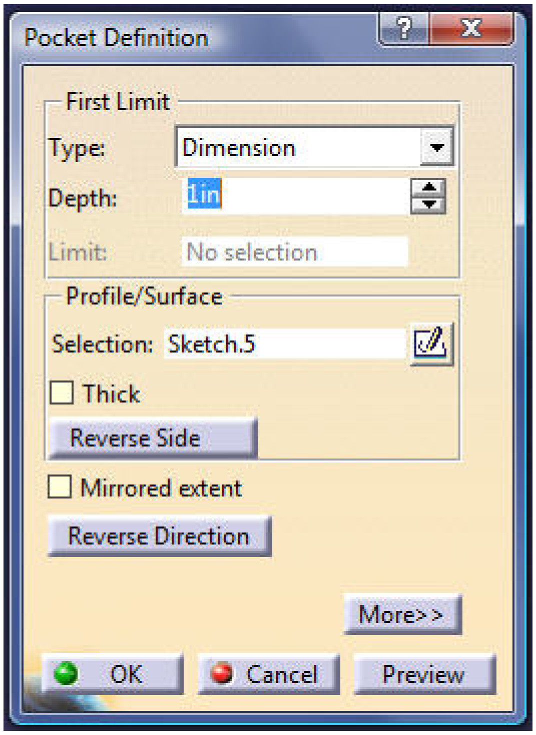
Hover the mouse pointer in the "Axis" area and pick the Axis. Add the dimension to the axis.

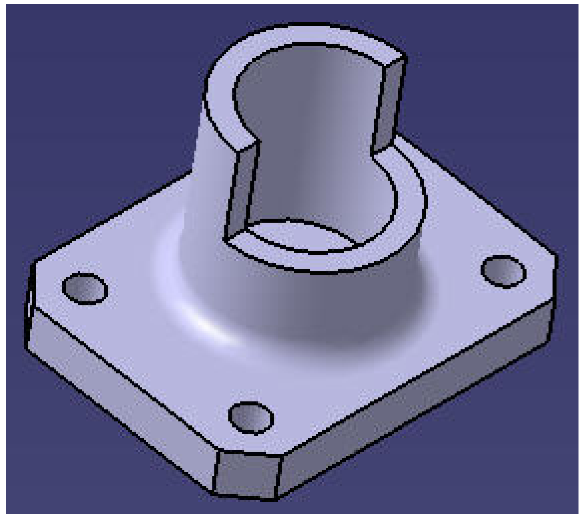




Double-click on the dimension and change it to zero. **Pick the "Pocket" tool shown below.**



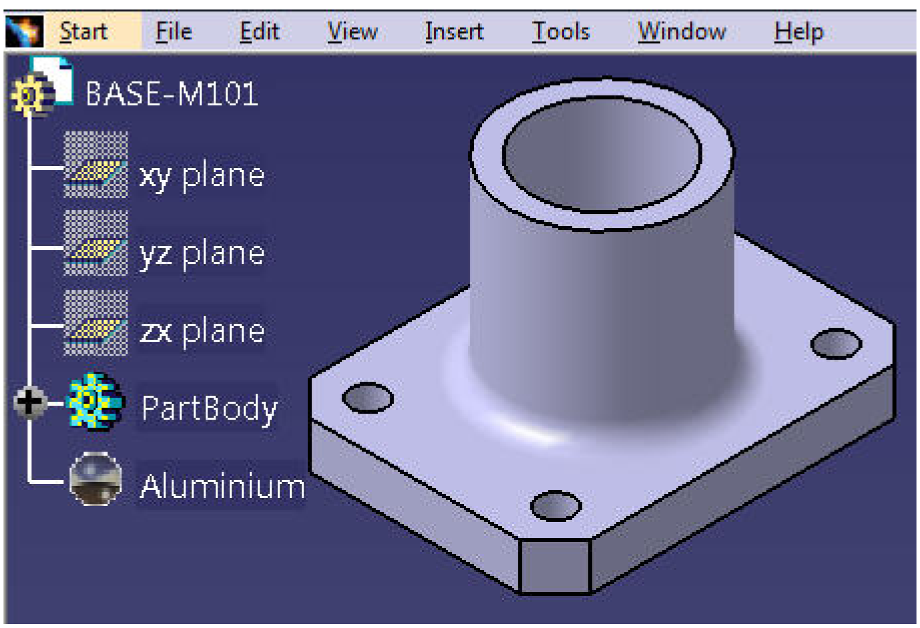




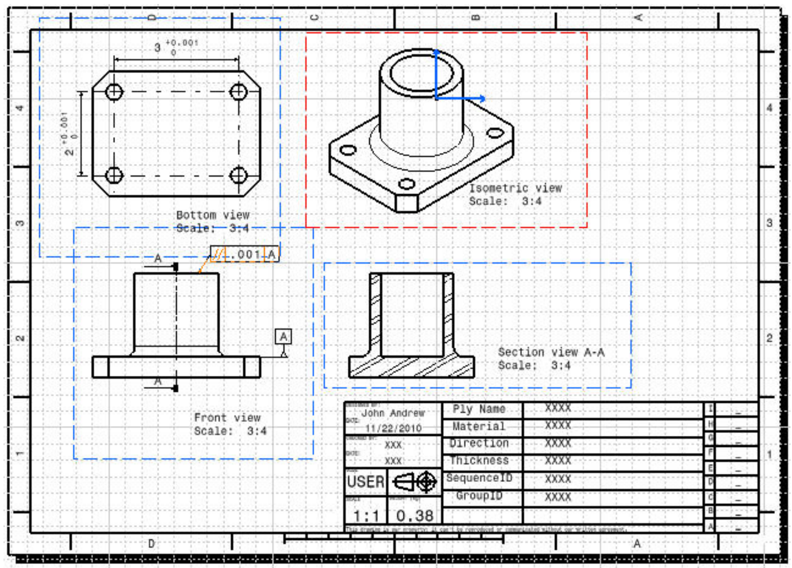
Edit Pocket depth to 1in. **Completed "Pocket".**

3. DIMENSIONED DRAWING

<u>Open a Catia part or assembly before creating the drawing.</u>



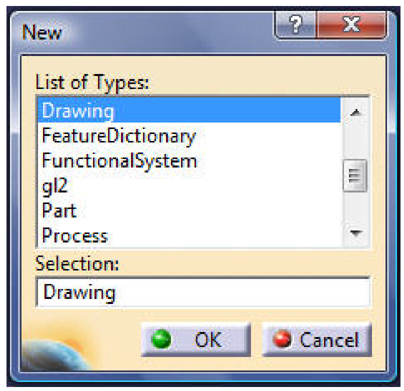
The Catia dimensioned drawing below of the BASE-M101 part above (Without Pocket) will be created as an example.



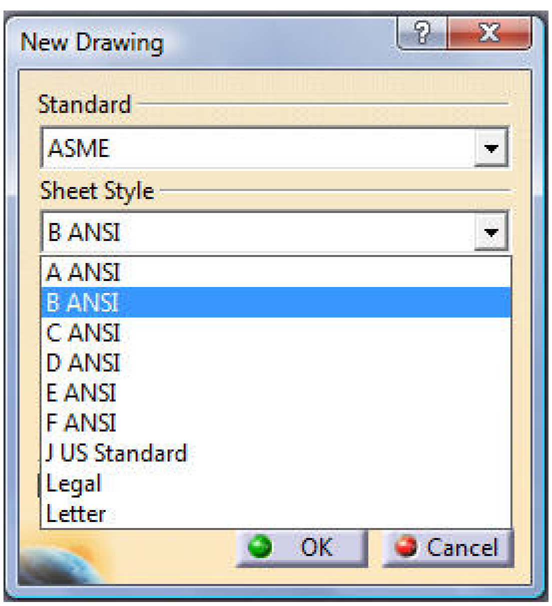
Pick the dropdown menu:

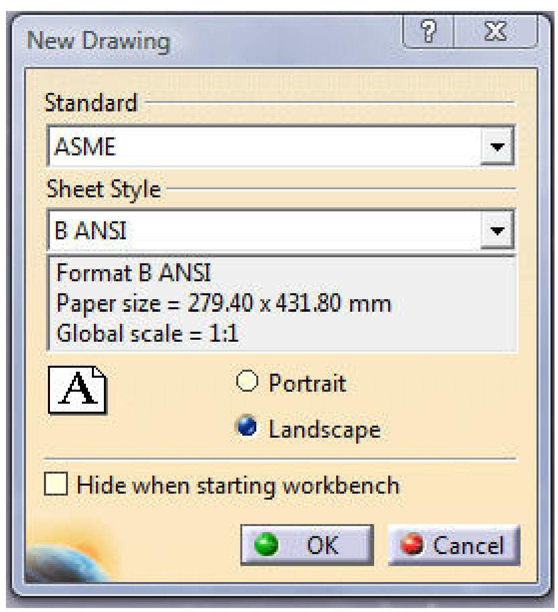
File >> New >> Drawing

The "New" box (right) will open.

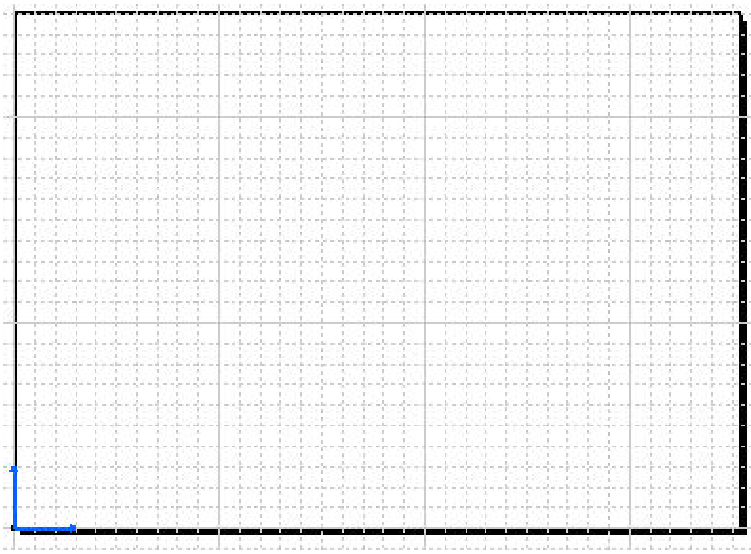


In the "New" box (above) select: Drawing

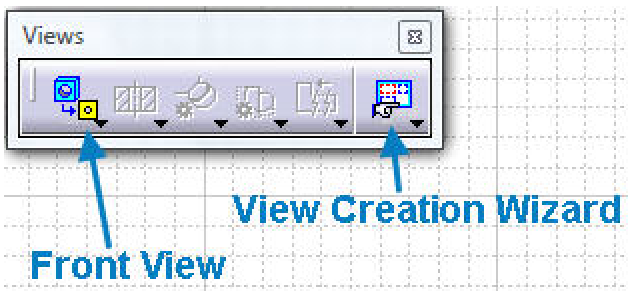




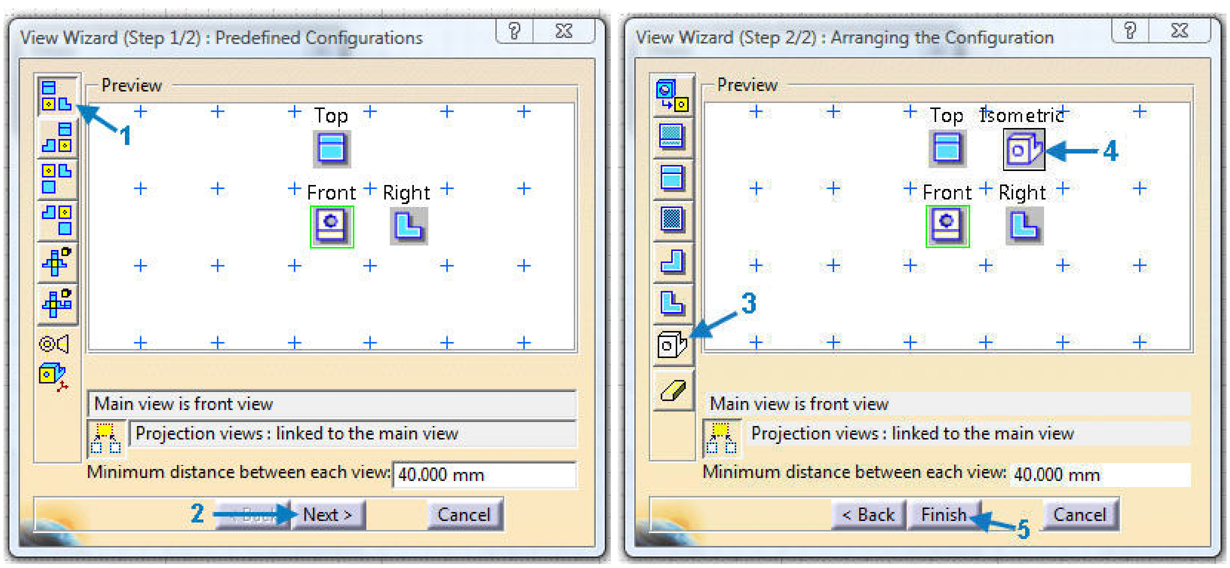
Pick the "Sheet Style": ANSI, ASME, ISO, or other drawing format >> OK. "New Drawing" >> OK.



A blank drawing will open as shown above.

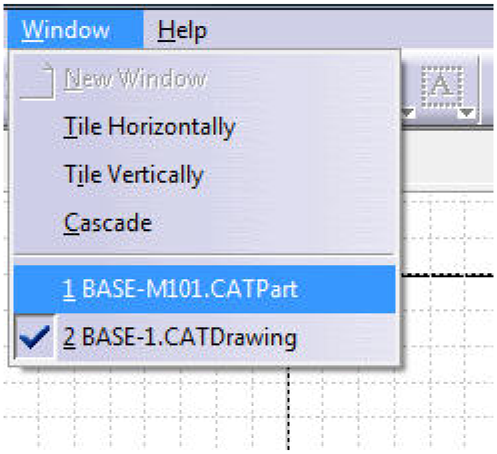


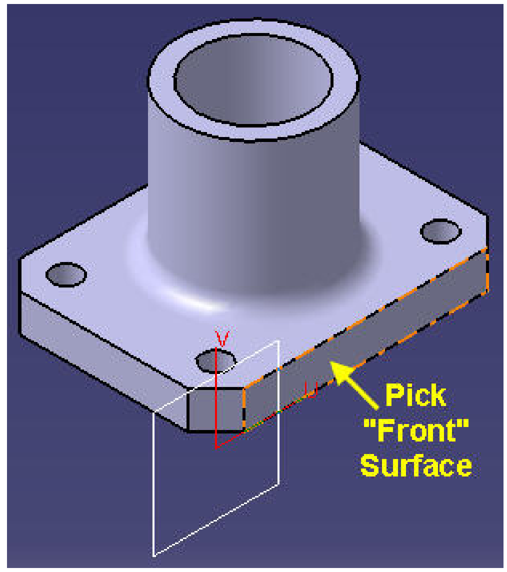
On the “Views” toolbar select, “View Creation Wizard”.



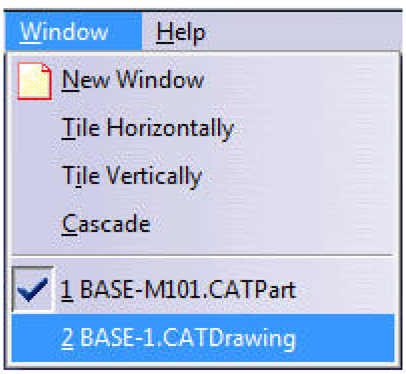
Pick-1 (3 Views) >> Next-2 >> Pick-3 (Iso View) >> Place (Iso View) at 4 >> Finish-5.

The drawing remains blank.

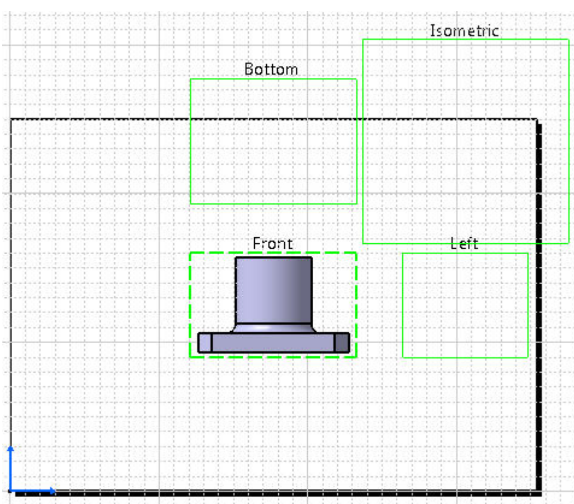


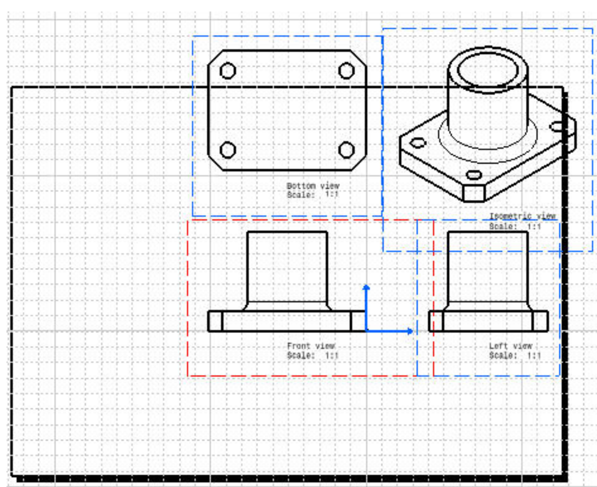


Pick: Window >> BASE-M101.CATPart. The part will open. Pick the “Front” surface.



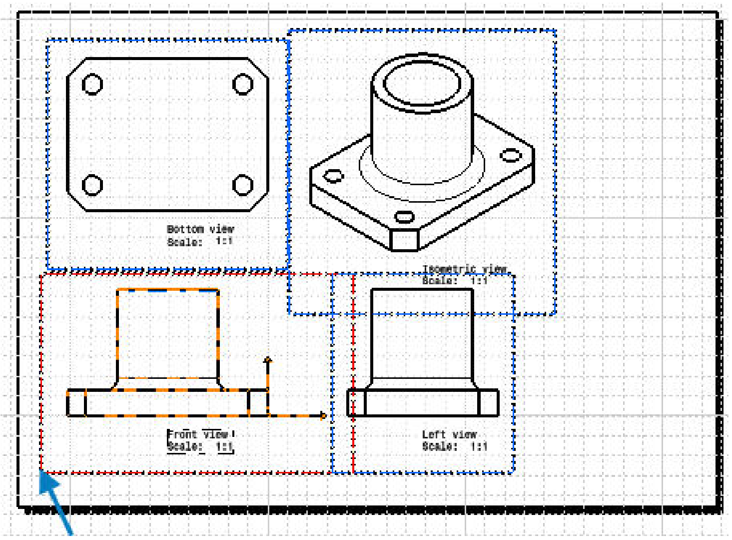
Pick: Window >> BASE-1.CATDrawing.





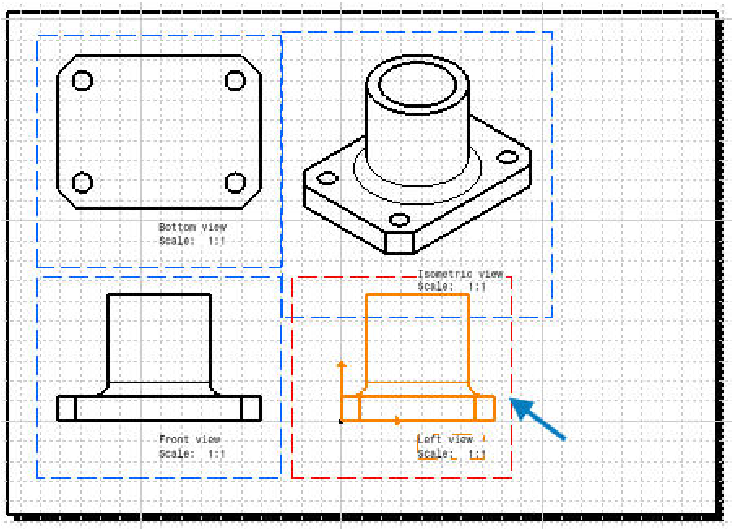
Catia places the “Front” view in the center of the drawing.

Click on the front view and all views appear.

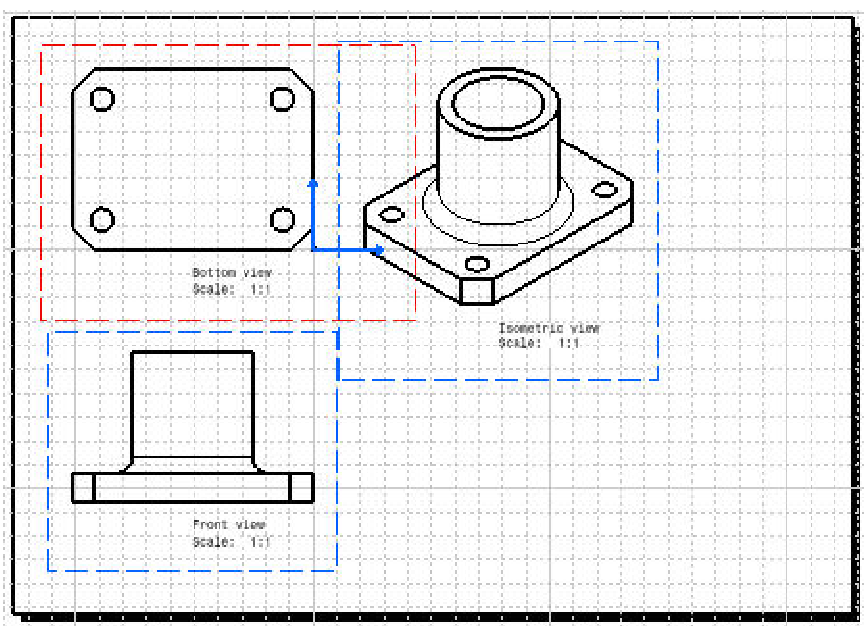


Pick an edge of the Front View and drag all views into the drawing, left.

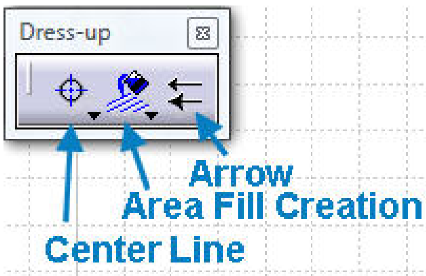
File >> Save As: >> Browse Files >> BASE-M101.CATDrawing.

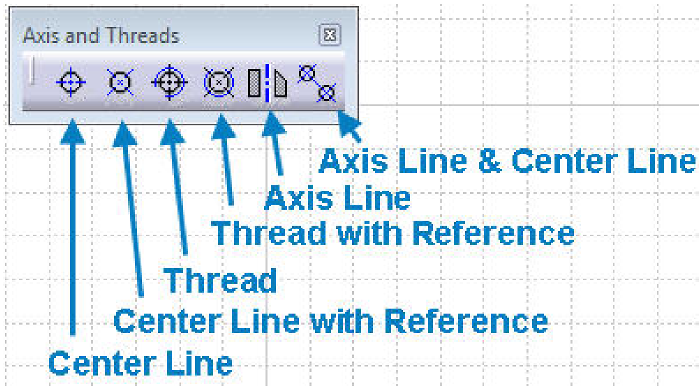


Double click on an edge of the Left View to make it the active view.



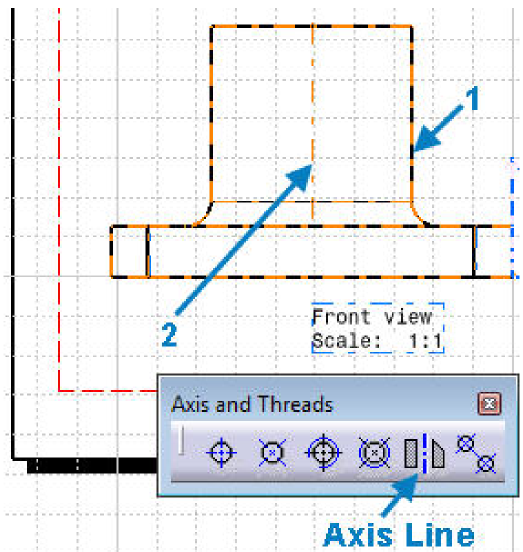
Press the delete key to remove the left view as shown above.





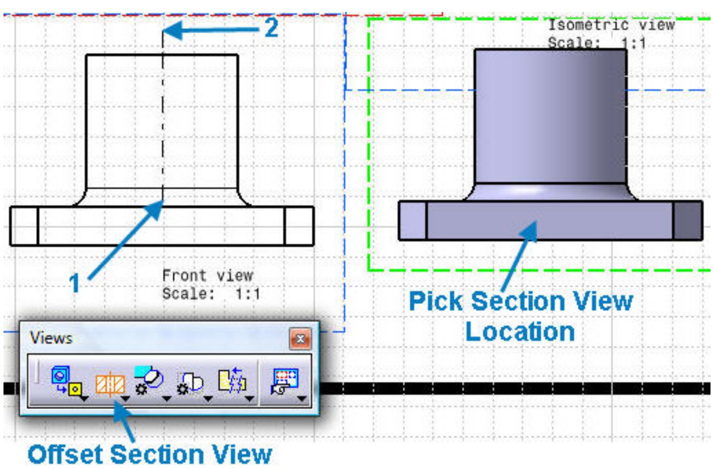
The "Dress-up" toolbar.

The "Axis and Threads" toolbar.



Double click the dashed line boarder of the front view to make it active (orange).

Pick the: Axis Line icon >> Edge-1 >> Part center line-2 will be added by Catia.



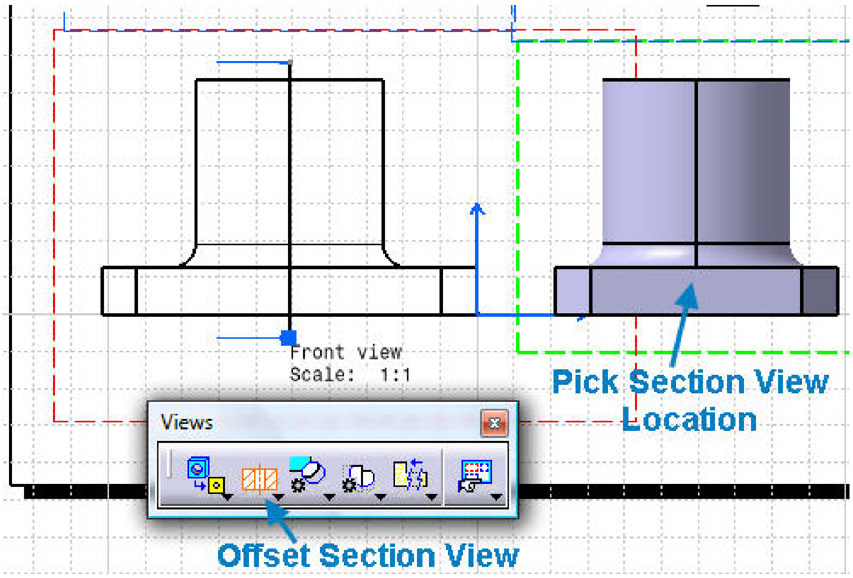
On the "Views" toolbar pick the:

"Offset Section View" icon >> Pick

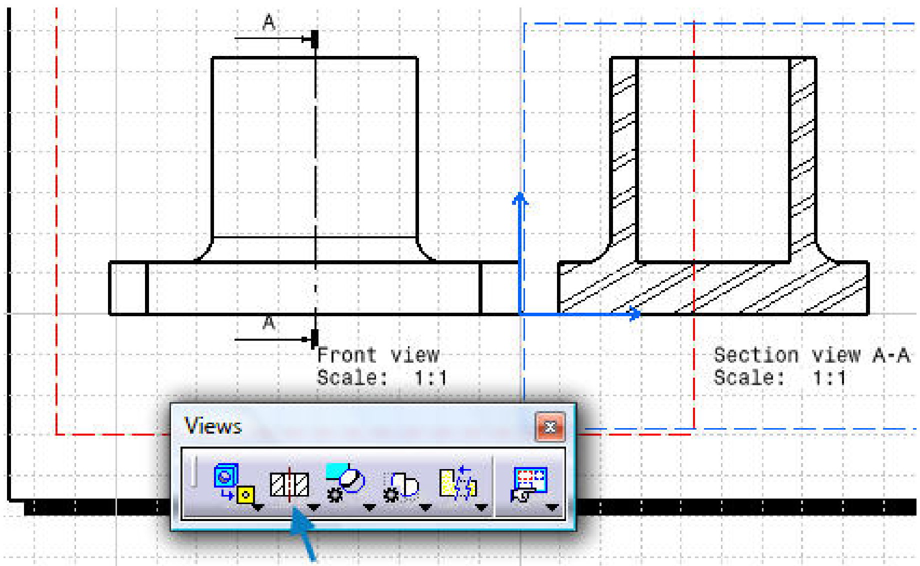
section line starting point-1 >>

Drag to section line end point-2 >>

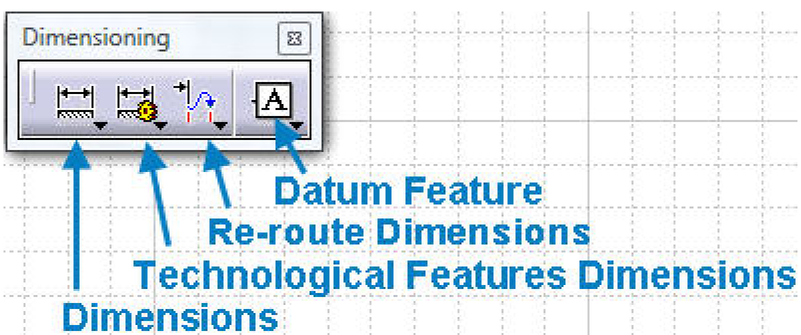
Double Click.



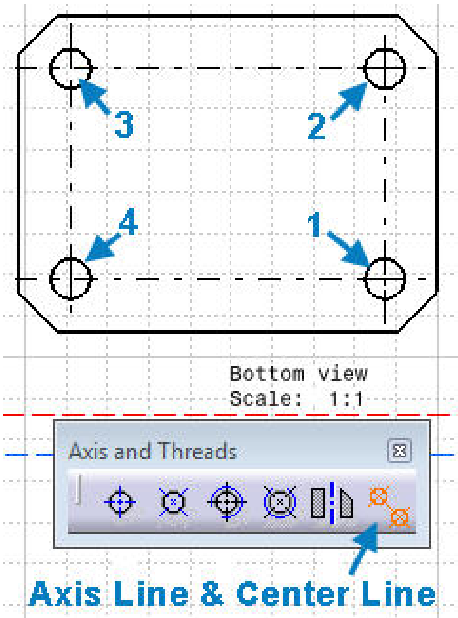
Pick the "Section View" location shown upper right.



Click on the "Section View" in the drawing and Catia will finish the section view (upper right).



The "Dimensioning" toolbar is shown above.

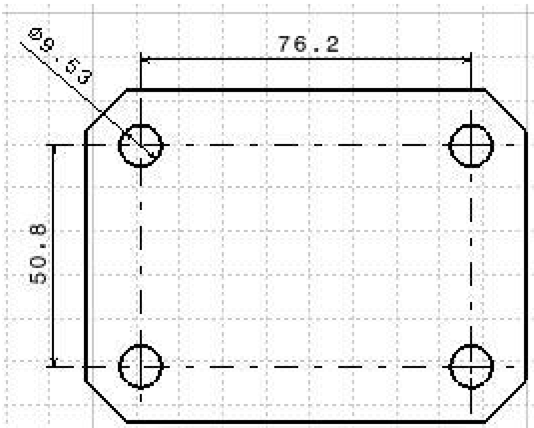


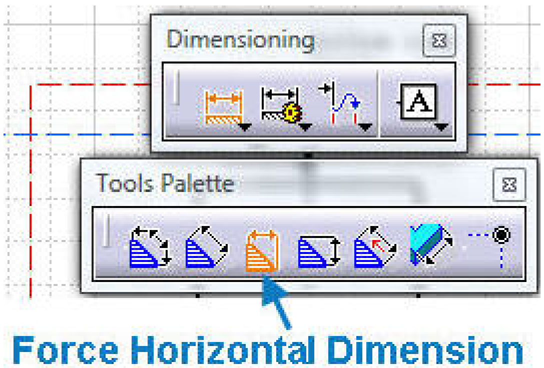
Create the hole centers and center lines between holes as shown above.

On the "Axis and Threads" toolbar double click the: Axis Line & Center Line icon >>

Pick circle-1 >> Pick circle-2 >>
Pick circle-2 >> Pick circle-3 >>
Pick circle-3 >> Pick circle-4 >>

Pick circle-4 >> Pick circle-1.





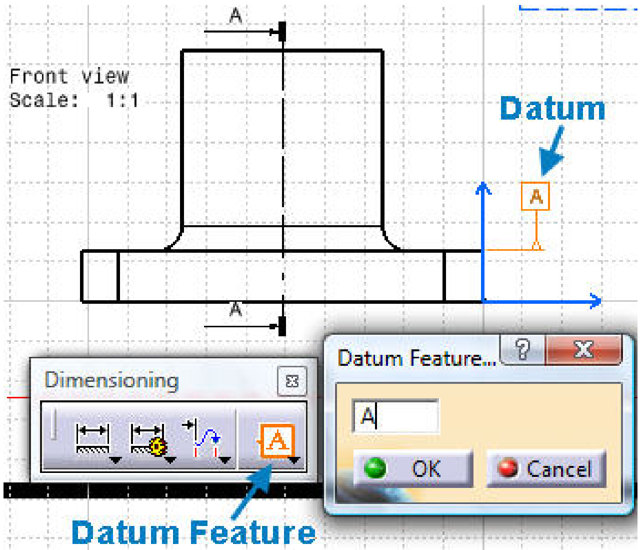
On the "Dimensioning" toolbar pick: Dimension >> Force Horizontal Dimension >> Pick the left and right vertical centerlines >> Place the 76.2 mm dimension.

Continue adding dimensions.

All dimensions will be converted from inch to millimeters below.

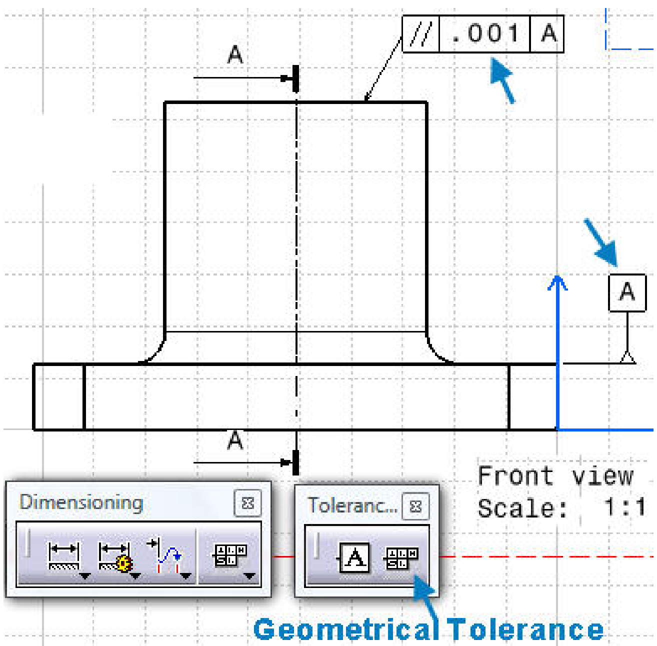
GEOMETRICAL TOLERANCES

Select the Geometrical Tolerance icon.

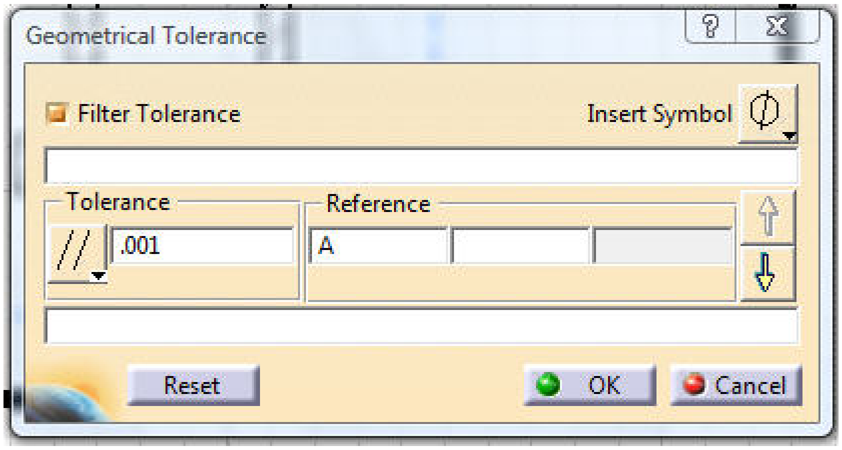


On the "Dimensioning" toolbar pick the: Datum Feature icon.

The datum letter (A) can be changed >> OK.



Next pick the "Geometrical Tolerance" icon as shown above.

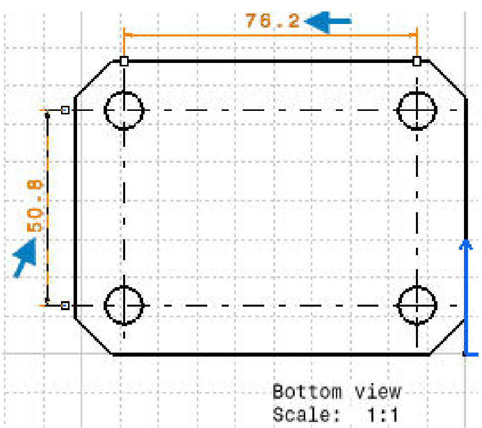


The “Geometrical Tolerance” box will open.

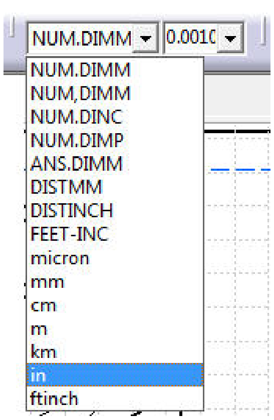
The “Parallel Symbol” has been selected from the drop down menu and the Tolerance has been set to .001 inch.

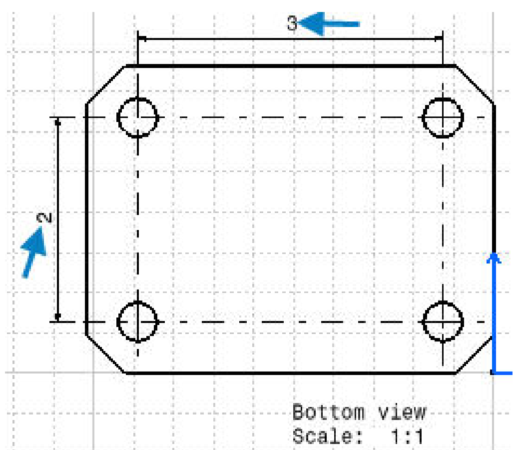
The reference letter (A) has been typed in the appropriate box >> OK.

CONVERT MILLIMETERS TO INCHES



Hold the Ctrl key down and pick each dimension needing to be changed from mm to inches.

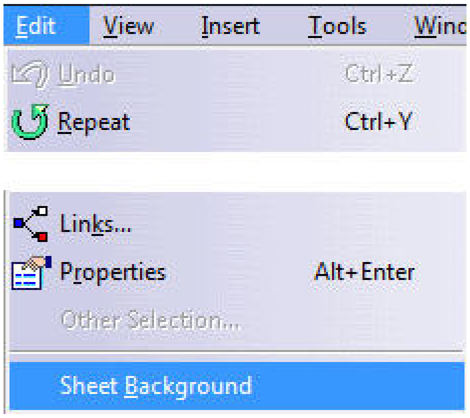




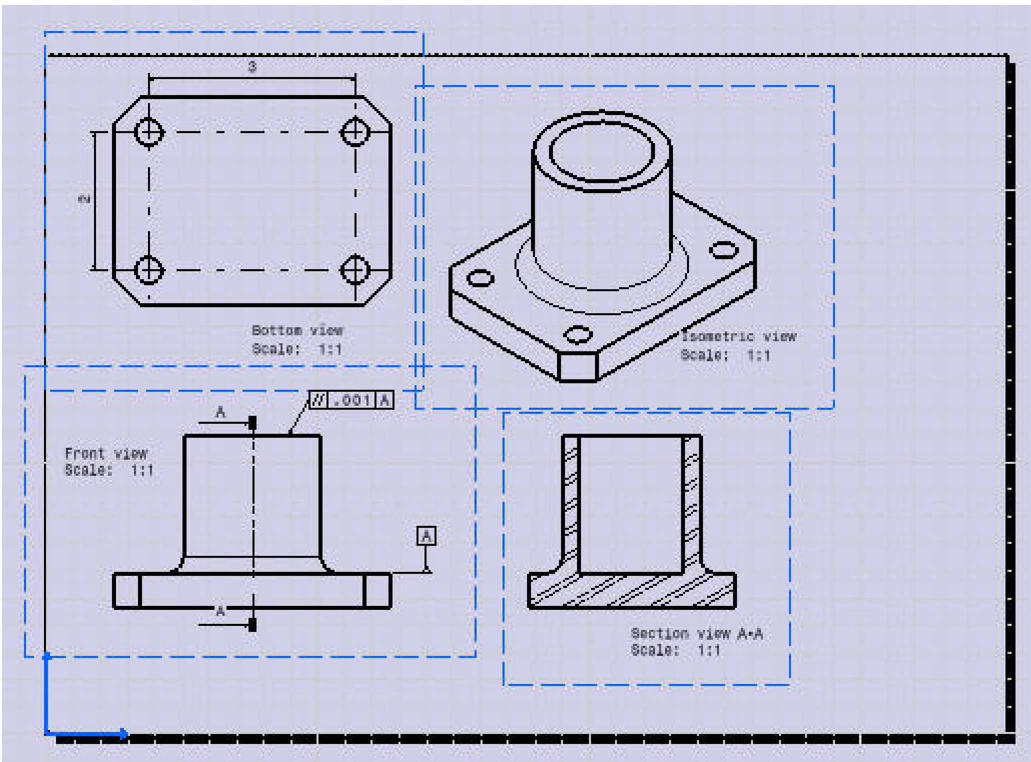
Pick the “NUM.DIMM” drop down menu >> Select “in”.

The selected dimensions changed from 50.8 and 76.2 mm to 2 and 3 inches respectively.

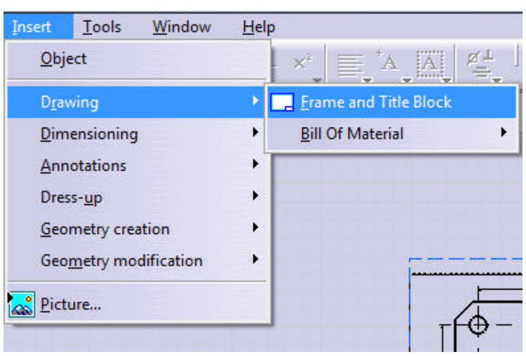
DRAWING SHEET BACKGROUND



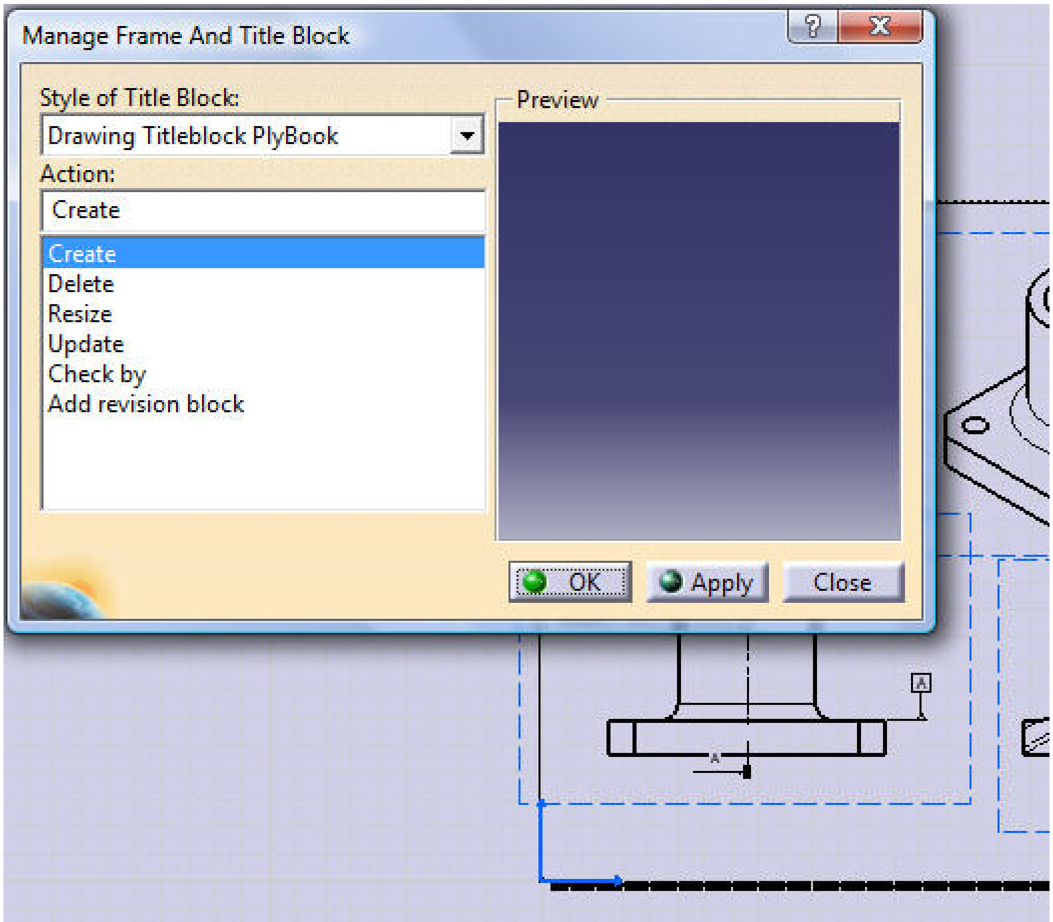
Pick: Edit >> Sheet Background.



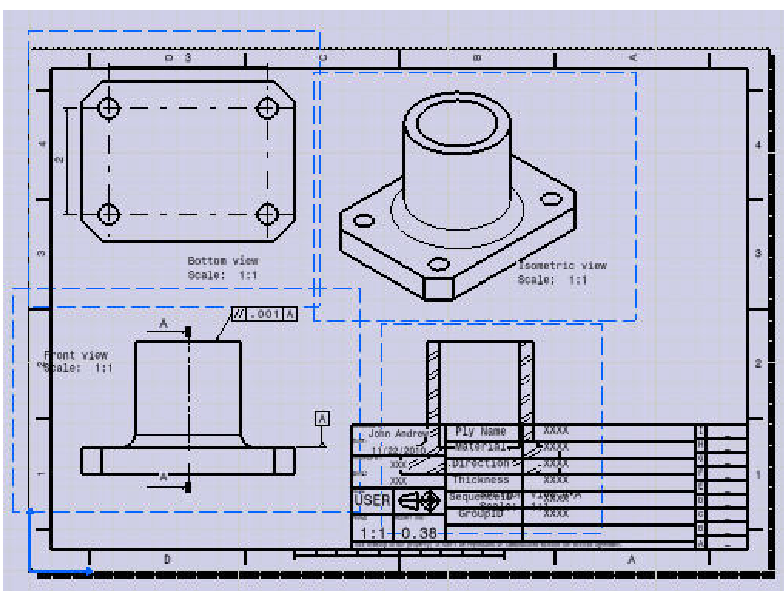
The grey color indicates sheet background.



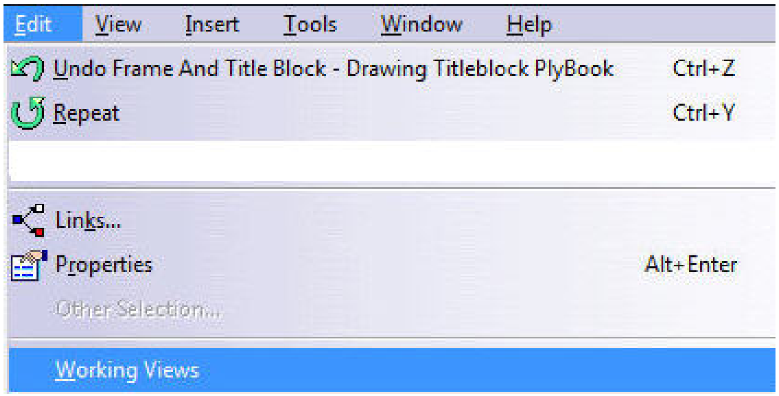
Pick: Insert >> Drawing >> Frame and Title Block.



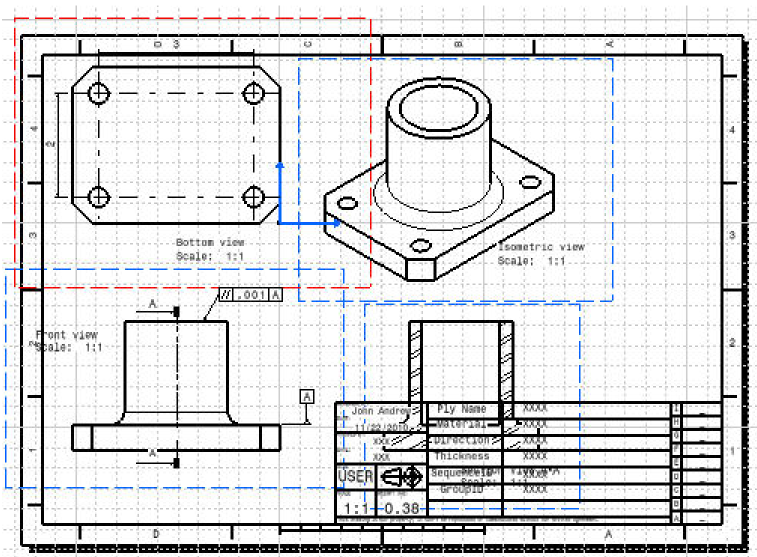
Pick: Create >> Apply >> OK.



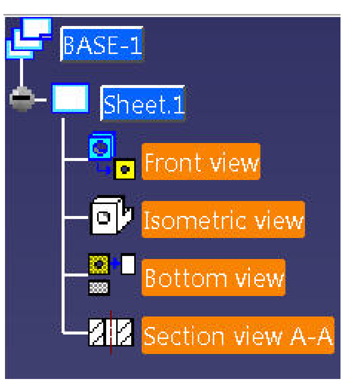
The Frame and Title Block are inserted but the views do not fit in the drawing blank area.

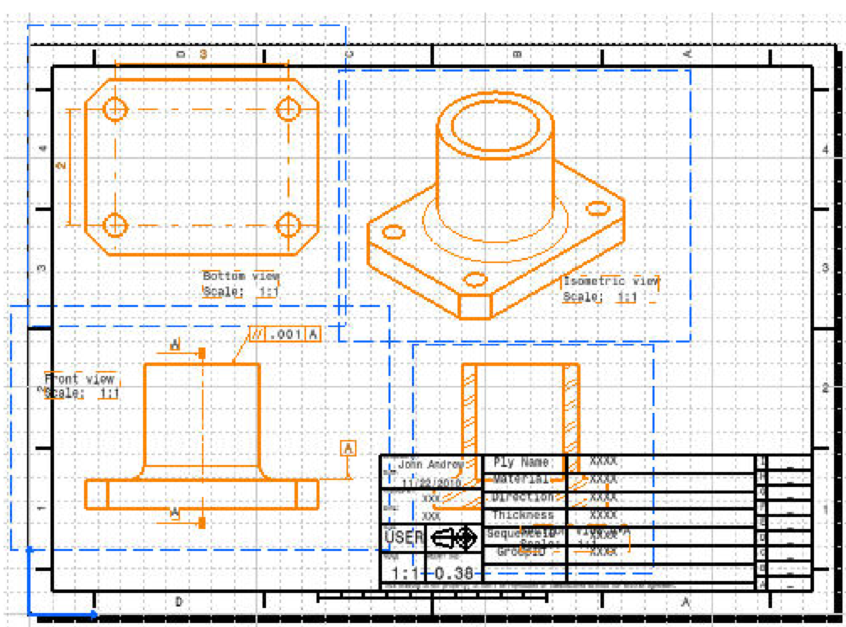


Pick: Edit >> Working Views >> see how the drawing has changed below.



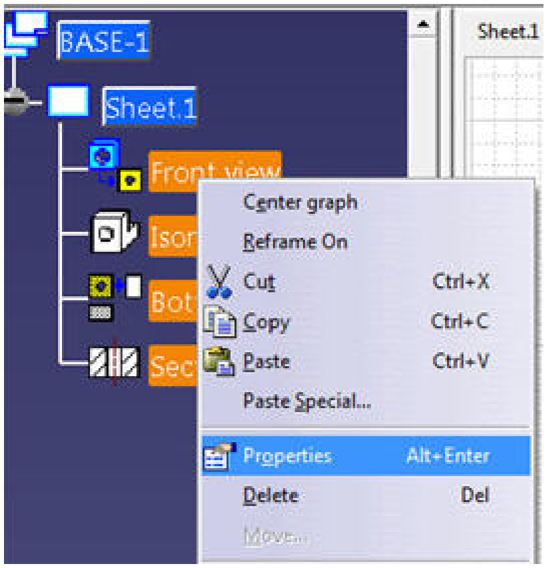
SCALE VIEWS





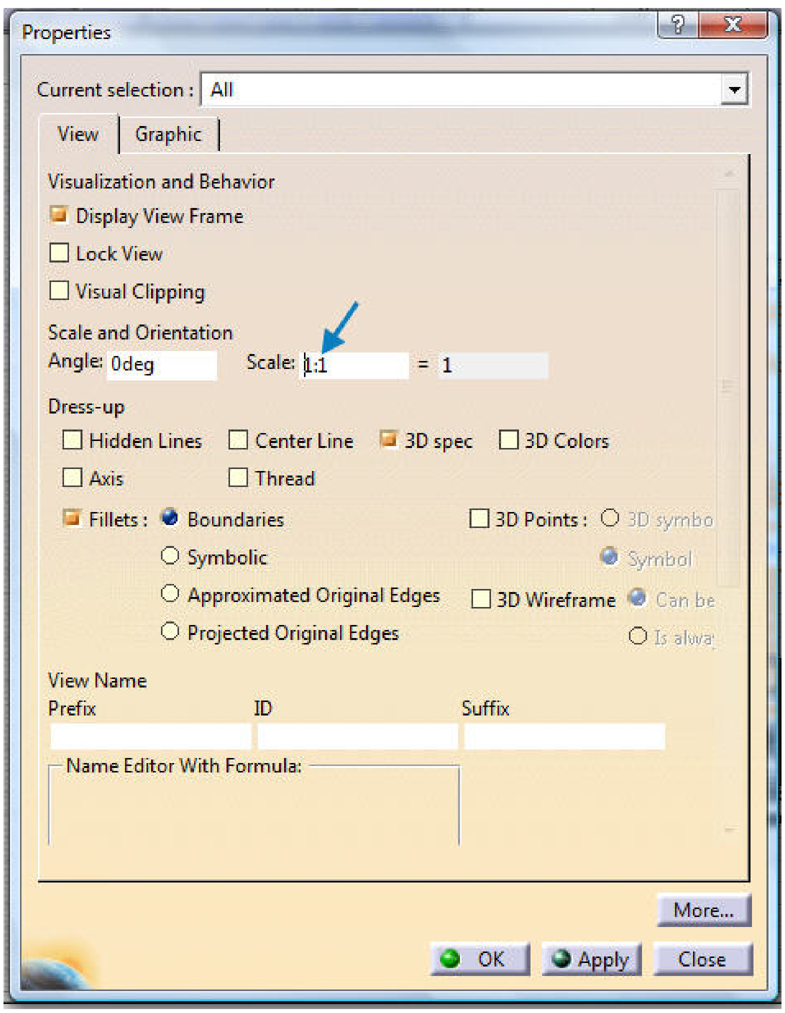
Hole the Ctrl key and pick each view needing to have its scale changed.

The selected views change color to orange.

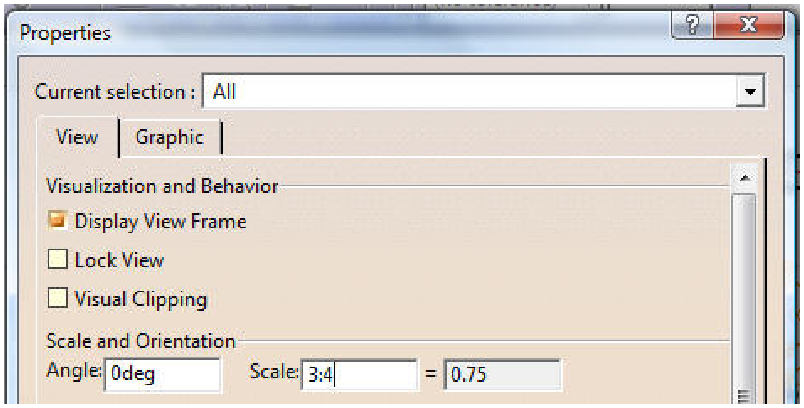


Right click on one of the selected views >> Select "Properties".

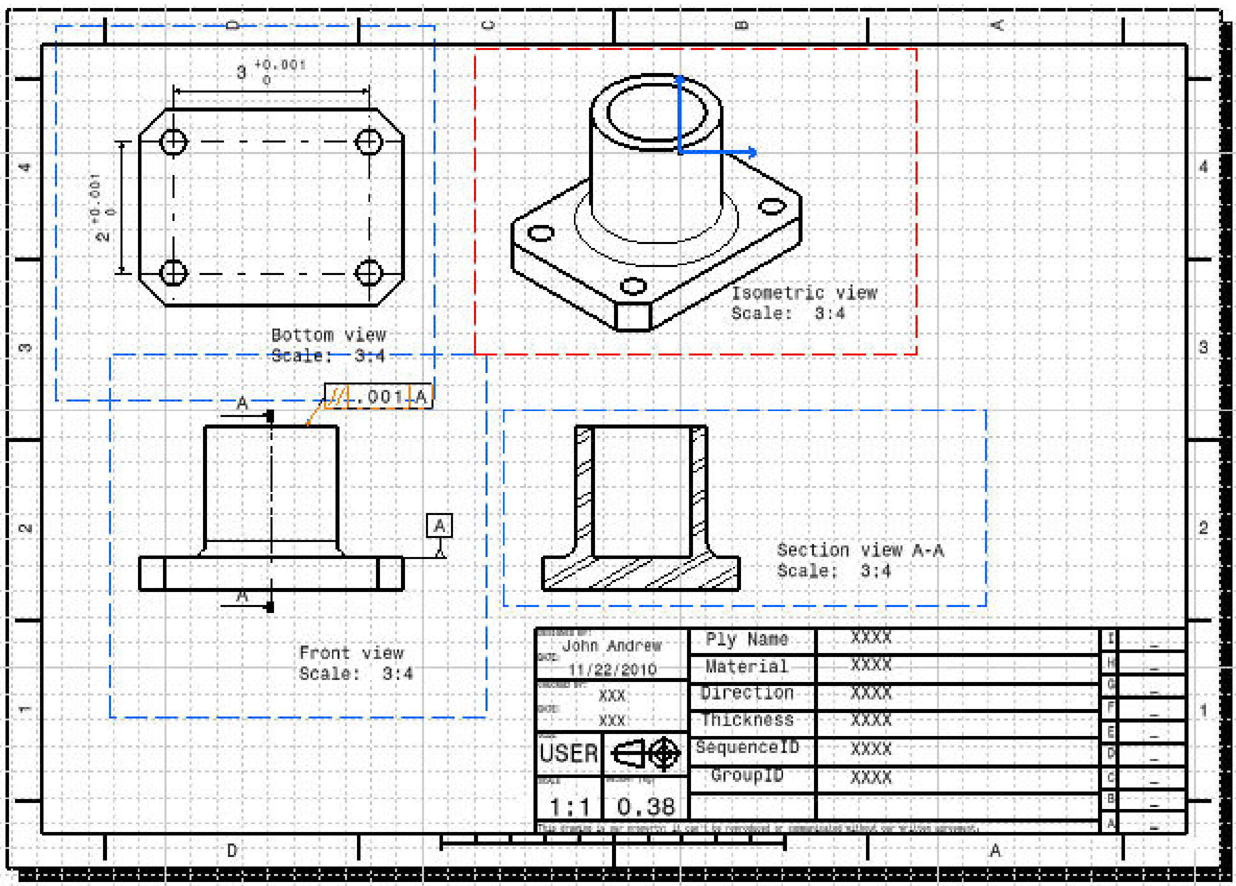
The "Properties" box will open as shown below.



The Scale is 1:1 or full size.



Type: “3/4” or 3:4 to change the scale to 0.75.



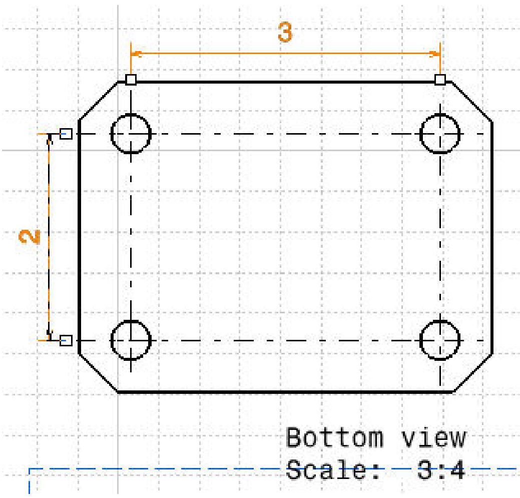
Now the views fit in the drawing.

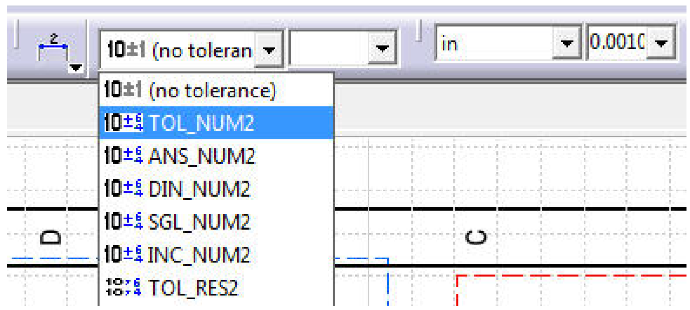
DIMENSIONAL TOLERANCES

Geometric tolerance is defined as the total amount that the dimension of a manufactured part can vary.

With increasing accuracy or less variation in the dimension, the labor and machinery required to manufacture a part is more cost intensive.

Any manufacturer should have a thorough knowledge of the tolerances to increase the quality and reliability of a manufactured part with the least expense possible.

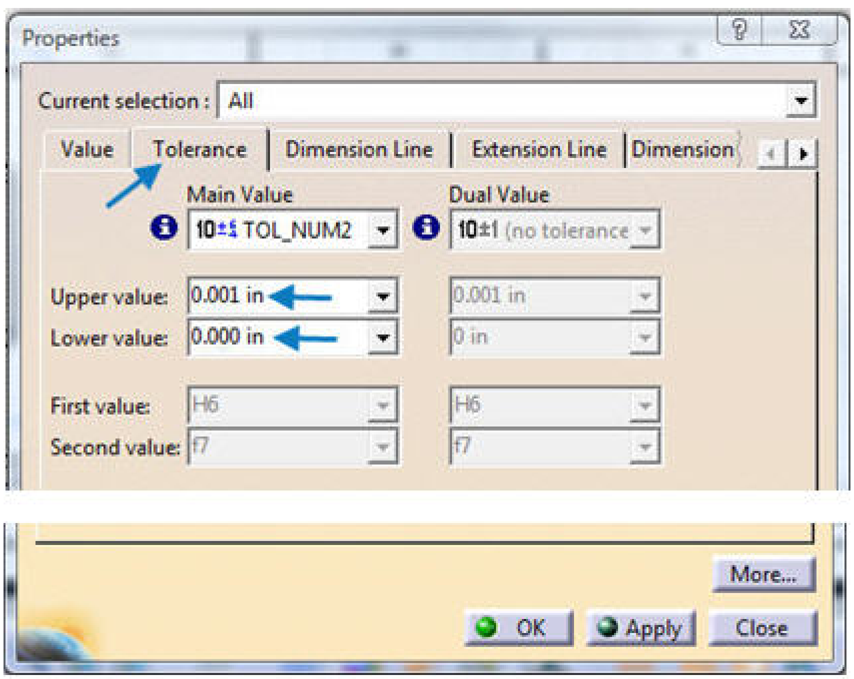




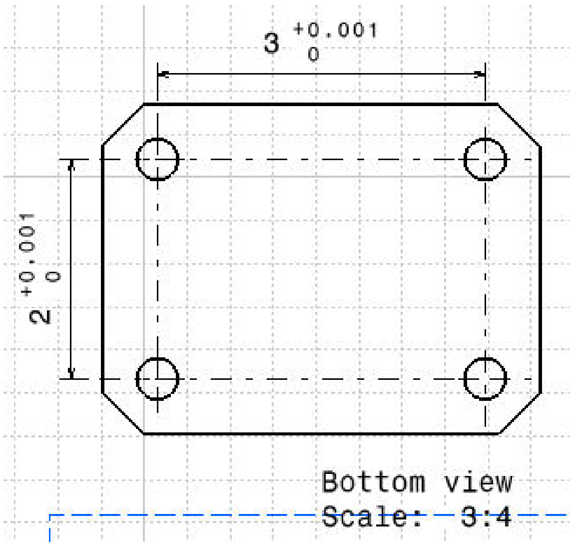
Hold: Ctrl and pick dimensions.

Pick “10± (no tolerance)” menu >> 10± ¾ TOL_NUM2”.

The “Properties” box will open.



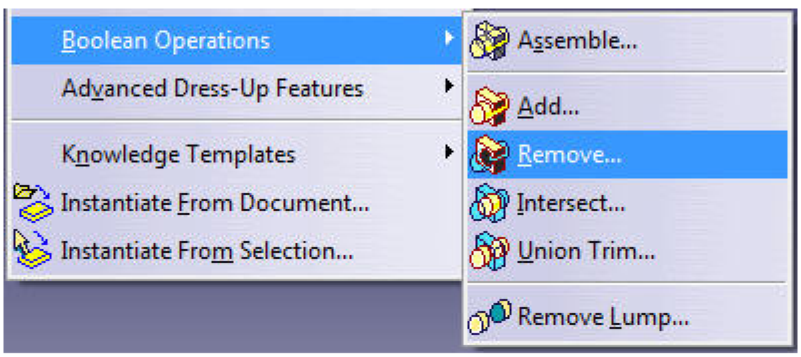
In the “Properties” box pick the “Tolerance” tab and edit the Upper and Lower values.



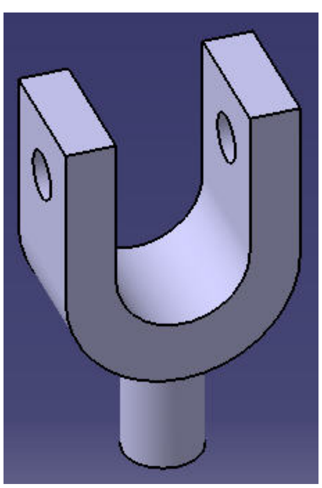
The dimensional tolerances are inserted by Catia.

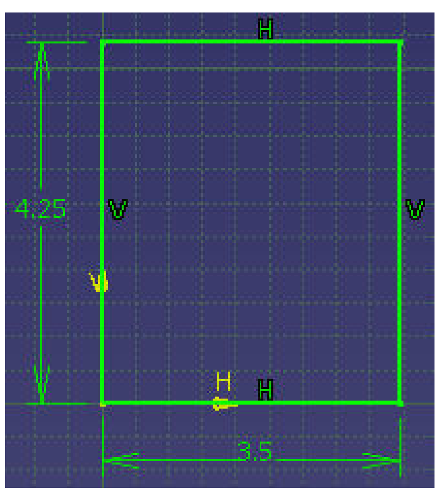
4- BOOLEAN PART MODELING

U-JOINT PART EXAMPLE



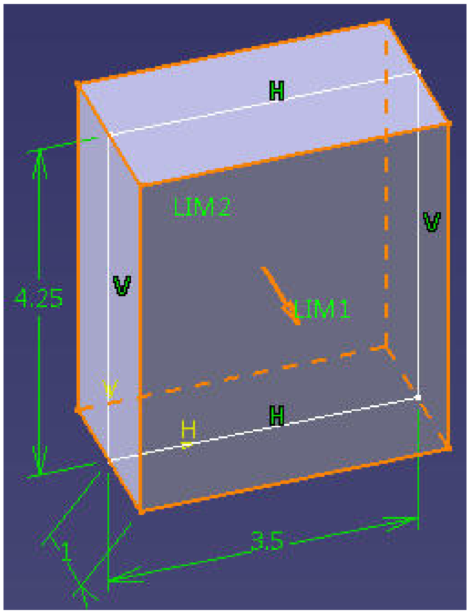
Boolean Operations tools will be used to create a part.

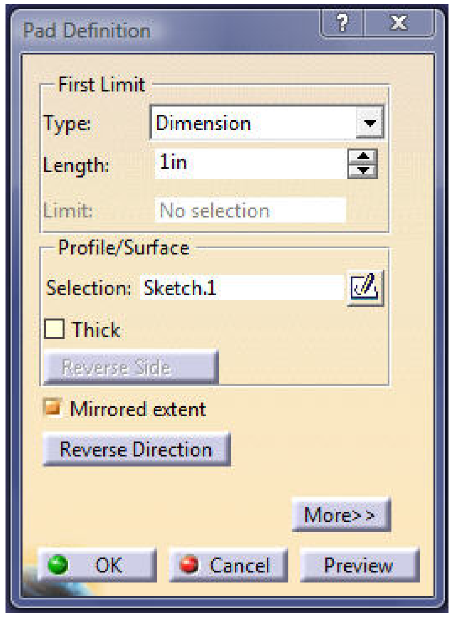




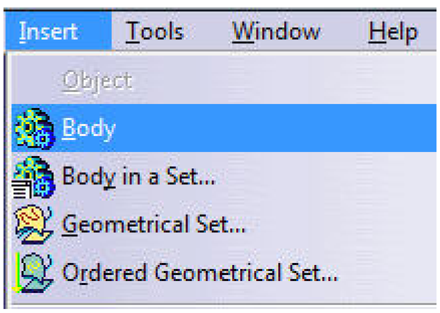
The U-Joint part above will be used as an example.

1. Sketch the above rectangle.

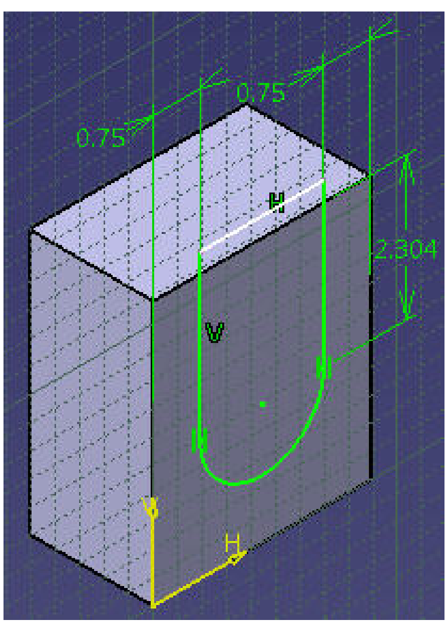


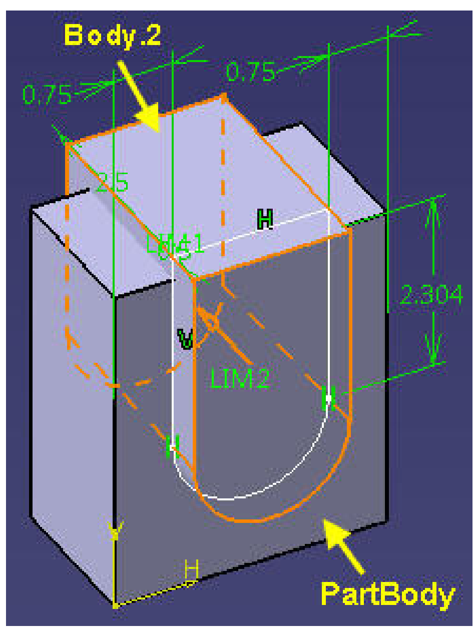


2. Pad the rectangle.

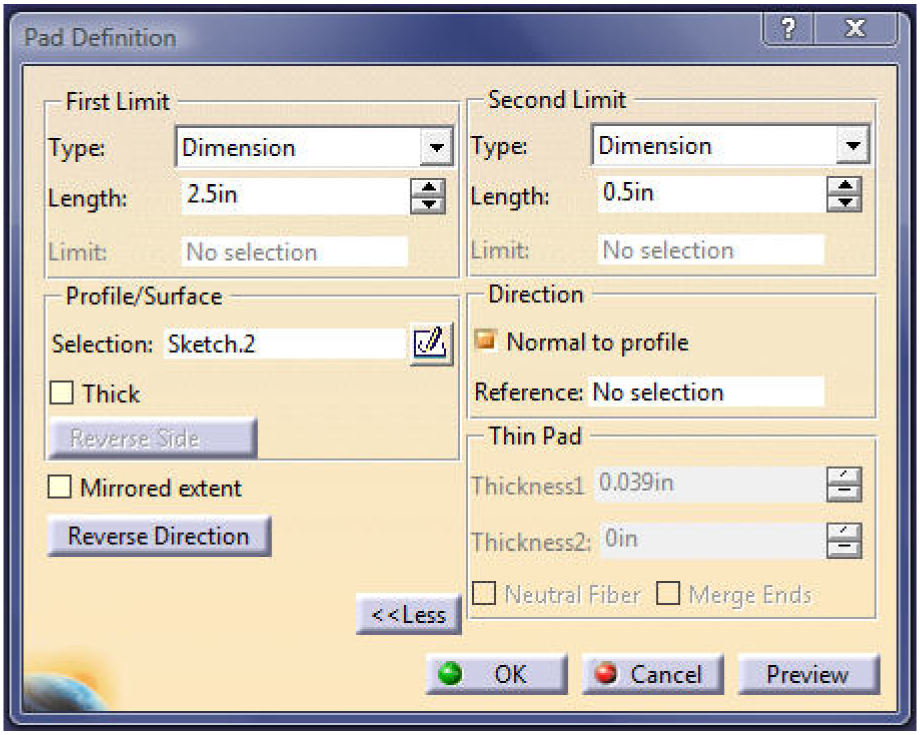


3. Insert >> Body >> Sketch

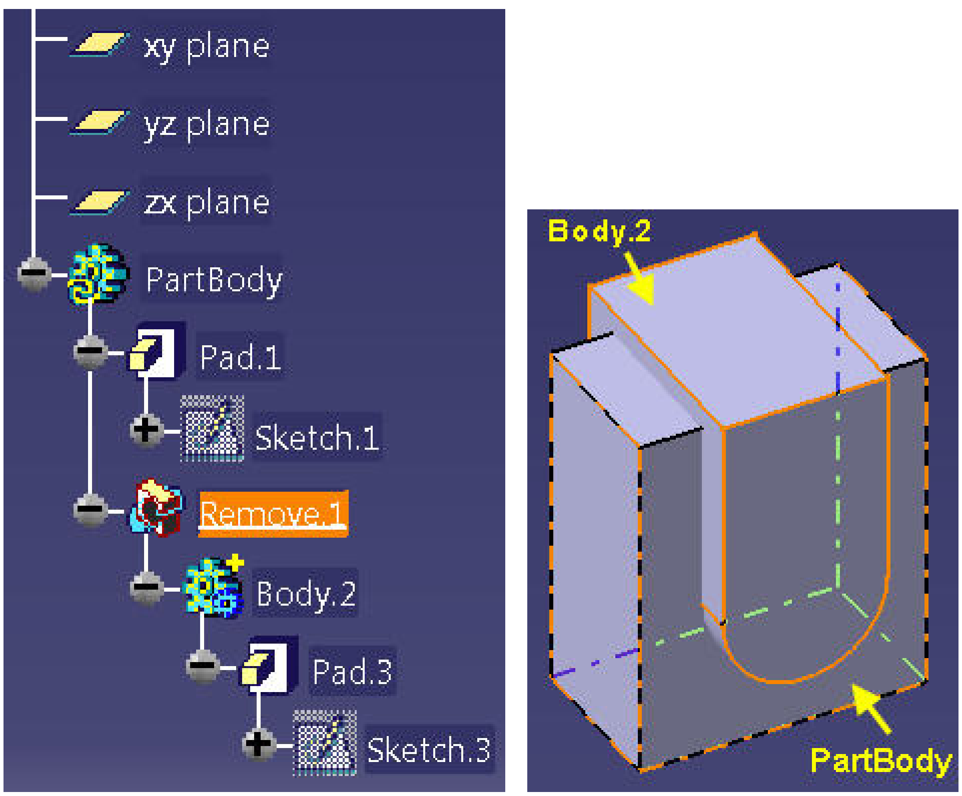




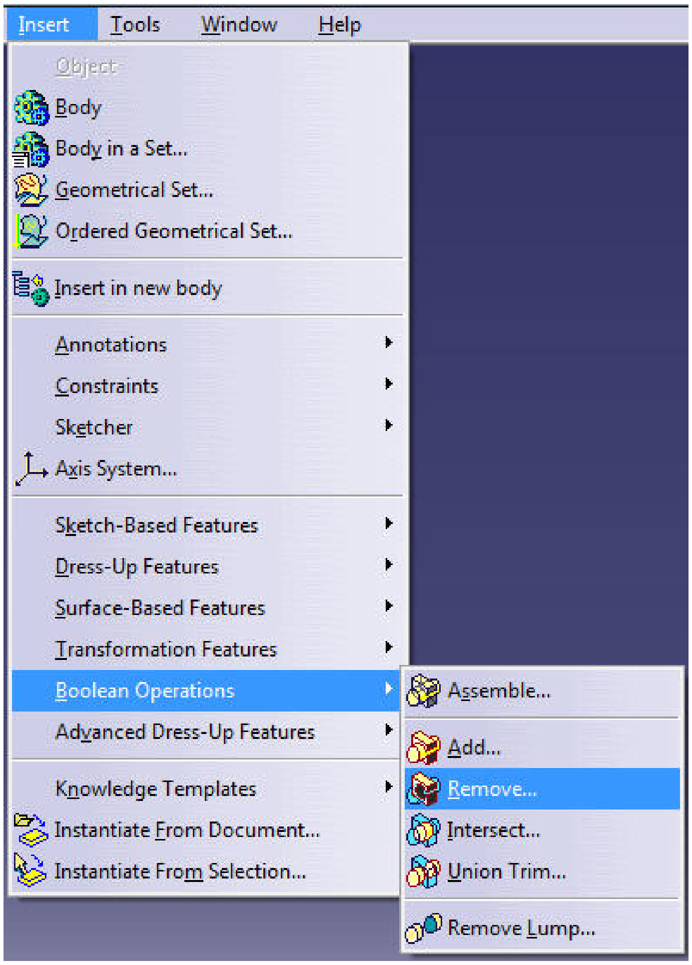
4. Pick on the front surface of “PartBody” >> Sketch >> Pad “Body.2”

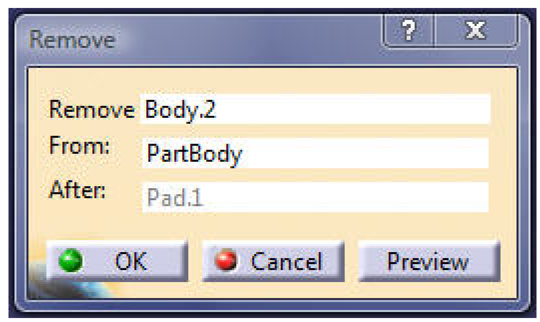


5. "Body.2" Pad definition



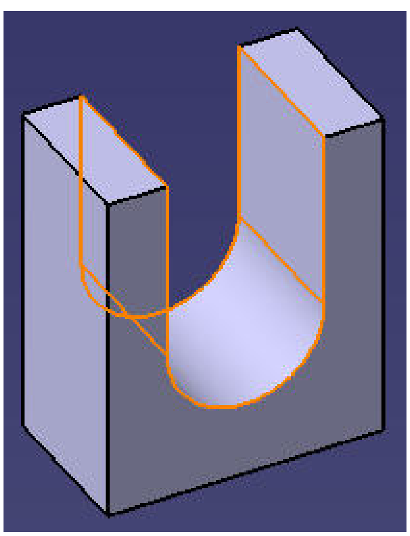
6. The Specification Tree shows "PartBody" and "Body.2"

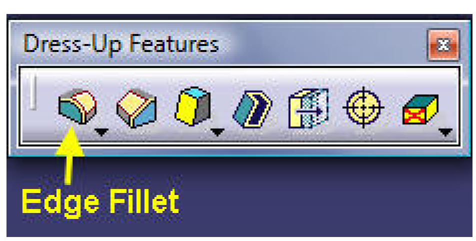


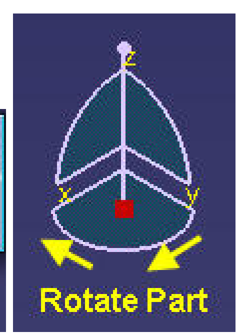


7. Insert >> Boolean Operation >> Remove. The "Remove" box will open.

8. Remove >> pick "Body.2" From >> pick "PartBody" >> OK.

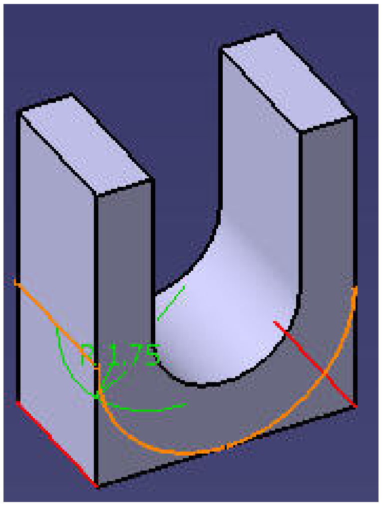


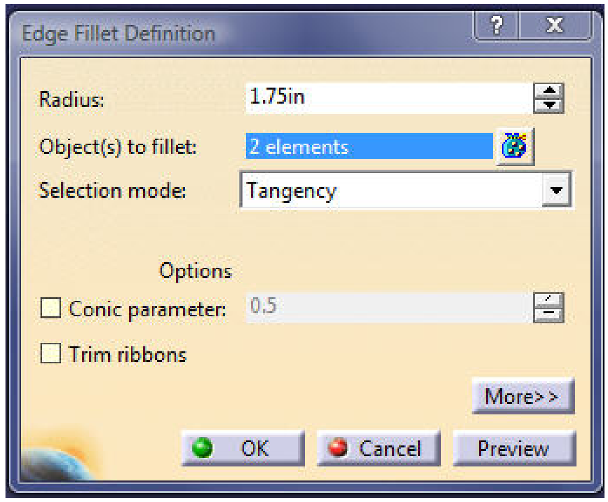




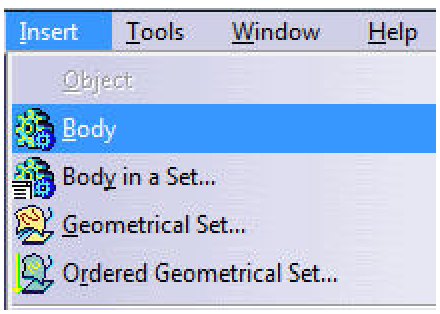
9. Body.2 has been removed from :PartBody".

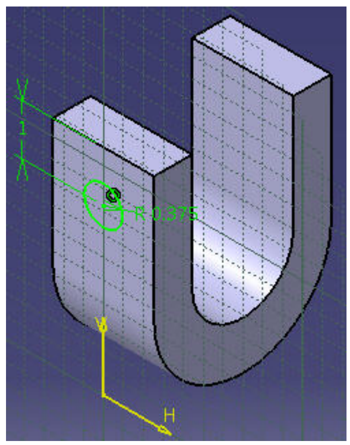
10. Pick the "Edge Fillet" tool.

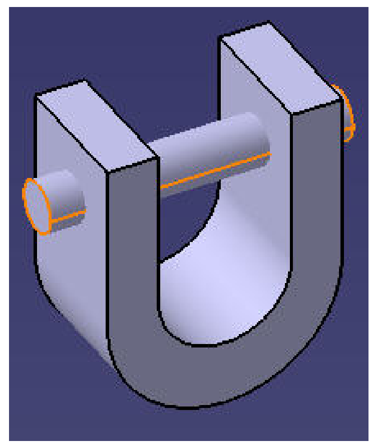




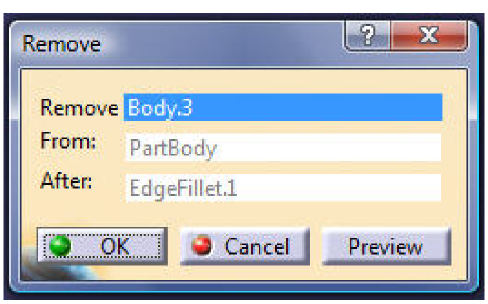
11. Pick the two bottom edges (red). Use the Compass to rotate the part.

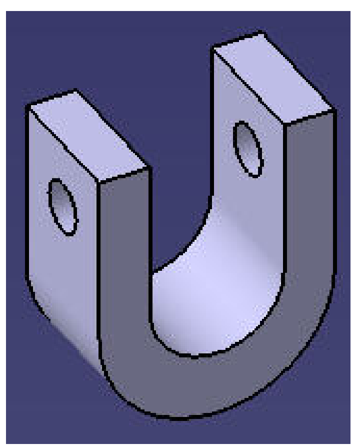




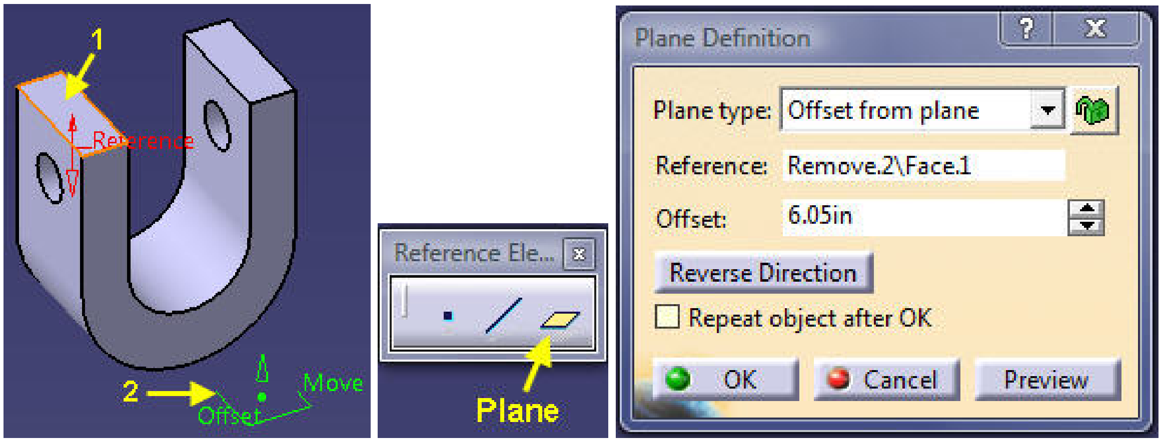


12. Insert >> Body >> Sketch the circle >> Pad the circle.

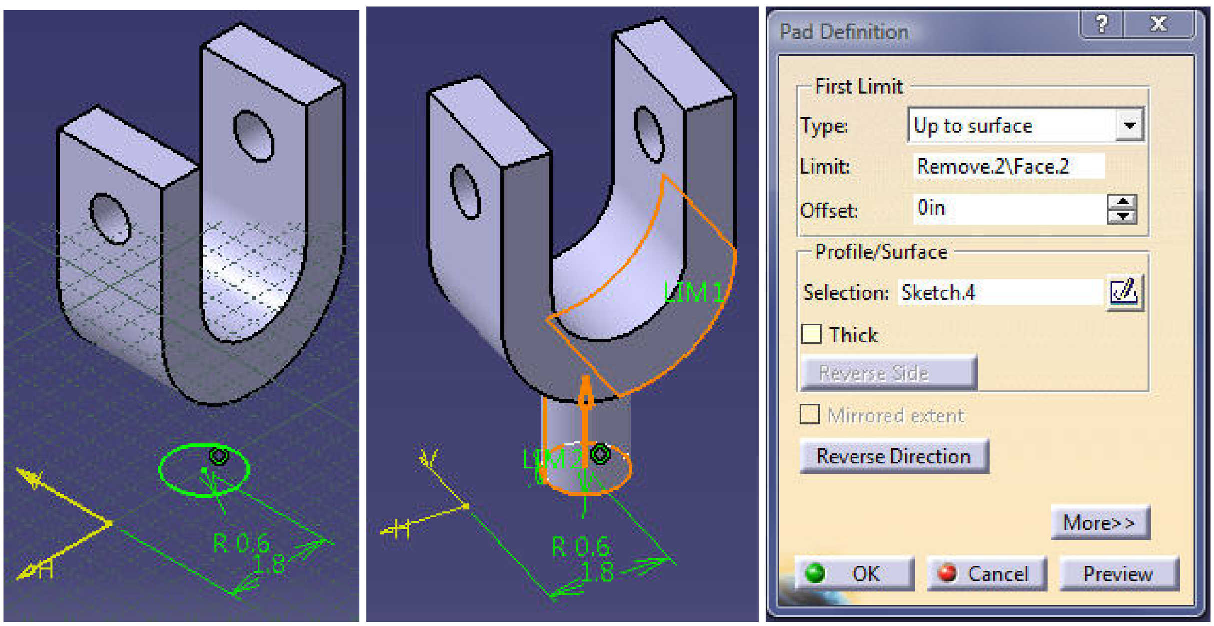




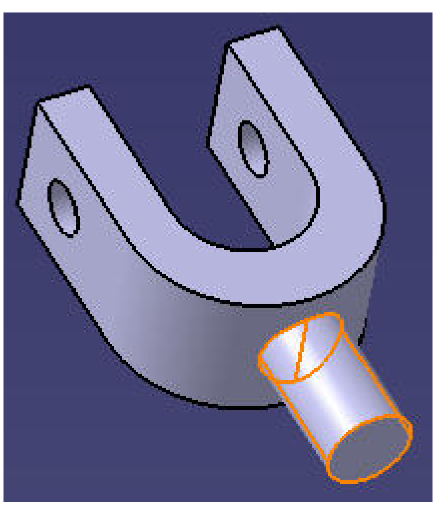
13. Rod "Body.3" >> Remove from "PartBody".



14. Pick top surface 1 >> Pick the "Plane" tool >> Offset >> 6.05in



15. Sketch icon >> Circle >> R0.6 >> 1.8in >> Pad >> Type: >> Up to surface

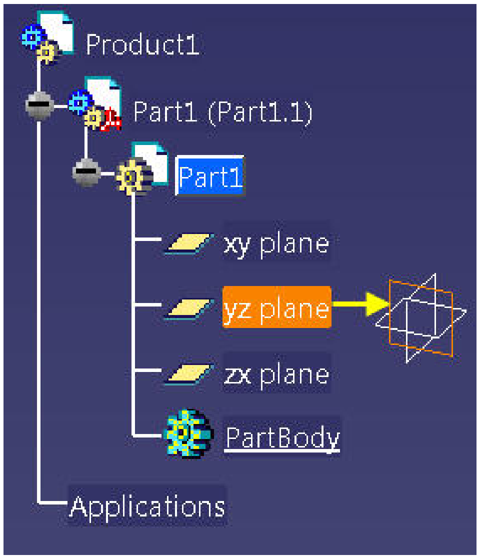


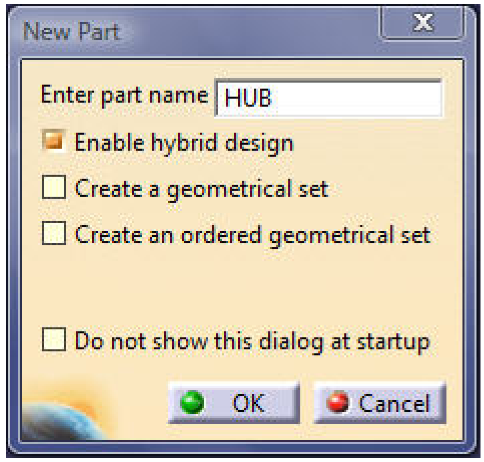
Complete U-Joint Part.

5- SHAFT & HUB DESIGN

All 3D solid parts begin with a sketch in the "Sketcher Workbench".

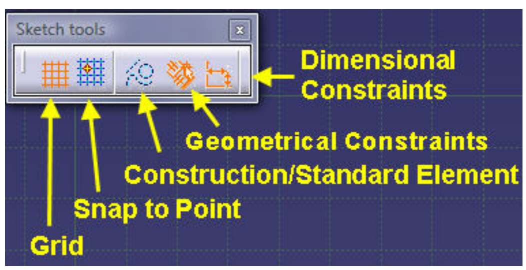
HUB EXAMPLE



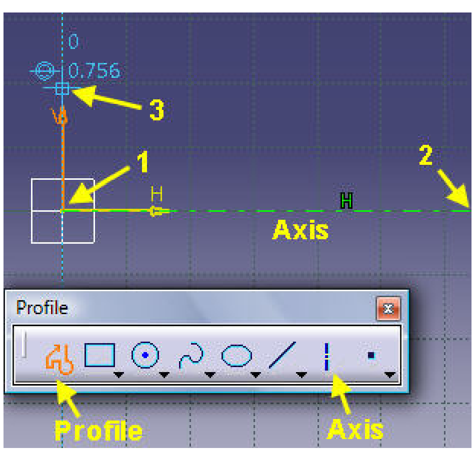


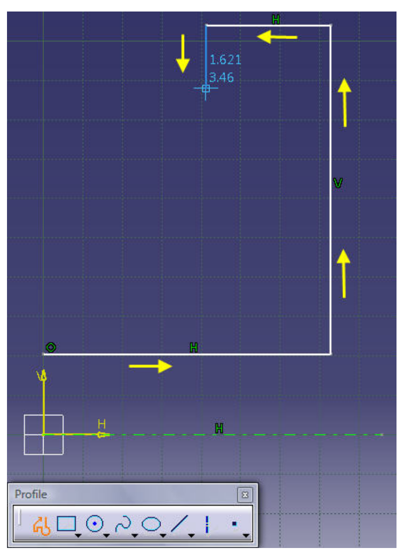
Start >> Mechanical Design >> Part Design >> Enter part name >> HUB.

Pick the yz plane.



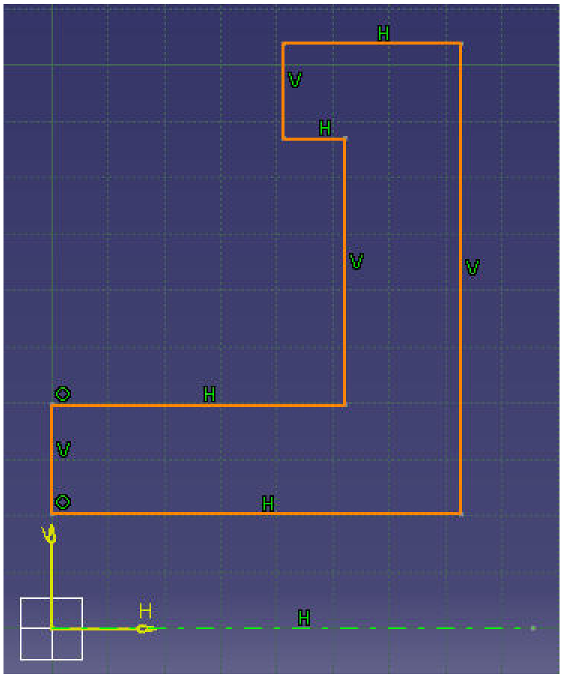
Pick the "Sketch" icon. **Pick "Snap to Point" to toggle blue color = Snap to Point Off.**

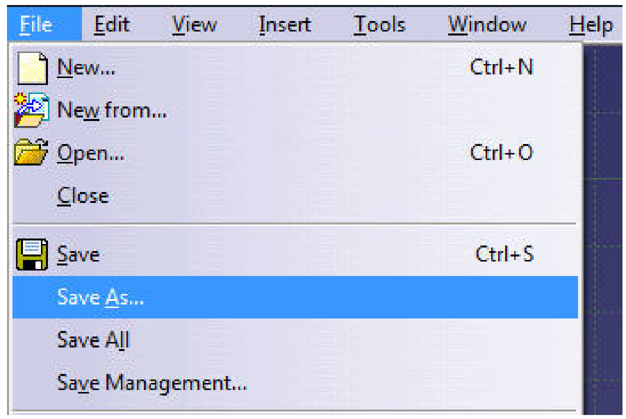




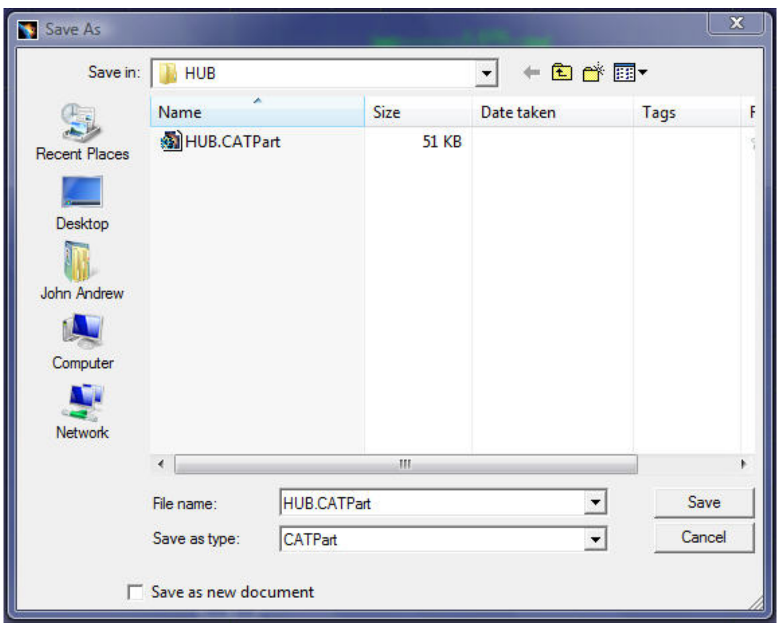
On the "Profile" toolbar pick the "Profile" icon shown above.

Pick point-1 >> Release mouse button >> Pick point-2 >> Release mouse button >> Continue this method to create the profile above right and below left.

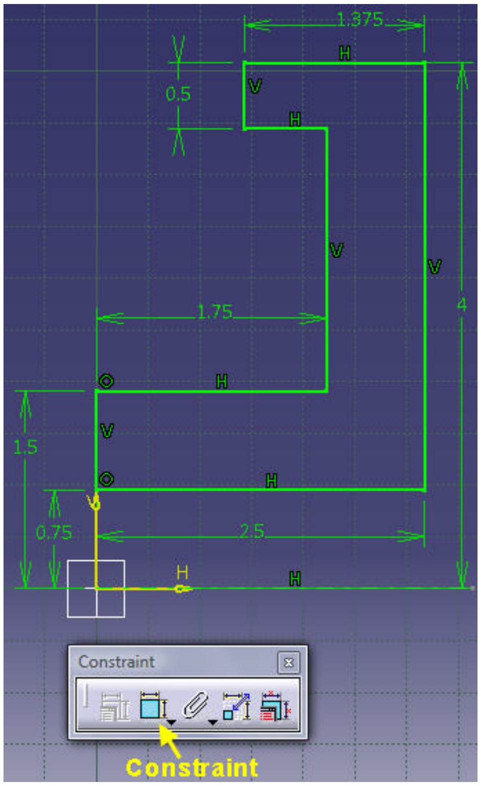


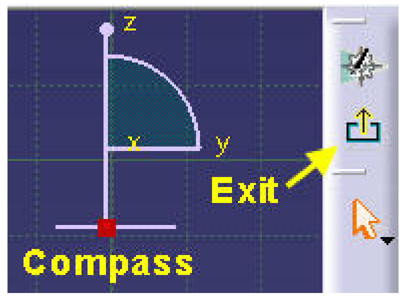


Save the above sketched profile as, "HUB". File >> Save As… >> Browse Files >> HUB.CATPart



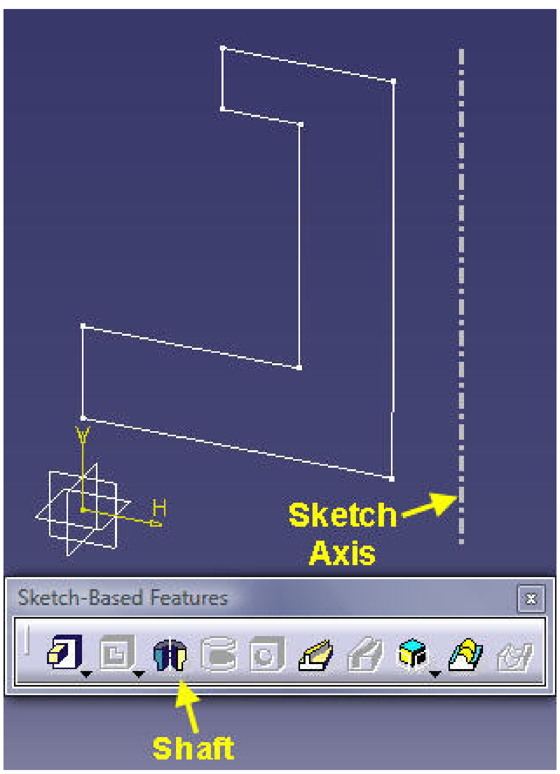
SKETCH DIMENSIONS



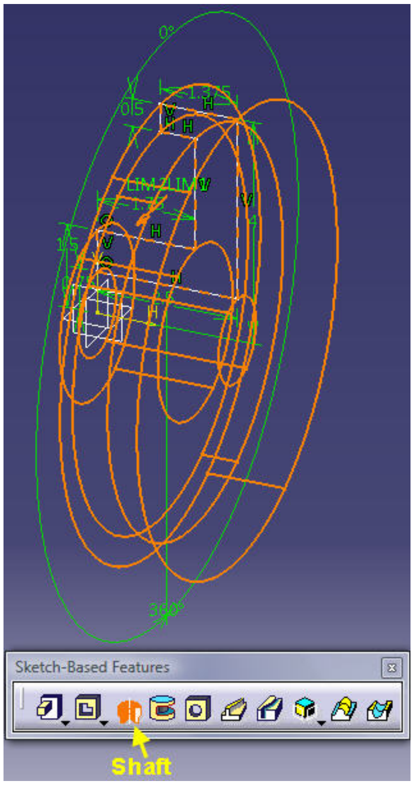


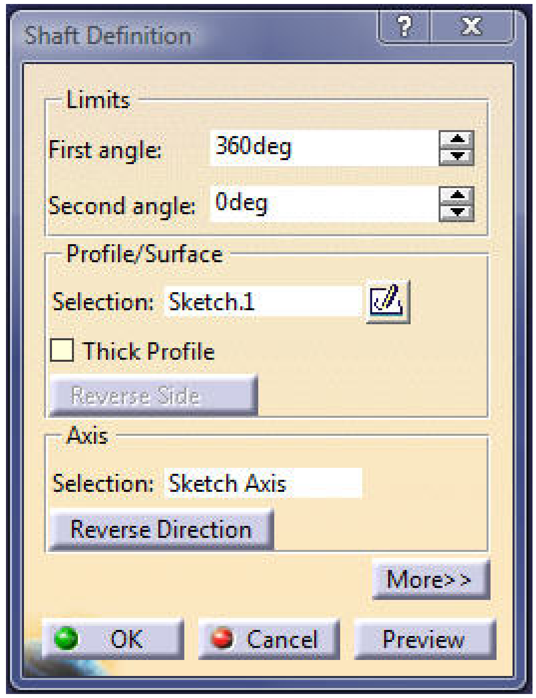
Place dimensions on the profile.

Next click "Exit" the Sketch Workbench.

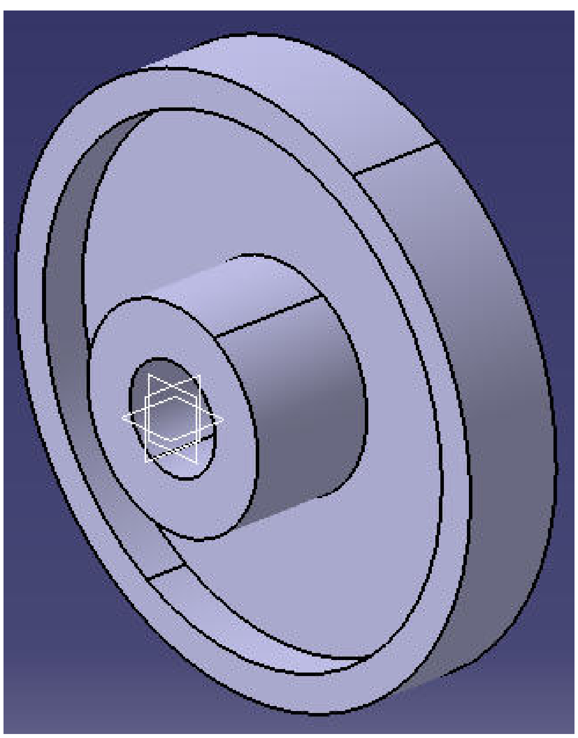


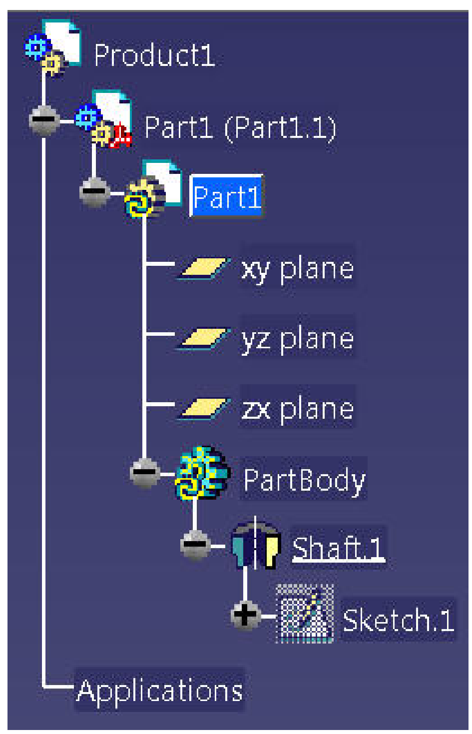
On the "Sketch-Based Features" toolbar select the "Shaft" icon.



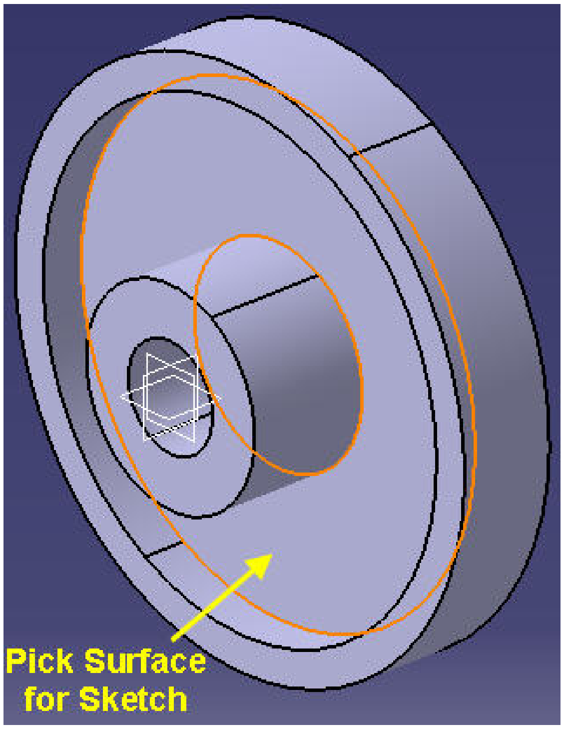


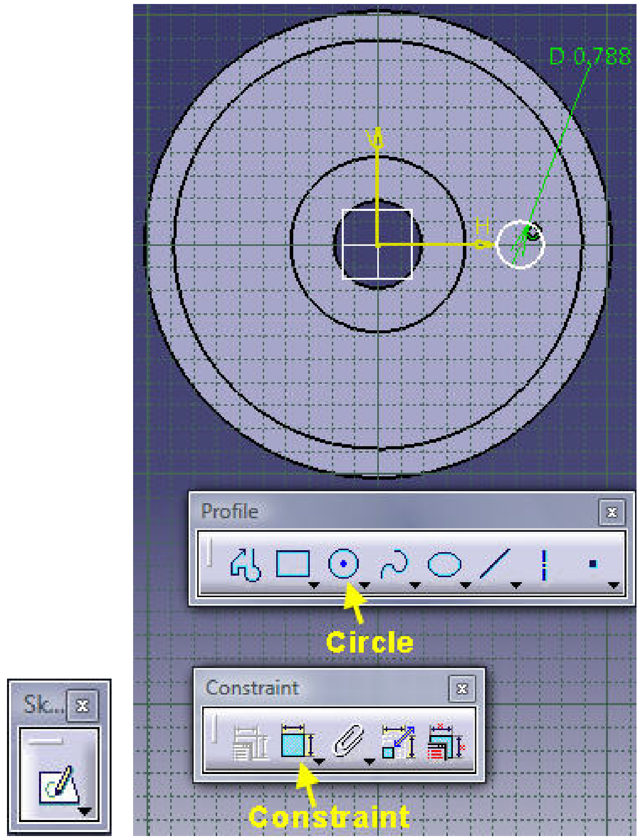
In the "Shaft Definition" box edit "First angle:" >> 360deg >> Axis >> Selection >> "Sketch Axis" >>OK



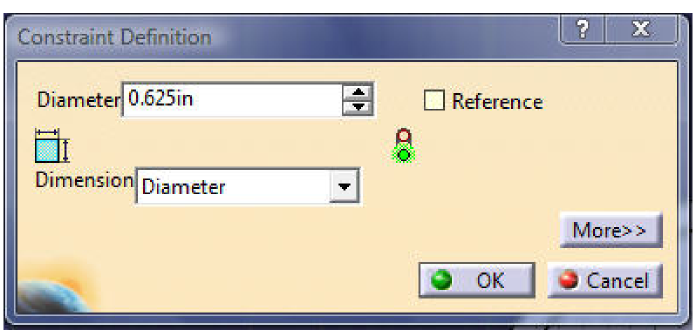


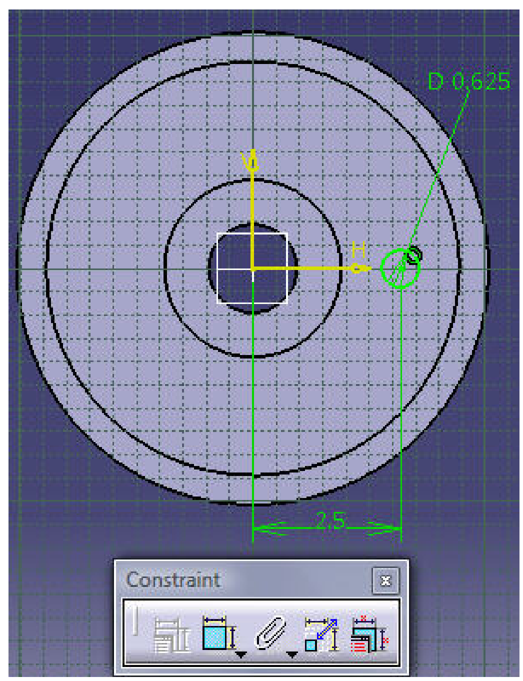
Basic Hub is now formed and added to the Tree.



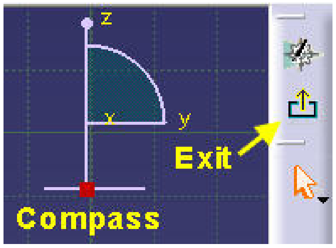


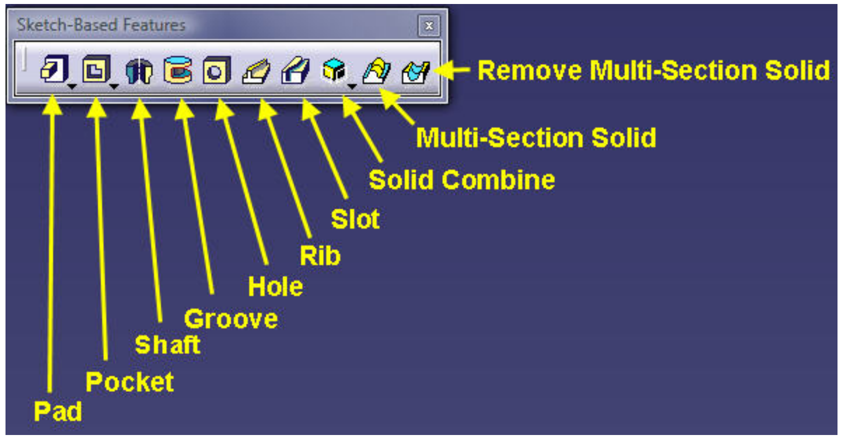
Pick the surface shown above for sketching. **Pick the "Sketch" icon and place the above circle.**



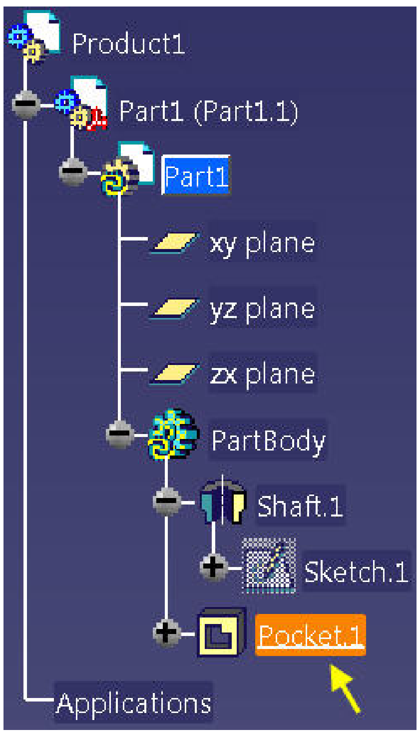


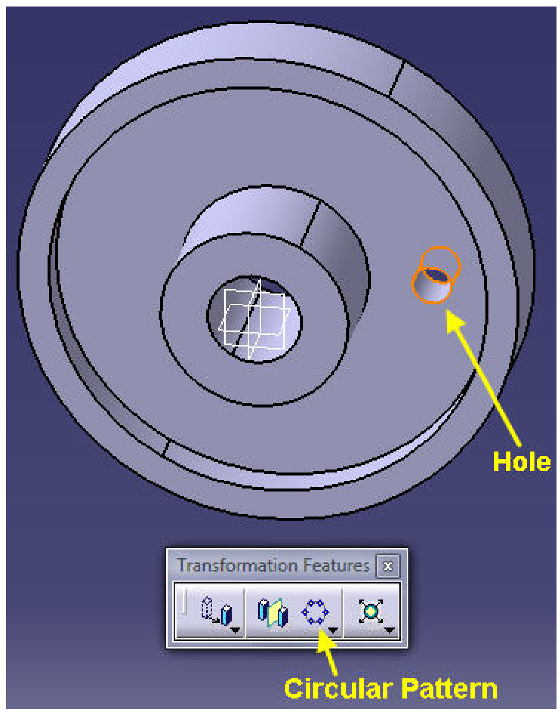
Set the circle diameter to 0.625in. **Constrain the circle location at 2.5in from the Hub center.**



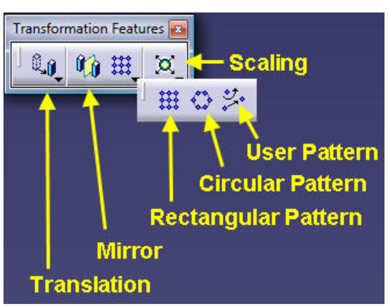


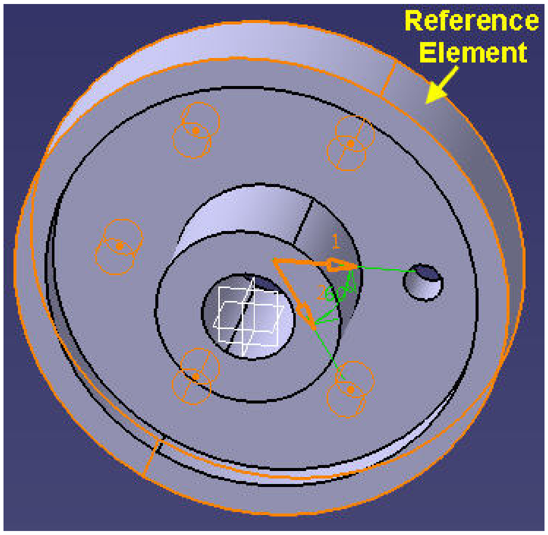
Pick "Exit" Sketch Workbench. On the "Sketch Based Features" toolbar select "Pocket".

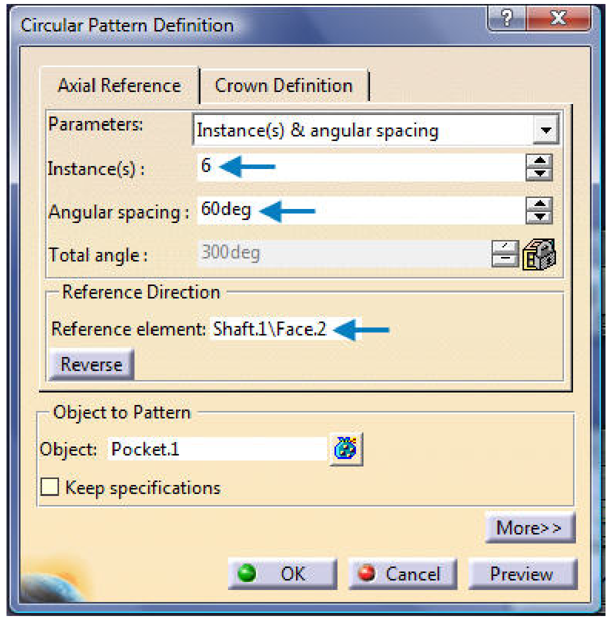




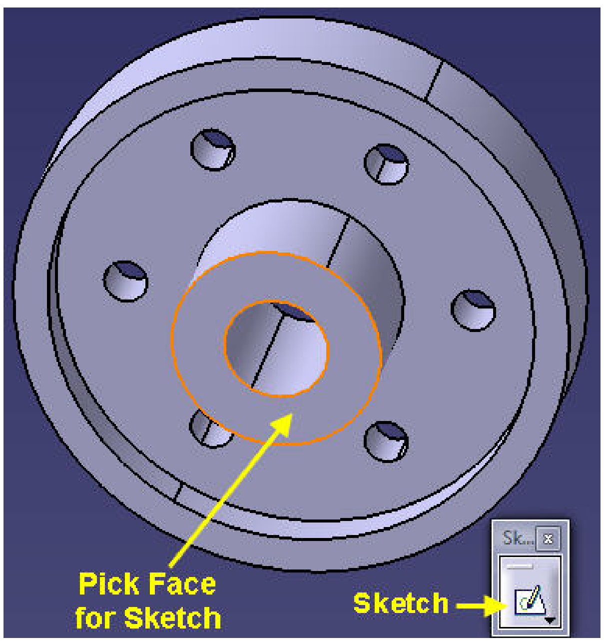
The hole "Pocket" is added to the tree. Pick "Circular Pattern" on "Transformation Features".

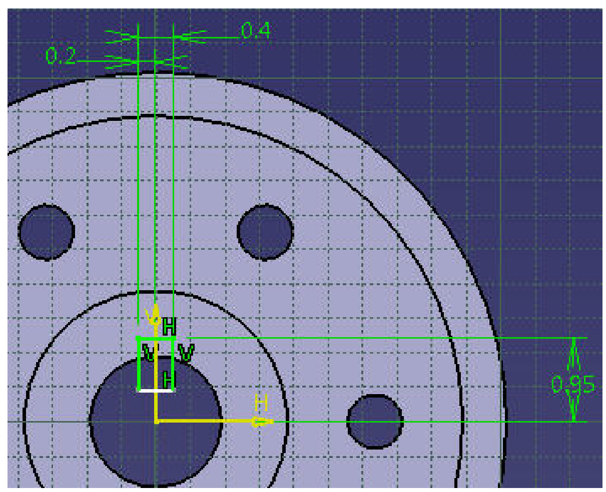






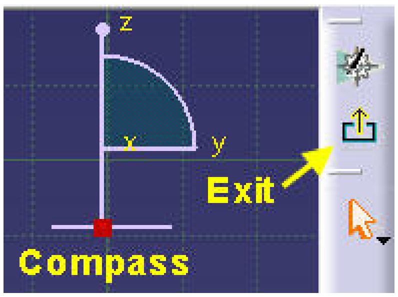
In the "Circular Pattern definition" box enter (6) Instances of the Hole spaced 60 degrees apart. Next pick the curved edge or curved surface as "Reference Element".

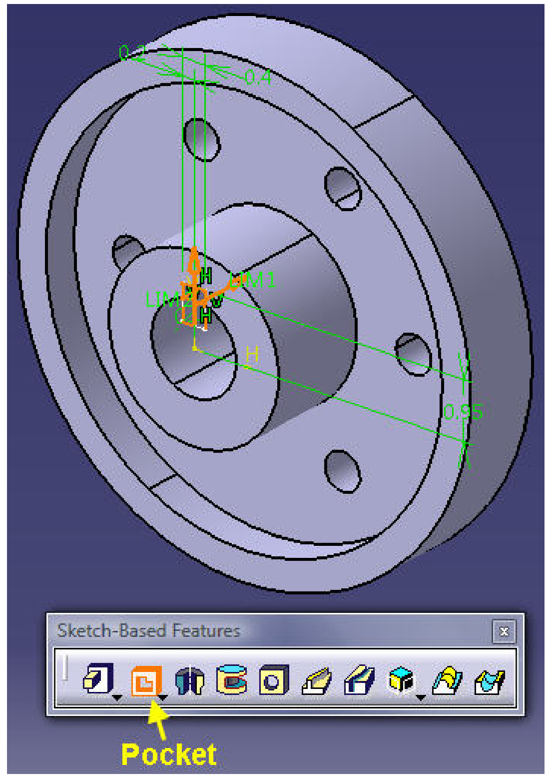




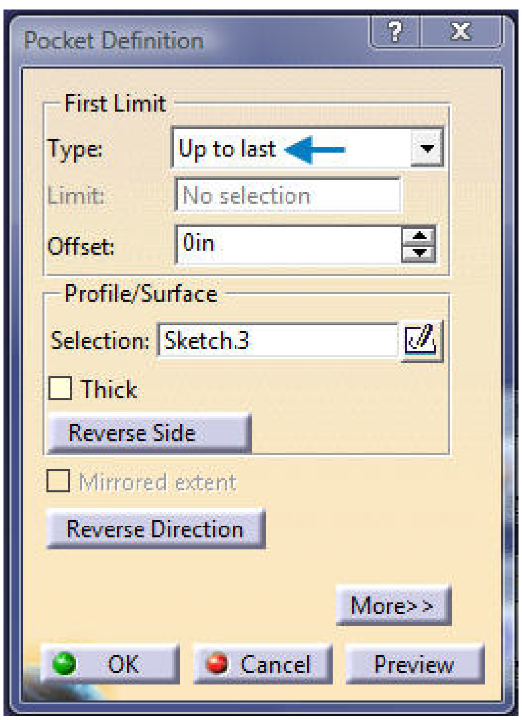
Pick the face for a sketch and Sketch icon. **Sketch the 0.4 x 0.95 rectangle above.**

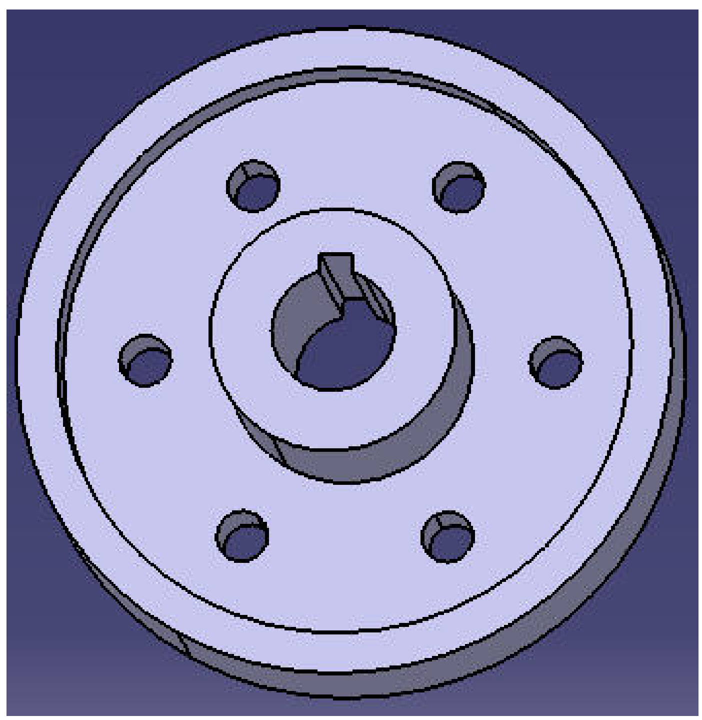
KEY SLOT



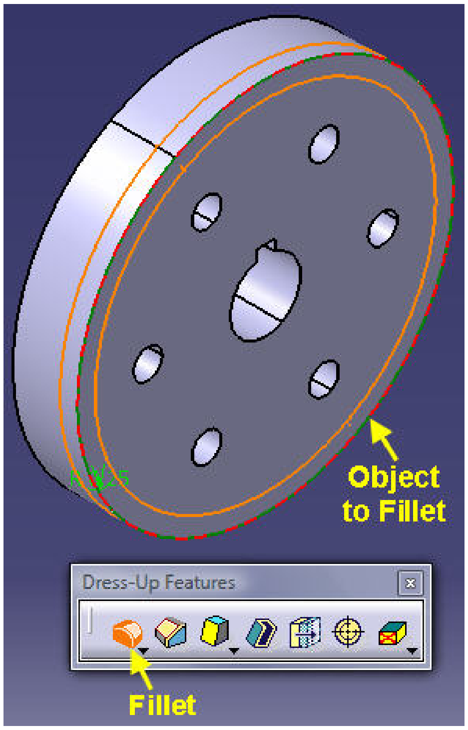


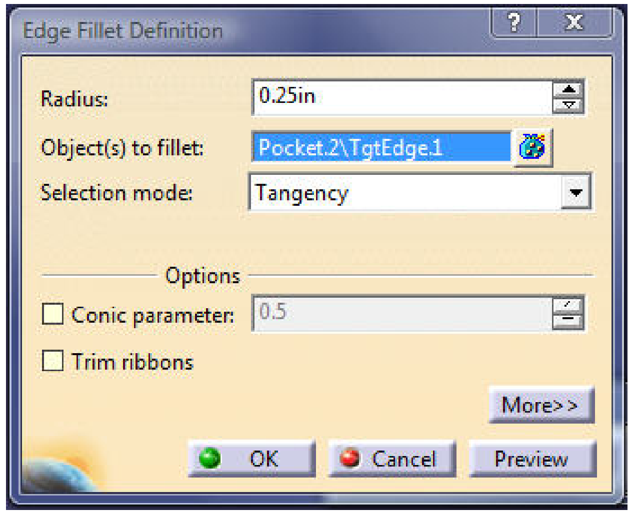
Pick "Exit" sketch workbench. **Pick "Pocket".**



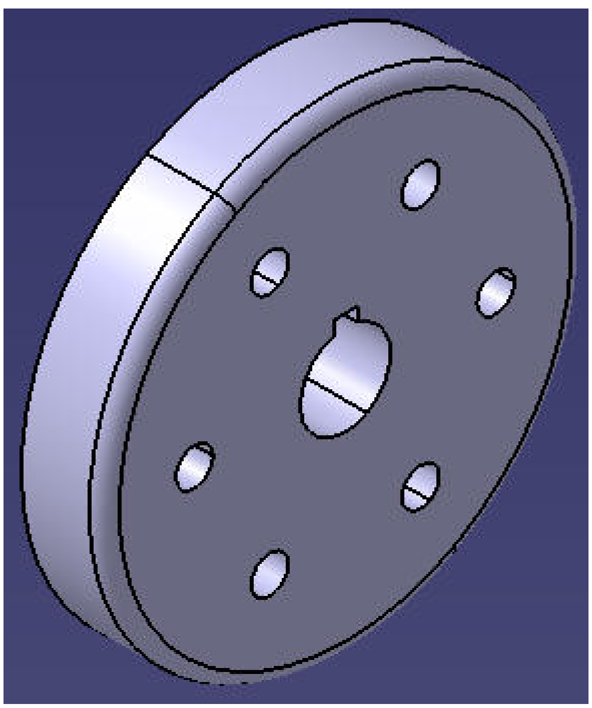


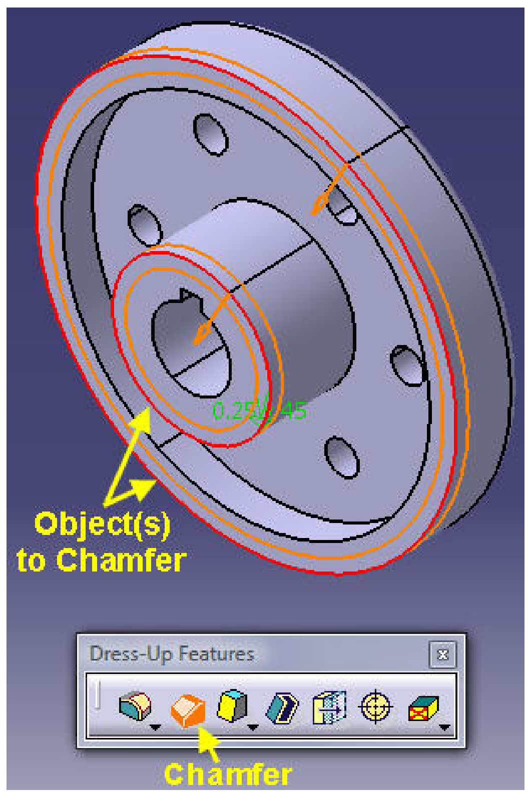
Select Type: "Up to last" >> OK. **The key slot is placed in the hub bore.**



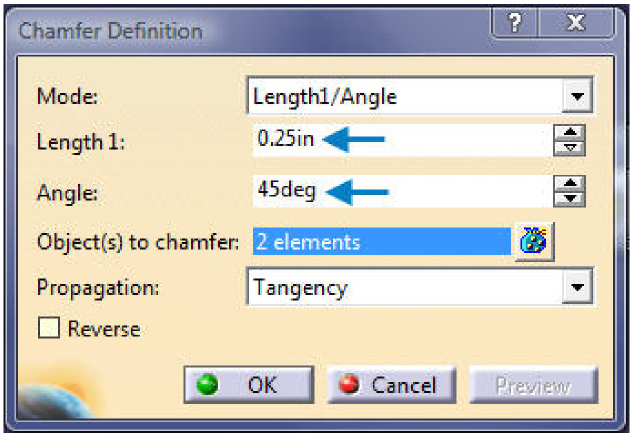


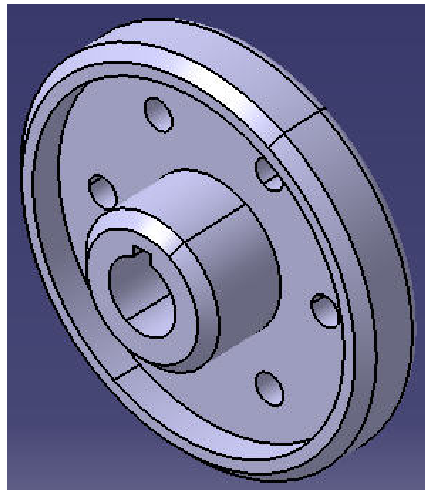
Pick "Fillet" >> Pick the hub circular edge. **Set Fillet radius to 0.25 inches >> OK.**



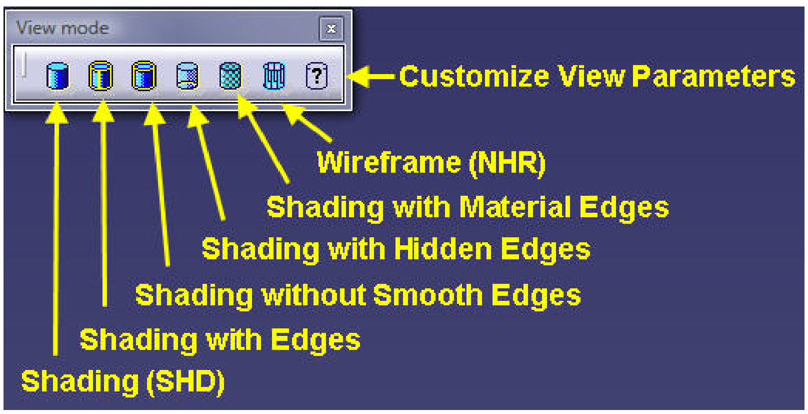


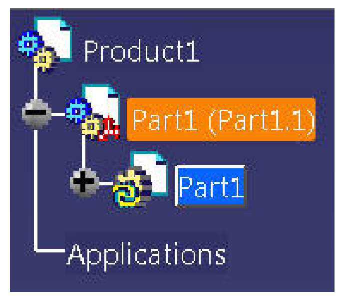
Fillet edge is complete. **Pick "Chamfer" >> Pick circular edges above.**

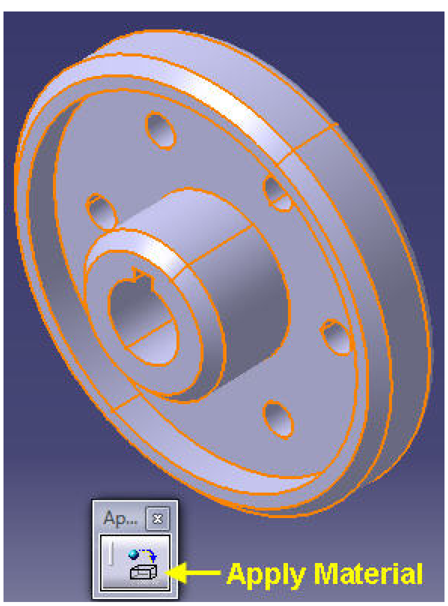


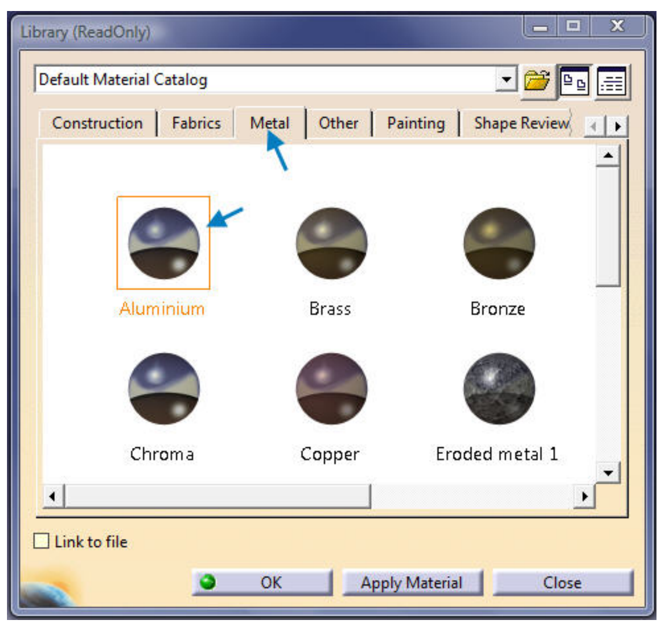


Set Chamfer Length 1: to 0.25 inches and Angle: to 45 degrees.







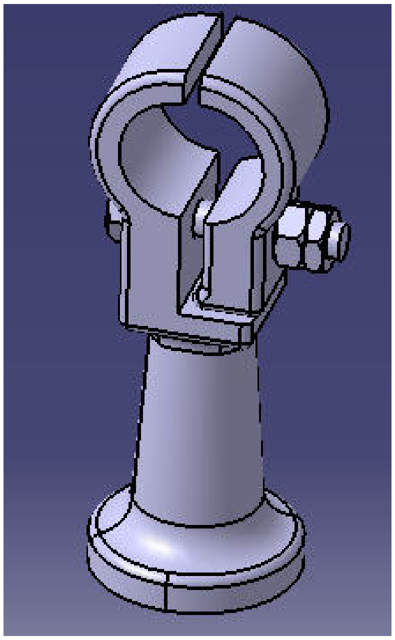


Pick the "Apply Material" icon.

Pick the "Metal" tab >> Aluminum >> Apply Material

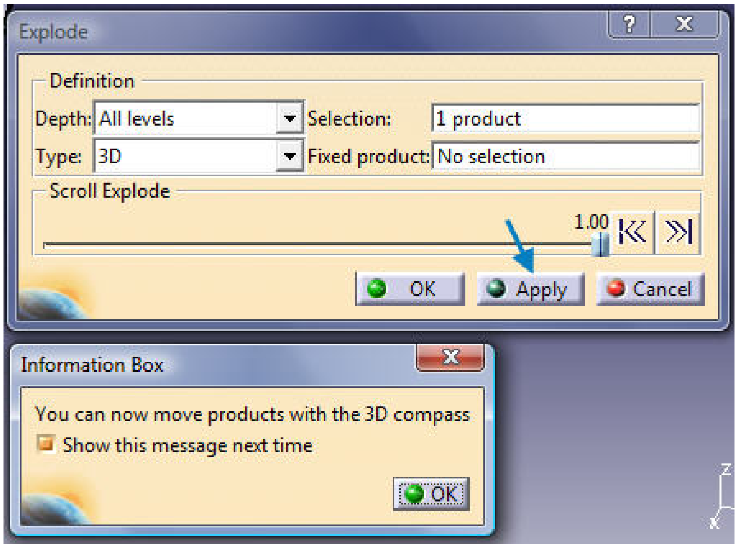
The completed HUB.

6- ASSEMBLY WITH BILL OF MATERIALS

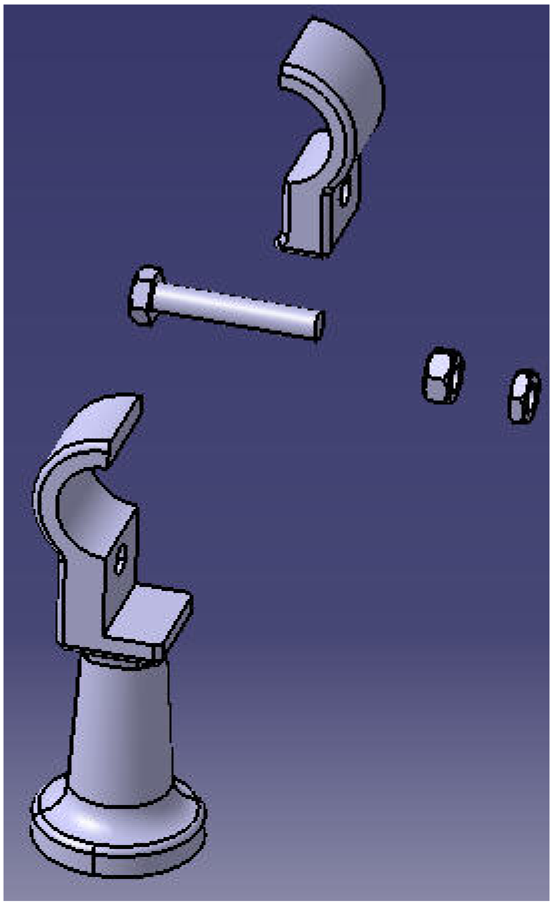


CLAMP ASSEMBLY EXAMPLE

Select: **Explode** on the **Manipulation** icon illustrated above.

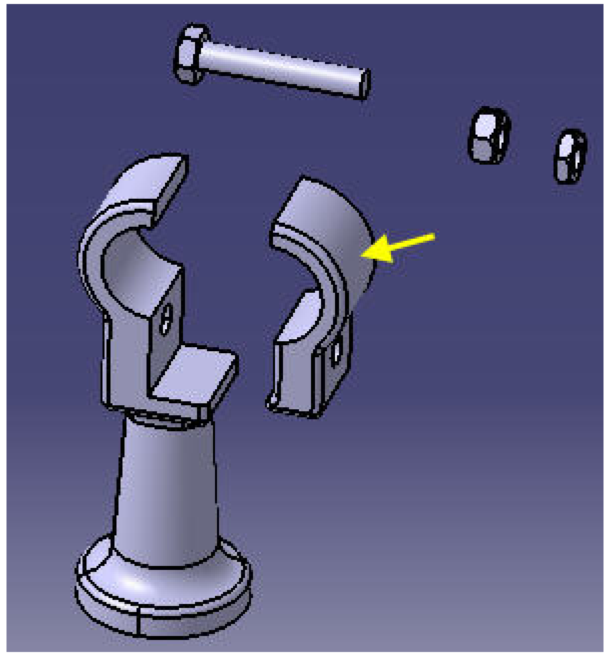


Select: **Apply**



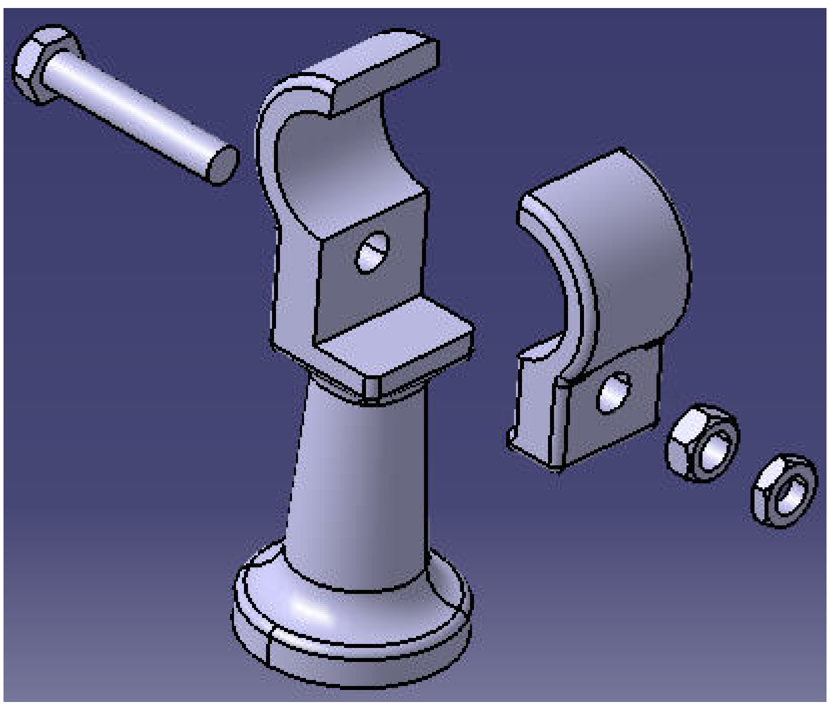
The assembly is exploded.

Select: **Manipulation.**

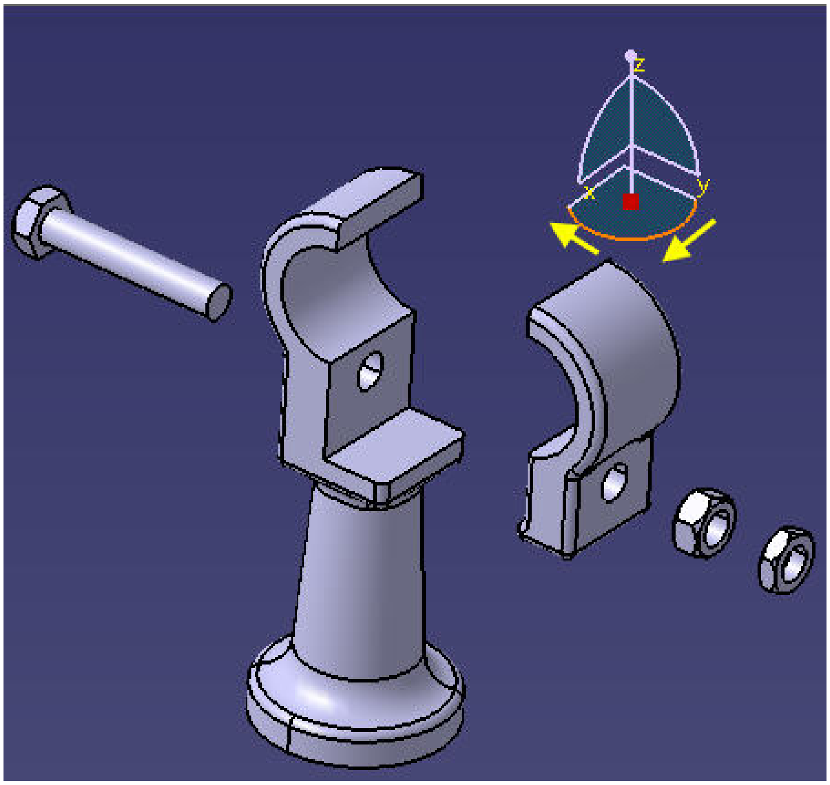


Pick one of the "Drag" direction icons. Select a part to drag.

Move parts as required to complete the exploded model as shown below.

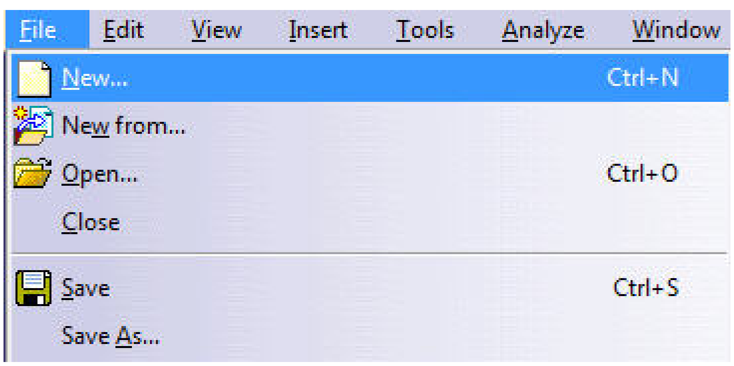


Each part is in its final exploded position.

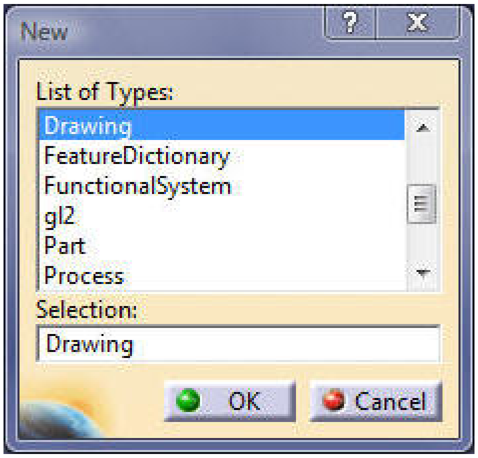


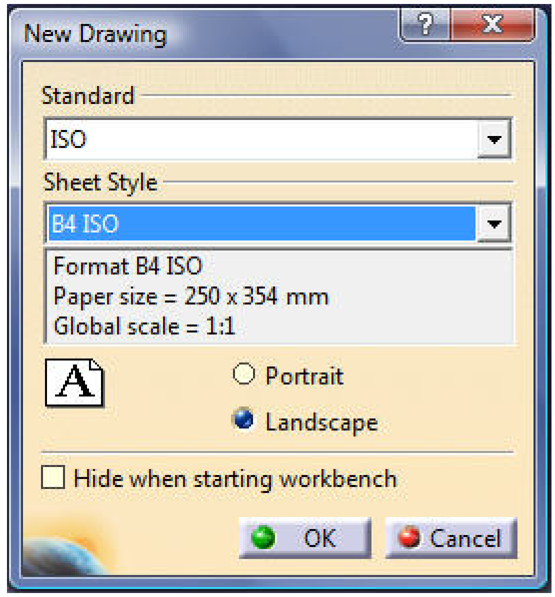
The "Compass" is used to rotate the exploded assembly.

CREATE A NEW DRAWING WITH BILL OF MATERIALS



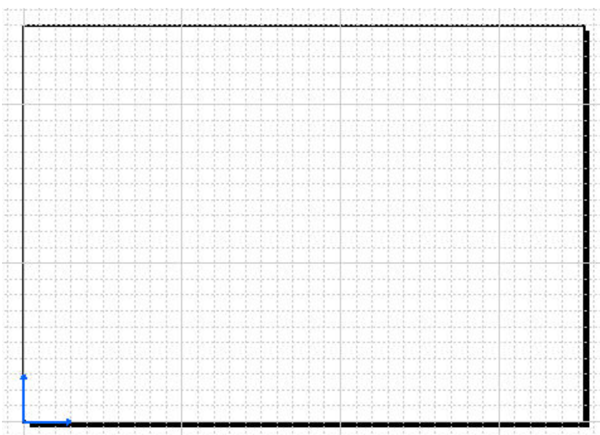
Select: **File** >> **New**





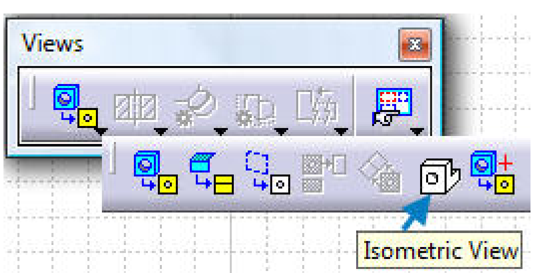
Select: **Drawing** >> **OK** above left.

Select a **Sheet Style** in the **New Drawing** dialog box above right >> **OK**

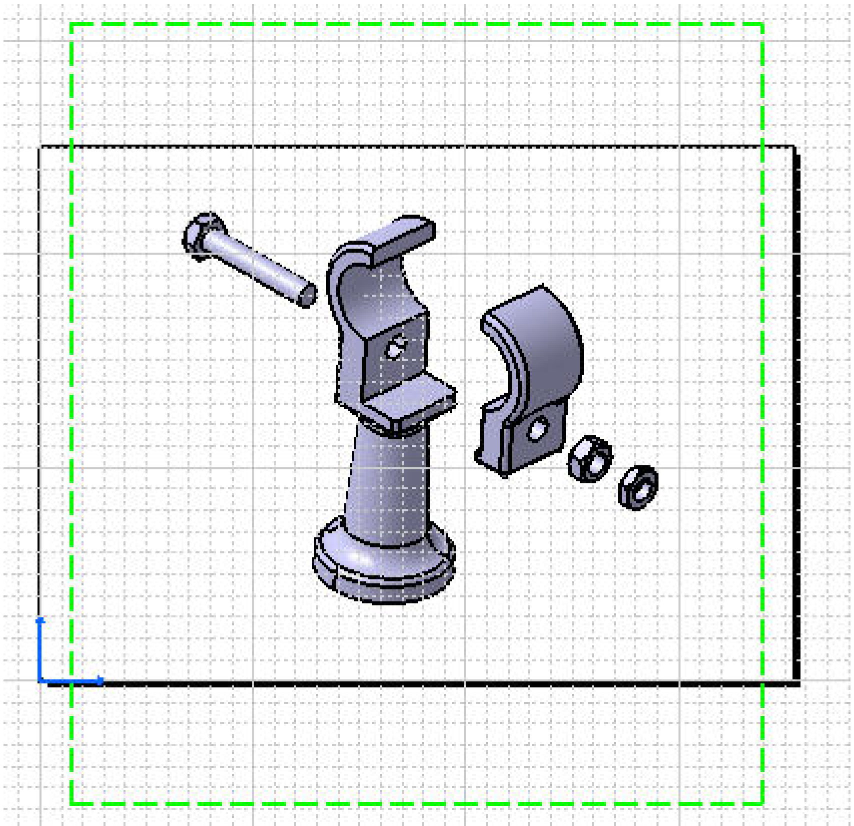


A blank drawing opens.

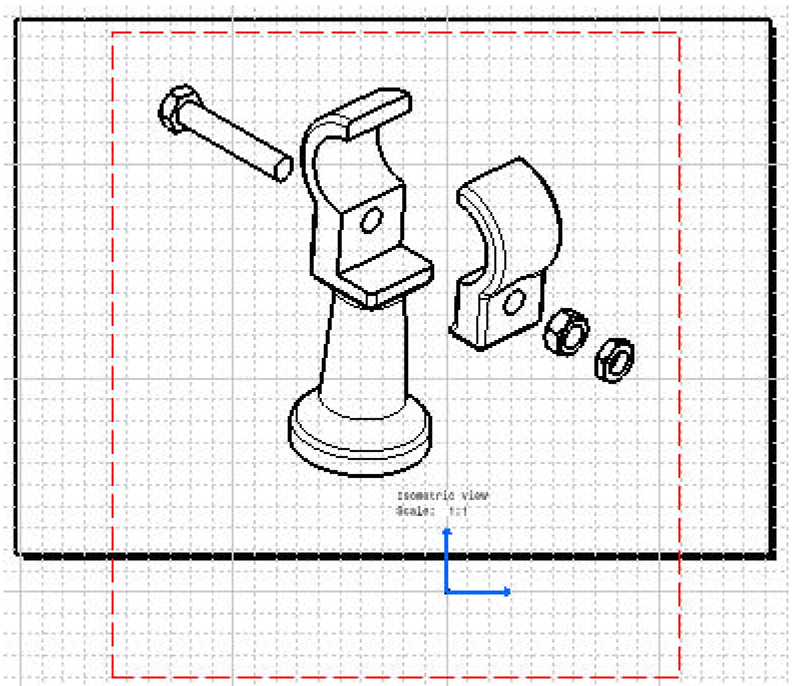
CREATE THE ISOMETRIC DRAWING VIEW



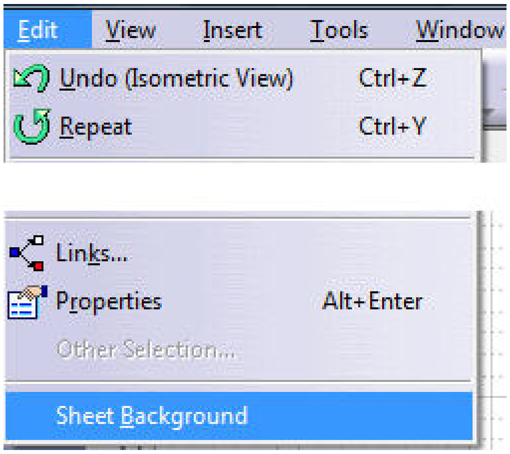
Select the **Views** toolbar >> Pick the drop-down menu >> Pick **Isometric View**

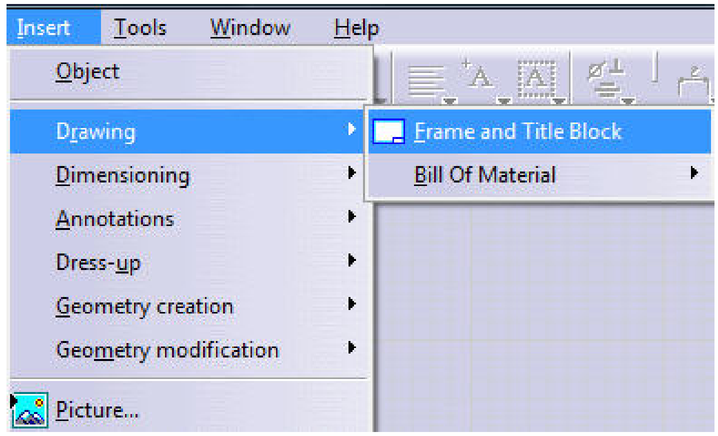


The Isometric Model is placed in the drawing sheet.



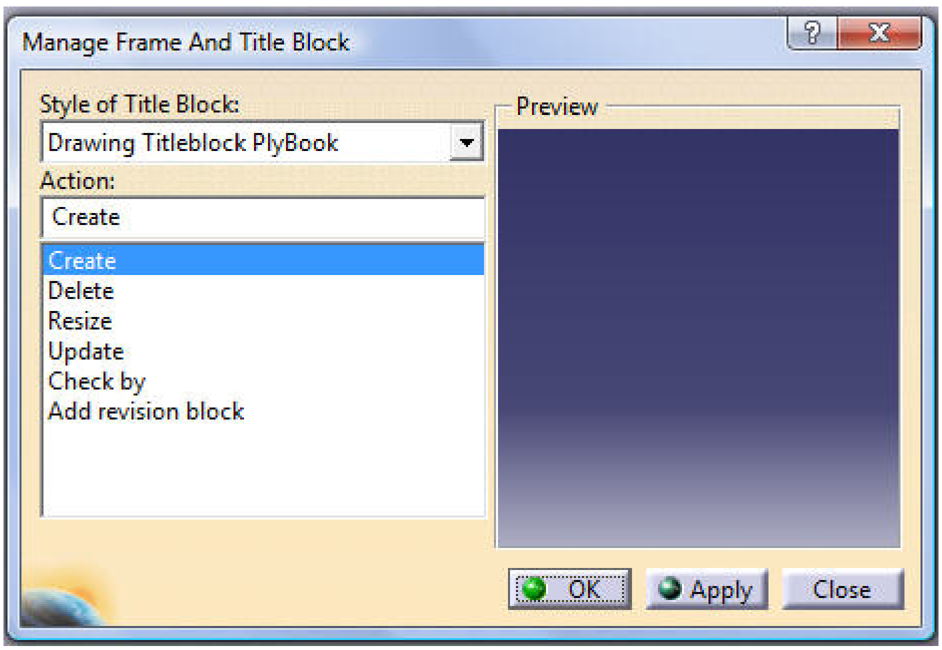
Click to convert to Isometric Drawing.



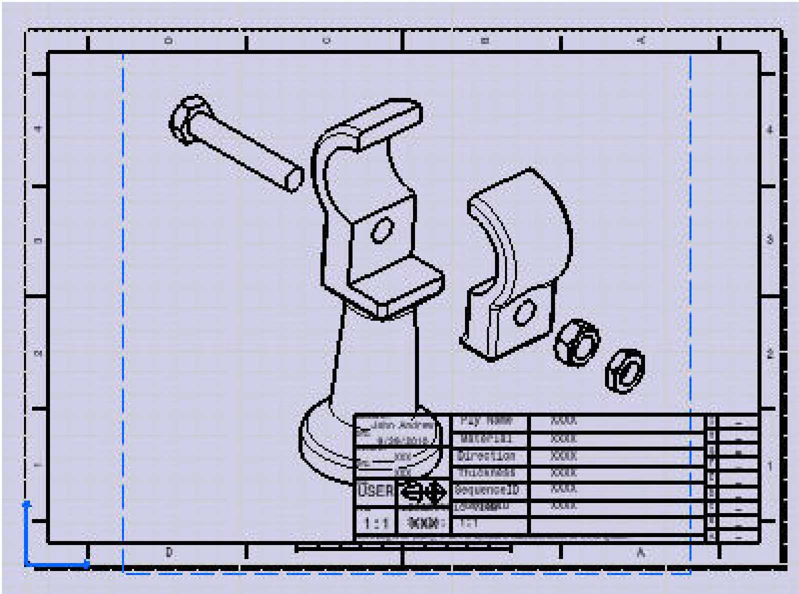


Select: **Edit** >> **Sheet Background**

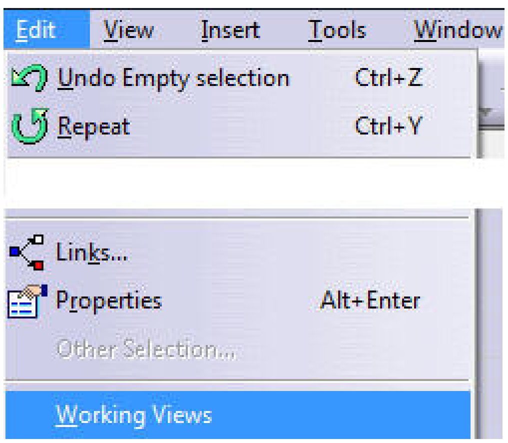
Select: **Insert** >> **Frame and Title Block**



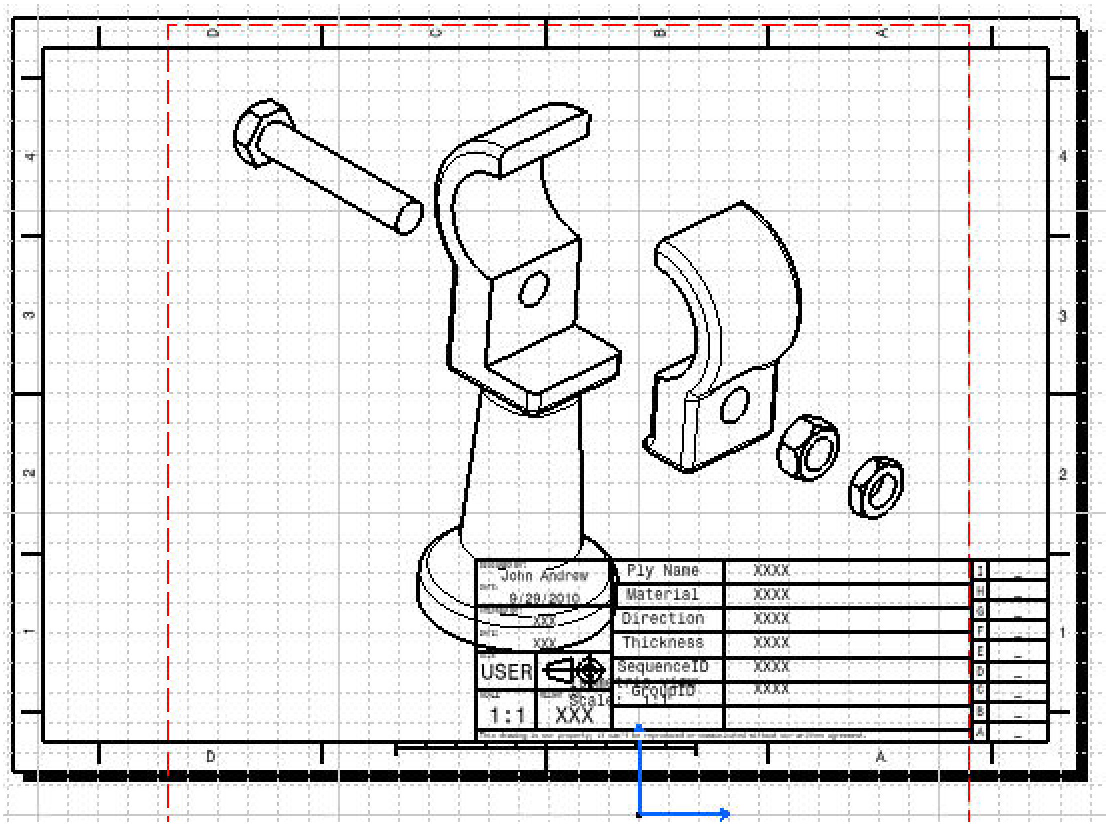
Select: **Create** >> **OK**



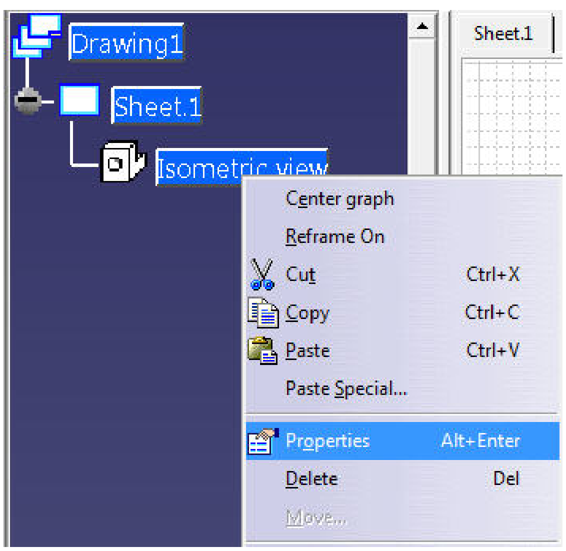
The frame and title block are inserted.



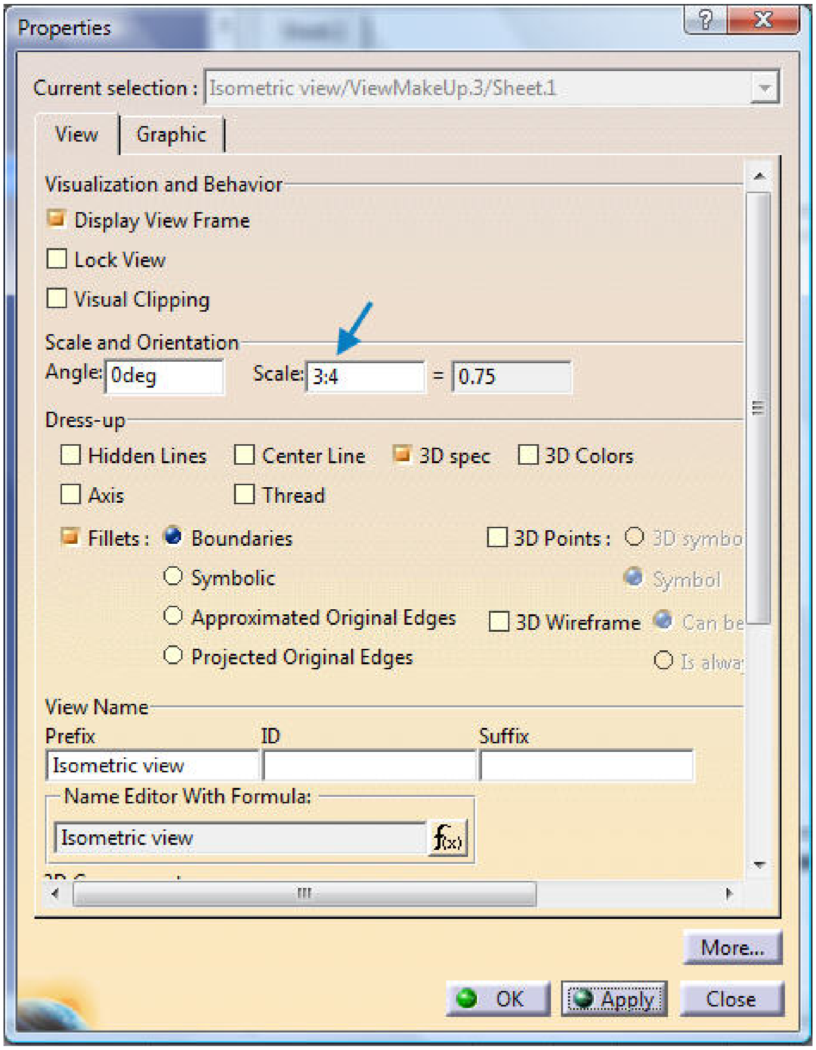
Select **Edit** >> **Working Views**



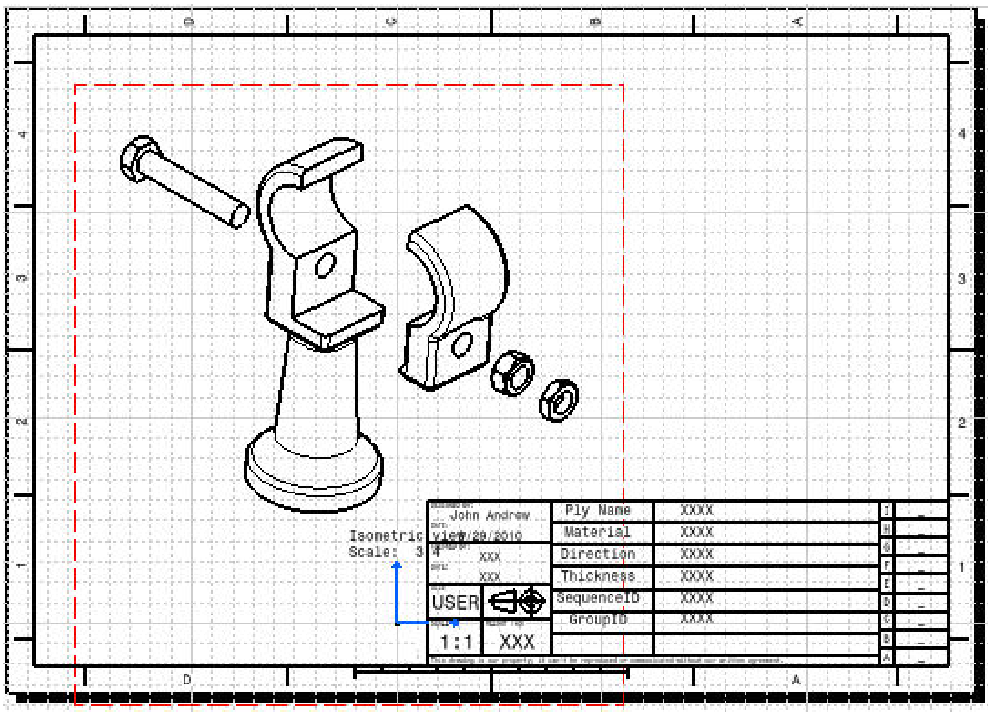
The assembly needs to be moved and scaled.



Right click on: **Isometric View** >> **Properties**

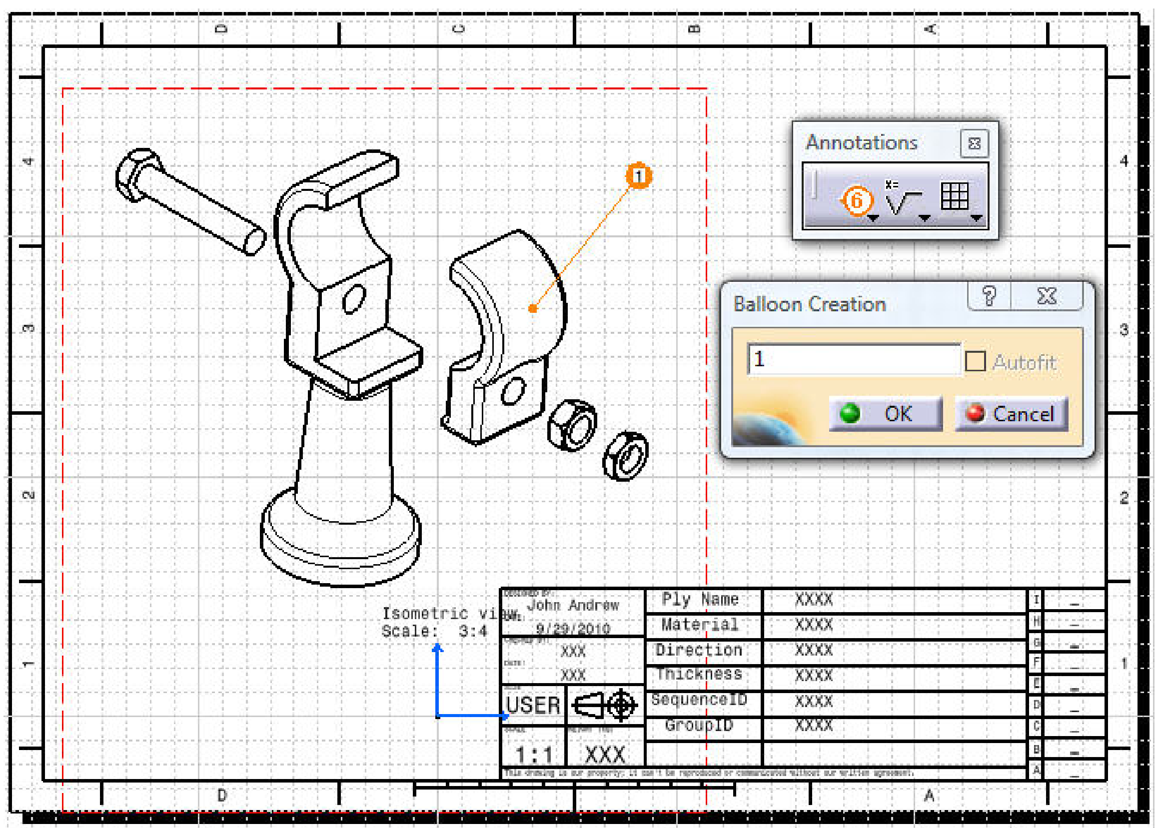


Edit Scale from (1:1) to (3:4) or other scale as required for your assembly.



Double click on the view boarder to make it active.

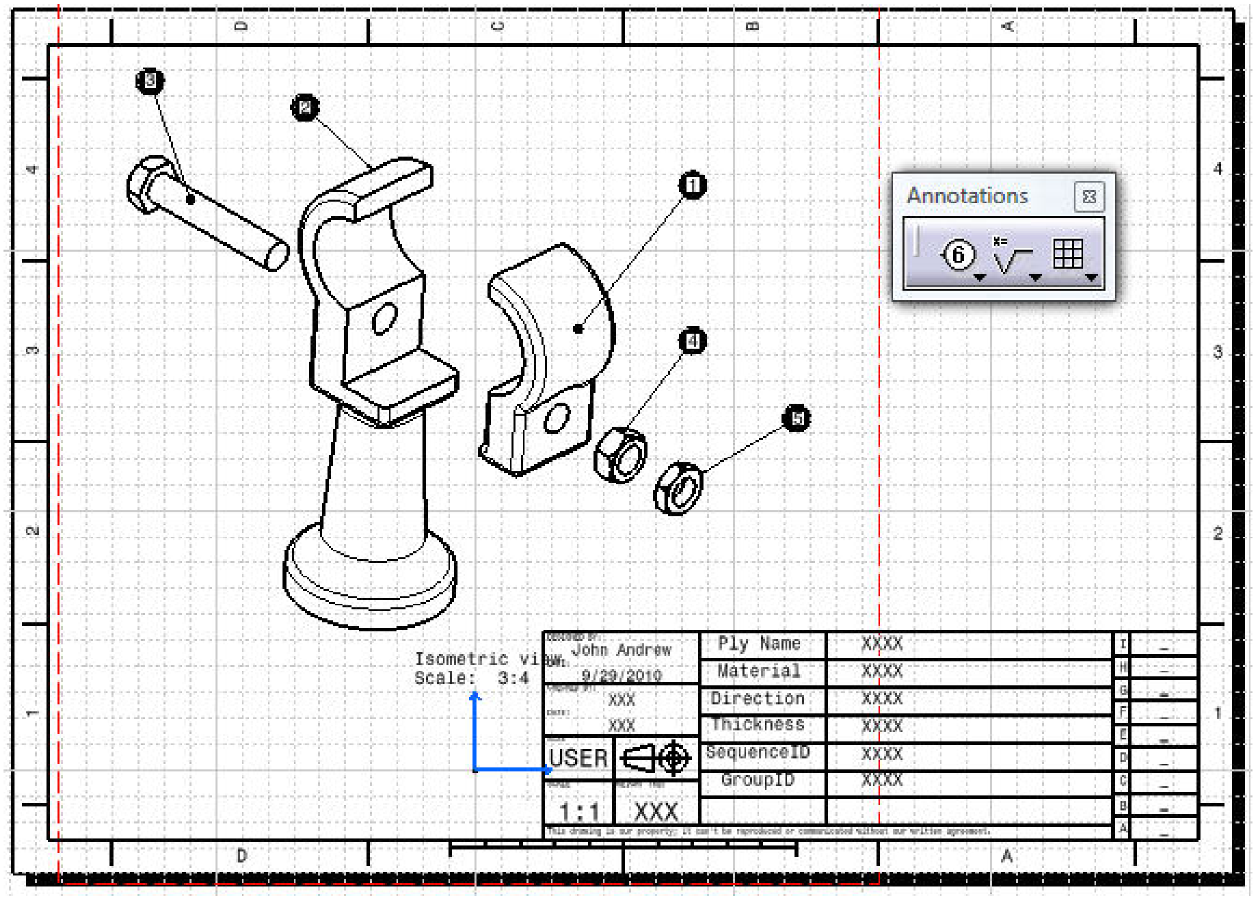
Place mouse pointer on the orange boarder to move the view.



Select the **Annotations** toolbar.

Pick the surface of any one of the parts.

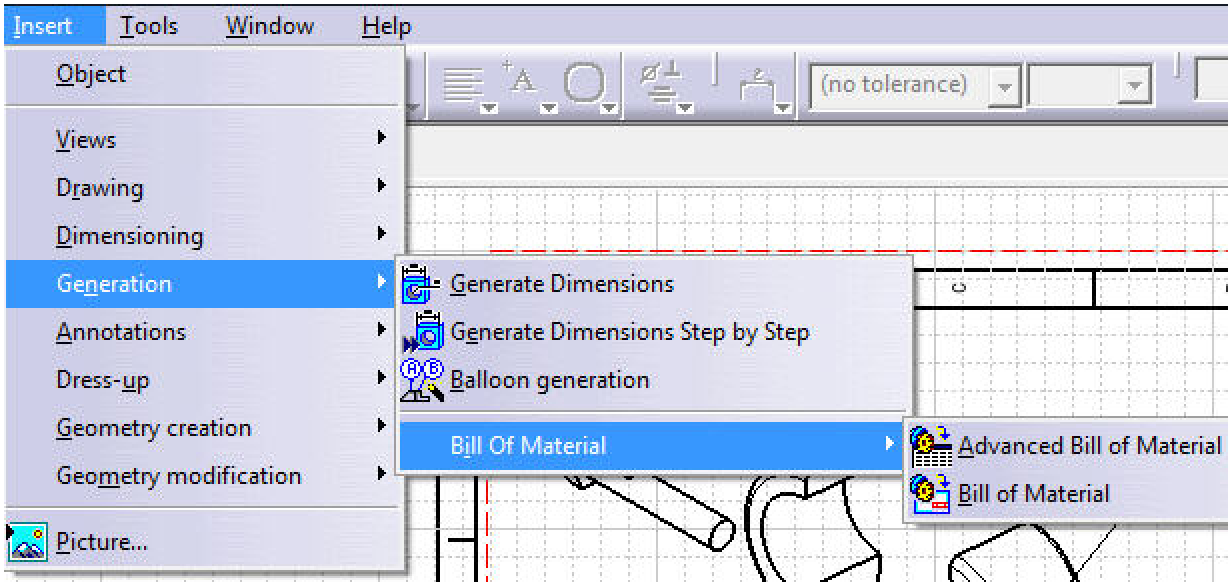
Place the numbered balloon.



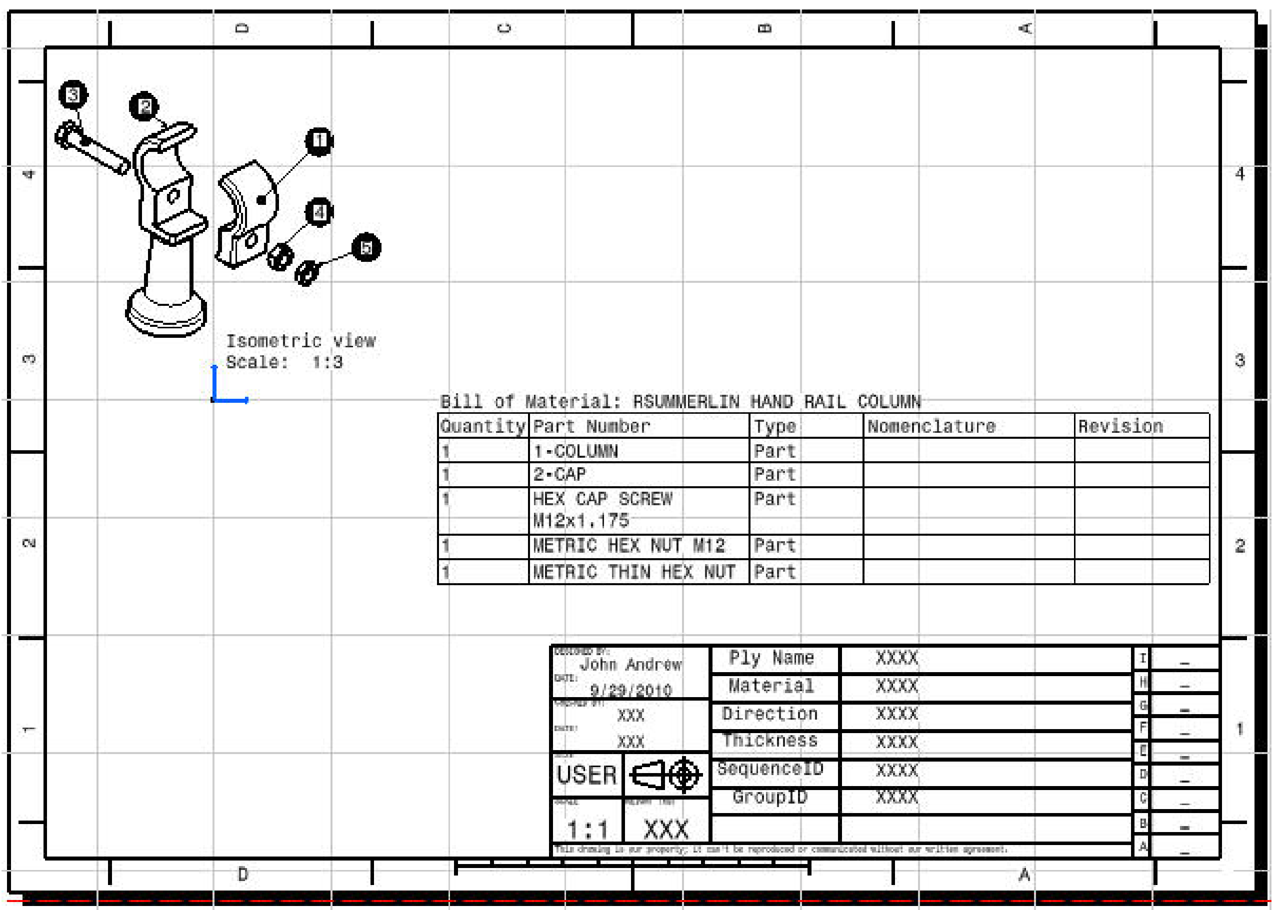
Repeat balloon placement for each part.

BILL OF MATERIALS

This Bill of Material, or parts list, consists of an itemized list of the several parts of an assembly shown on a CATDrawing.



Select: **Insert** >> **Generation** >> **Bill of Material**

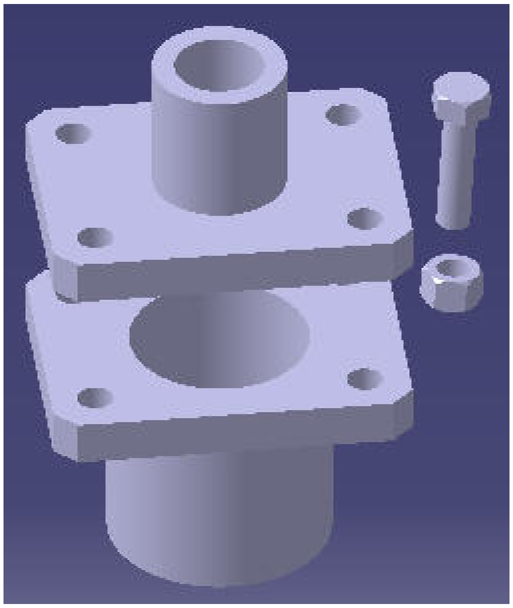


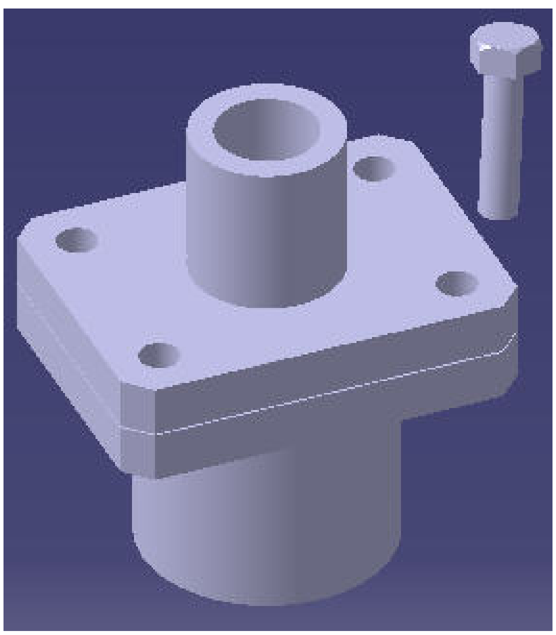
Double-click on the Bill of Materials >> pick its boarded

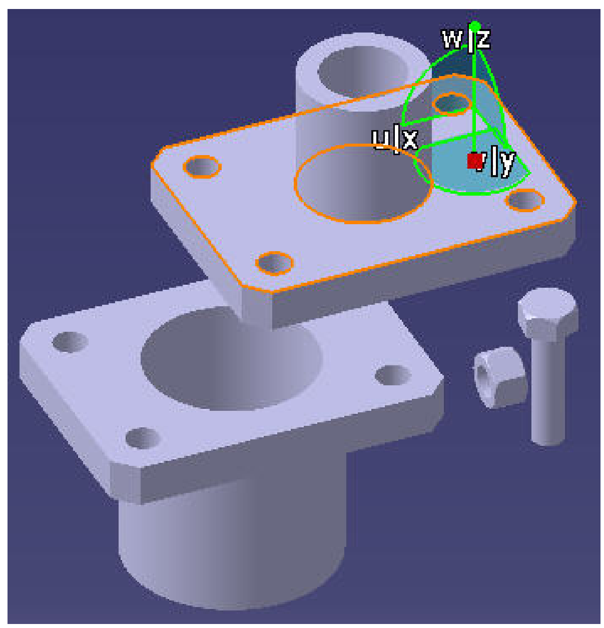
>> Drag to desired location.

Double-click on an item >> Edit if necessary.

7. ASSEMBLE OF PART INSTANCES





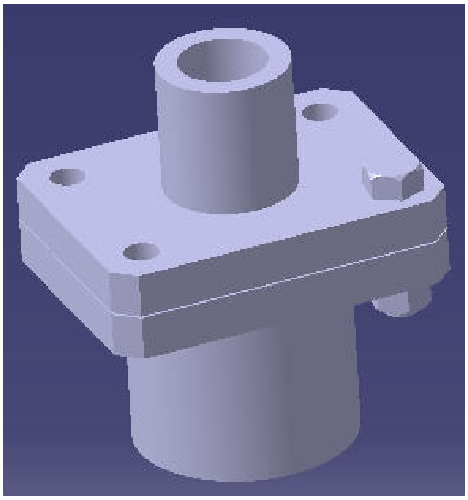


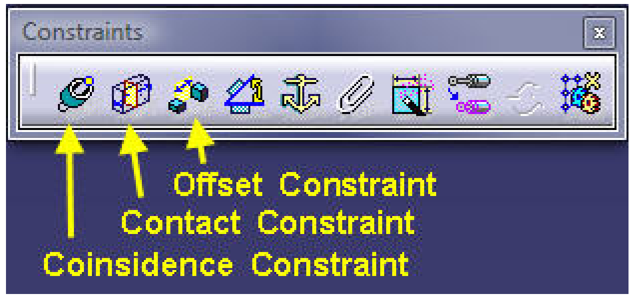
Assemble the 1in Pipe Flange on top of the 2in Pipe Flange.

The gap between flanges is 0.040 inches for a gasket (not shown).

The first hex head cap screw and nut have been inserted in the Pipe Reducer Assembly above.

The top reducer has been moved with the compass to reveal the hidden nut.

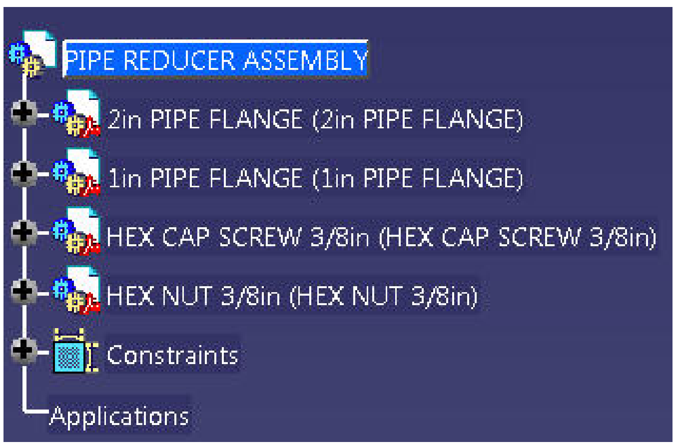




Press the Ctrl key & U to update and re-assemble.

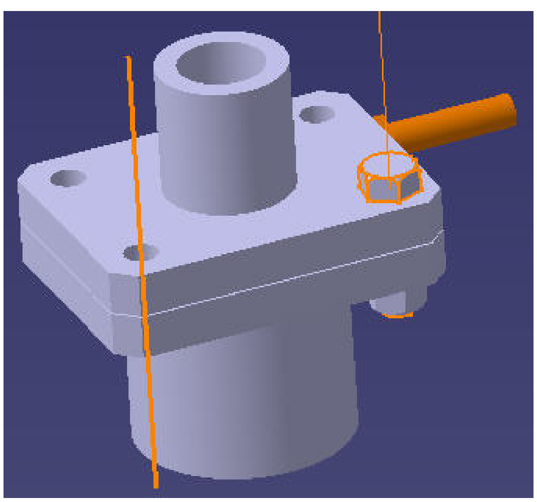
Constrain the Hex Head Cap Screw in the top flange hole as shown above.

Constrain the nut on the bolt.

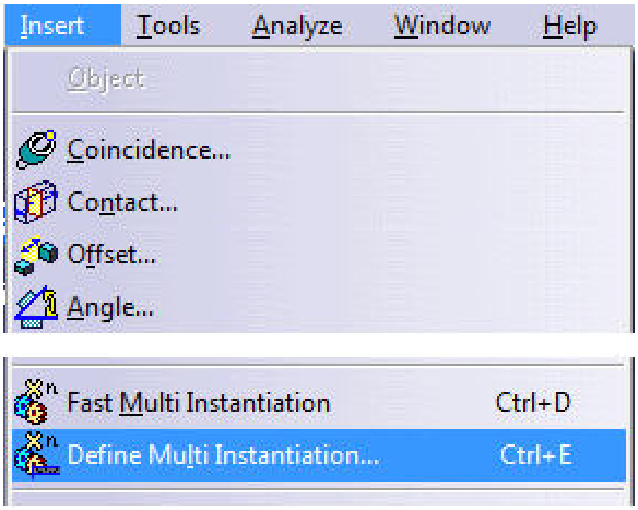


4 parts have been assembled as shown in the tree.

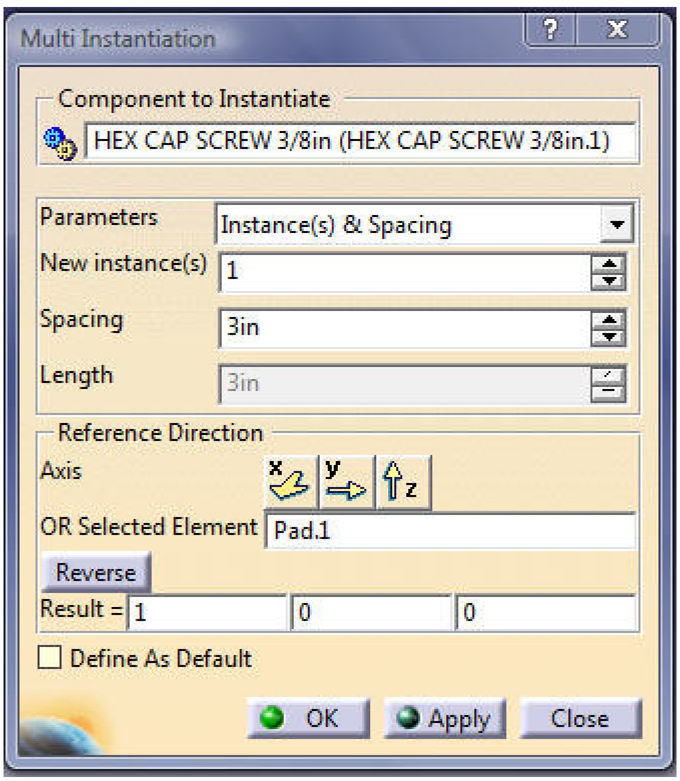
INSTANCES OF A PART

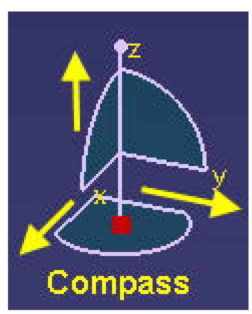


1. Pick the 1st Hex Head Screw



2. Insert >> Define Multi Instantation >> Spacing >> 3in (see below) >> Pick the empty hole center shown above.

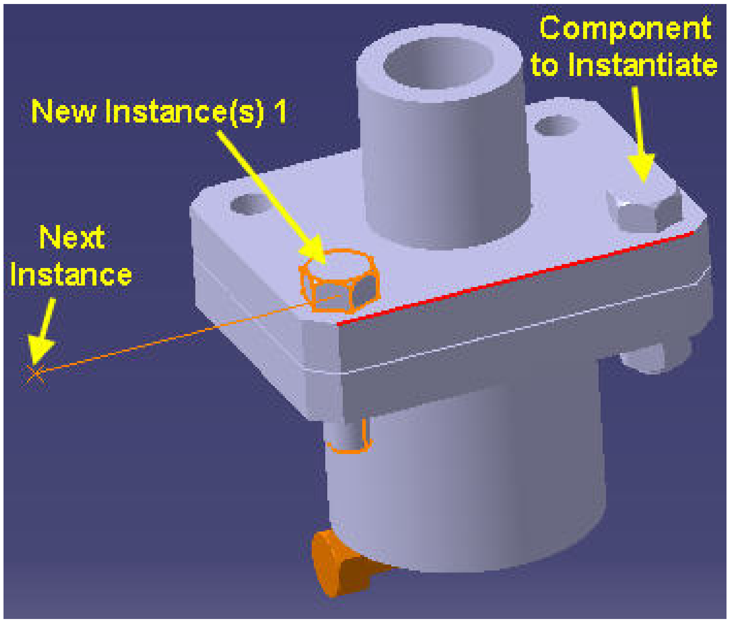




3. Pick the x or y direction as needed to place the second machine screw.

Find the x or y directions in the Compass.

Reverse spacing direction if necessary.



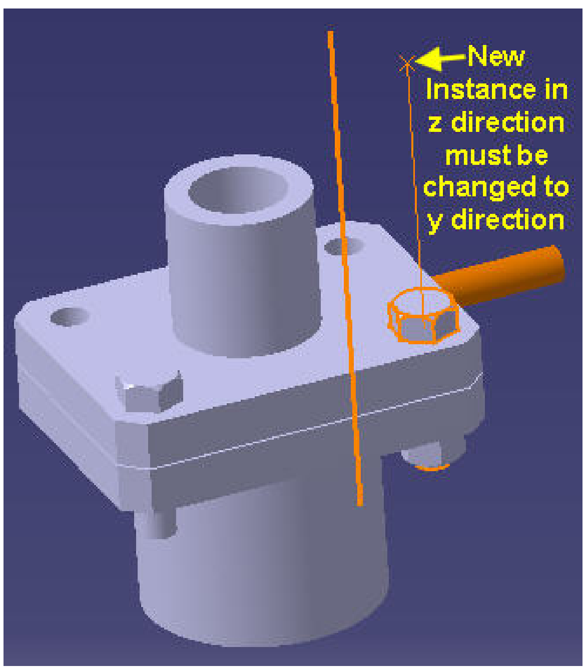
Apply >> Close.

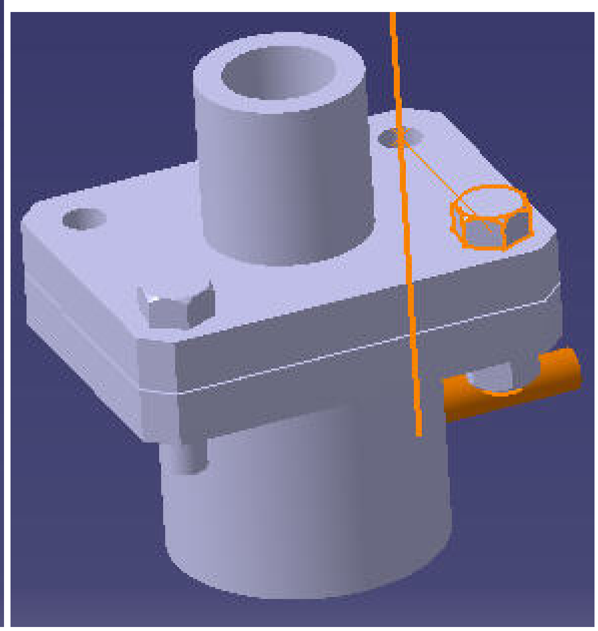
Not: Apply >> OK, because the OK will create one more instance.

Delete the extra Instance of the machine screw if you press OK and it appears.

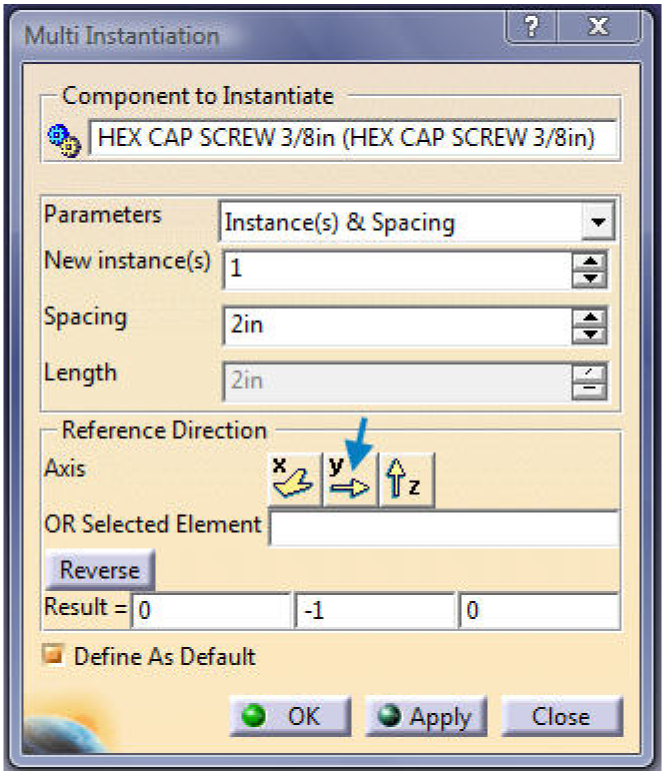
4. File >> Save

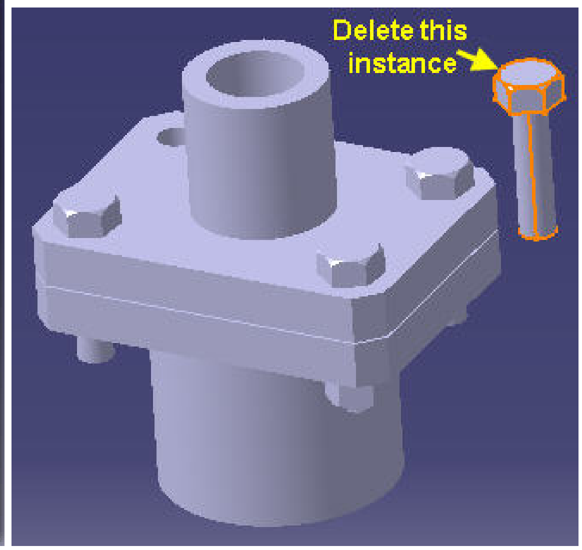
The "Component to Instantiate" has been changed by Catia to the new (HEX HEAD CAP SCREW 3/8in.1)





5. Pick the 1st Hex Head Screw

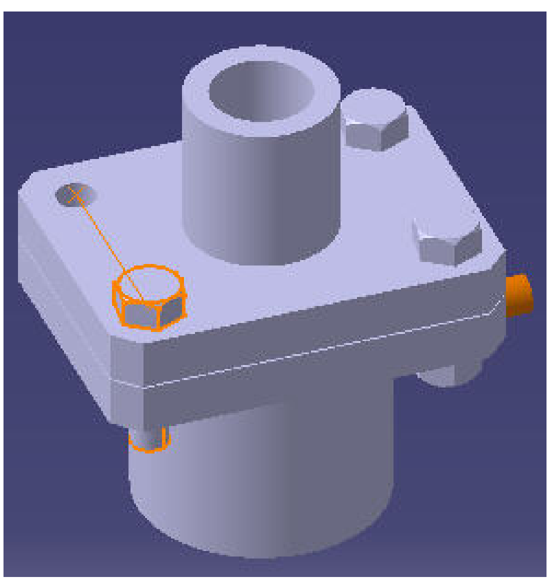


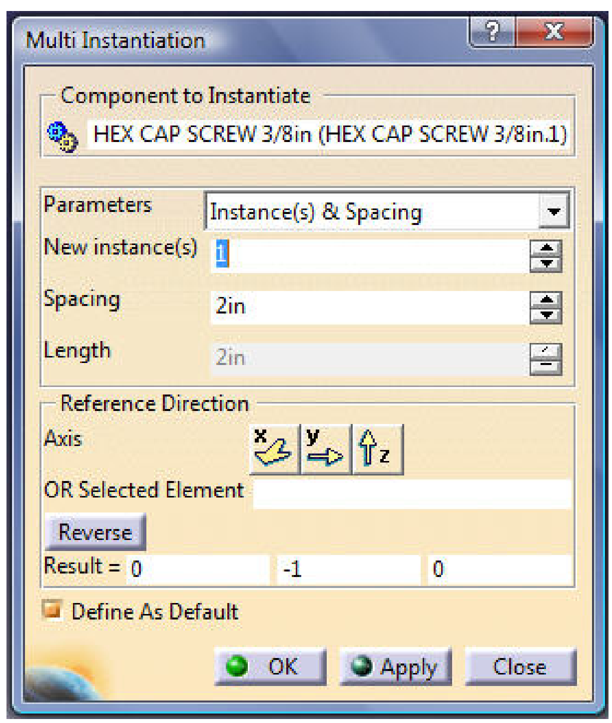


6. Apply >> Close.

Not: Apply >> OK, because the OK will create one more instance.

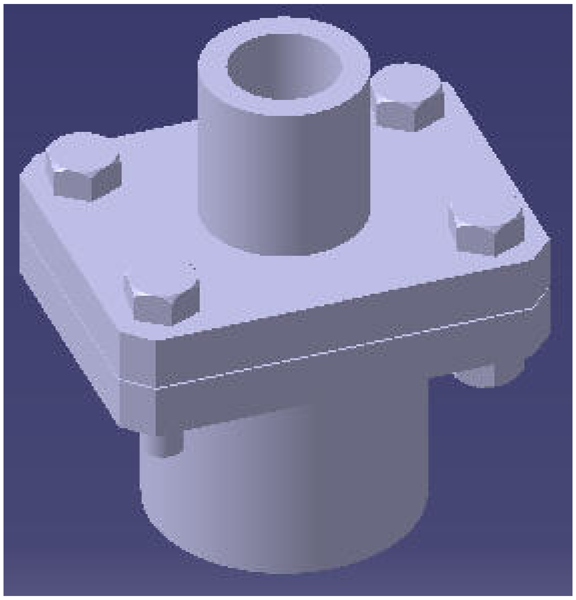
Delete the extra Instance if you press OK and it appears.

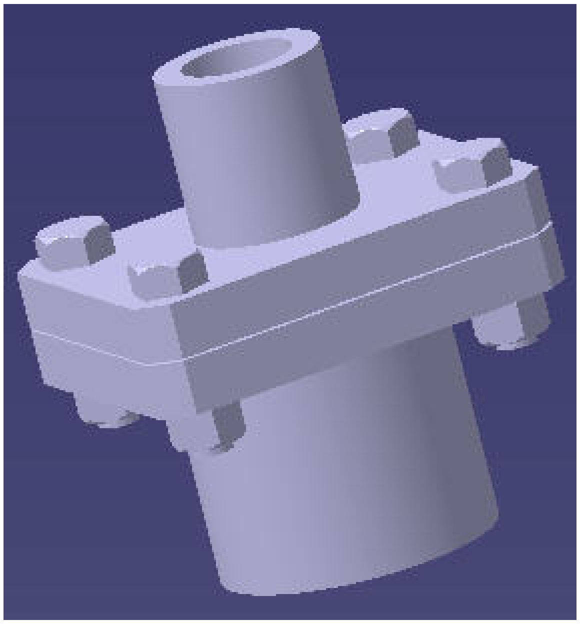




6. Pick the 2nd Hex Head Screw >> Define Multi-Instantation >> Spacing >> 2in >> Pick the empty hole center shown above.

Apply >> Close.



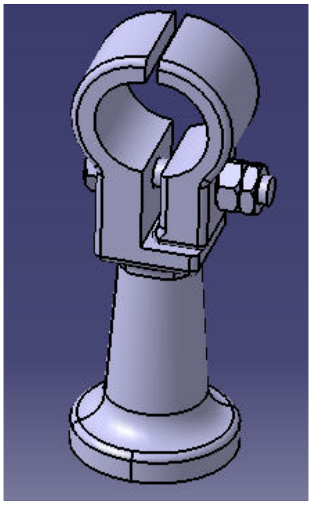


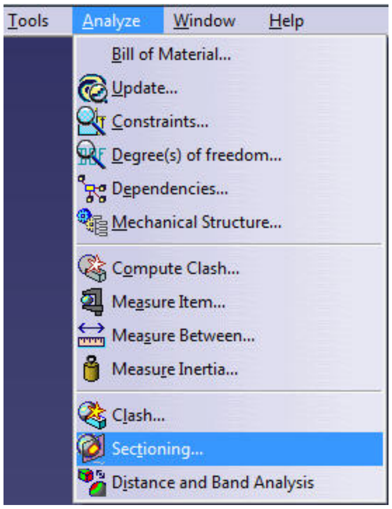
All 4 Hex Head Screws assembled.

Repeat the method above to Define Multi- Instantation the 4 Hex Head Nuts.

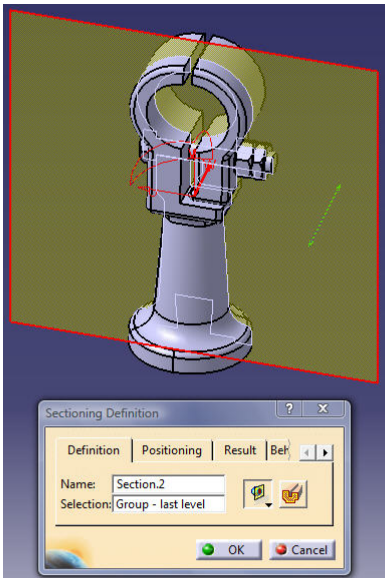
The complete "Pipe Reducer Assembly" is shown above right.

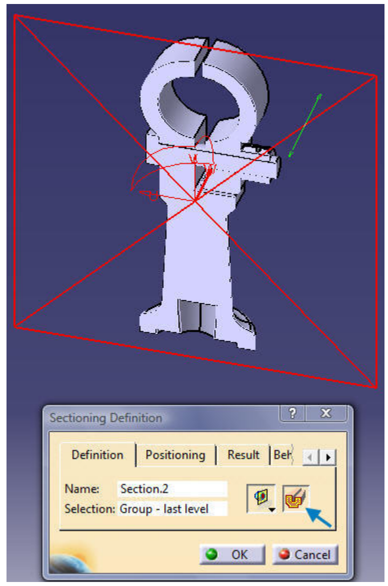
8. DYNAMIC SECTION VIEW





Open a part or assembly. **Pick: "Analyze >> Sectioning…**





Click at the approximate green arrow location to change section direction.

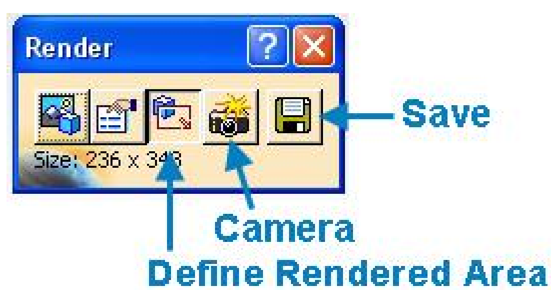
Pick Volume Cut icon at blue arrow.

Move Section dynamically with mouse pointer.

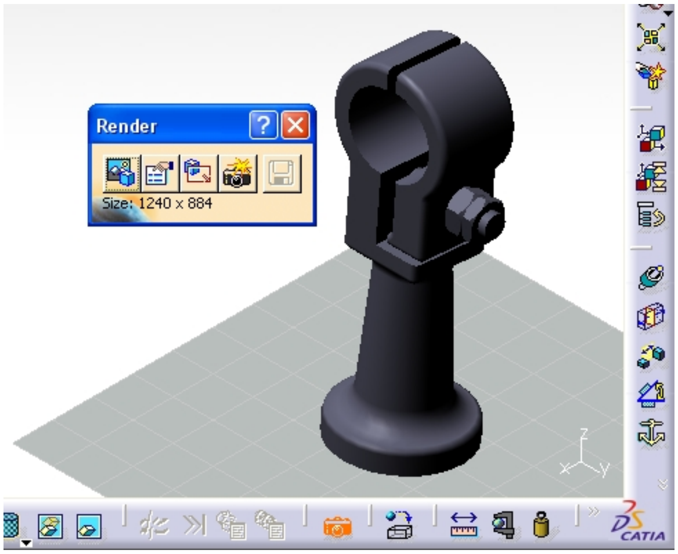
9. CATIA CAMERA JPG PICTURES

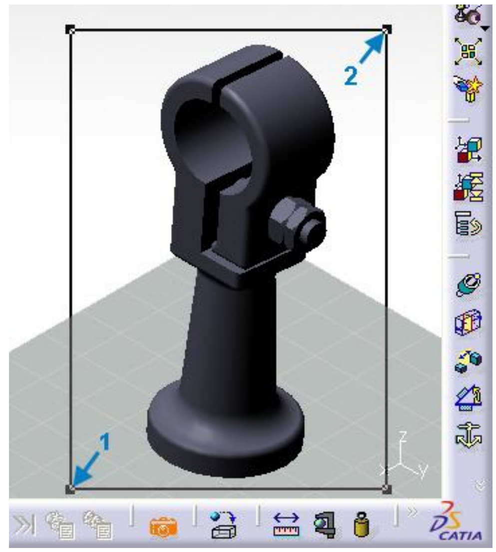
Objective:

Make a .jpg picture file of the Catia assembly left.



Pick the "Camera" icon and the "Render" toolbar above will open.

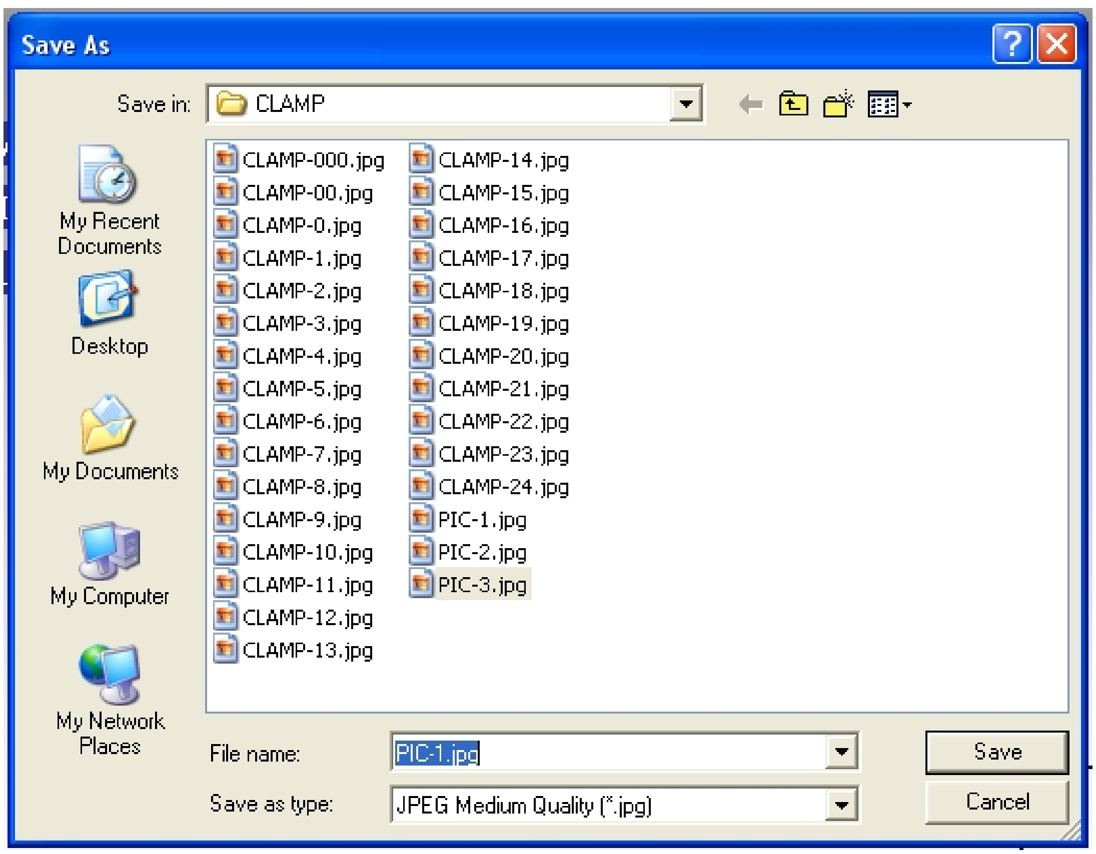




Pick the "Define Render Area" icon.

Pick point 1 and drag the mouse pointer to point 2.

Pick the camera icon again.



Save as a .jpg or other picture format.

END OF COURSE CONTENT