



PDHonline Course G351 (10 PDH)

**CATIA-5 PART-B: 3D CAD,
Mechanisms and Finite Element
Analysis**

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2020

PDH Online | PDH Center

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CATIA-5 PART-B: 3D CAD, Mechanism and Finite Element Analysis

John Andrew, P.E.

SOME OF THE 20,000+ COMPANIES USING CATIA

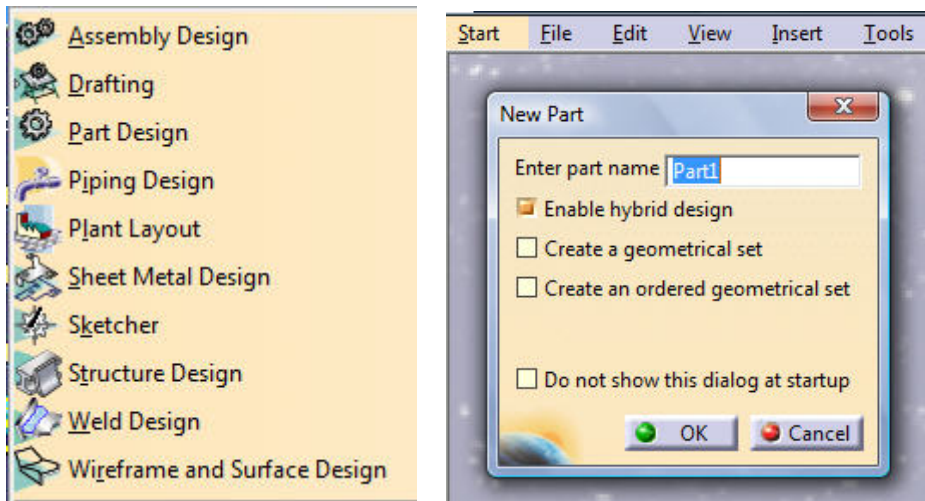
Boeing, Air Bus, KelseyHayes, Lear Jet, Northrop Grumman
BMW, Daimler Chrysler, Volvo, Toyota, Ford, Honda Hyundai
Ferrari, Lockheed Martin, Porsche, Fiat Peugeot, Mercedes-Benz
Freightliner, Allied Signal, Volkswagen, Pratt Whitney, United Airlines
Black and Decker, Goodyear

- 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.

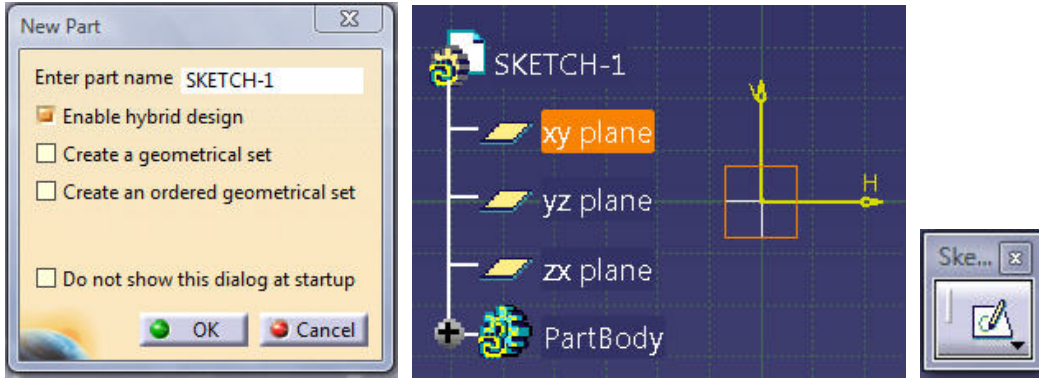
1- CATIA SKETCH WORKBENCH

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

START A CATIA SKETCH

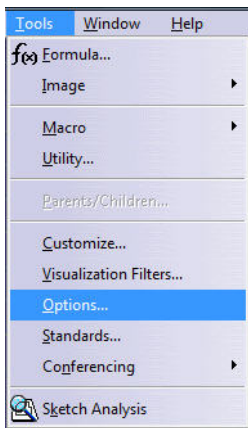


Open Catia and pick: Start >> Part Design >> Edit "Part1" >> SKETCH-1.



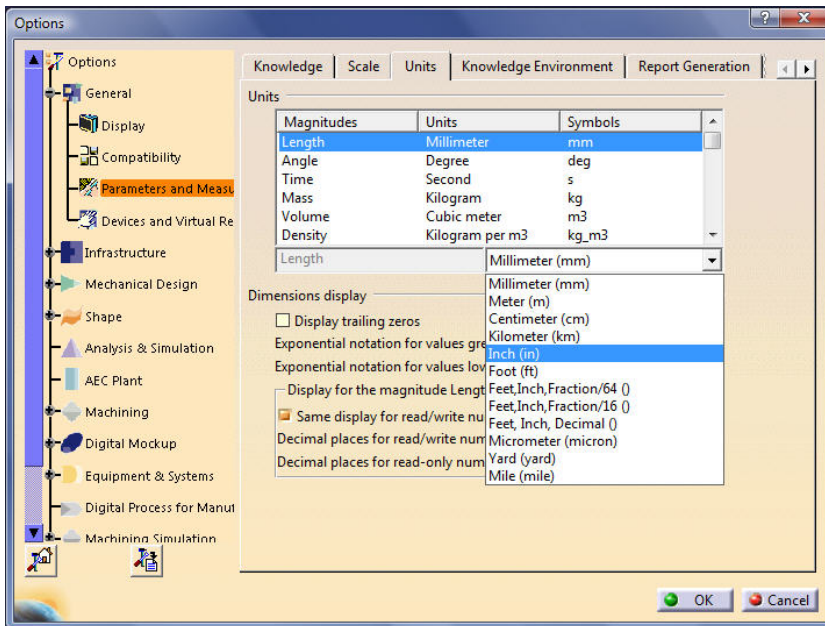
Pick the “xy plane” in the Tree with the left mouse pointer. Pick “Sketch” tool.

Always pick or drag with the left mouse pointer unless stated otherwise.



Pick drop down menu: Tools >> Options.

D. CHANGE UNITS OF LENGTH



Select drop-down menu:

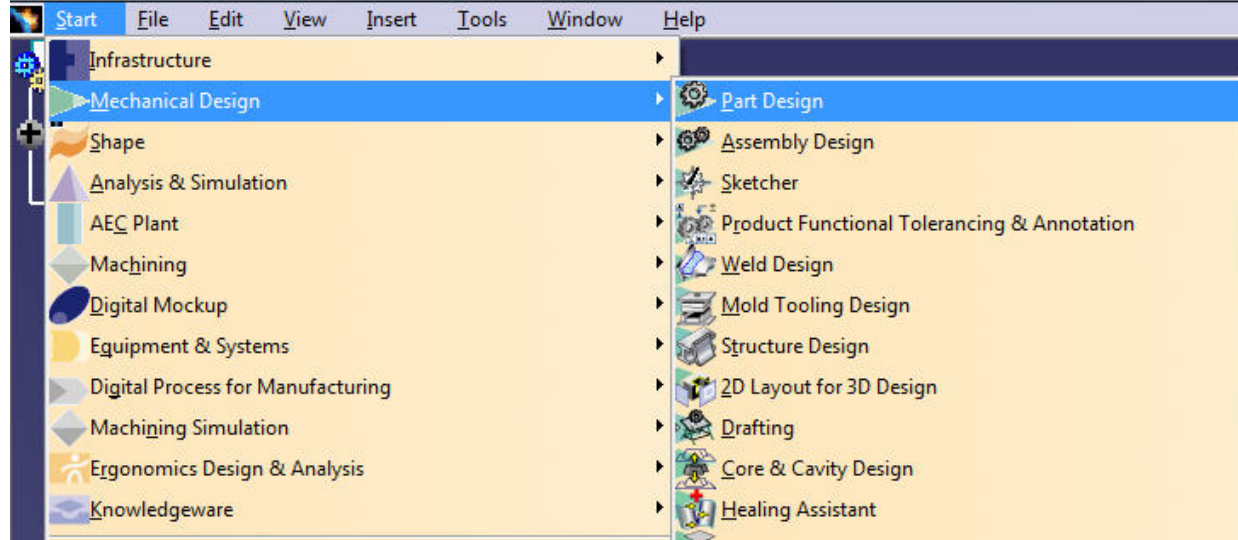
Tools >> Options >> Parameters & Measure >> Units >>
Select Inches or Millimeters, as shown above.

E. CHANGE AUTOMATIC BACKUP INTERVALS

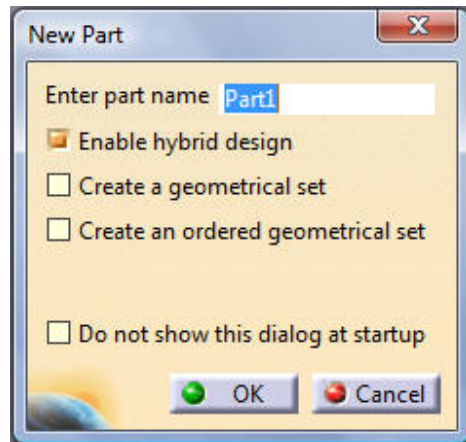
Automatic save is obtained at: Tools >> Options >> Automatic backup every >> 10 minutes.

START A NEW PART

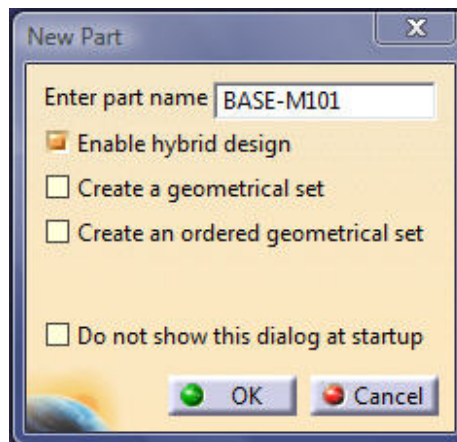
Create a new component of one assembly or a new part that may be inserted into any assembly.



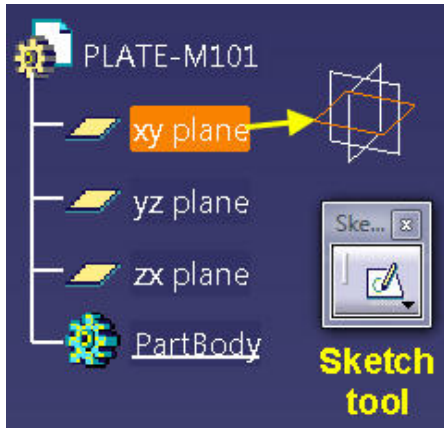
Select: Start >> Mechanical Design >> Part Design



Edit "Part1" above to BASE-M101

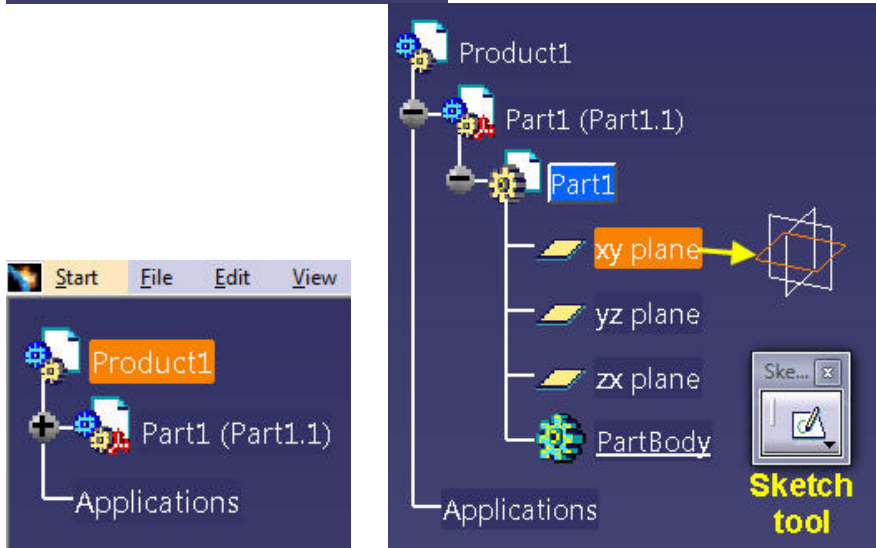


"BASE-M101" has been entered



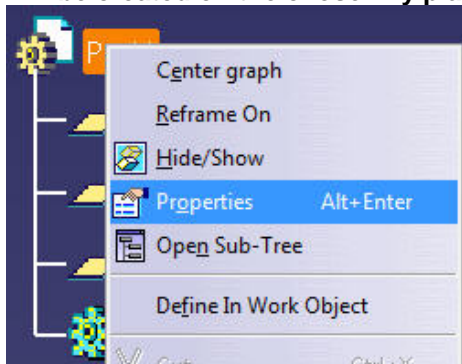
If the part name is not changed
Catia will provide the name "Part1".

The part name can be changed
later as shown below.

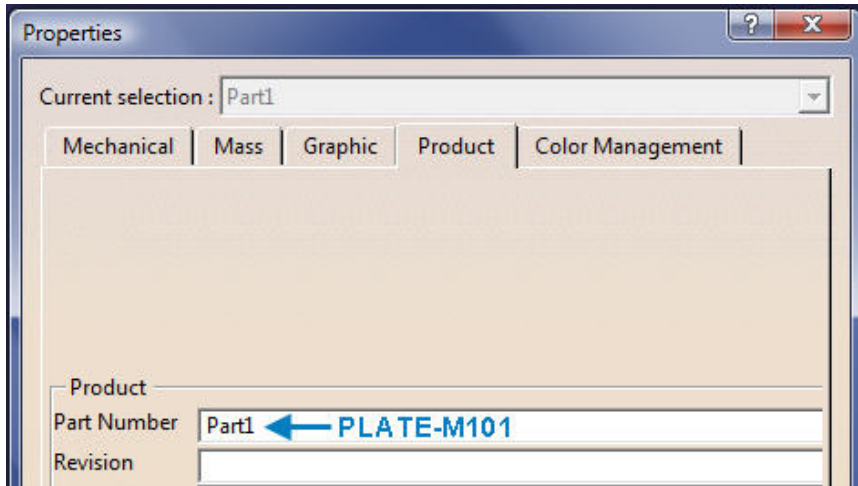


Pick the "+" sign to expand the "Specification Tree"
Pick one of the 3 planes to make a sketch on.
Pick the "Sketch" tool >> "xy" plane in this example.

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

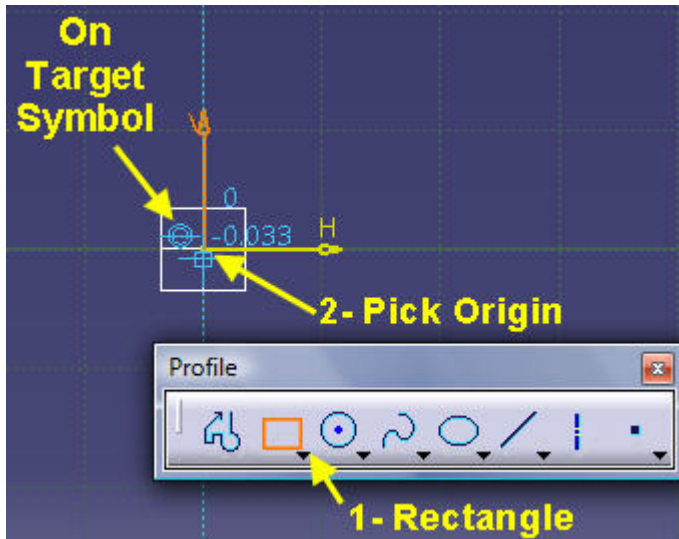


To change the Part name at any time, right click on the title at the top of the specification tree >>
Click on "Properties" >>

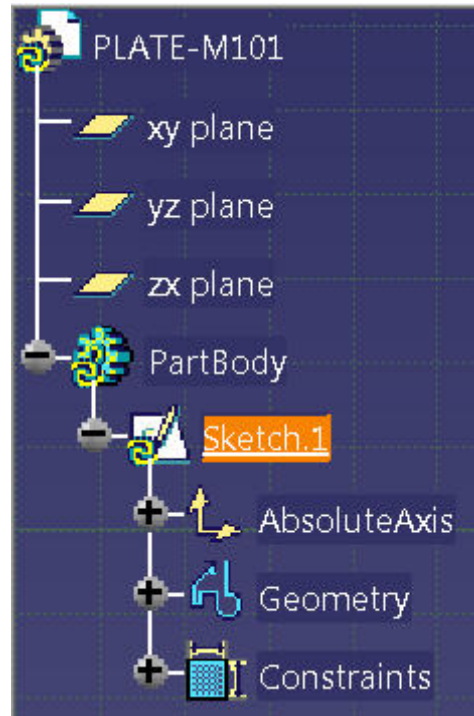


>> Select top tab “Product” >> edit Part Number >> PLATE-M101.

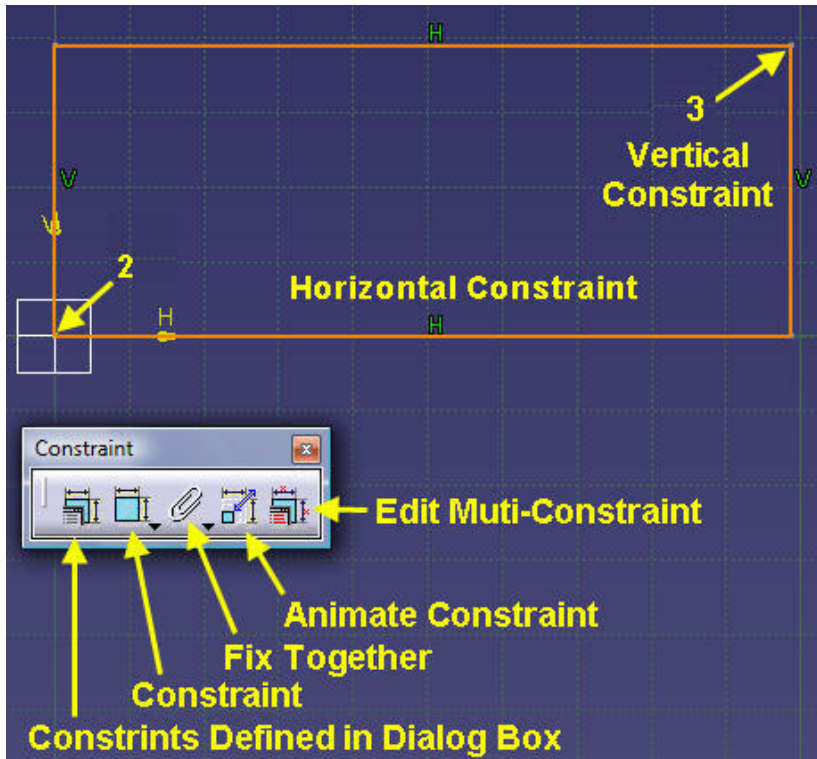
SKETCH A RECTANGLE



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.

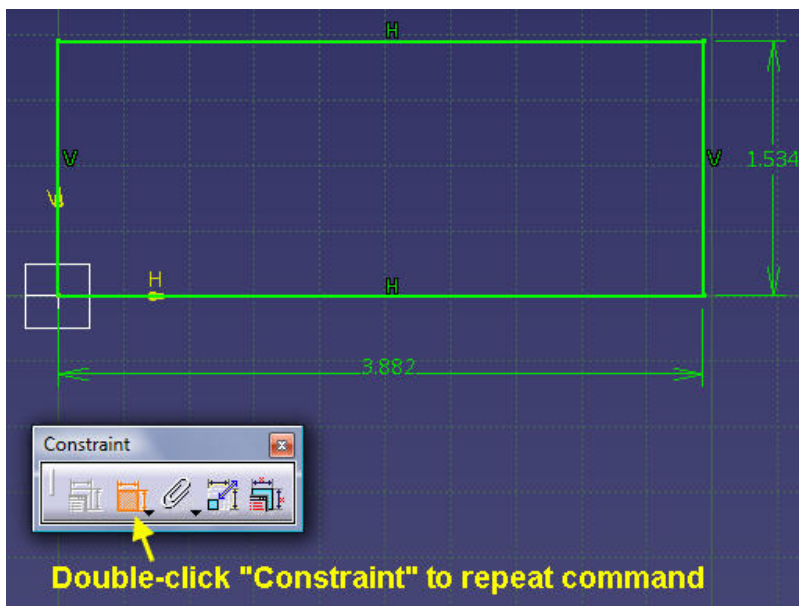


Create the above rectangle by dragging the mouse pointer from point 2 to 3.

AUTOMATIC CONSTRAINTS:

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

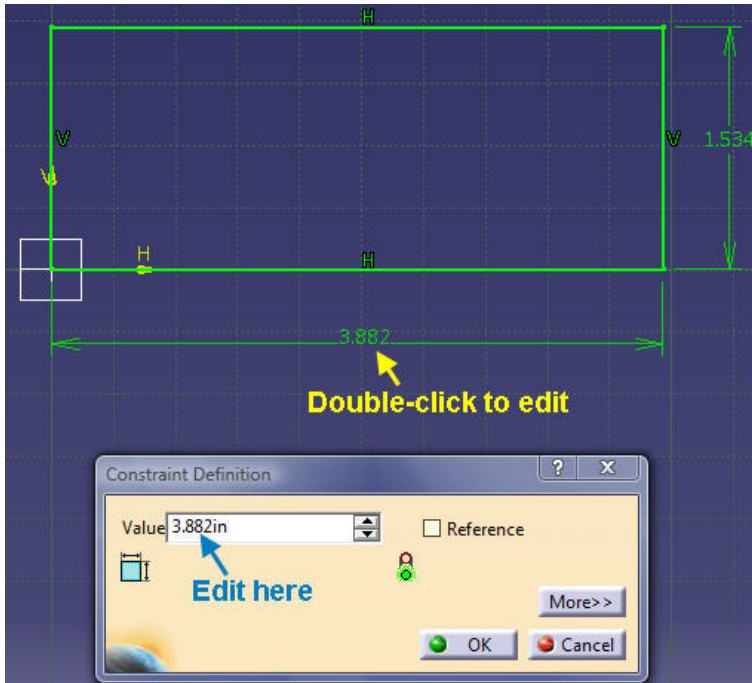
ADD DIMENSIONS



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

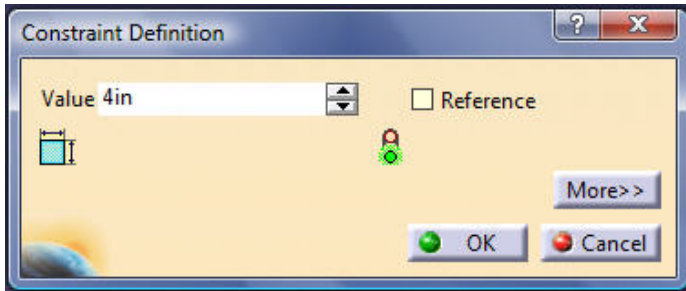
Pick the bottom horizontal line and place its dimension below.

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

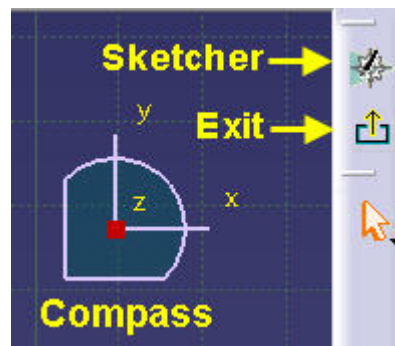
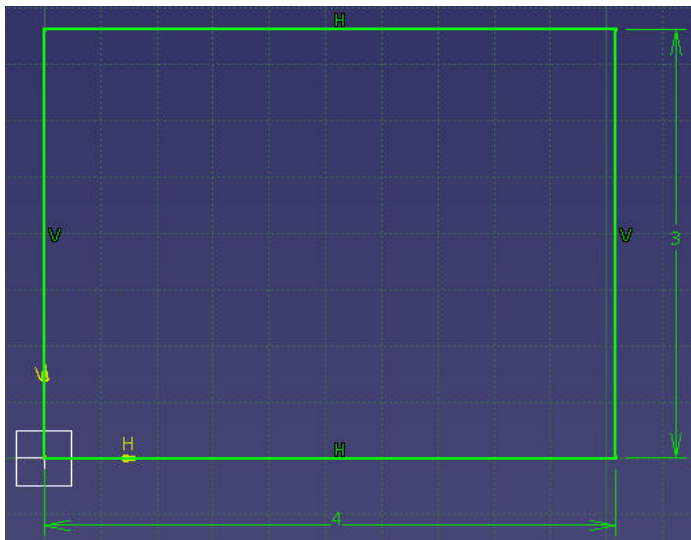


EDIT DIMENSIONS: Double-click the horizontal dimension (3.882 inches left).

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



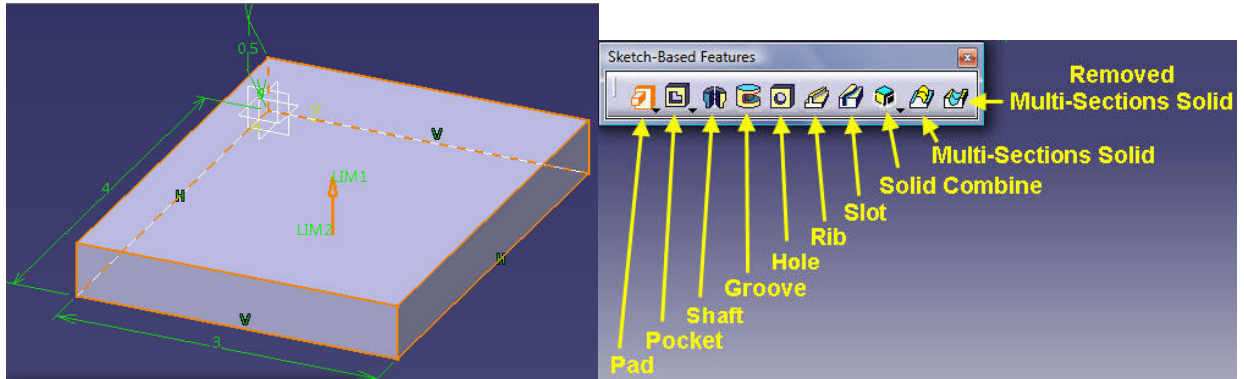
The 3.882 inch dimension has been changed by typing, "4" >> OK.



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

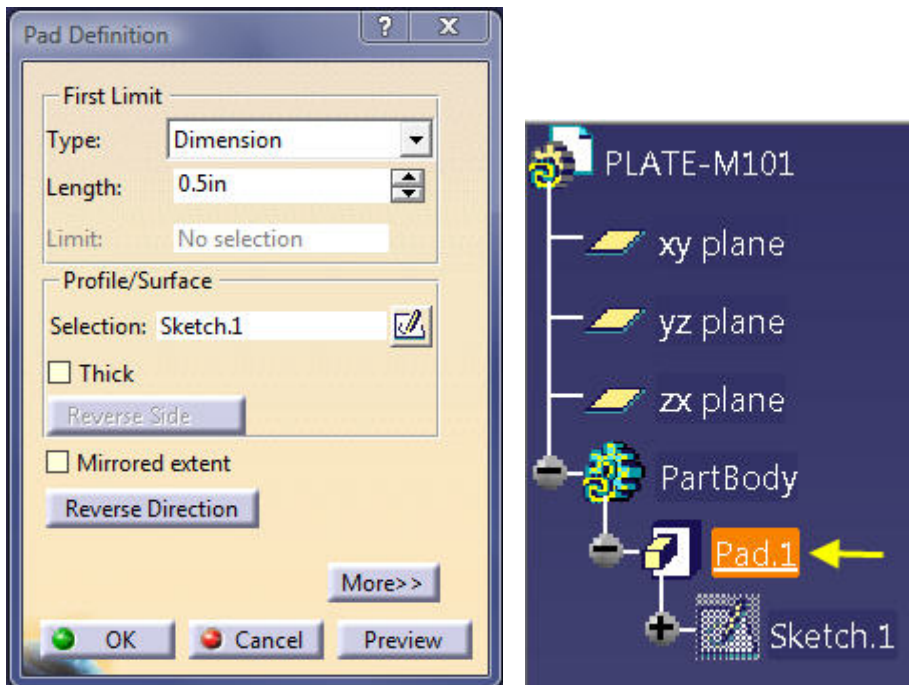
Pick the "Exit" tool to leave the "Sketcher Workbench".

ENTER THE 3 DIMENSIONAL ZONE

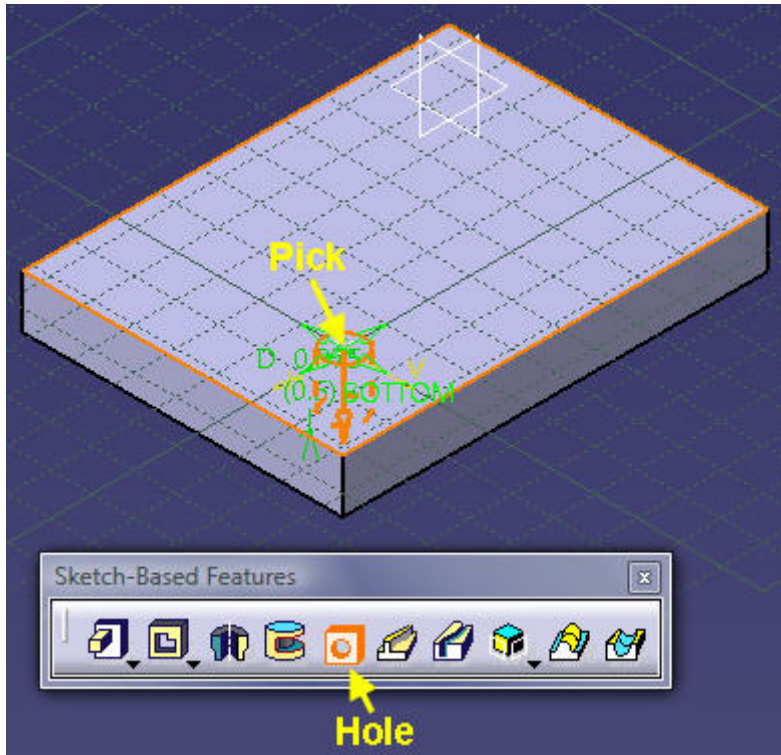


With the mouse pointer select “Pad” on the “Sketch-Based Features” toolbar above.

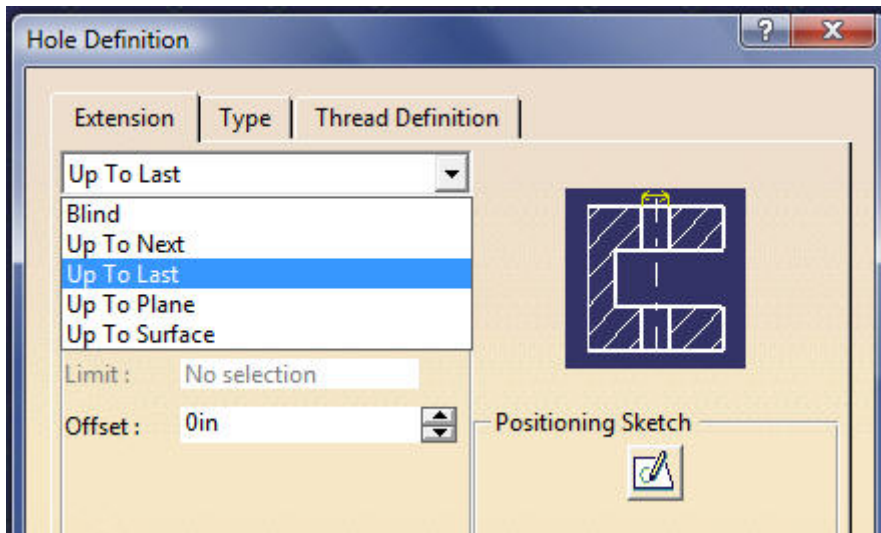
Each tool on the “Sketch-Based Features” toolbar represents its function.



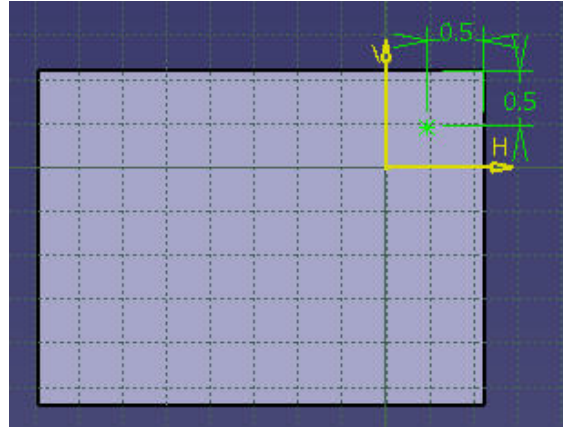
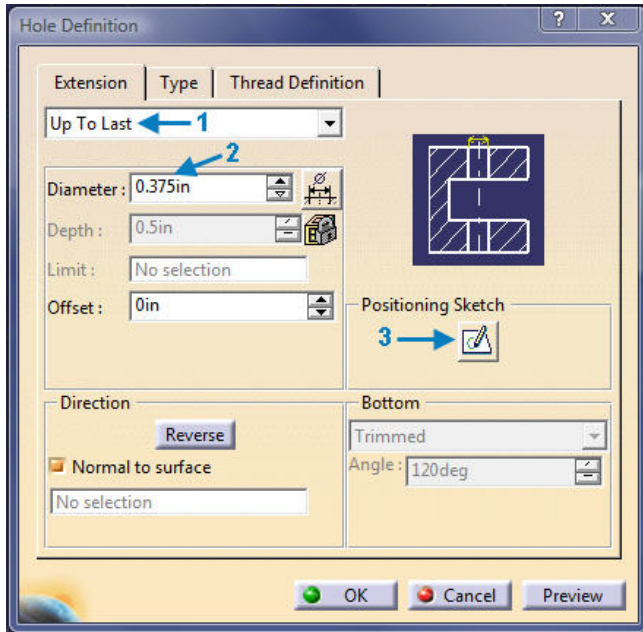
Set the PAD Length to 0.5in above.



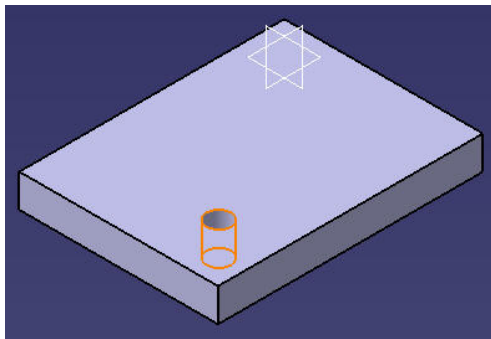
Pick Pad.1 in specification tree. Select: Hole >> Pick hole location

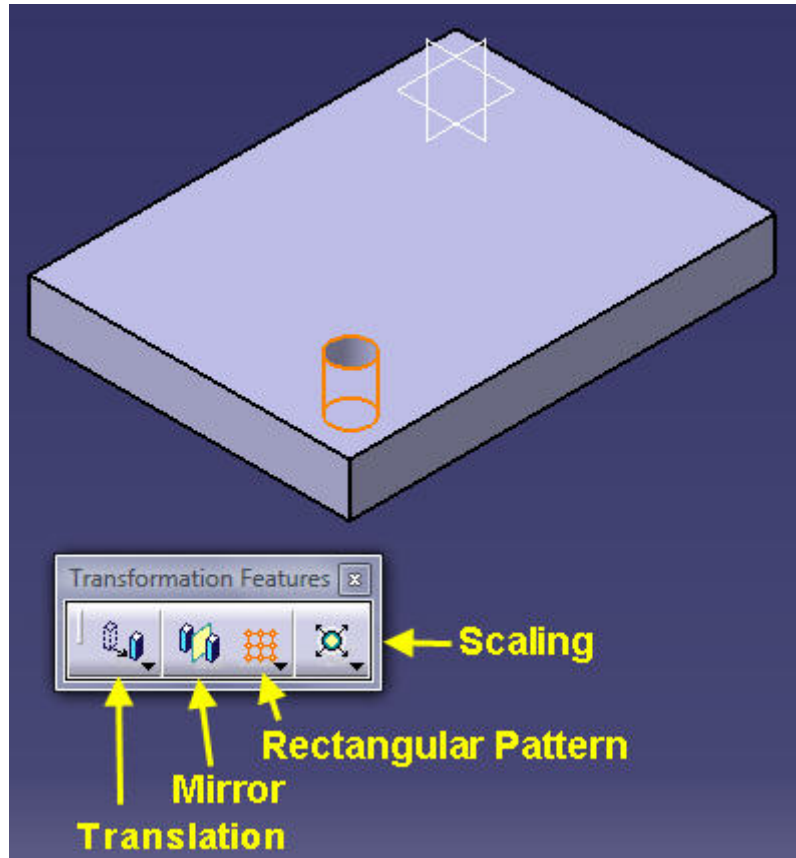


1. Hole Definition box opens >> Drop down "Extension" >> select Up To Last

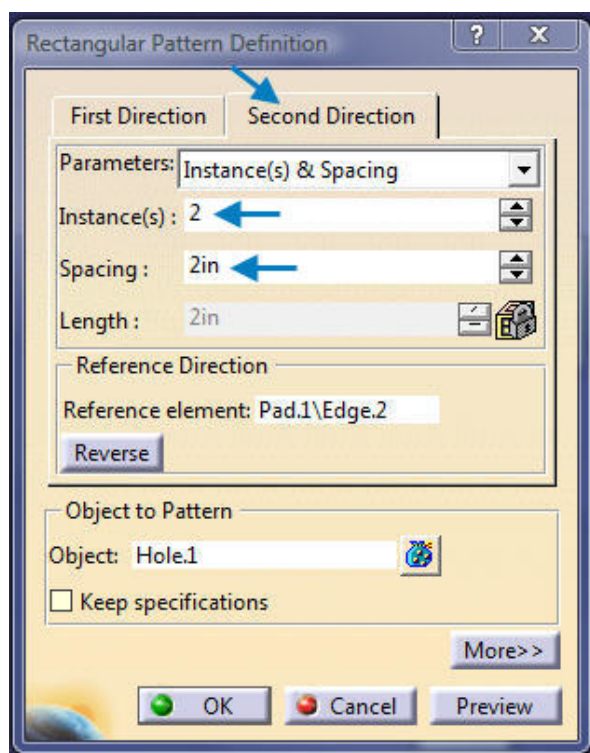
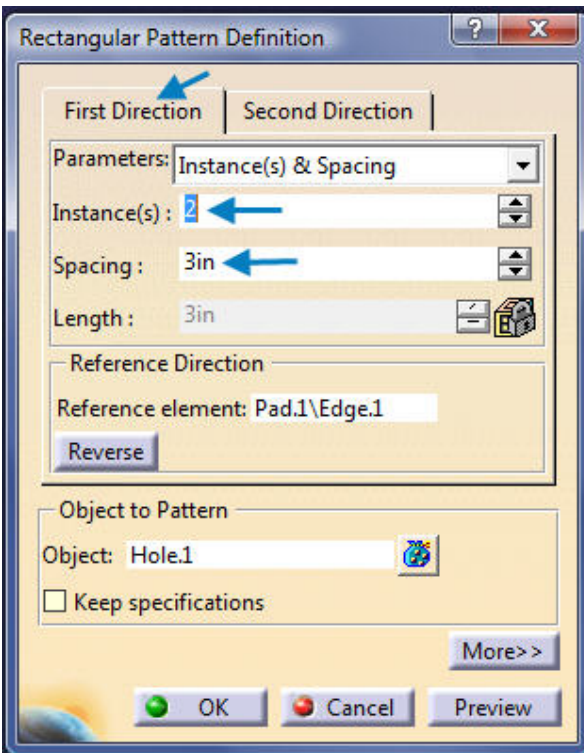


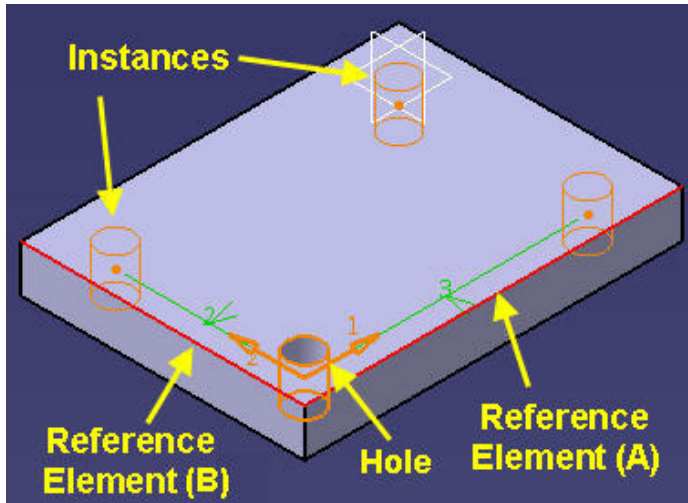
- 2. Edit diameter to: 0.375in as above.
- 3. Pick: "Positioning Sketch" and add hole location dimensions 0.5 and 0.5 as above right.





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

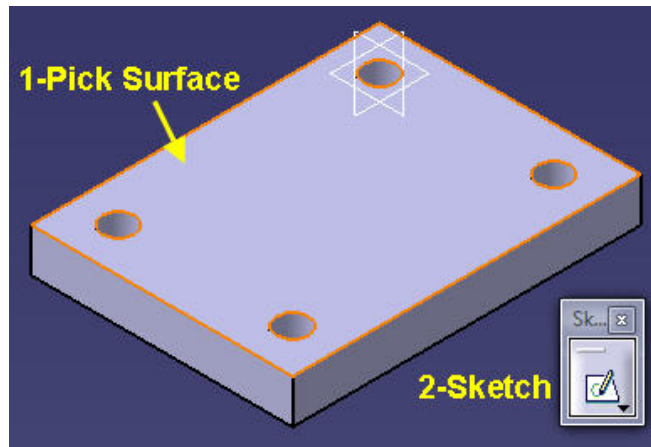
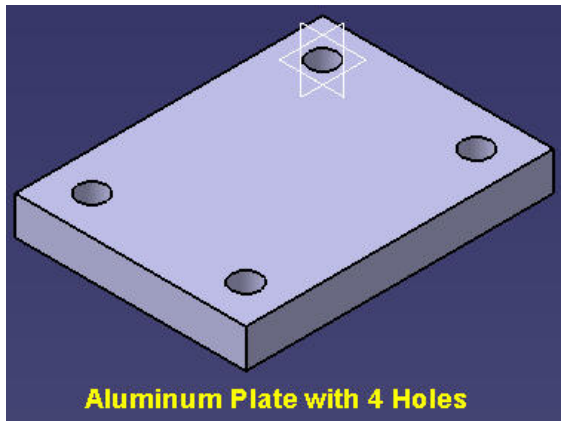




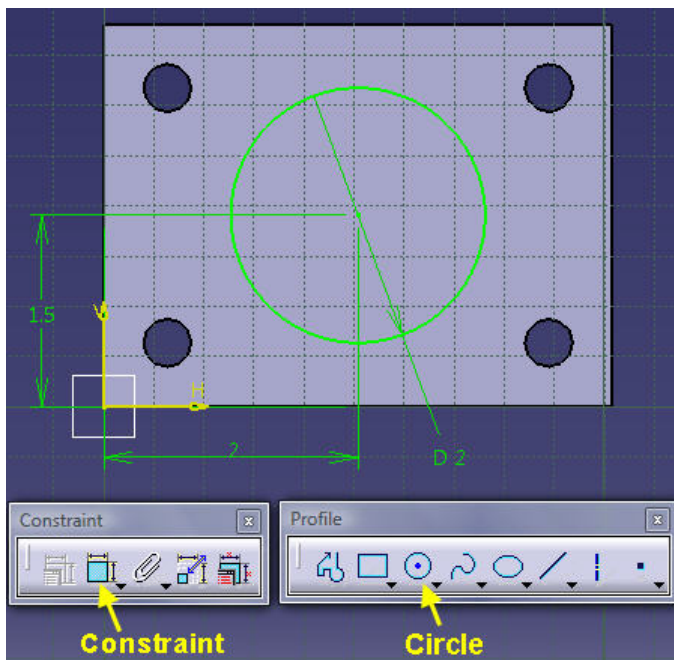
Pick: First Direction >>
Instances >> 2 >>
Spacing >> 3in >>
Reference Element (A)

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

Preview >> OK

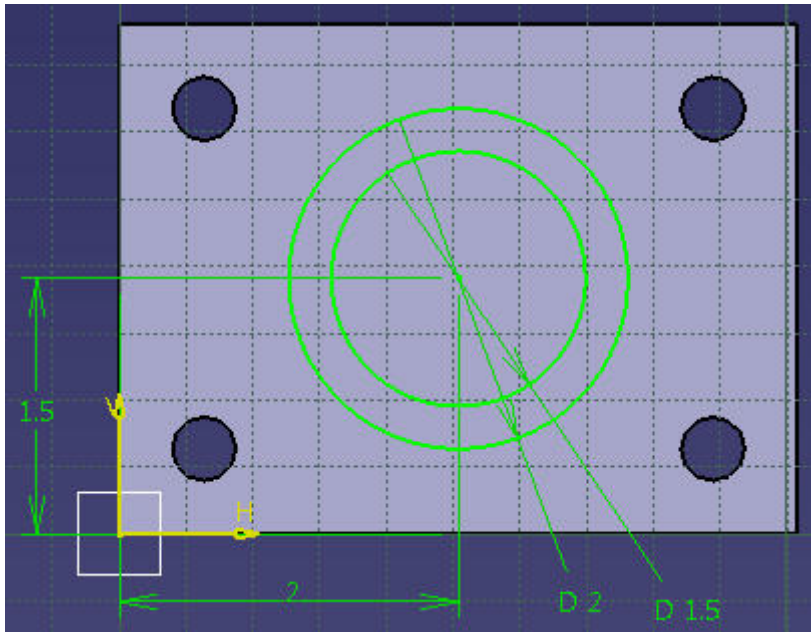


Pick any point on the plate top surface >> Pick the "Sketch" tool.



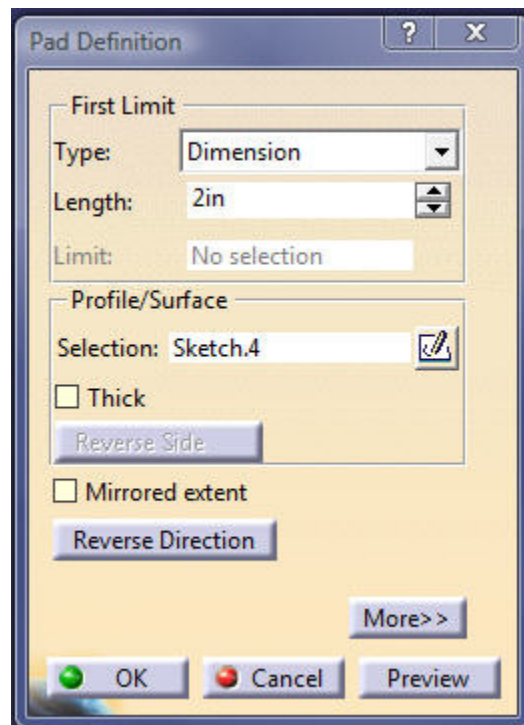
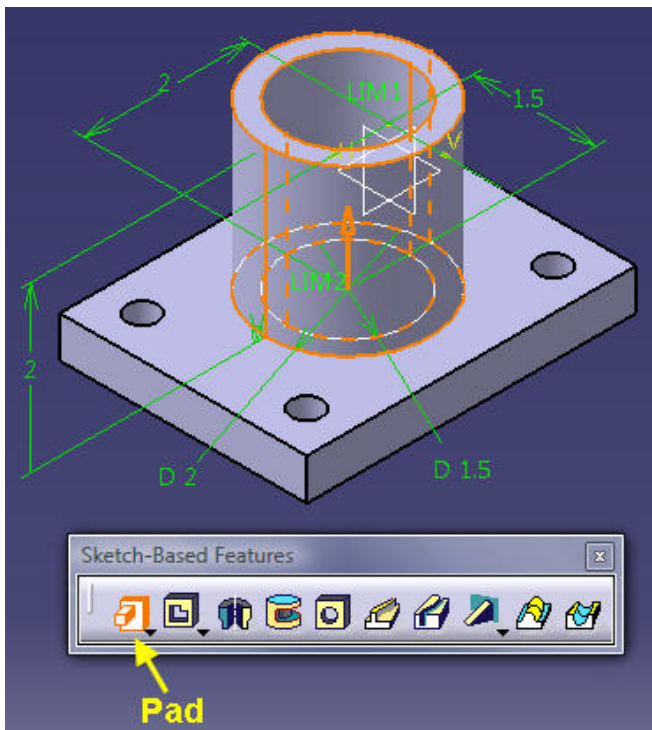
Pick the "Circle" tool on the Profile toolbar.

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

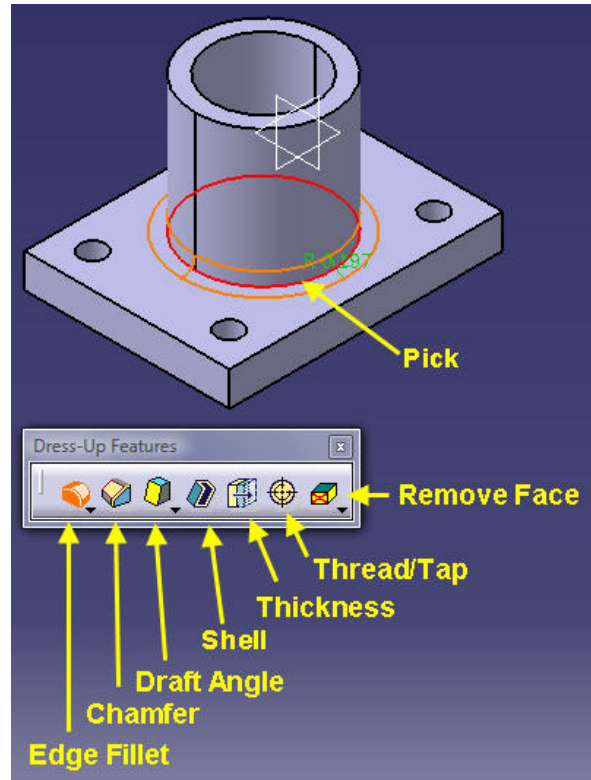
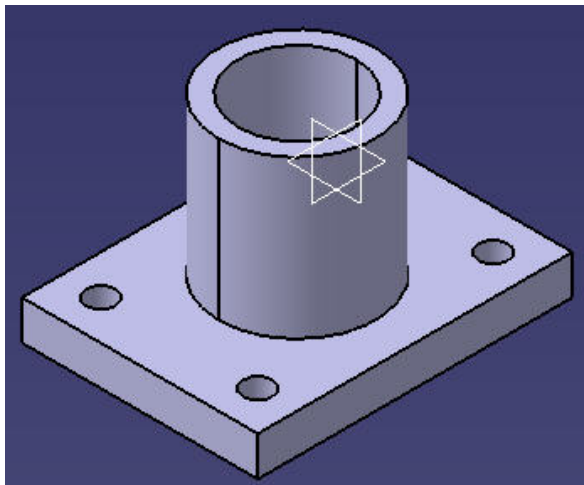


Double click the “Constraint” tool >> Pick the circle center >> Pick the plate bottom edge >> Position the vertical dimension. Create the hole horizontal dimension.

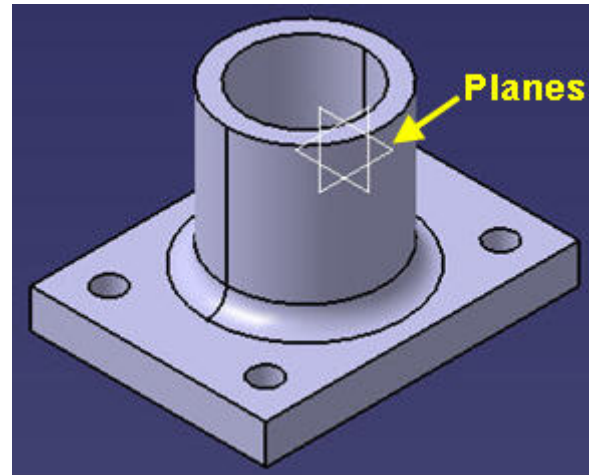
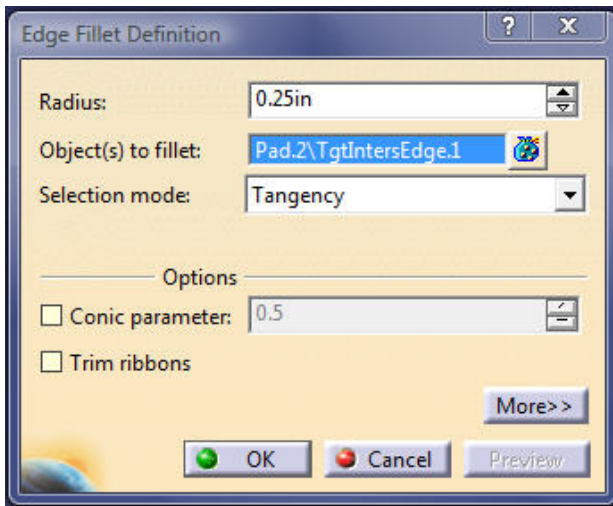
Double click on one dimension and edit as shown above. Edit the other dimension.



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

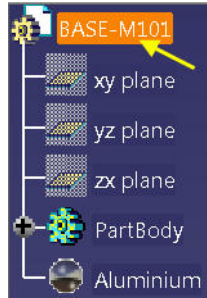
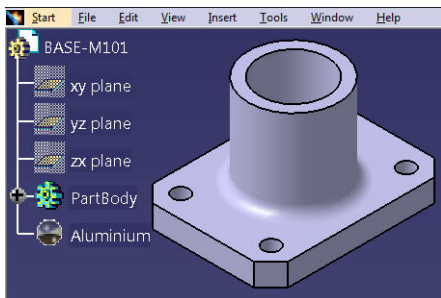


On the "Dress-Up Features" toolbar select: Edge Fillet >> Pick the circle shown above.

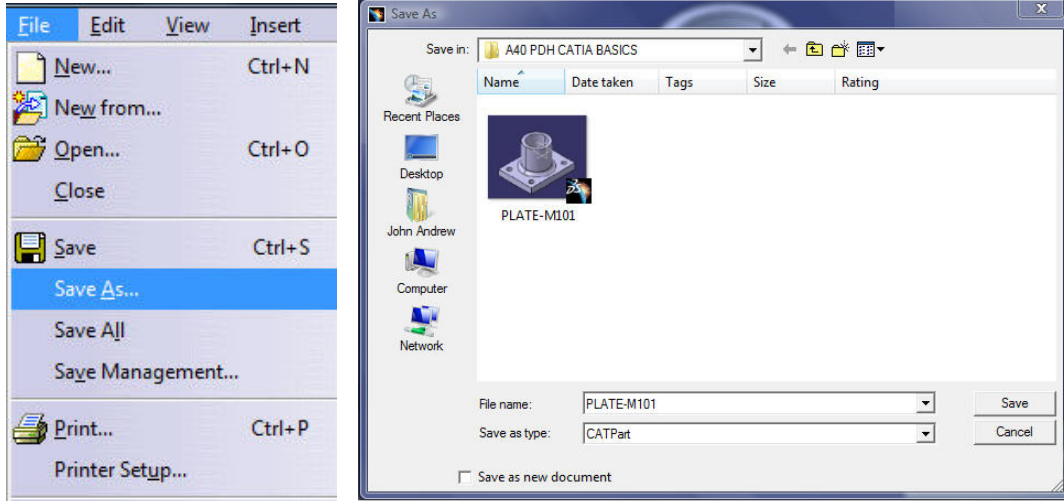


Edit the fillet radius to 0.25in.

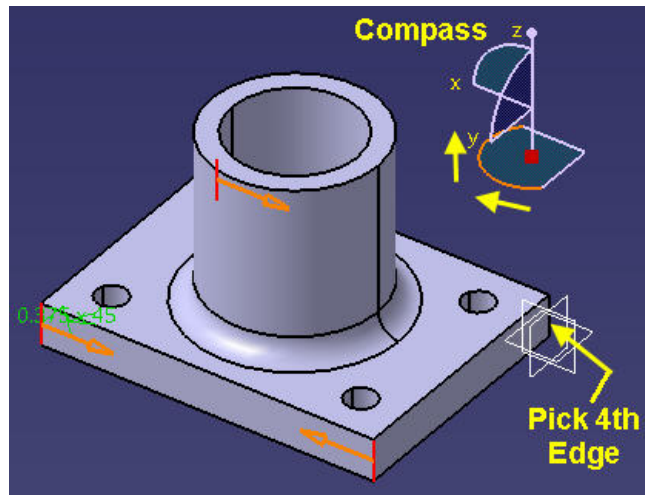
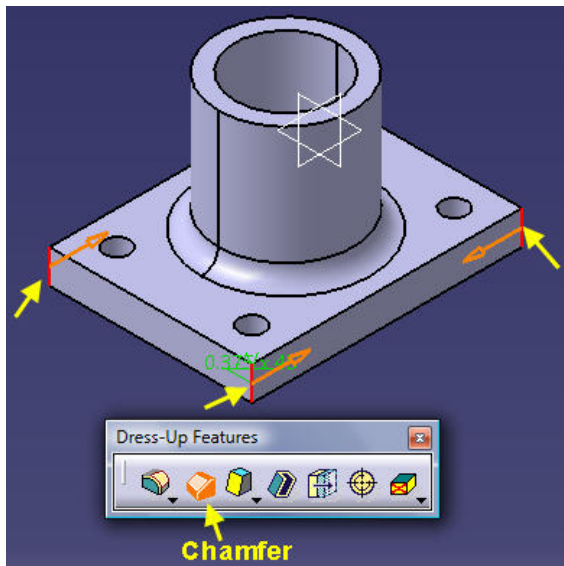
Finished fillet.



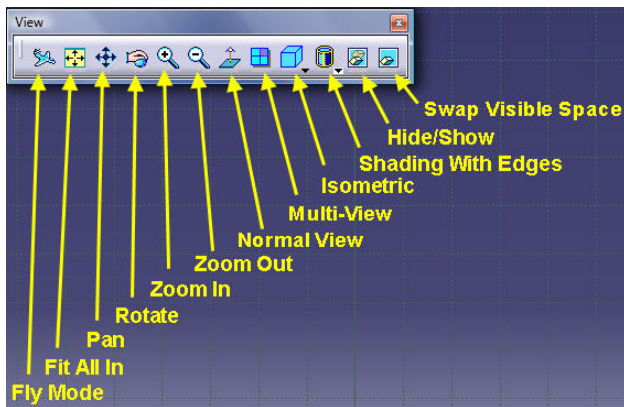
ERASE Planes above.
Hold Ctrl key >> Pick each plane >> Hide / Show.



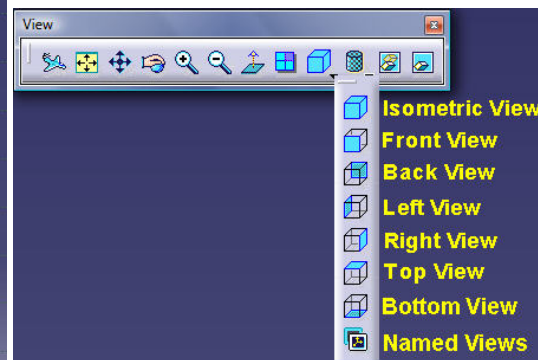
File >> Save As... >> Browse >> Folder >> BASE-M101



VIEW TOOLBAR

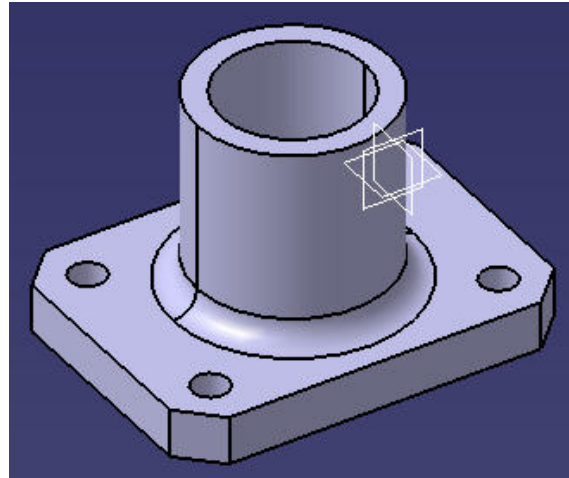
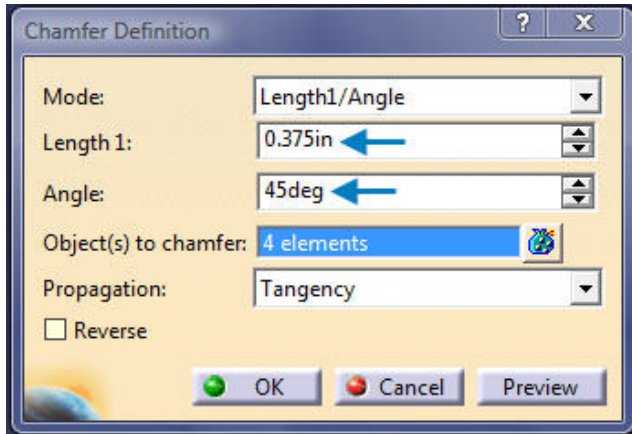


The "View" toolbar tools are listed above.

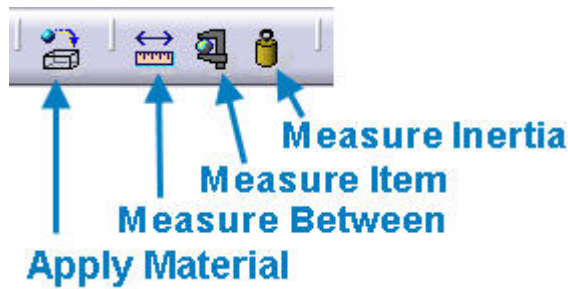
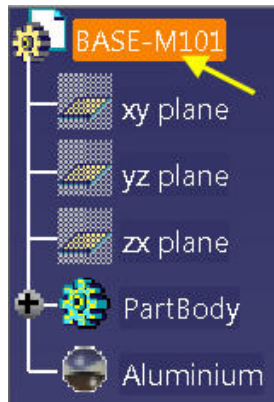


The "Isometric View" drop-down menu.

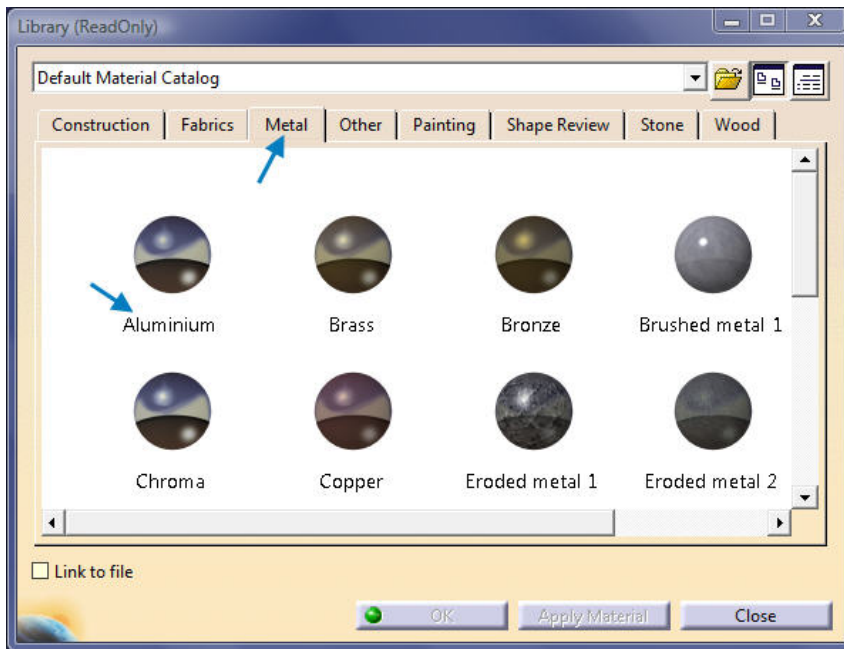
Tools with a “Down Arrow” have additional tools.

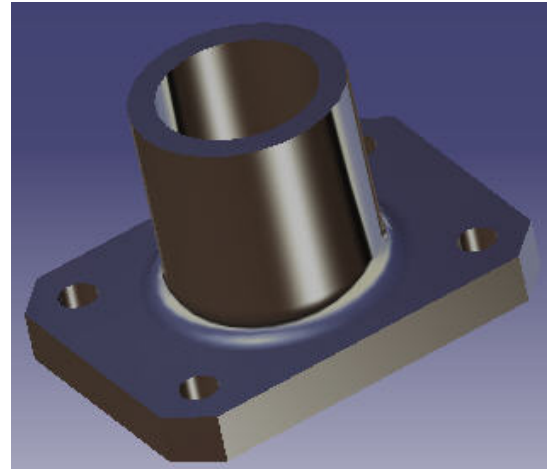
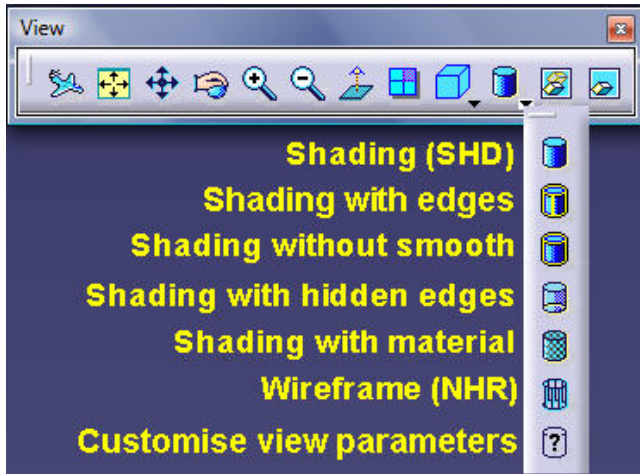


On the “Dress-Up Features” toolbar select: Chamfer >> Length 1: >> 0.375in >> Angle >> 45deg



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

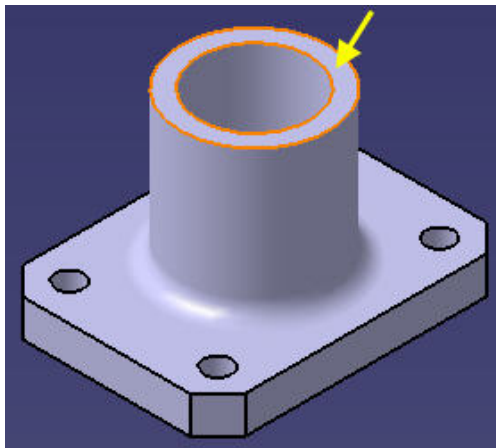




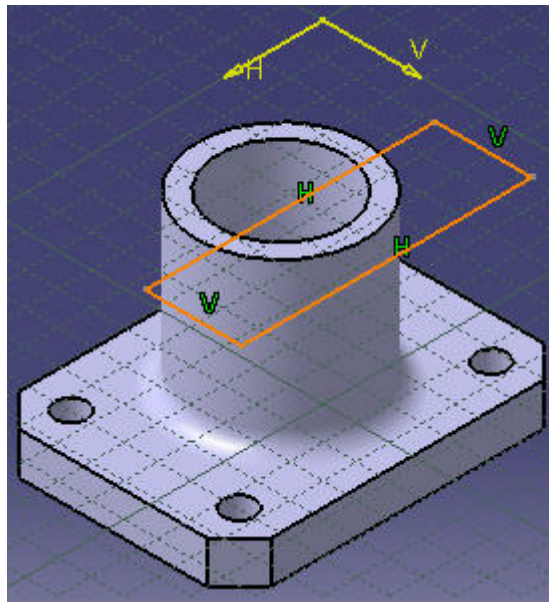
Select "Shading with material" on the "View" toolbar. Result is above right.

POCKET

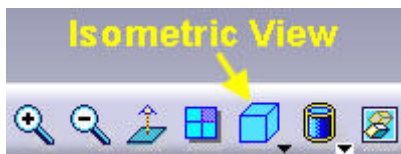
SKETCHING in Isometric View can improve design visualization.



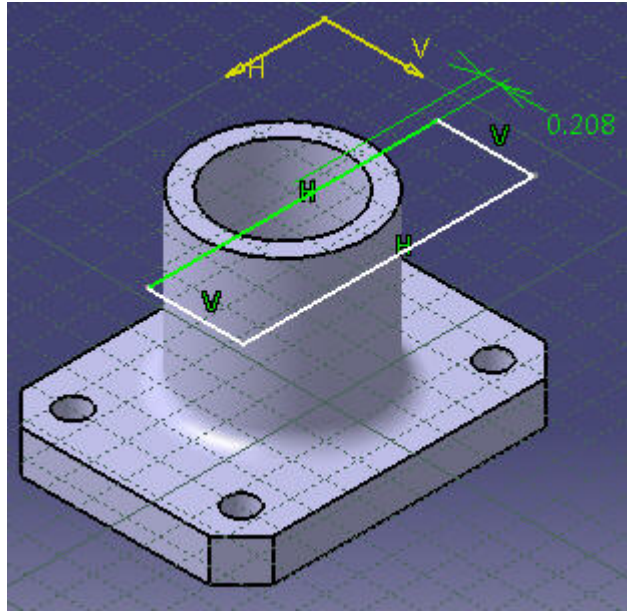
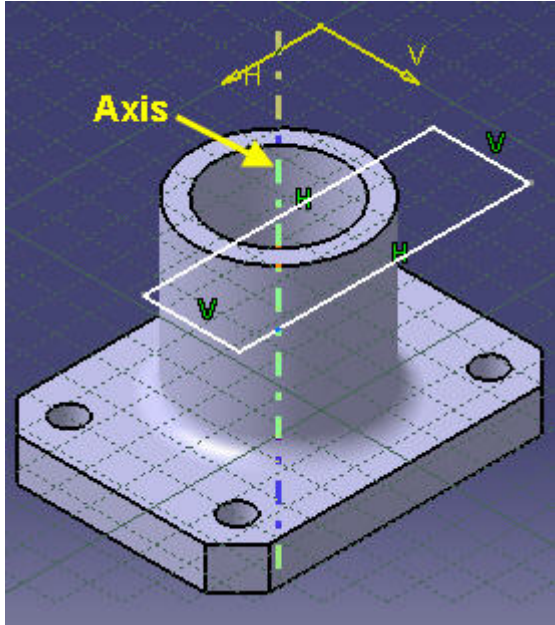
Pick the top surface of the tube.



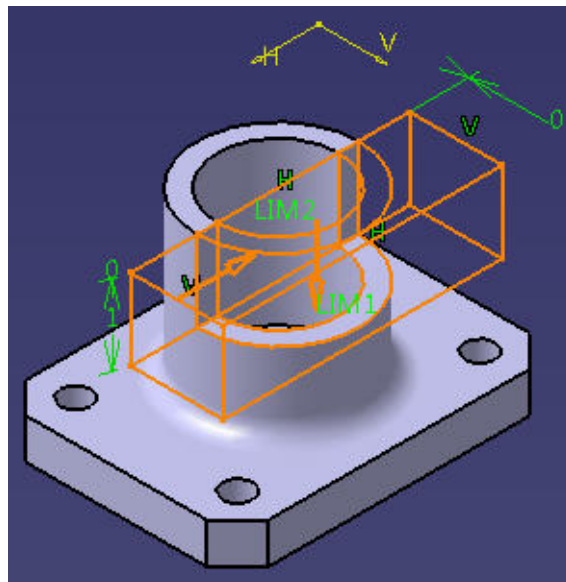
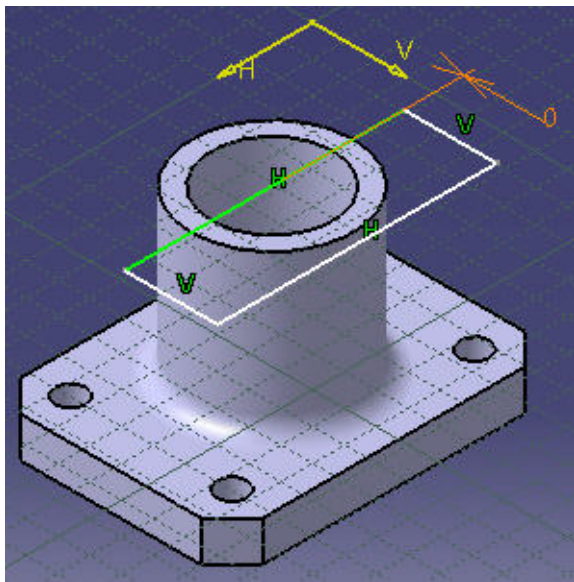
Pick: Sketch tool >> Isomeric View tool >>Rectangle



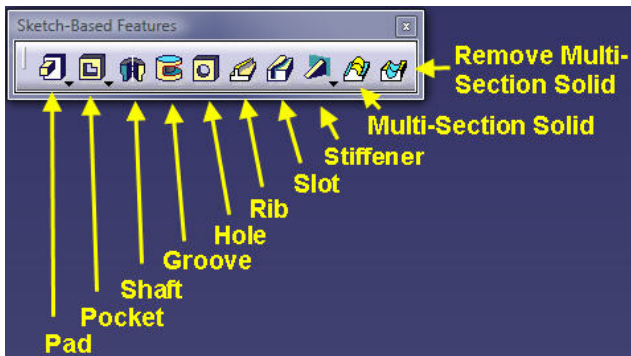
Sketch the rectangle shown above.

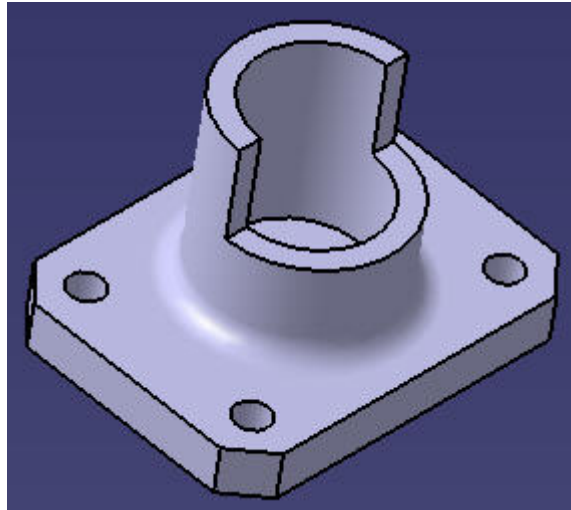
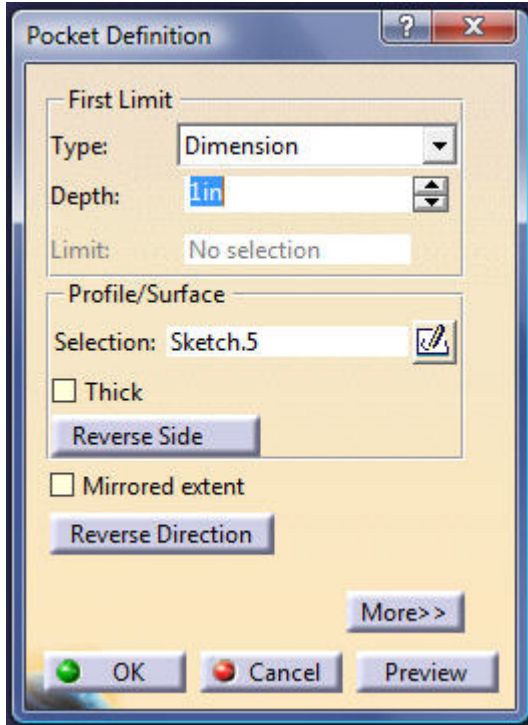


Hover the mouse pointer in the "Axis" area and pick the Axis. Add the dimension to the axis.



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



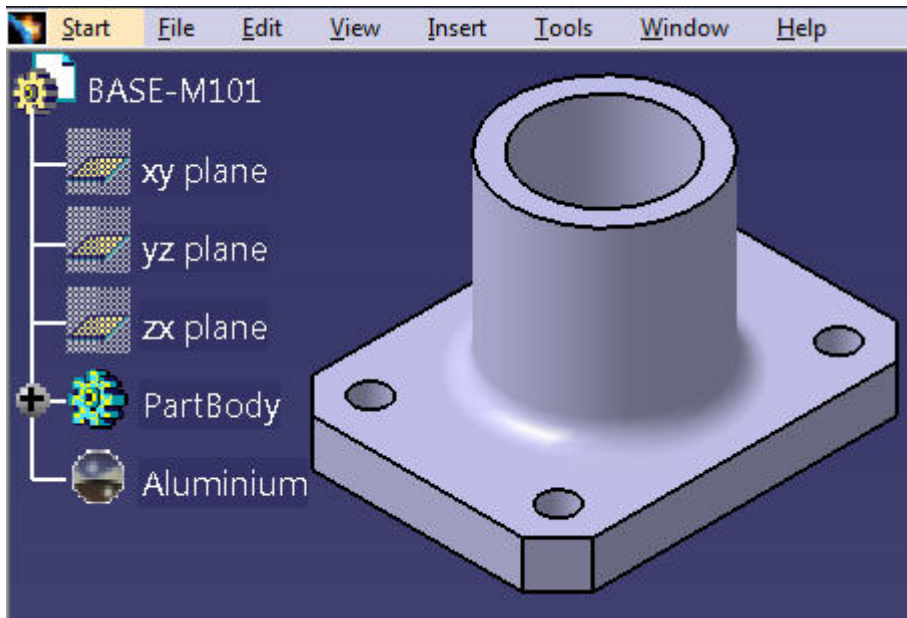


Edit Pocket depth to 1in.

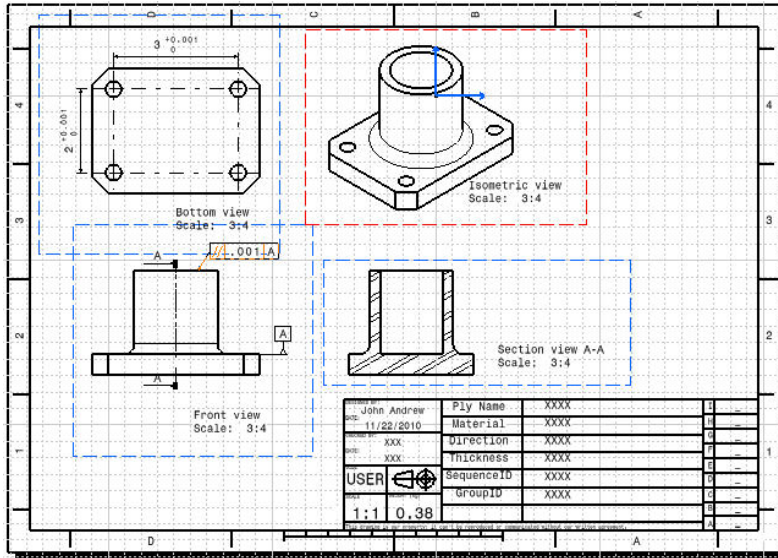
Completed "Pocket".

3- DIMENSIONED DRAWING

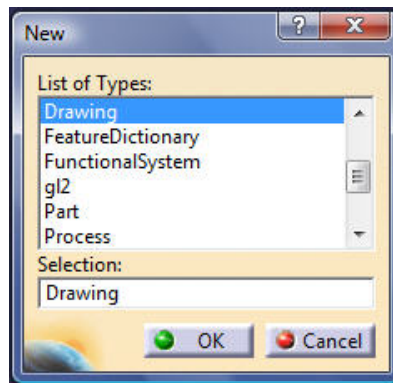
Open a Catia part or assembly before creating the drawing.



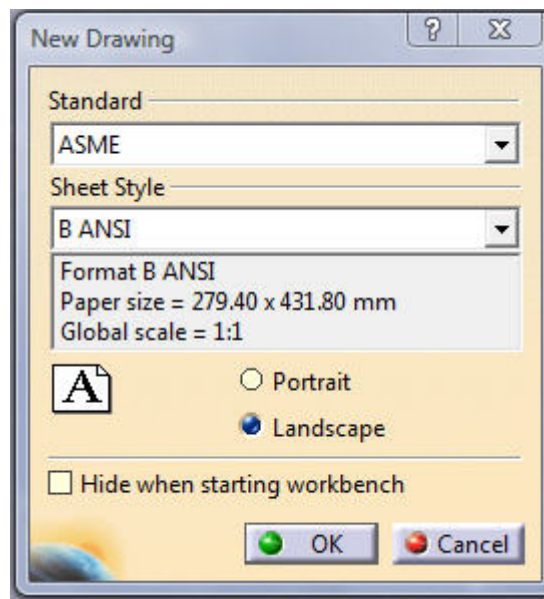
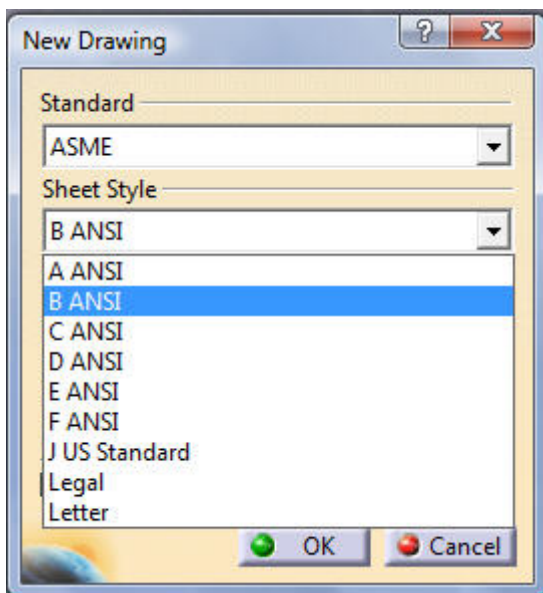
The Catia dimensioned drawing below of the BASE-M101 part above (Without Pocket) will be created as an example.



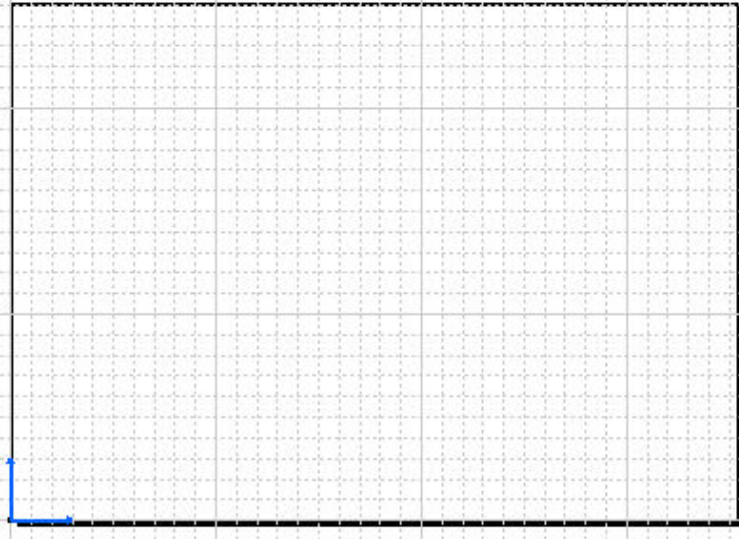
Pick the dropdown menu:
File >> New >> Drawing
The "New" box (right) will open.



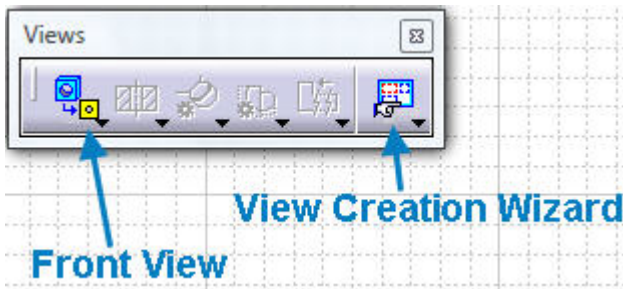
In the "New" box (above) select: Drawing



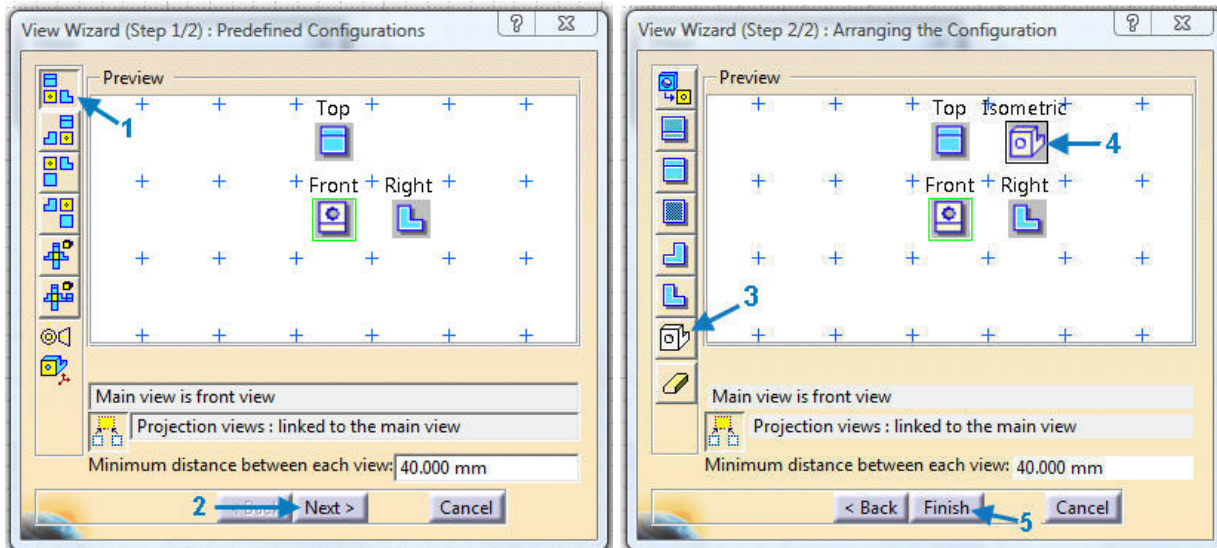
Pick the "Sheet Style": ANSI, ASME, ISO, or other drawing format >> OK. "New Drawing" >> OK.



A blank drawing will open as shown above.

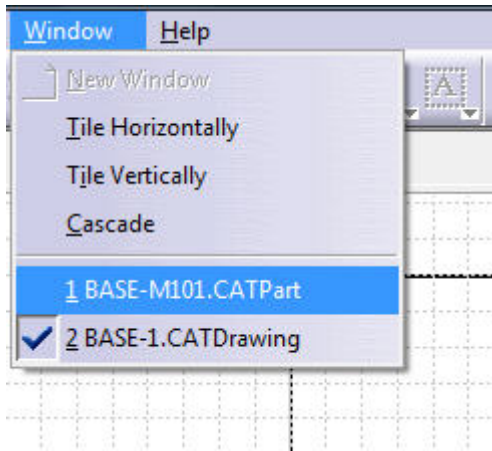


On the "Views" toolbar select, "View Creation Wizard".

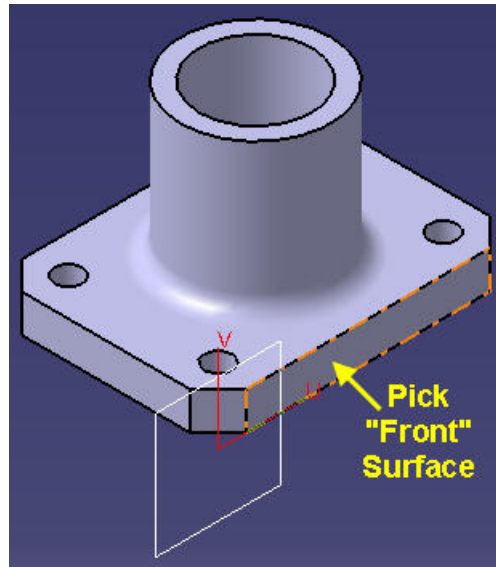


Pick-1 (3 Views) >> Next-2 >> Pick-3 (Iso View) >> Place (Iso View) at 4 >> Finish-5.

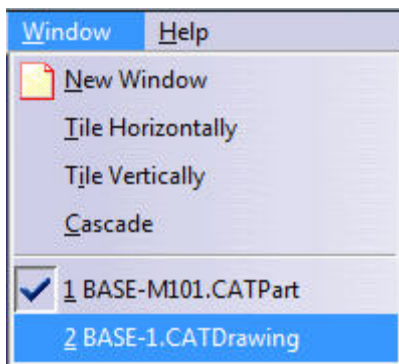
The drawing remains blank.



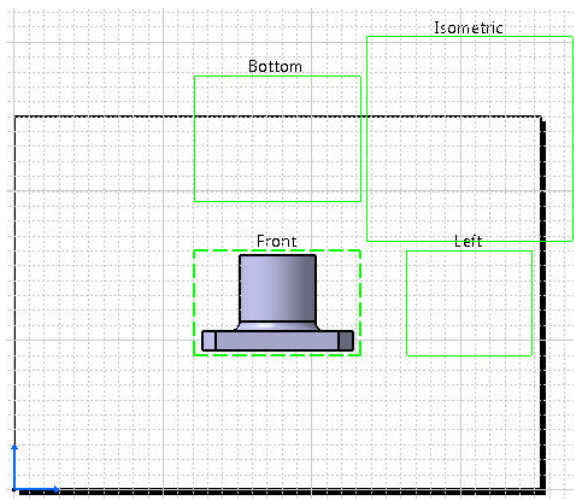
Pick: Window >> BASE-M101.CATPart.



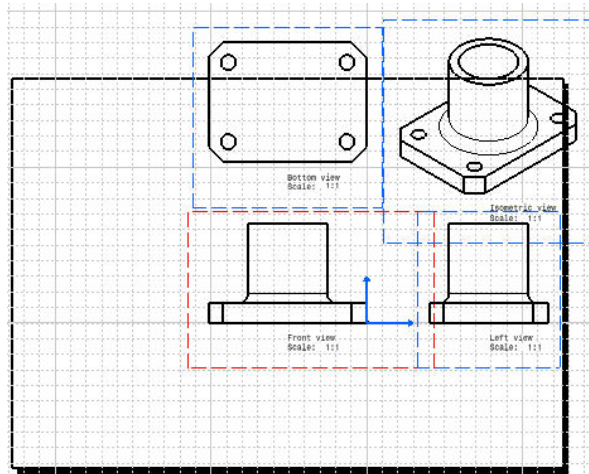
The part will open. Pick the "Front" surface.



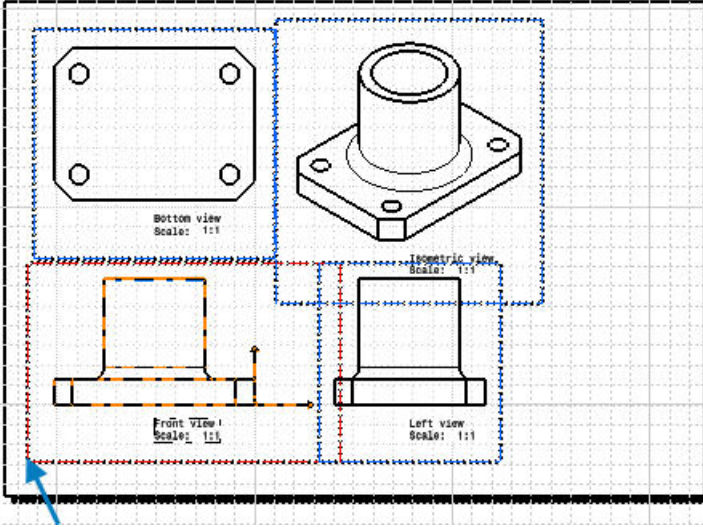
Pick: Window >> BASE-1.CATDrawing.



Catia places the "Front" view in the center of the drawing.

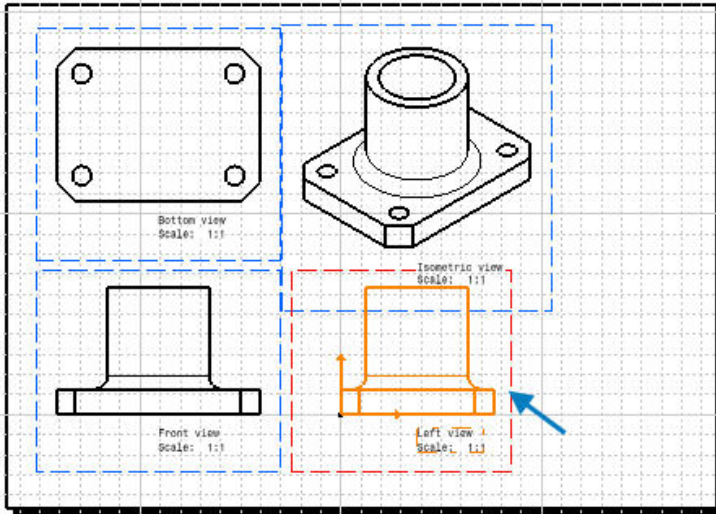


Click on the front view and all views appear.

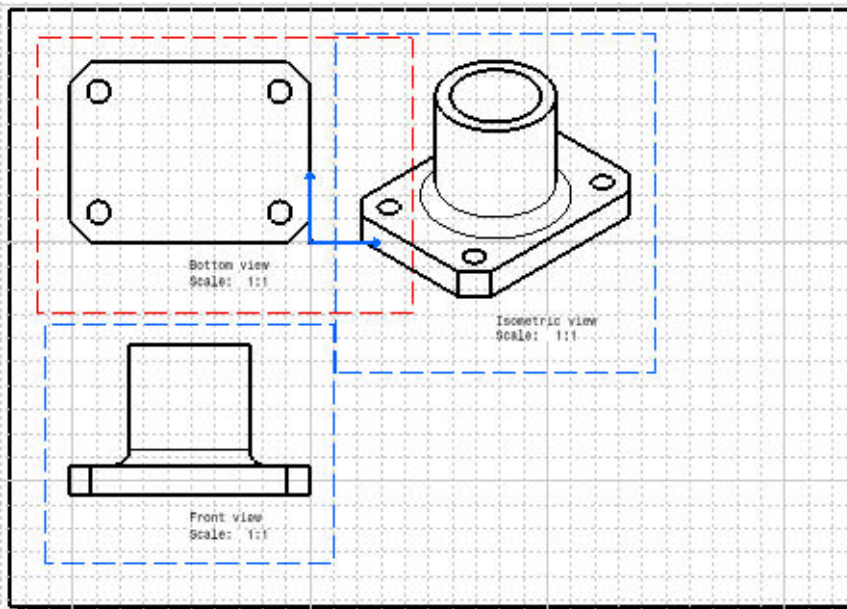


Pick an edge of the Front View and drag all views into the drawing, left.

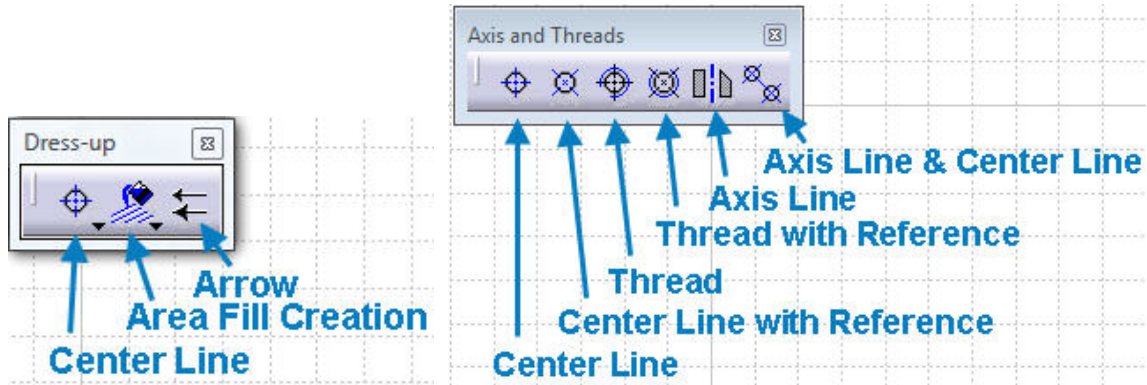
**File >> Save As: >>
Browse Files >> BASE-M101.CATDrawing.**



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

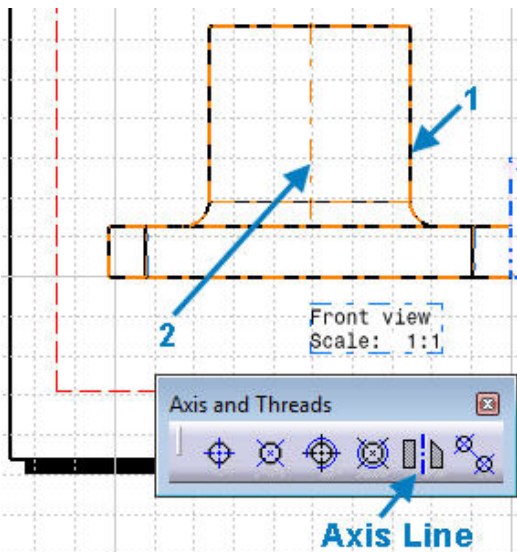


Press the delete key to remove the left view as shown above.



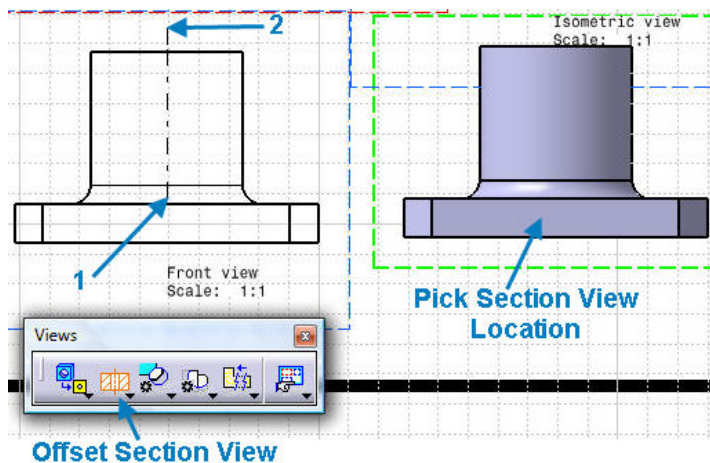
The "Dress-up" toolbar.

The "Axis and Threads" toolbar.

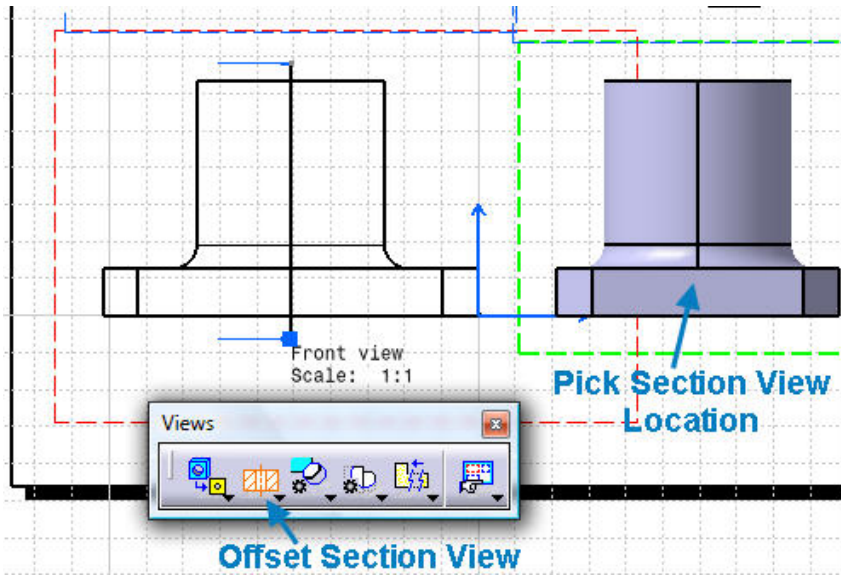


Double click the dashed line border of the front view to make it active (orange).

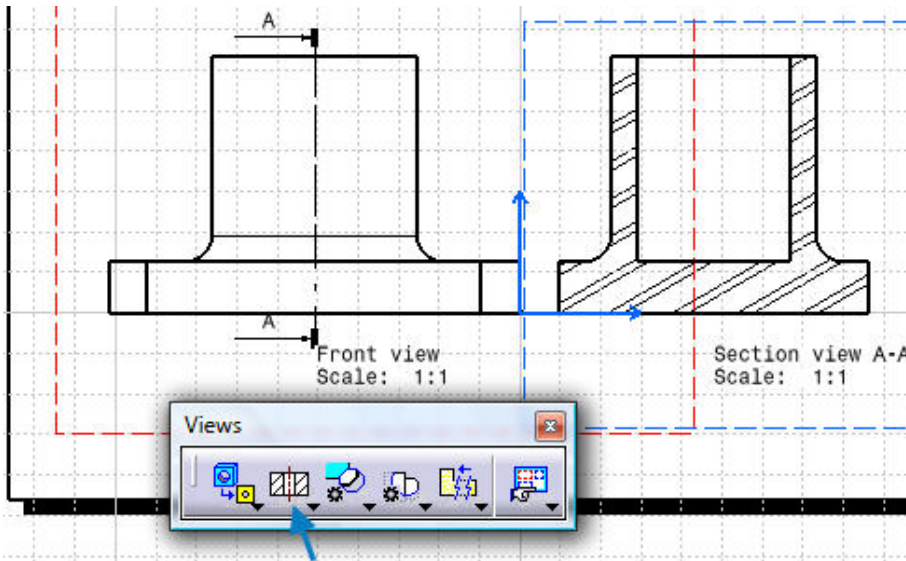
Pick the: Axis Line tool >> Edge-1 >> Part center line-2 will be added by Catia.



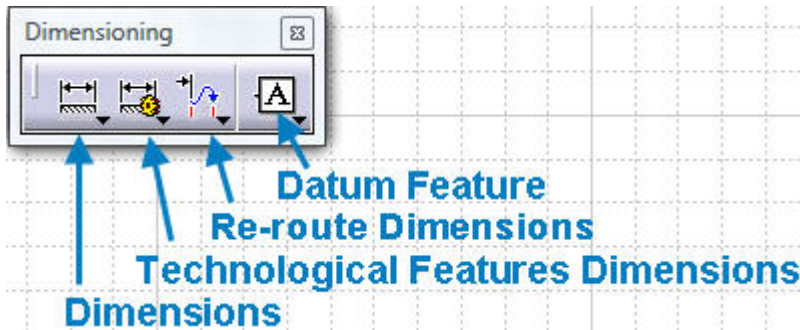
On the "Views" toolbar pick the: "Offset Section View" tool >> Pick section line starting point-1 >> Drag to section line end point-2 >> Double Click.



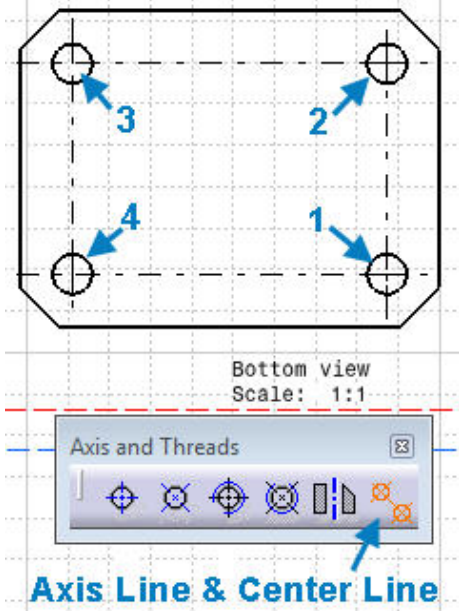
Pick the "Section View" location shown upper right.



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



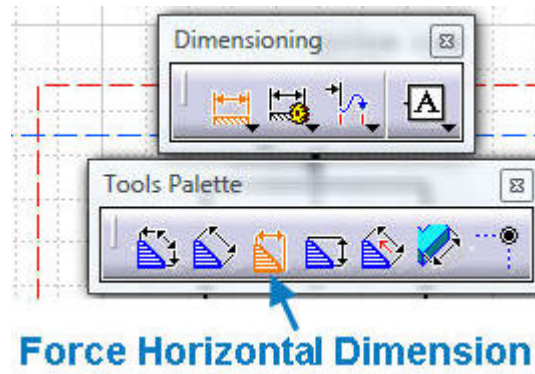
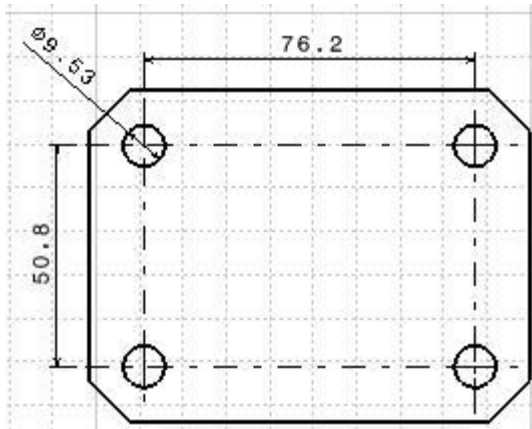
The "Dimensioning" toolbar is shown above.



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 tool >>

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.

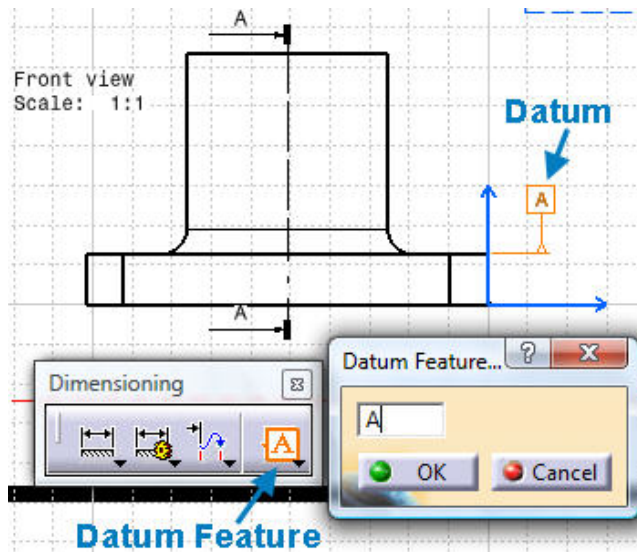
Continue adding dimensions.

All dimensions will be converted from inch to millimeters below.

GEOMETRICAL TOLERANCES

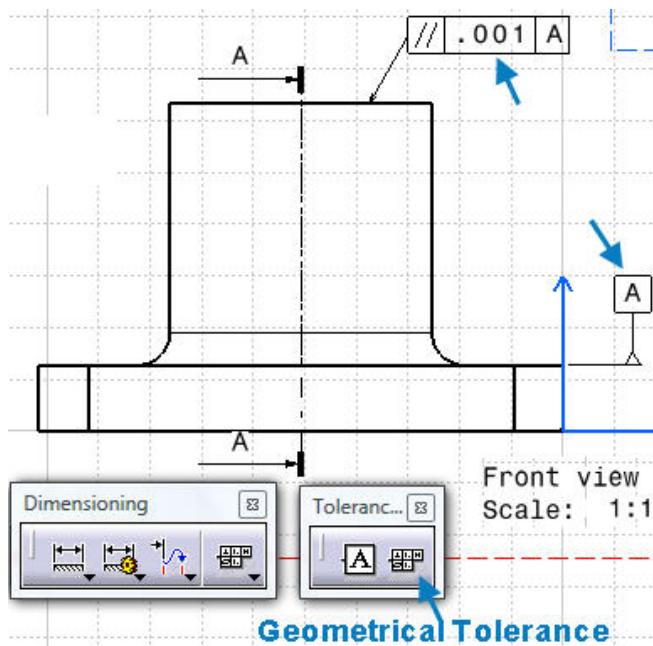
Select the Geometrical Tolerance tool.



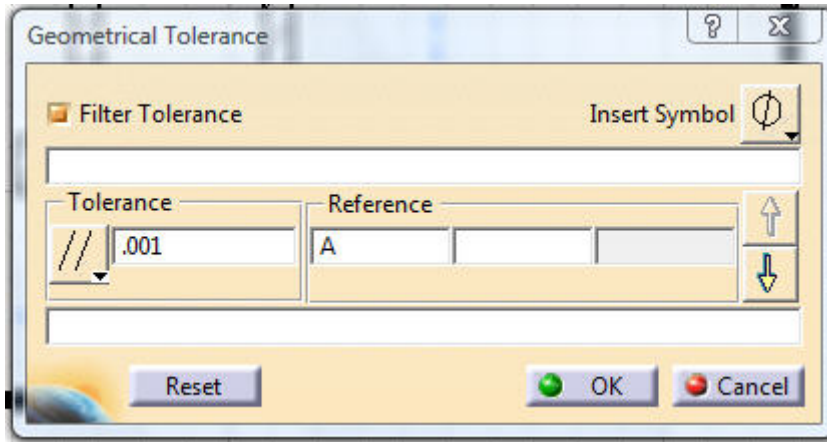


On the "Dimensioning" toolbar pick the: Datum Feature tool.

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



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

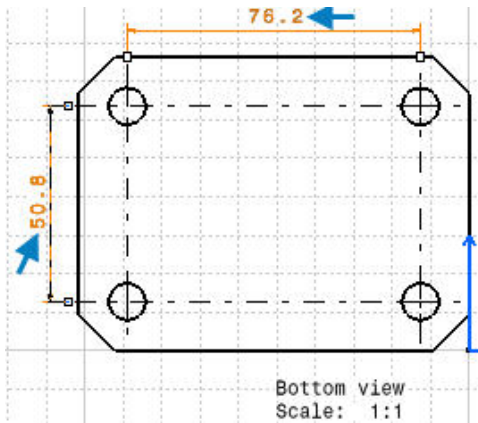


The “Geometrical Tolerance” box will open.

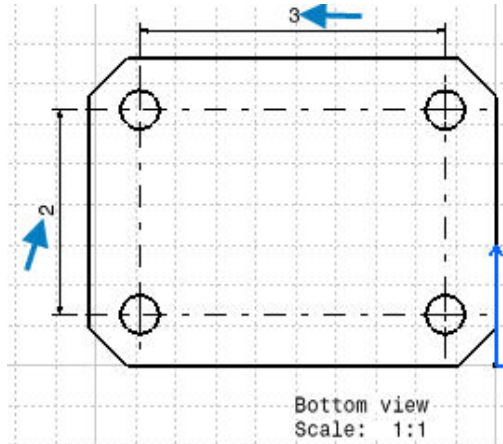
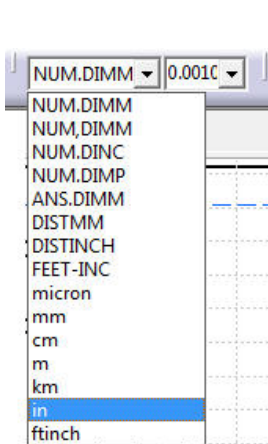
The “Parallel Symbol” has been selected from the drop down menu and the Tolerance has been set to .001 inch.

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

CONVERT MILLIMETERS TO INCHES



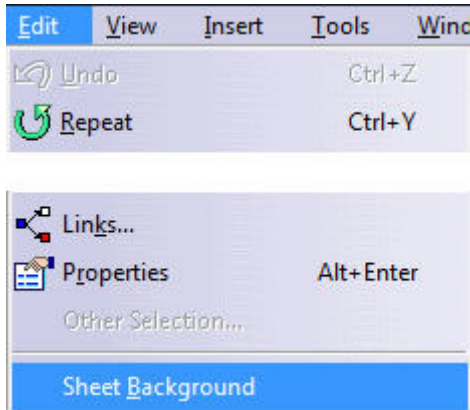
Hold the Ctrl key down and pick each dimension needing to be changed from mm to inches.



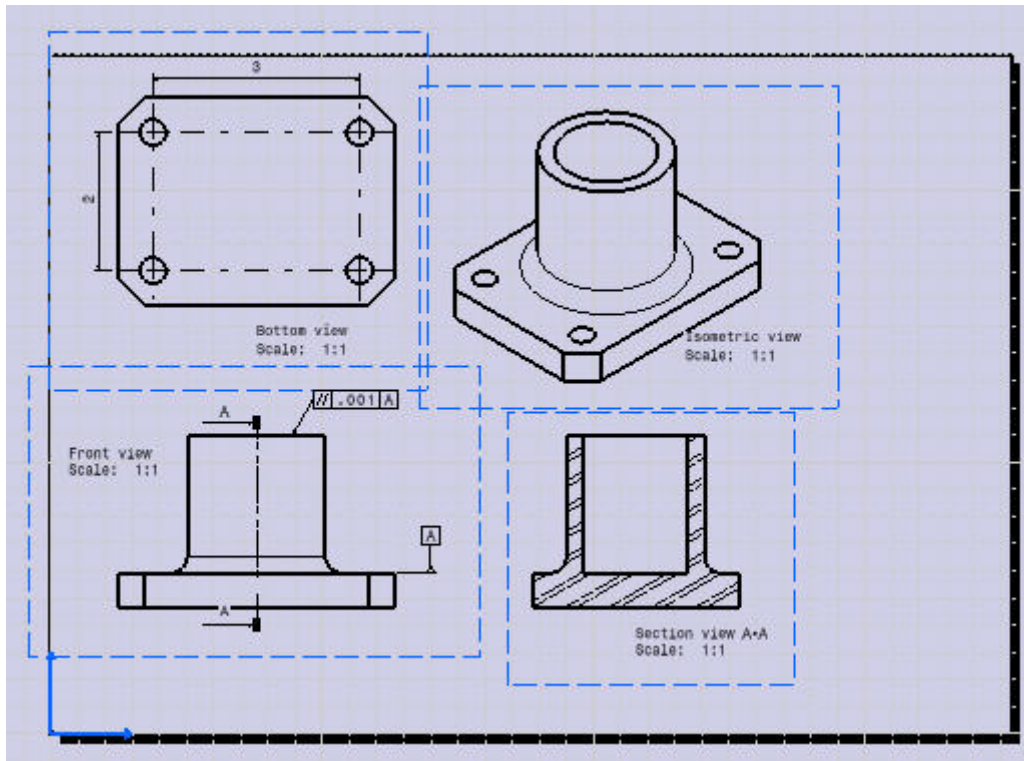
Pick the “NUM.DIMM” drop down menu >> Select “in”.

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

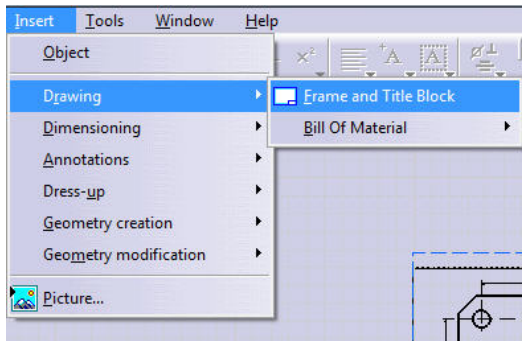
DRAWING SHEET BACKGROUND



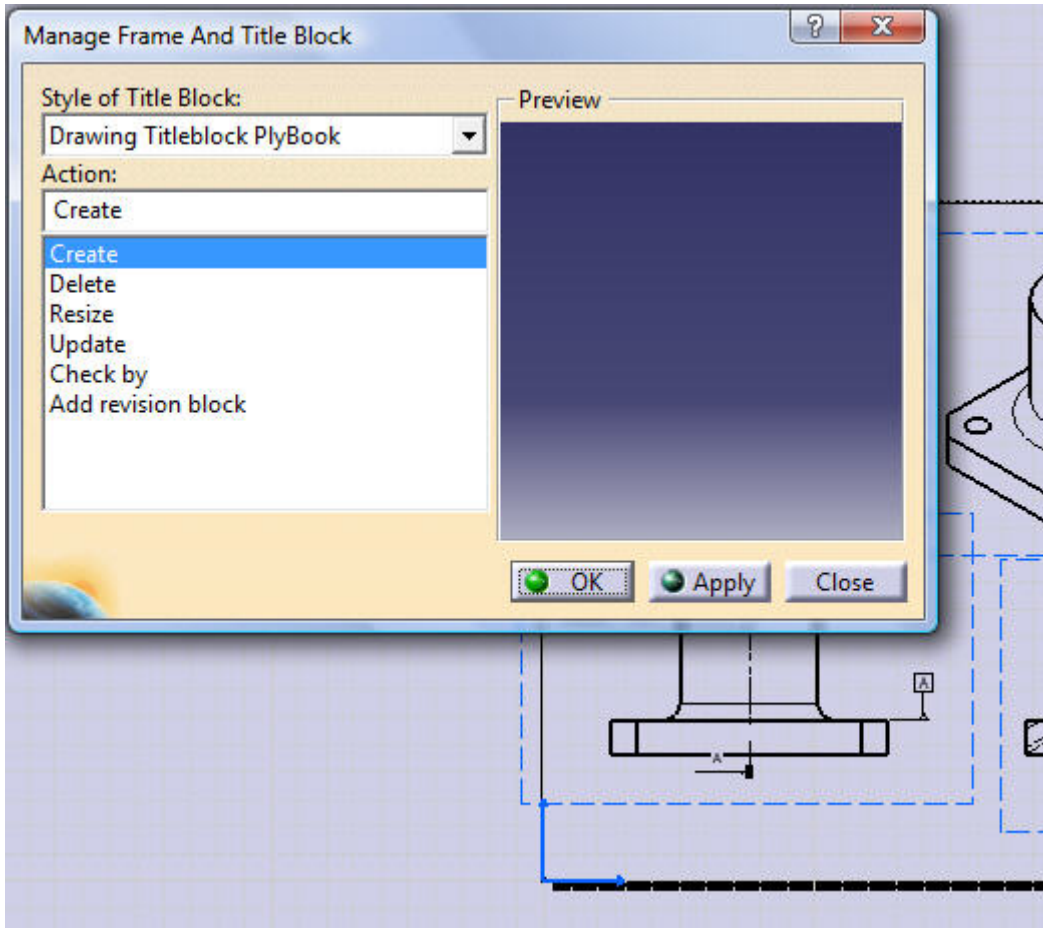
Pick: Edit >> Sheet Background.



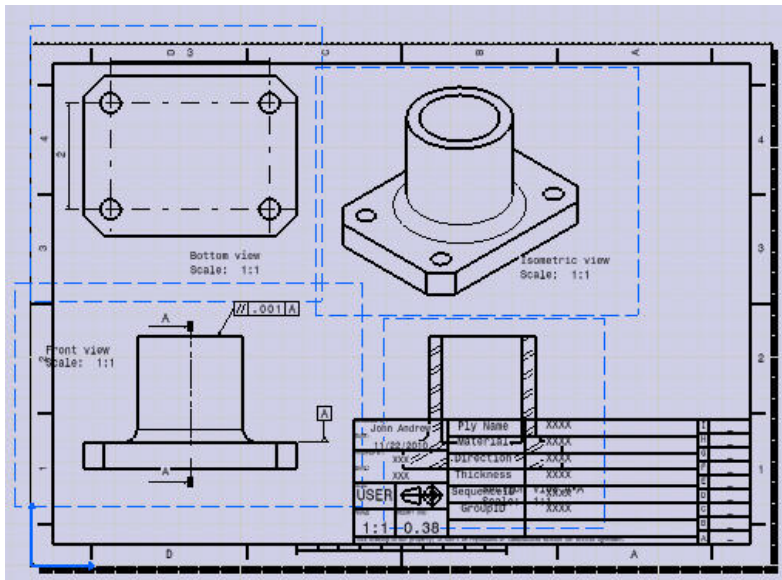
The grey color indicates sheet background.



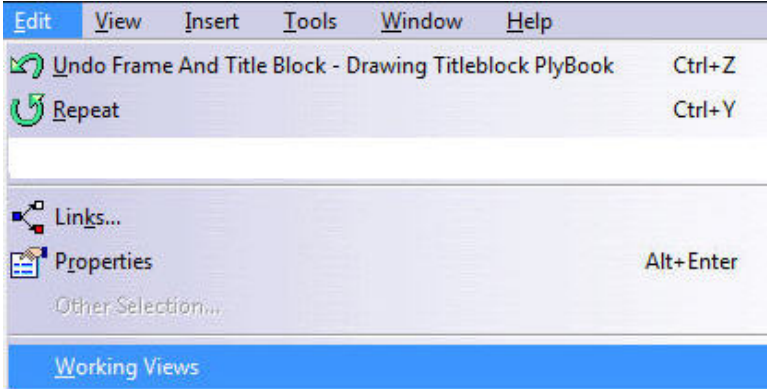
Pick: Insert >> Drawing >> Frame and Title Block.



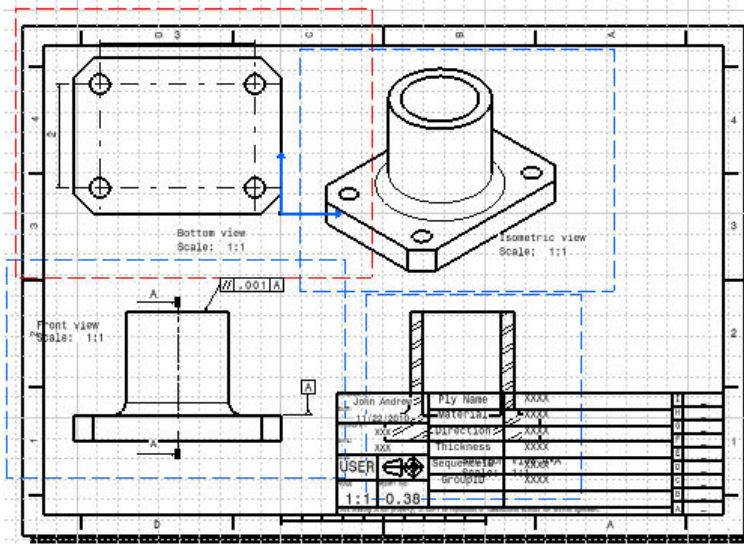
Pick: Create >> Apply >> OK.



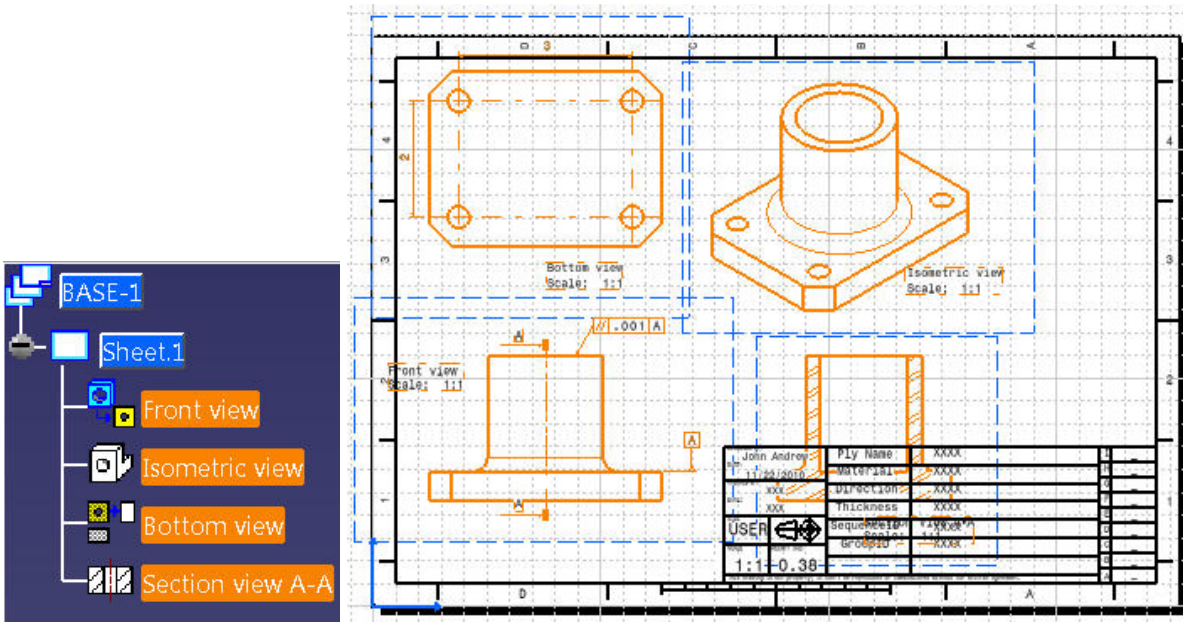
The Frame and Title Block are inserted but the views do not fit in the drawing blank area.



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

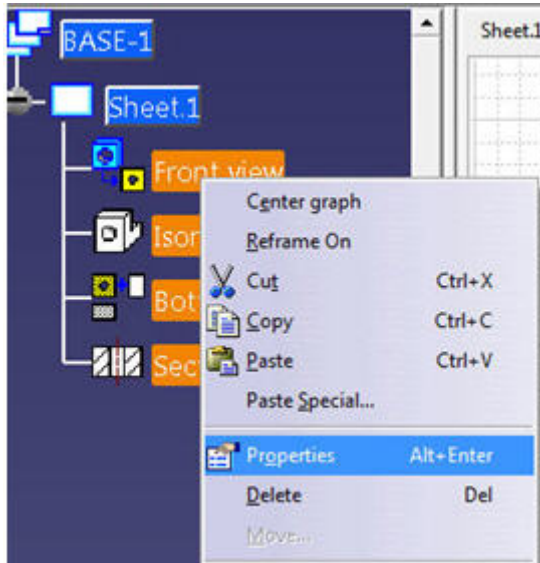


SCALE VIEWS



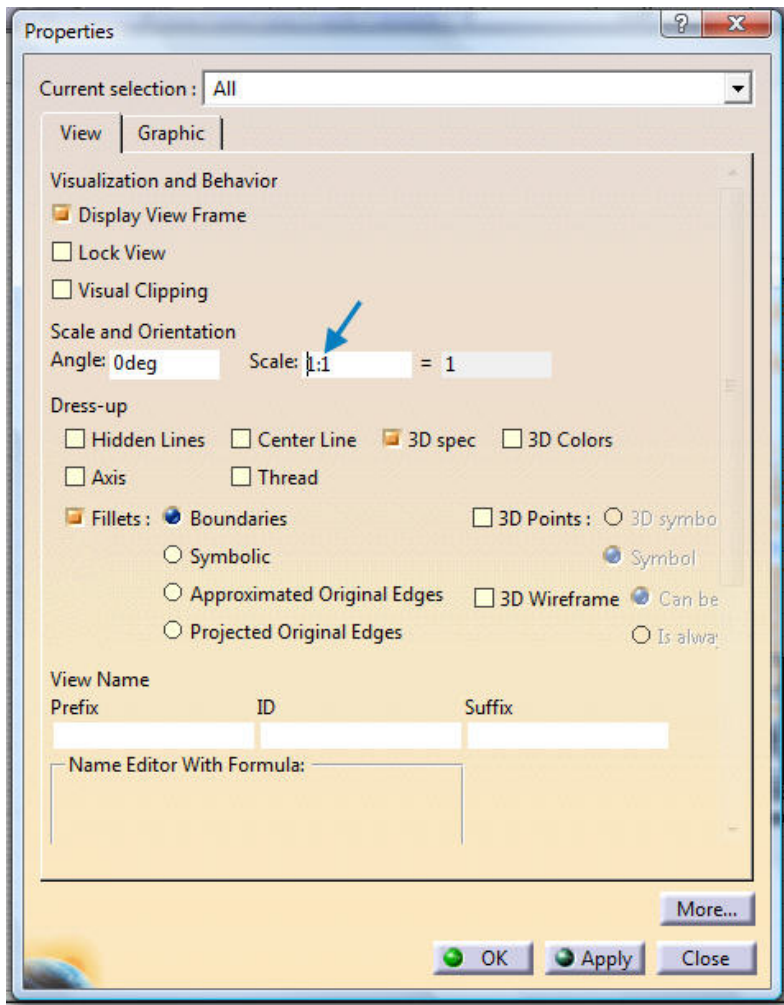
Hold the Ctrl key and pick each view needing to have its scale changed.

The selected views change color to orange.

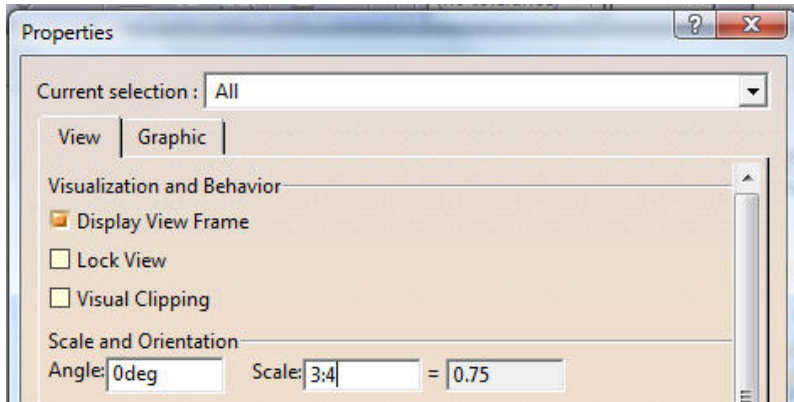


Right click on one of the selected views >> Select "Properties".

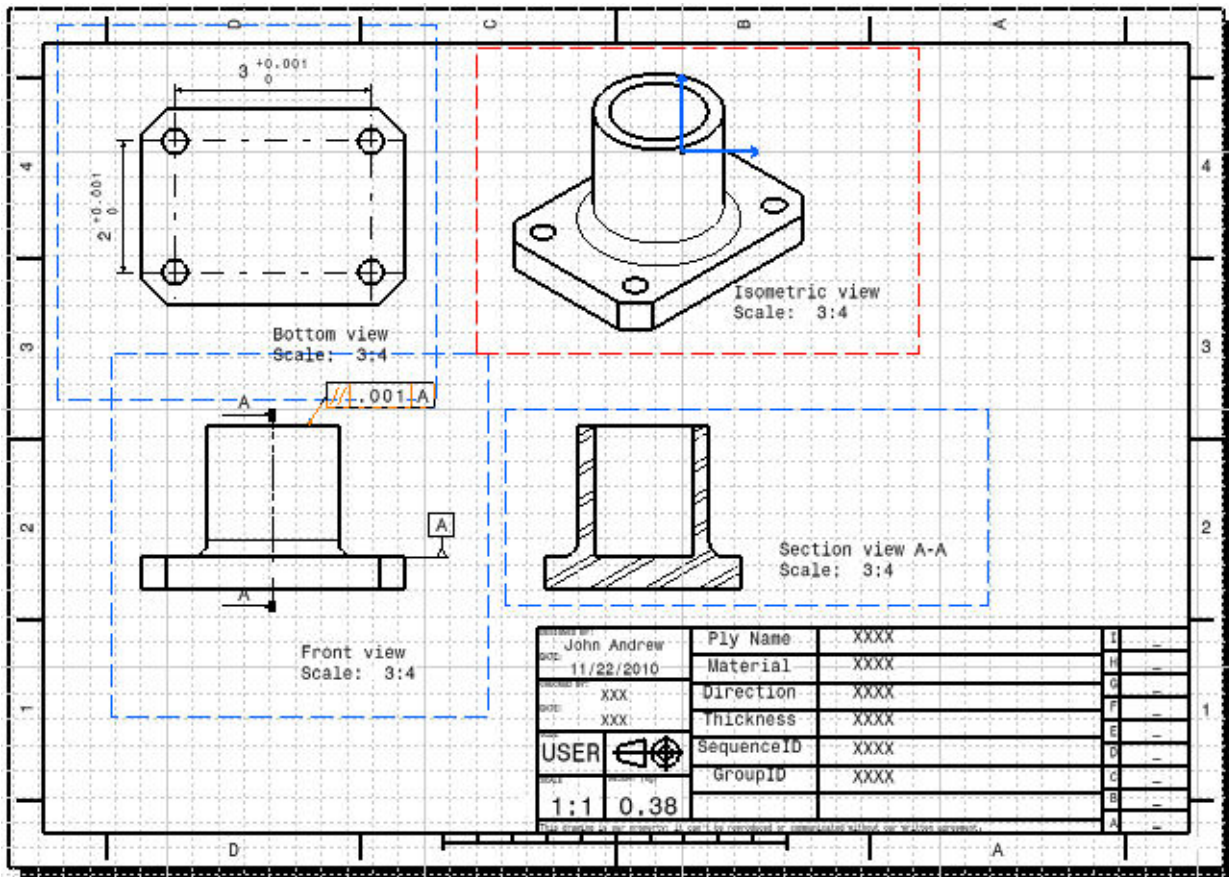
The "Properties" box will open as shown below.



The Scale is 1:1 or full size.



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

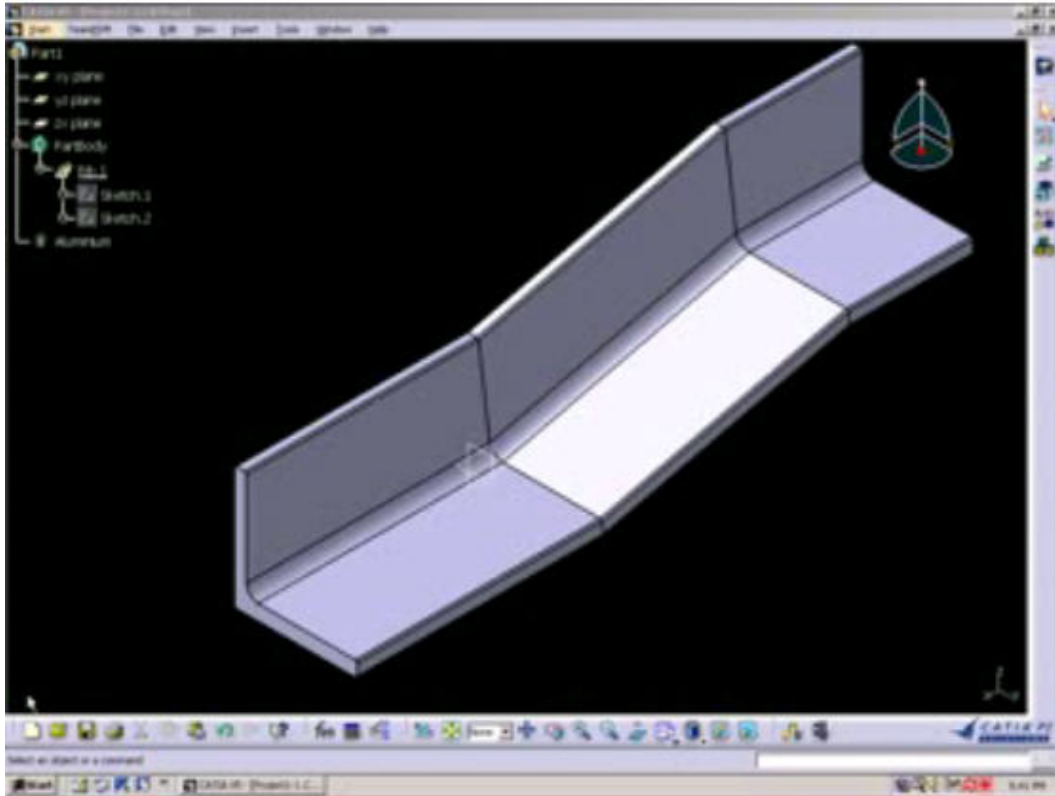


Now the views fit in the drawing.

2. JOGGLED EXTRUSION PART

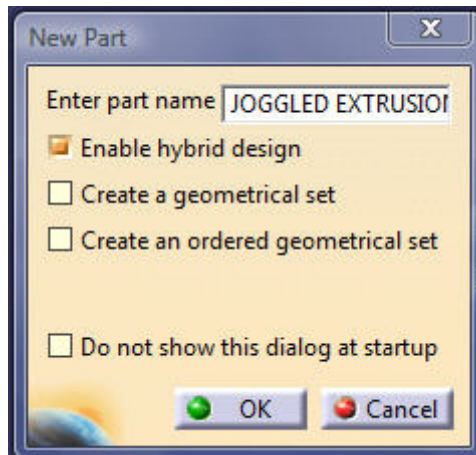
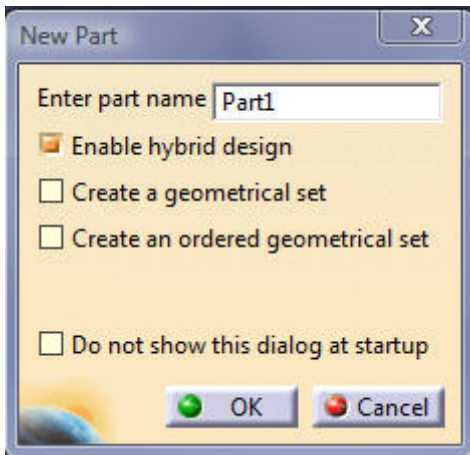
All 3D solid parts begin with a sketch in the "Sketcher Workbench".

Objective: Create the "Joggled Extrusion" part below.

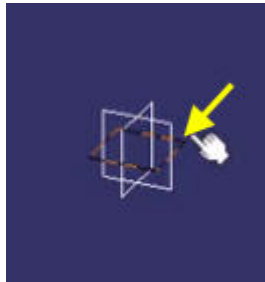
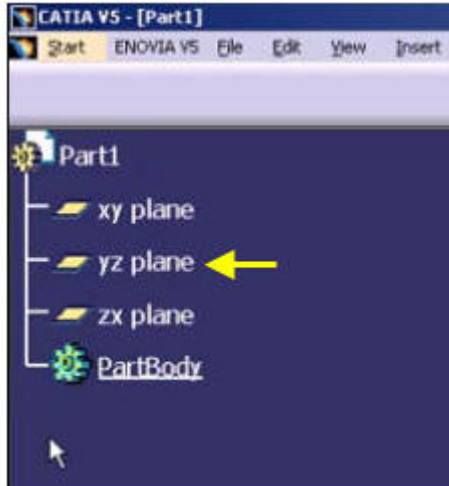


The finished Catia “Joggled Extrusion” aluminum aircraft part is shown above.

Pick, Start >> Mechanical Design >> Part Design.

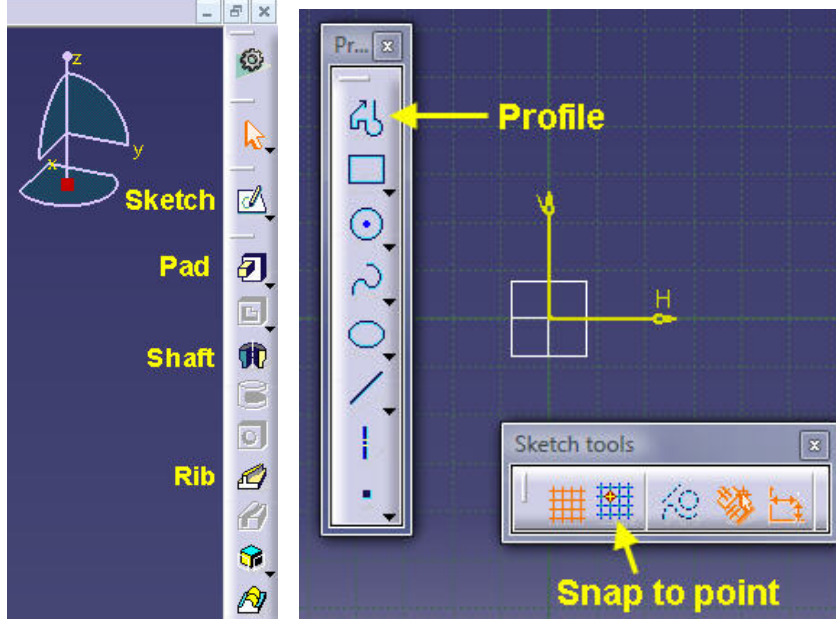


Change the Part1 name to “JOGGLED EXTRUSION” >> OK.



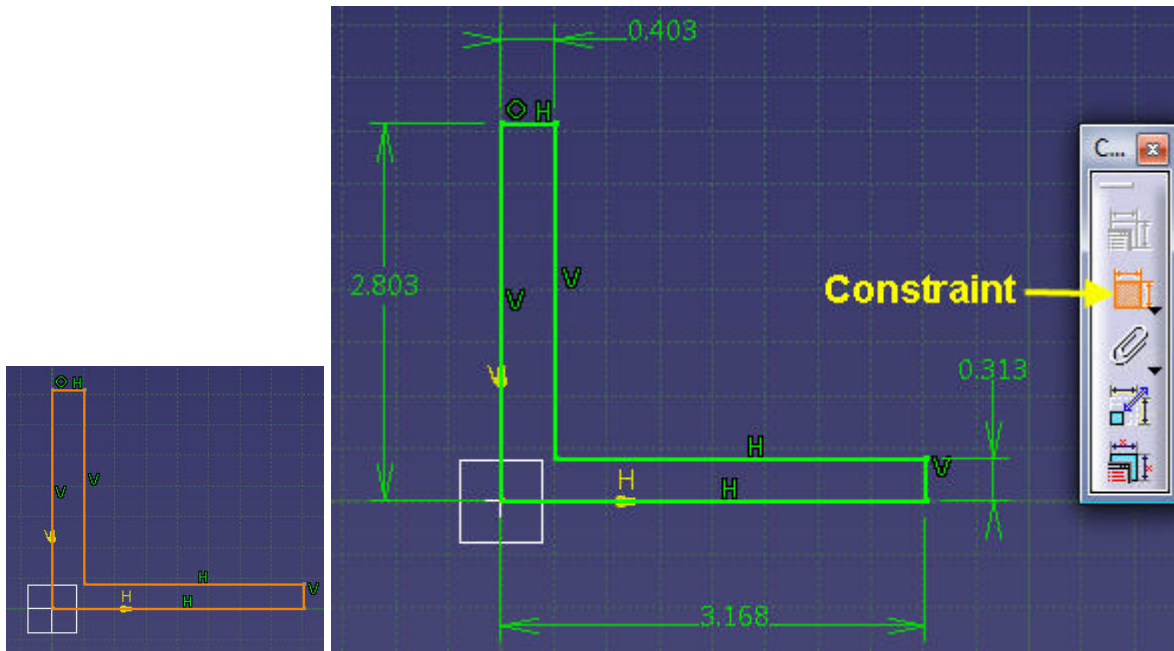
Pick the "YZ plane" tool.

XY plane has been selected as an example.



Pick the "Sketch" tool.

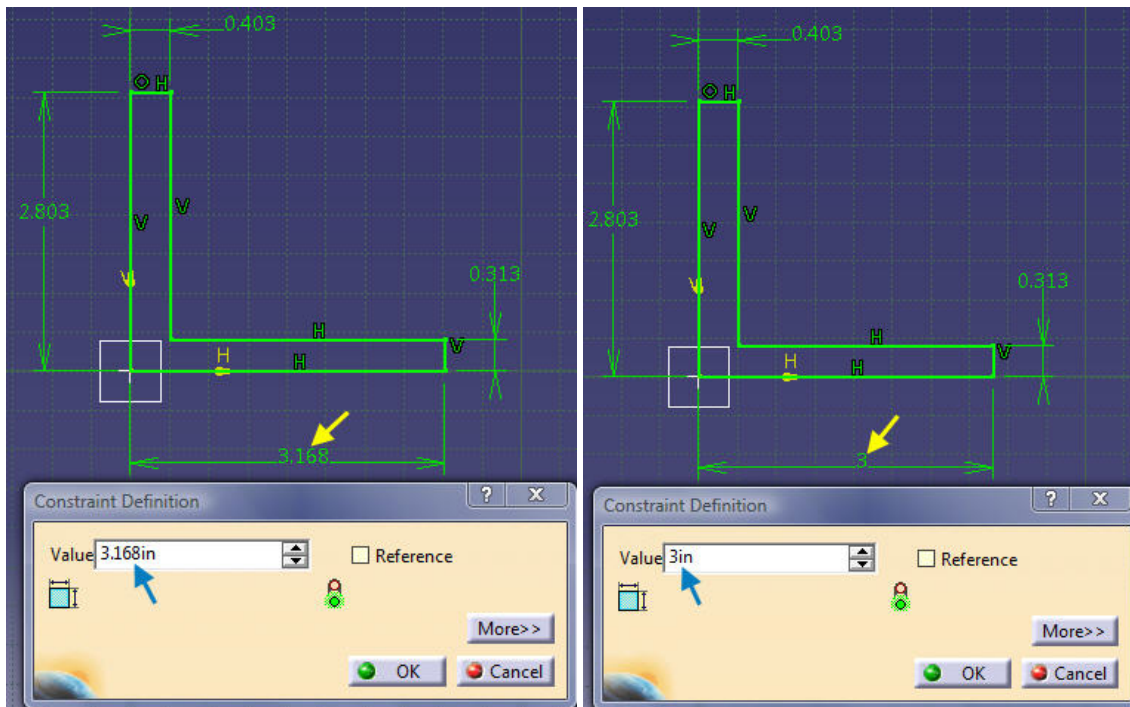
Pick "Snap to point" to toggle snap to the "off" condition as above.



Red lines indicates under constrained.

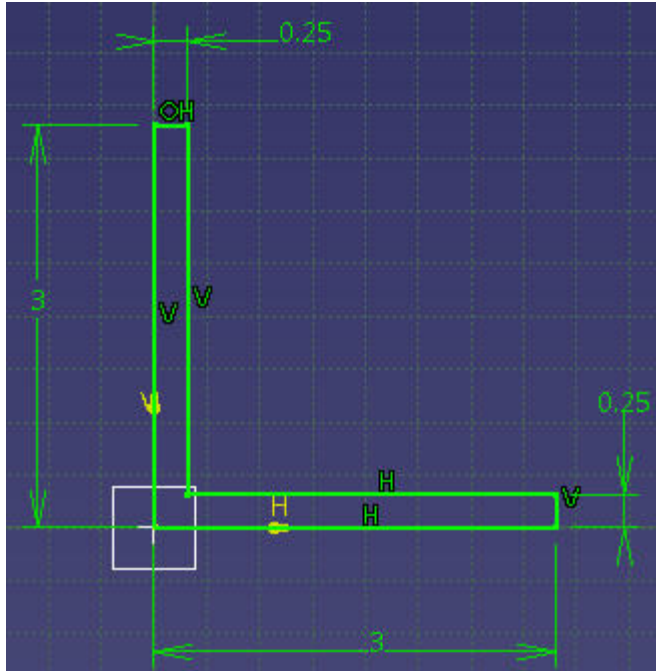
Double-click "Constraint" and pick a line and place dimension.

Repeat until all lines are green indicating "Fully Constrained" shape.

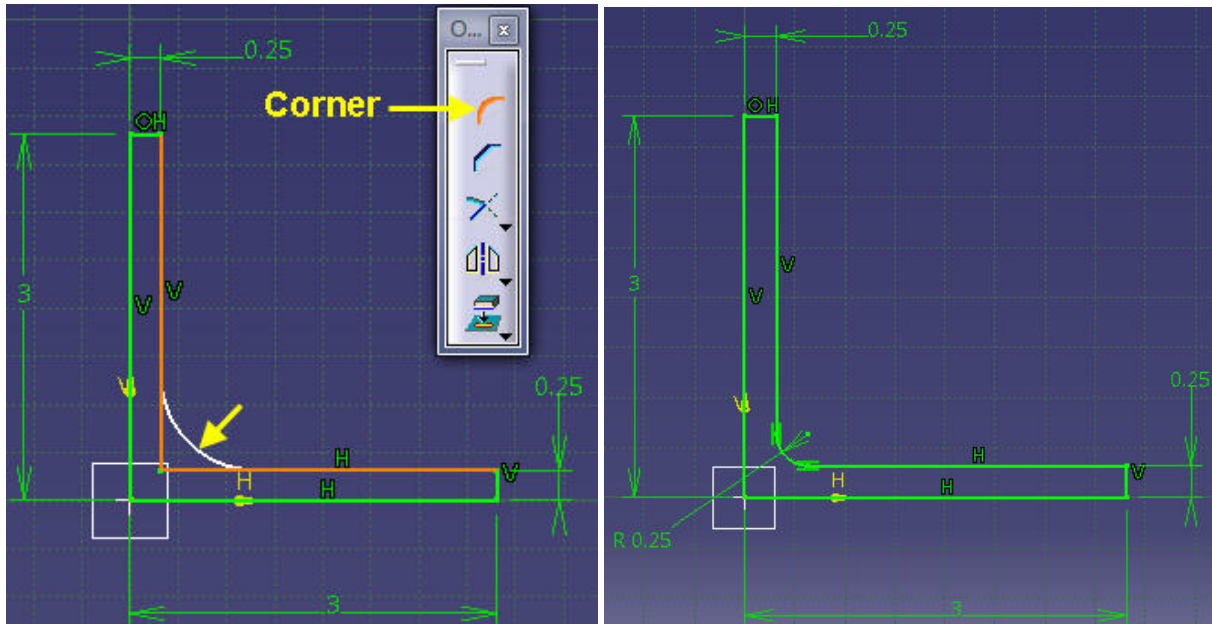


Double-click on a dimension 3.168 inches in the example above.

Change dimension 3.168 inches to 3 >> OK.



Edit the dimensions to the values shown above.



Pick the "Corner" tool >> Pick one intersecting line >> Pick second line >>

Pick radius center location.

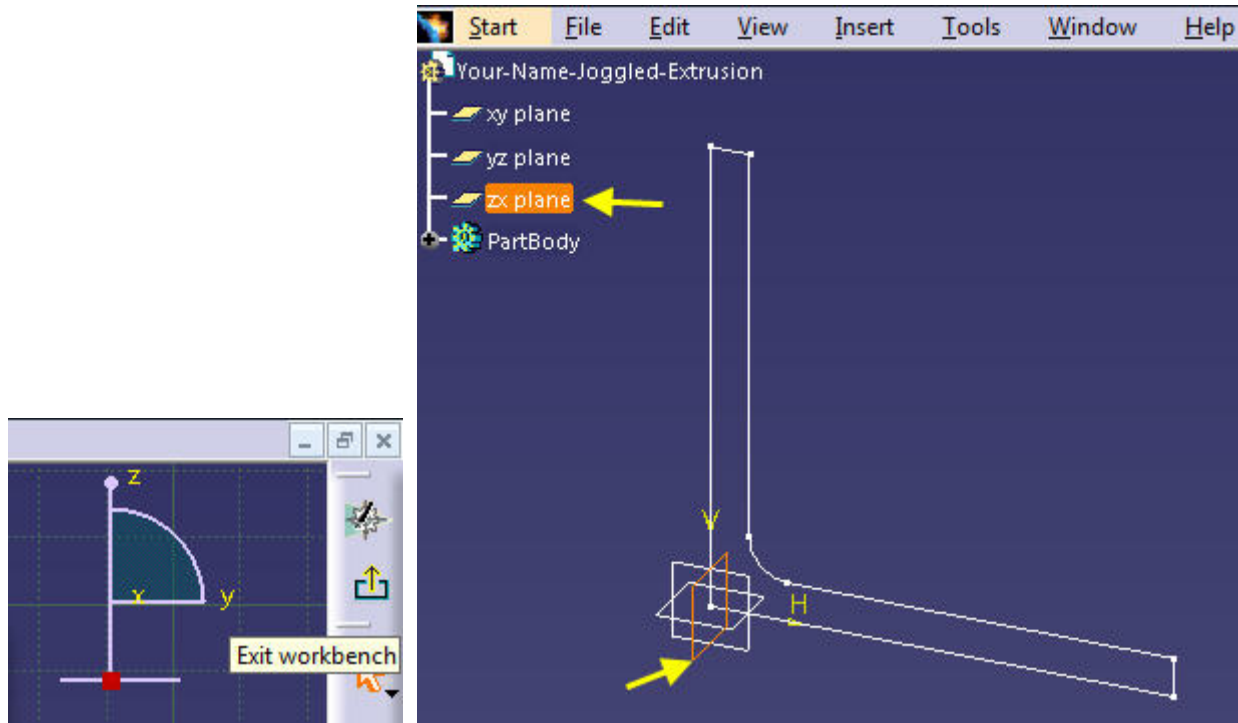
Double click on the corner radius and change its value to 0.25 inches.

The left end profile of the extrusion is sketched as shown above.

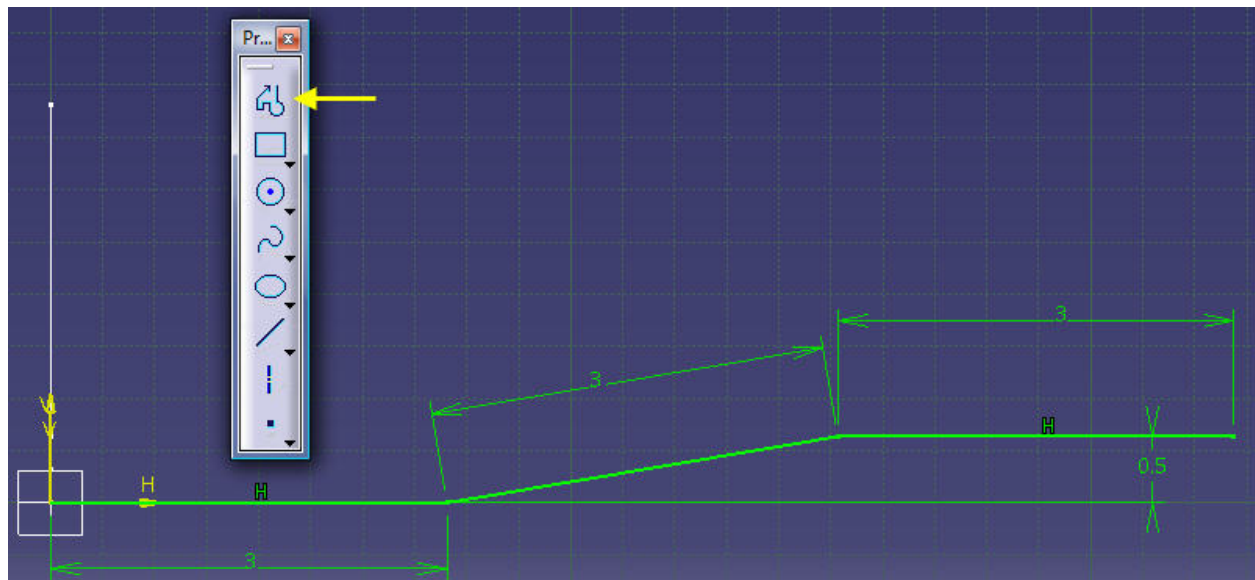
Save the sketched "Profile".

File >> Save As... >> Browse Files >> JOGGLED EXTRUSION.CATPart

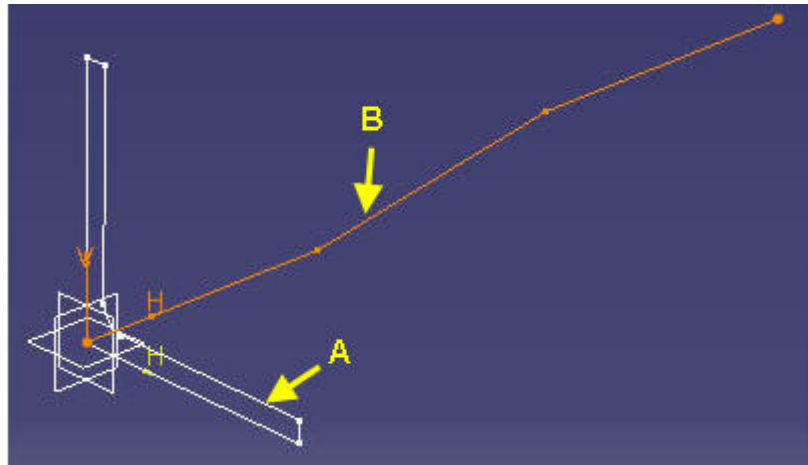
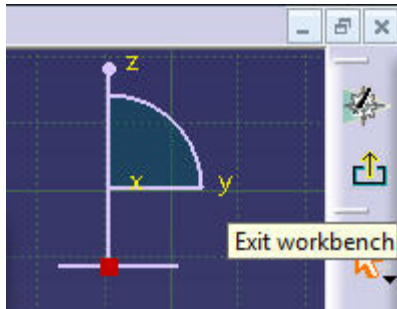
>> OK.



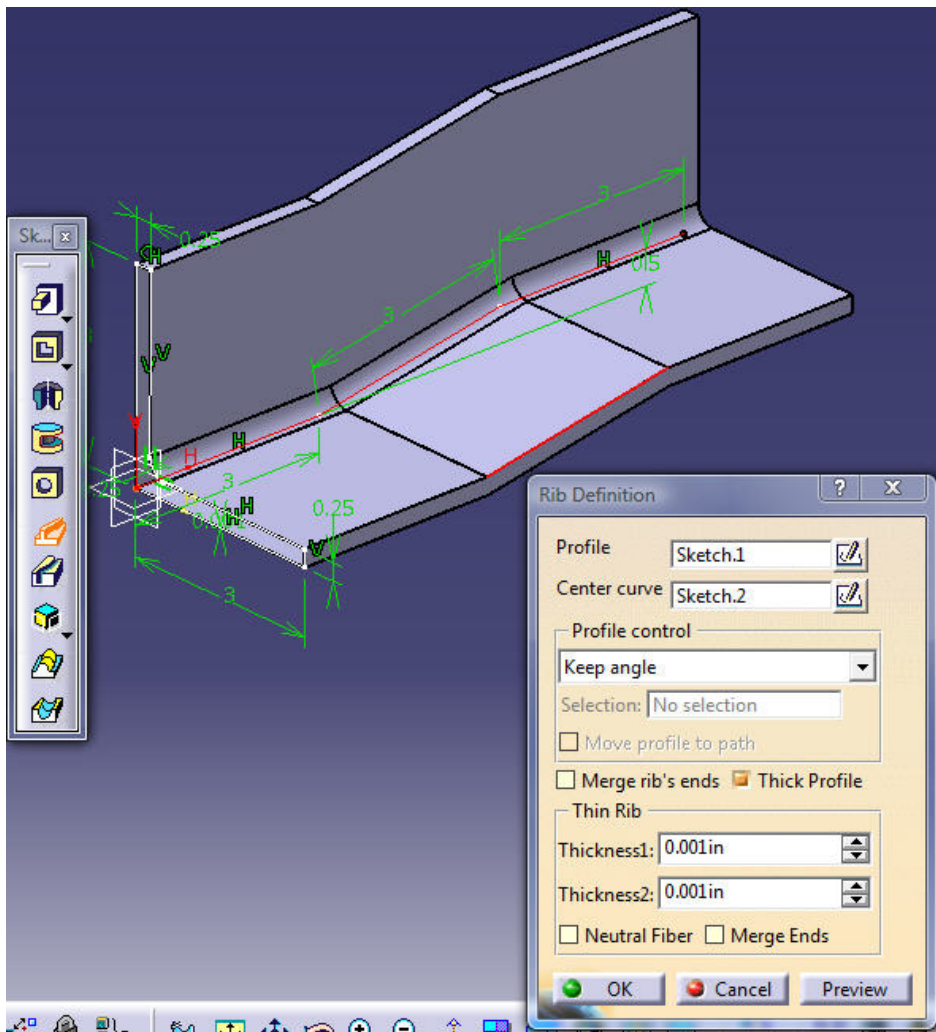
Exit “Sketch Workbench”
Pick the “ZX plane” tool.
Pick the “Sketch” tool.



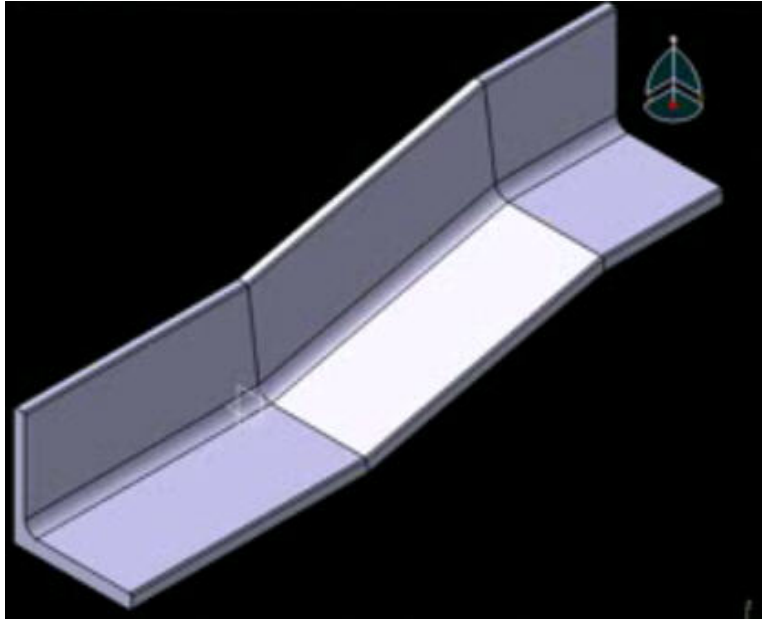
Sketch the extrusion profile in the “ZX plane as shown above.



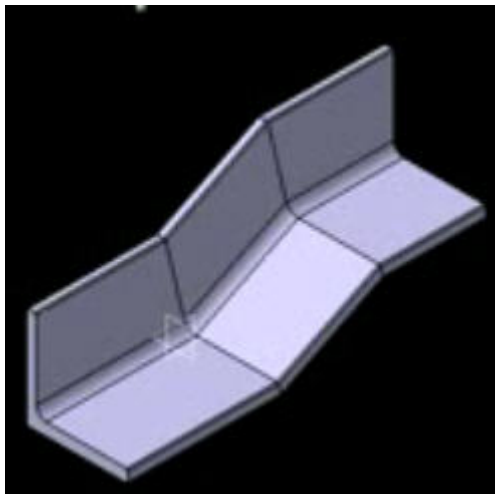
Exit "Sketch Workbench". A is the profile. B is the extrusion path.
The sketched extrusion path is shown above.



On the "Sketch" toolbar pick the "Rib" tool >> Profile >> Sketch.1 >> Center curve >> Sketch.2.



The extruded CATPart is illustrated above is saved as JOGGLED EXTRUSION.CATPart

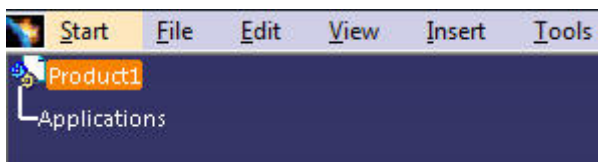


The extrusion path above has been changed by double clicking on the middle 3 inch dimension and changing it to 2 inches.

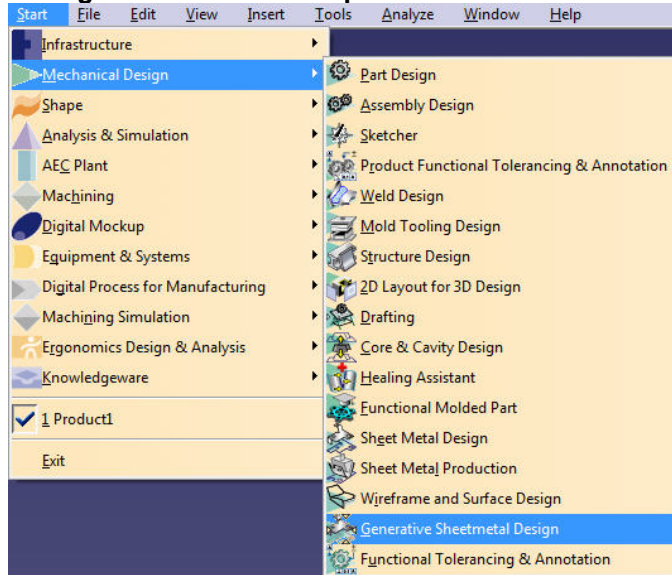
The modified part has been saved as: JOGGLED EXTRUSION-A.CATPart.

Multiple versions: A, B, C, etc. of the part may be required in an application.

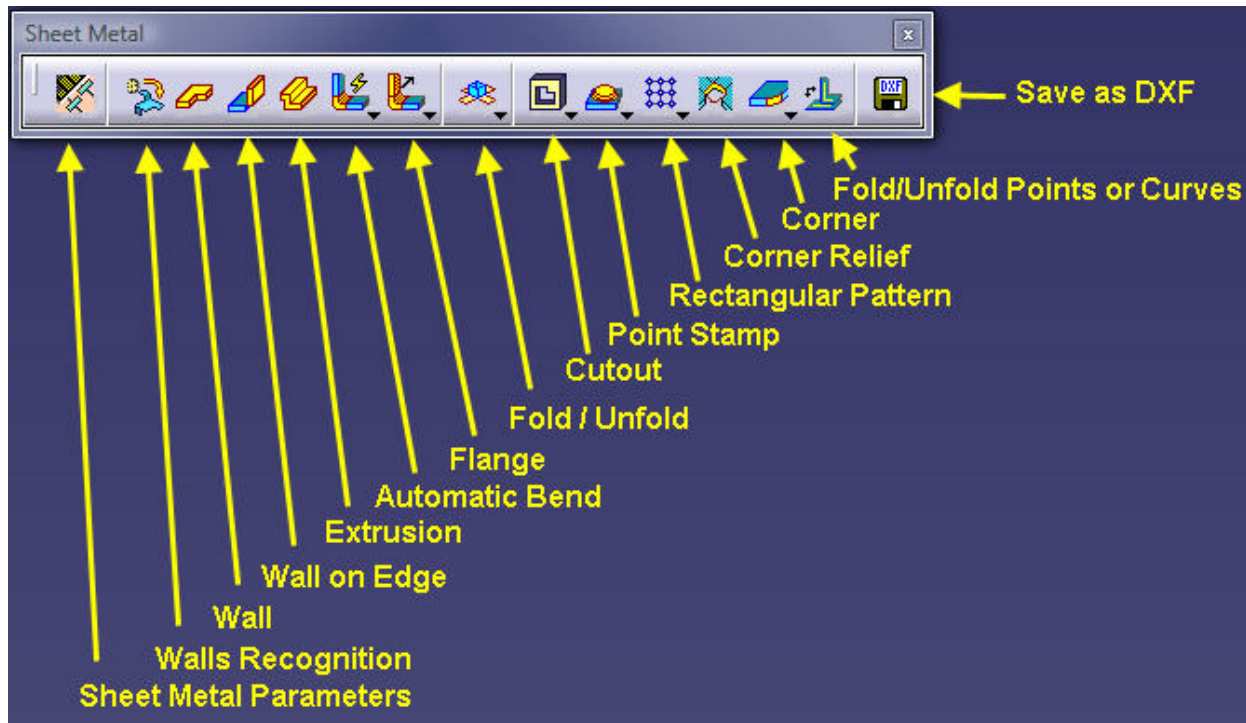
4. SHEET METAL DESIGN



Configure the sheet metal parameters.

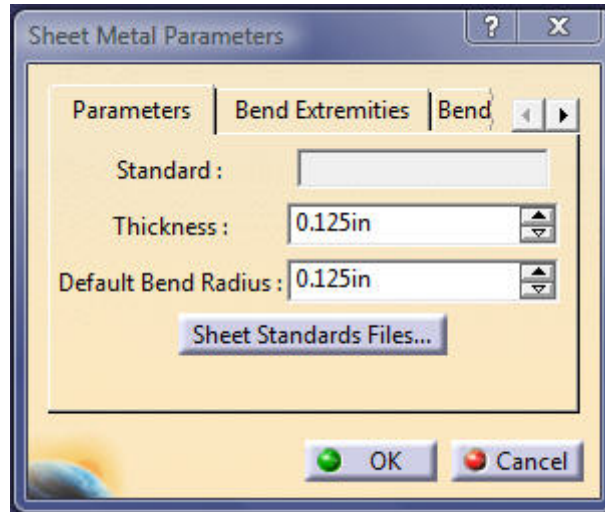
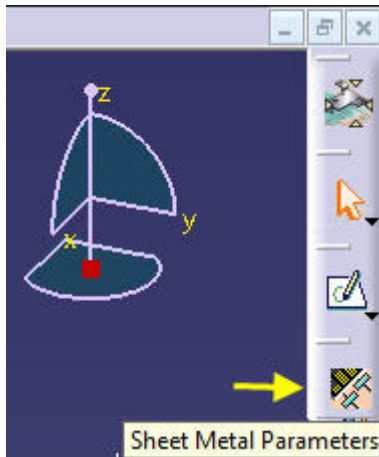


Pick: Start >> Mechanical Design >> Generative Sheet metal design.

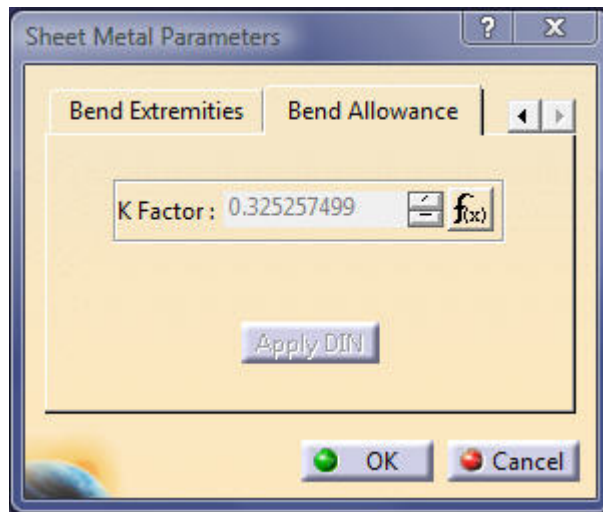
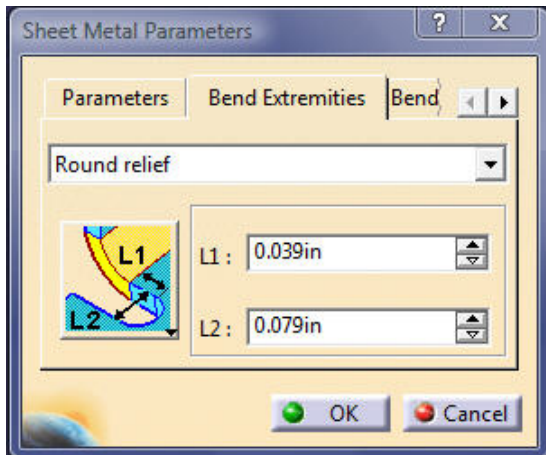


The "Sheet Metal" toolbar is shown above.

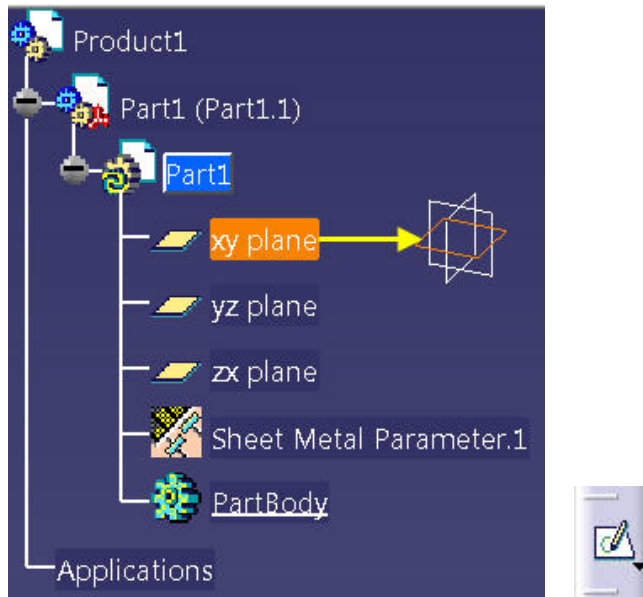
If the "Sheet Metal" toolbar is not visible pick: Start >> Sheet Metal Design.



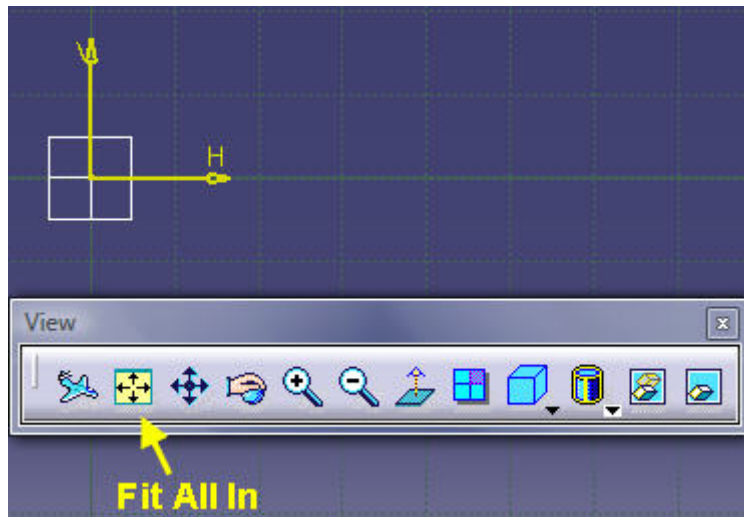
1. Click the Sheet Metal Parameters tool shown above.
The Sheet Metal Parameters dialog box is displayed.
2. Enter 0.125in the Thickness field.
3. Enter 0.125in in the Bend Radius field.
4. Select the Bend Extremities tab.



5. Select Tangent in the Bend Extremities combo list.
An alternative is to select the bend type in the graphical combo list.
 6. Click OK to validate the parameters and close the dialog box.
- The Sheet Metal Parameters feature is added in the specification tree.



Click on the “Sketch” tool

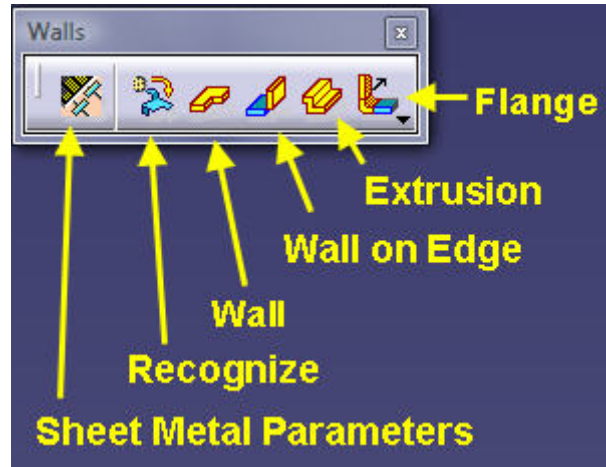
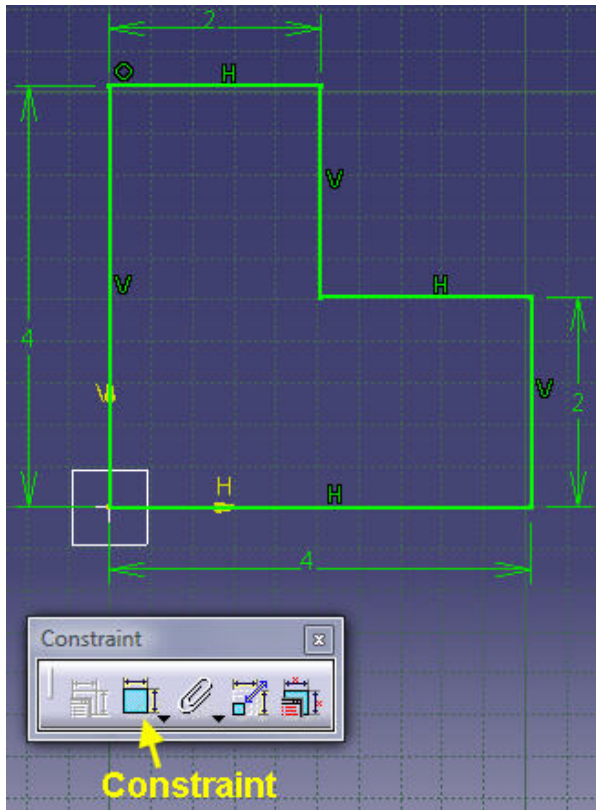


If the V-H Origin above is off the screen click on the “Fit All In” tool as above.

Creating the First Wall

This task shows how to create the first wall of the Sheet Metal Part.

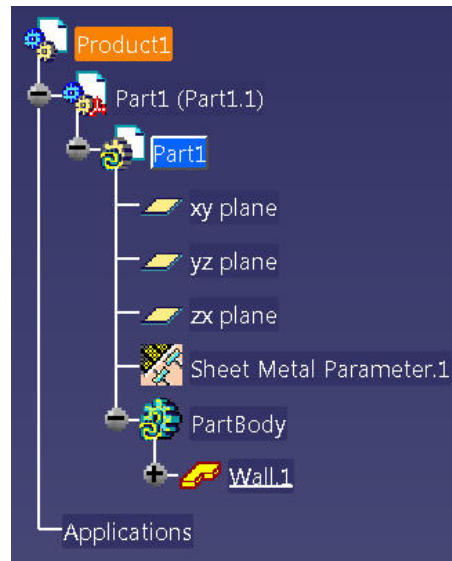
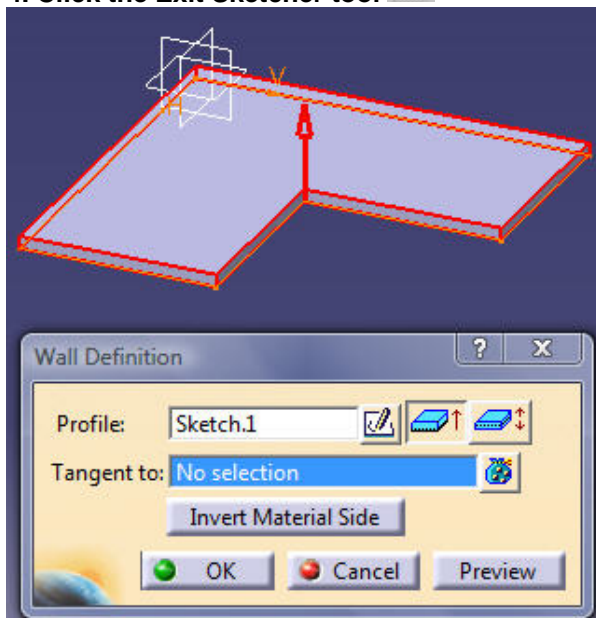
1. Click the Sketcher tool  then select the xy plane.
2. Select the Profile tool .



3. Sketch the contour as shown above.

4. Click the Exit Sketcher tool 

5. Click the Wall tool 



The Wall Definition dialog box opens.

By default, the Material Side is set to the top.

6. Click OK.

The Wall.1 feature is added in the specification tree.

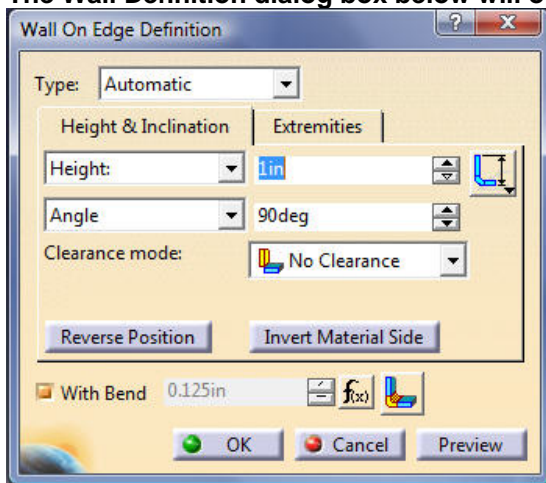
The first wall of the Sheet Metal Part is known as the Reference wall.

Creating the First Side Wall

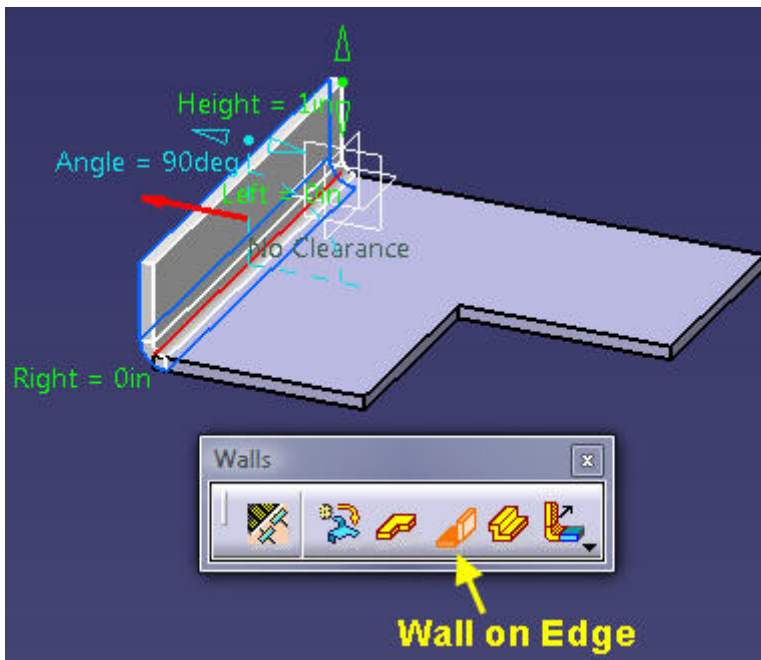
1. Select the Wall on Edge tool .

2. Select the left edge.

The Wall Definition dialog box below will open.



3. Enter 1in in the Length field.



The application previews the wall.

By default, the Material Side is set to the outside and the Sketch Profile to the top.

4. Reverse the Sketch Profile.

5. Click OK.

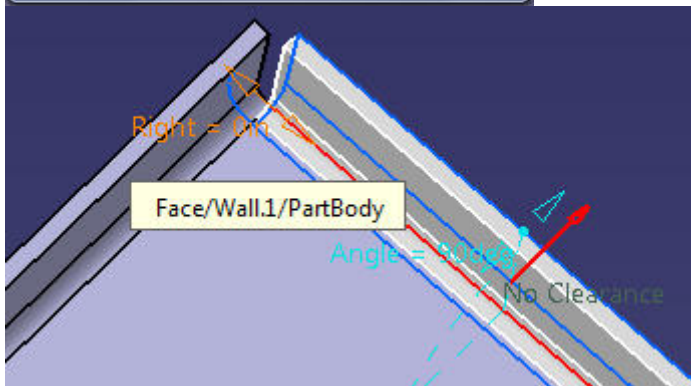
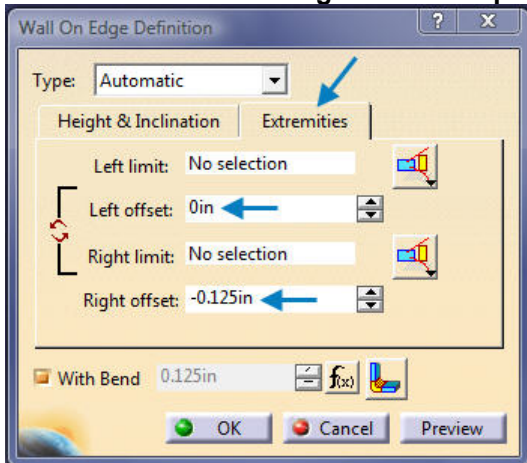
The first wall is created.

Creating the Second Side Wall

1. Select the Wall on Edge tool .

2. Select the right edge.

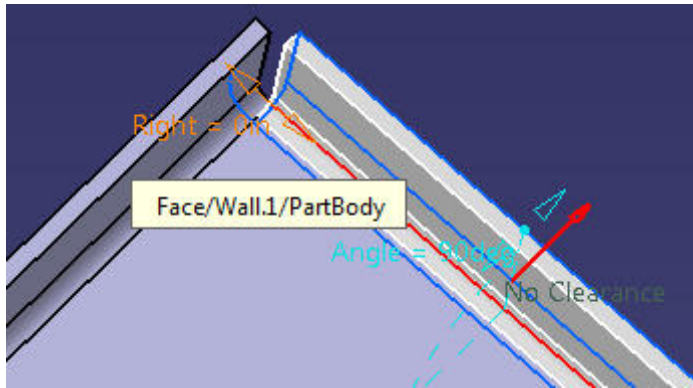
The Wall Definition dialog box below opens with the parameters previously selected.



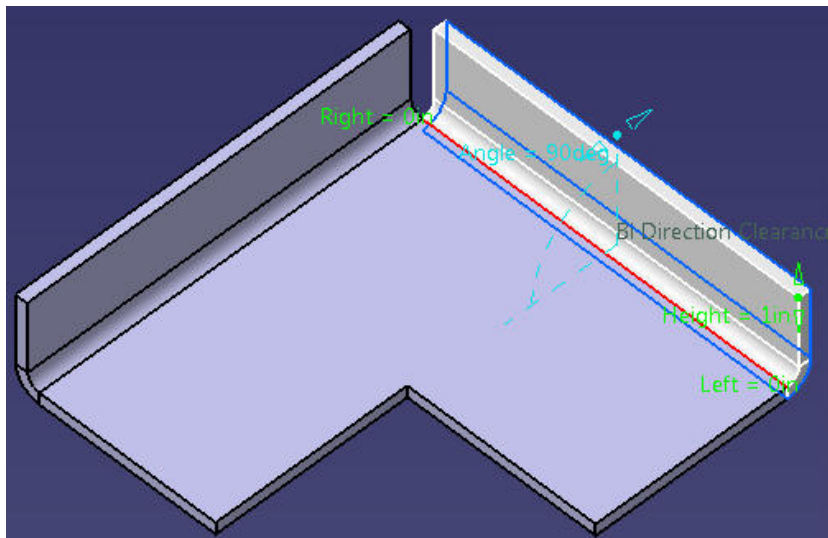
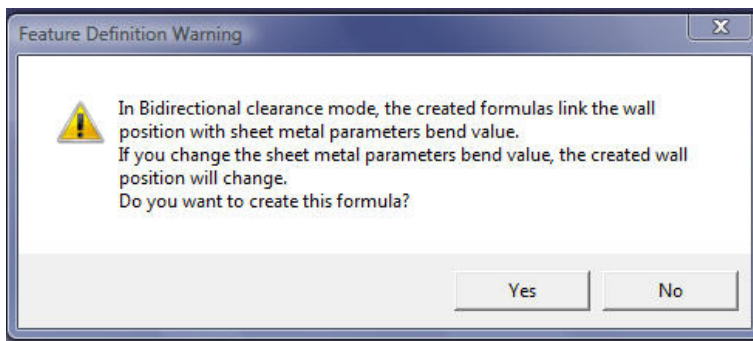
7. Select the right edge as shown above.

8. Press OK to validate.

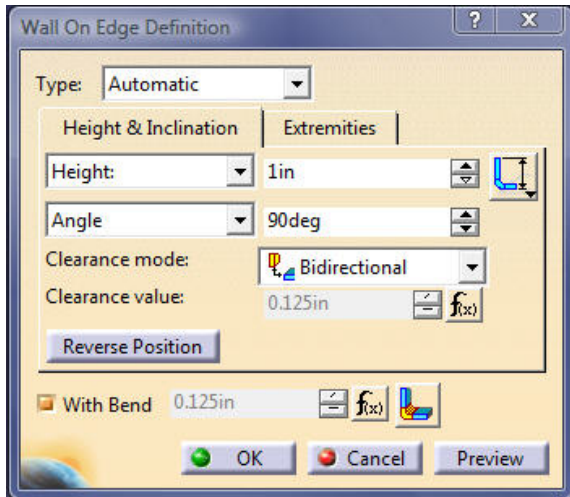
The second wall is created.



There is interference at the top corner and the “Feature Definition Warning” box will open below.

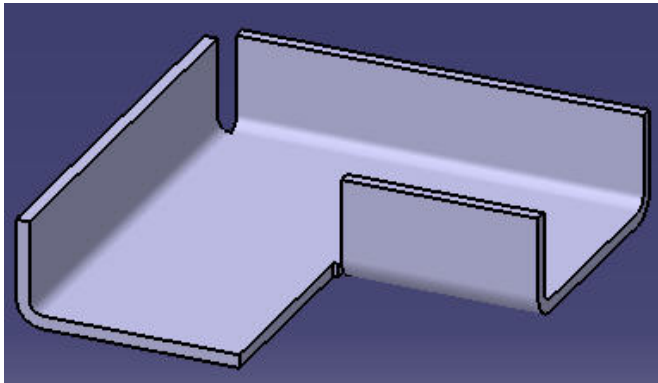


Change “Clearance mode:” below to “Bidirectional” >> OK.




Creating the Third Side Wall

1. Select the Wall on Edge tool .
2. Select the right near edge shown below and create the 3rd wall.




Creating a Cutout

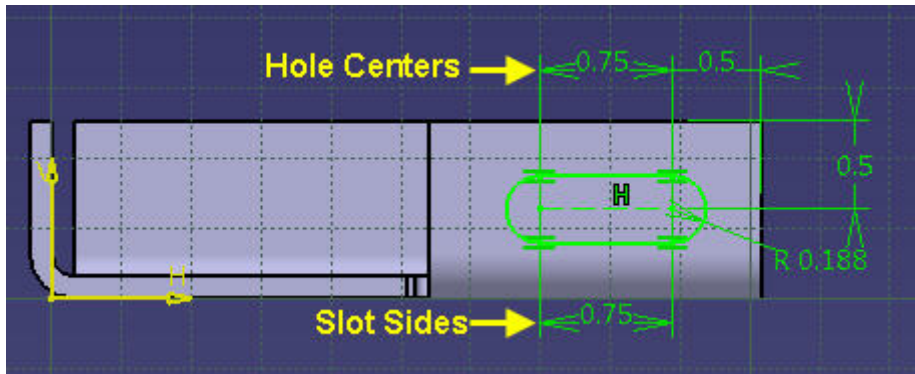
In this task, you will learn how to:
open a sketch on an existing face
define a contour in order to create a cutout.

1. Select the wall on the right (Wall.3) to define the working plane.
2. Click the Sketcher tool 



3. Click the Oblong Profile tool  to create the contour.
4. Click to create the first point and drag the cursor.
5. Click to create the second point.

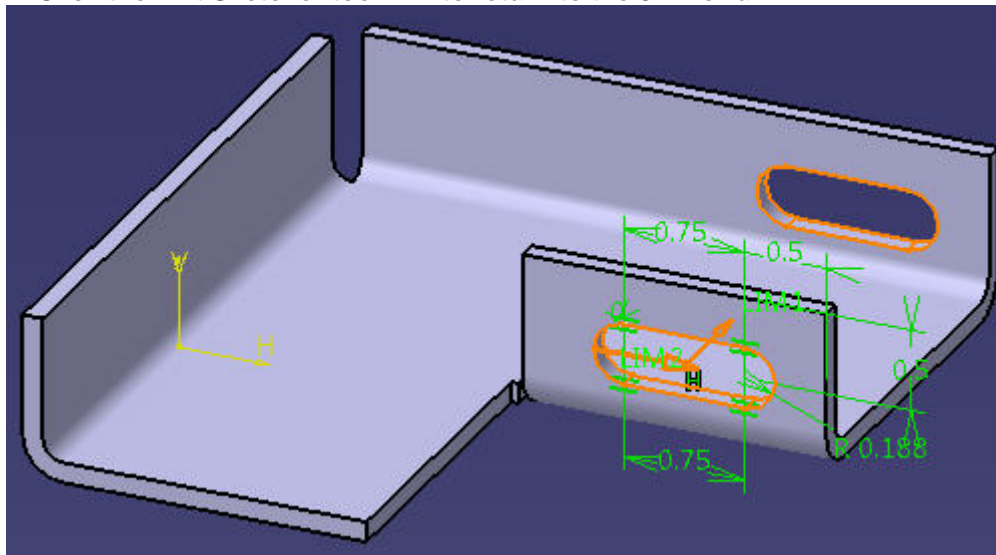
The first semi-axis of the profile is created.



6. Drag the cursor and click to create the third point.

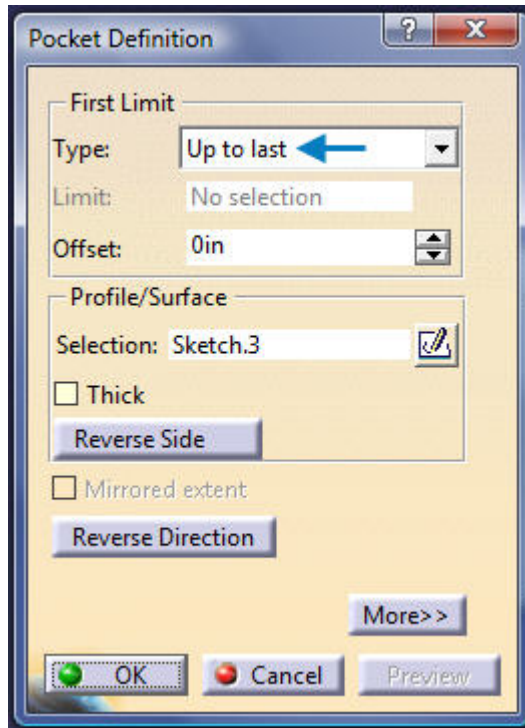
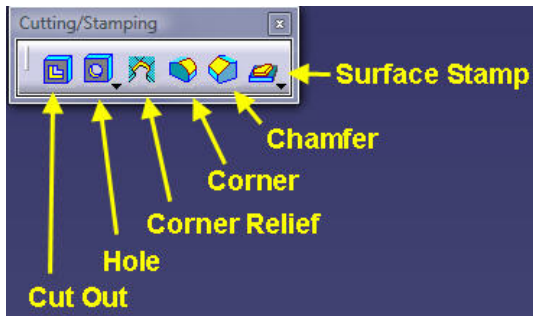
The second semi-axis is created and CATIA displays the oblong profile.

7. Click the Exit Sketcher tool  to return to the 3D world.



8. Select the Cutout tool .

The Pocket Definition dialog box right is displayed and CATIA previews a cutout with default parameters.

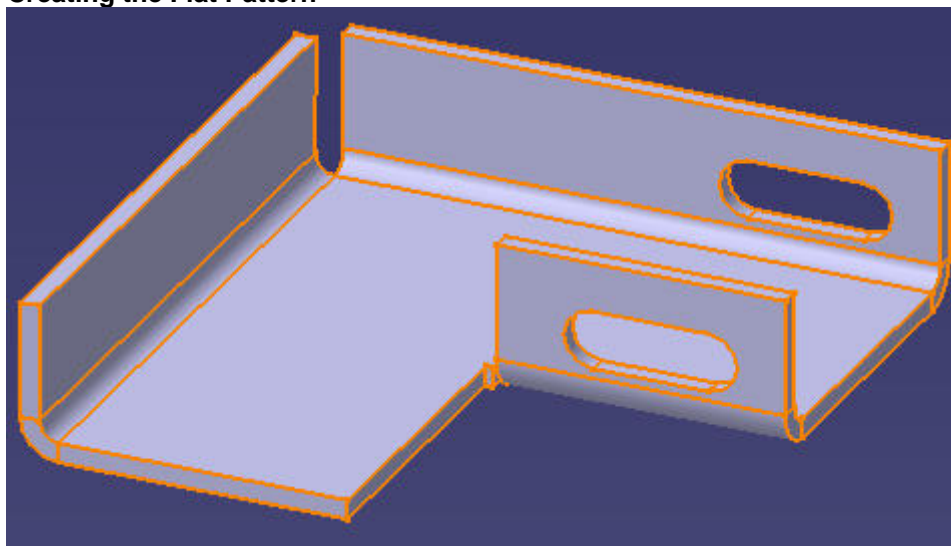


9. Set the Type: "Up to last" option to define the limit of the cutout length.

This means that the application will limit the cutout onto the last possible face, that is the opposite wall.

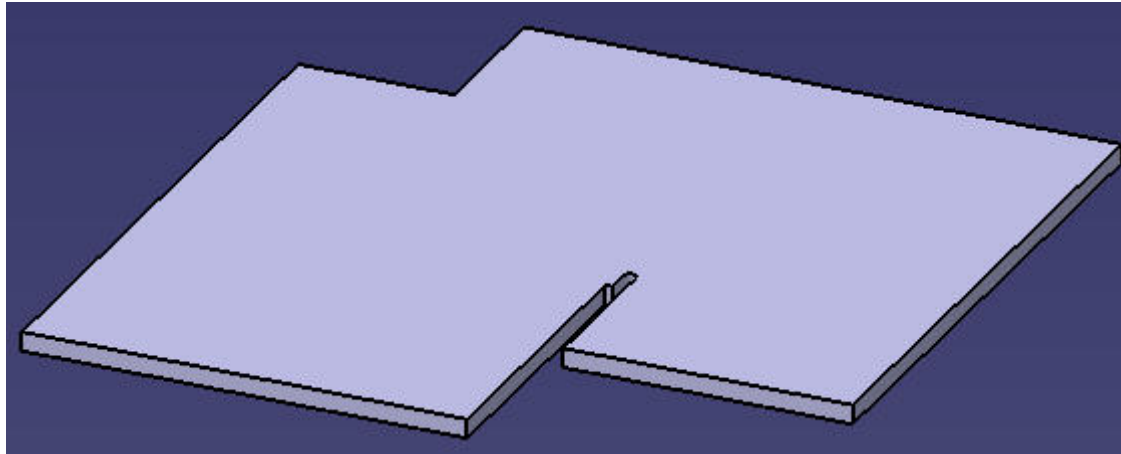
10. Click OK.
This is your cutout:

Creating the Flat Pattern

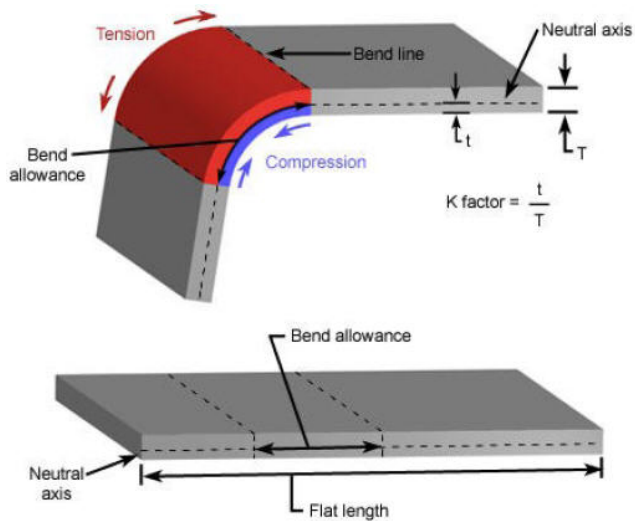




Pick: "Fold / Unfold" to obtain the Flat Pattern below.



The Flat Pattern above includes "Bend Allowance" material.



Copyright © 2009 CustomPartNet

Above is the image at: www.custompartnet.com/wu/sheet-metal-forming.

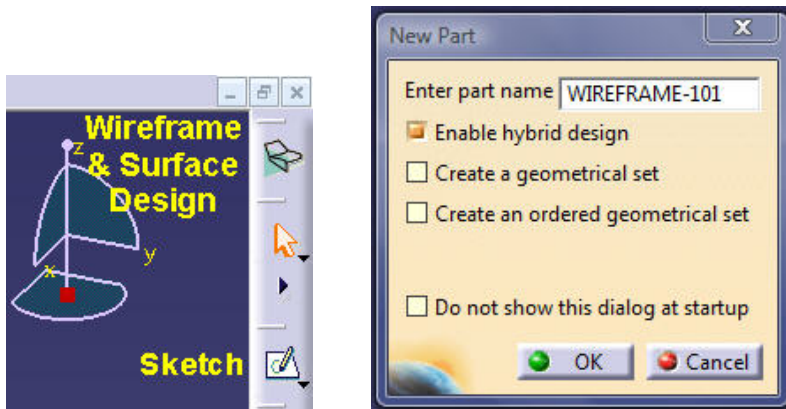
4. WIREFRAME AND SURFACE

CONTENTS

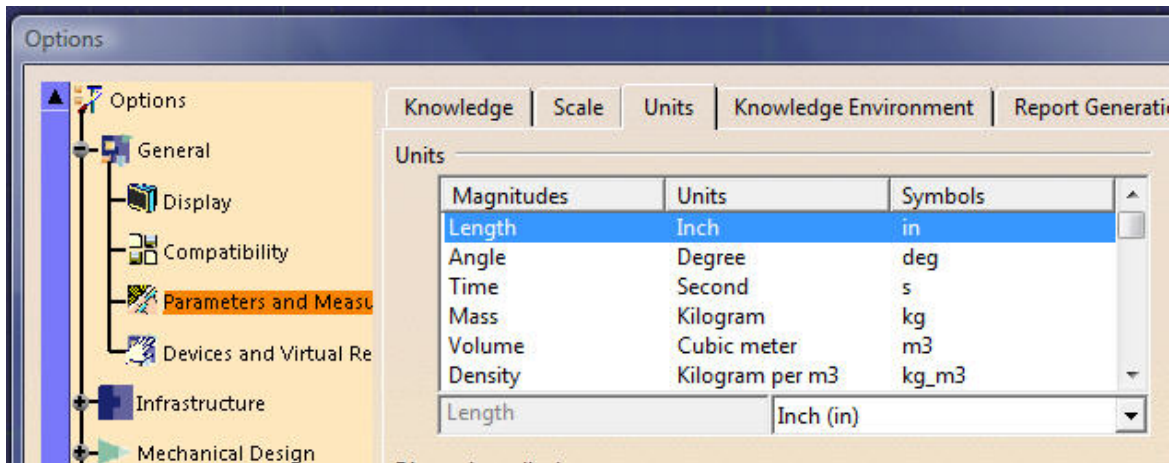
- 4.1 Extruded
- 4.2 Revolved
- 4.3 Swept
- 4.4 Offset
- 4.5 Split
- 4.6 Blend
- 4.7 Loft

Starting Wireframe and Surface Design Workbench
Open Catia

Pick: Start >> Mechanical Design >> Wireframe and Surface Design.



“Enter Part Name” change from “Part1” to “WIREFRAME-101” as shown above.

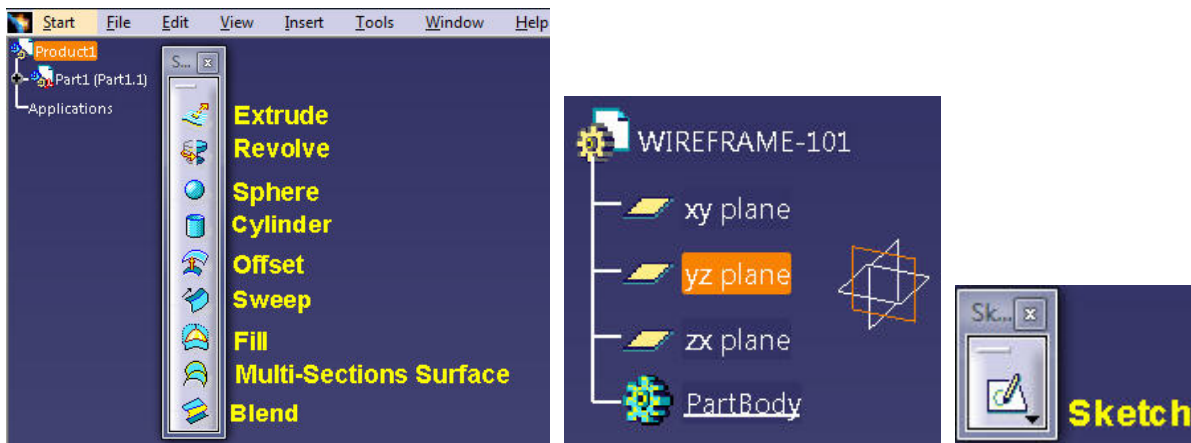


Pick: Tools >> Options >> Parameters and Measure >> Units >> Inch (in) or Millimeters (mm).



Wireframe Toolbar >> Planes. Operations Toolbar.

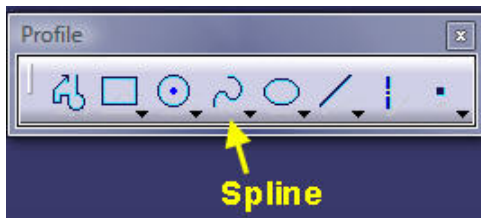
4.1 EXTRUDE SURFACE



The "Surface" toolbar is above.

Pick: yz plane >>

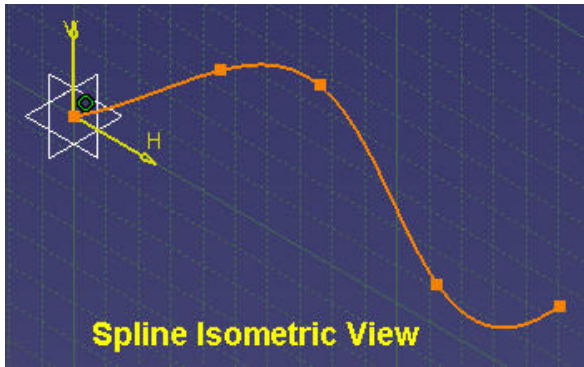
Sketch tool.



Pick the "Spline" tool >>



Sketch the Spline >>



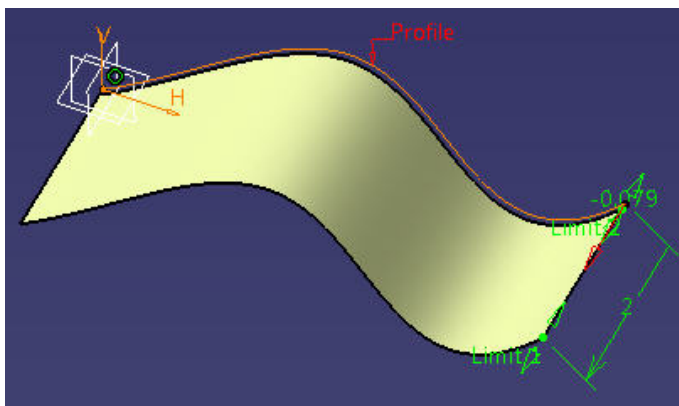
Pick the "Isometric View" tool.



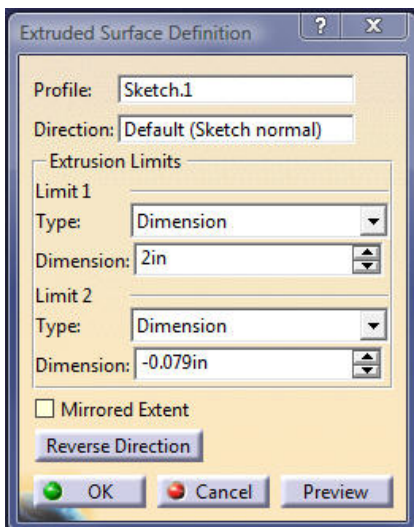
Exit Sketch.



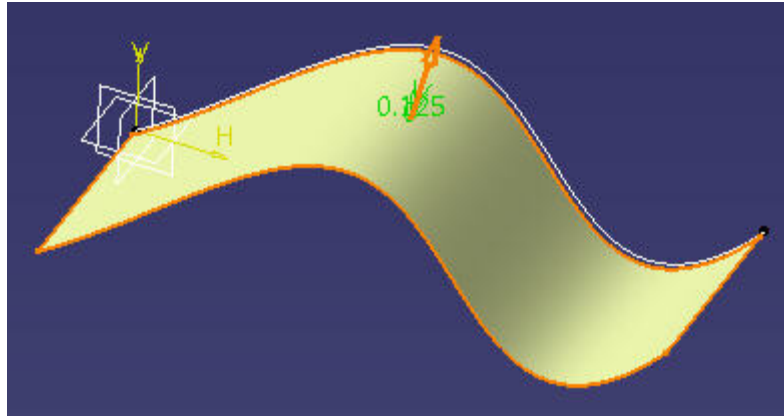
Pick: Extrude >>



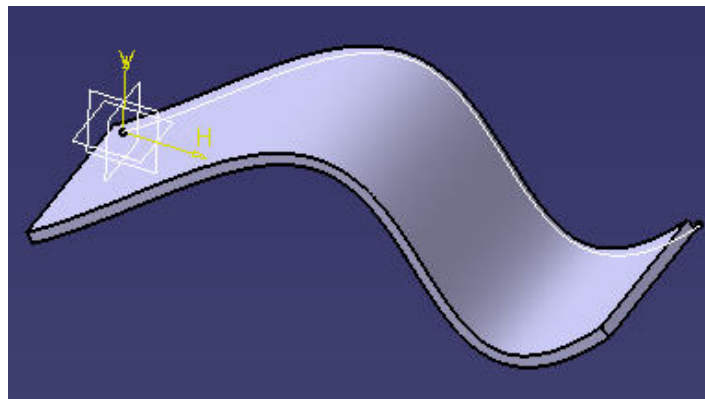
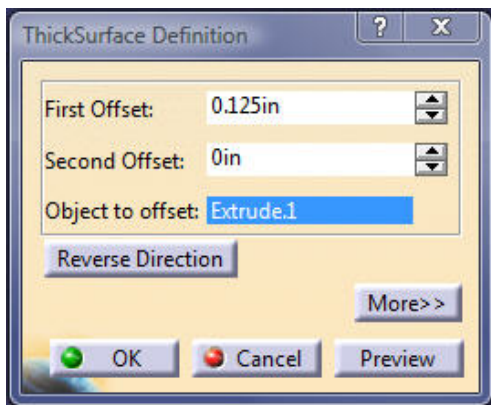
Pick the "Spline" profile.



Pick: Extrude >> Dimension >> 2in >> OK. Extrude.1 has been added to the tree.



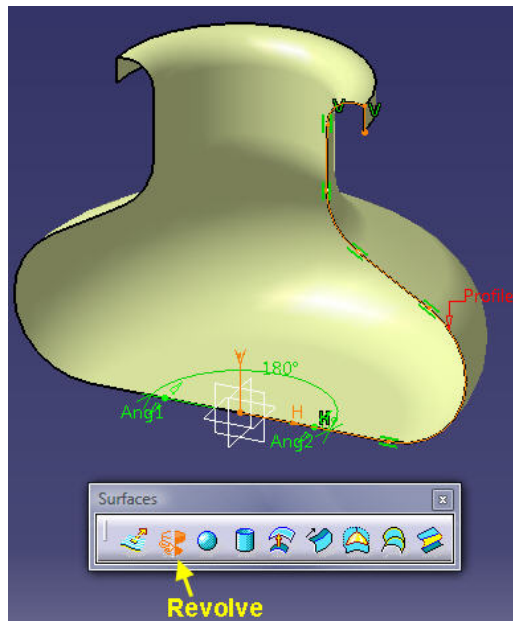
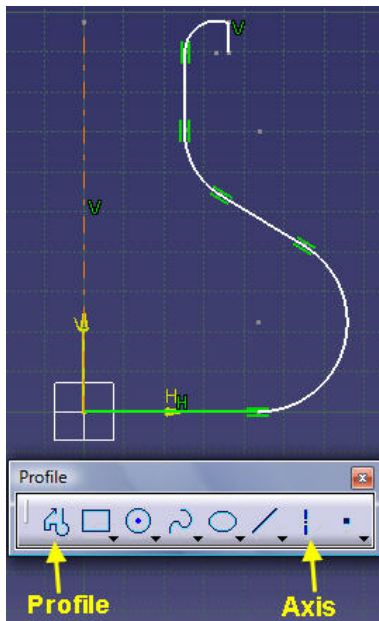
Start >> Mechanical Design >> Part Design >> Thick Surface >> Pick the extruded surface



First Offset (thickness above profile) >> 0.125in >>

Second Offset (thickness below profile) >> 0in >> OK.

4.2 REVOLVED SURFACE

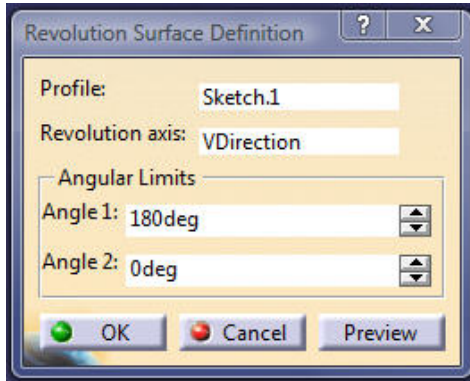


Pick: yz plane >> Sketch Profile (sketch as above) >> Axis (add to sketch) Exit Sketch.

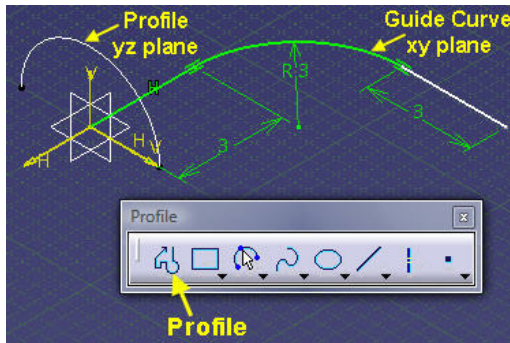
Start >> Wireframe and Surface.

Pick: Revolve >> Pick the profile above "Sketch.1" >> Pick the Revolution axis: >> VDirection.

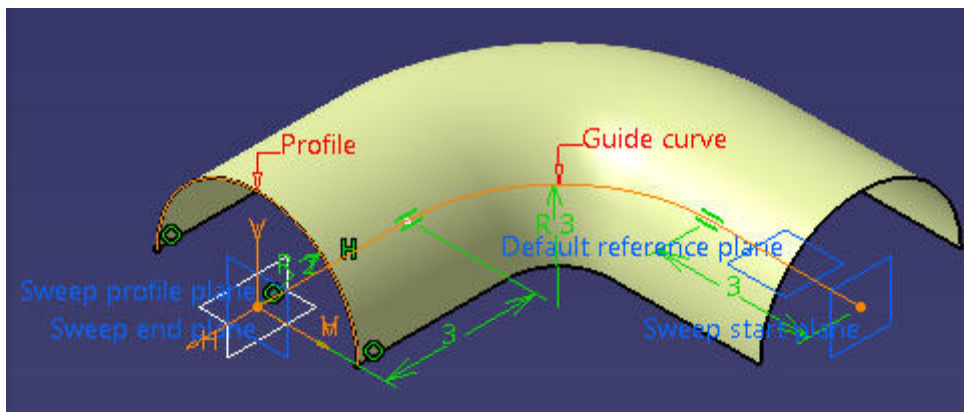
The "Revolution Surface Definition" box below will open automatically.



4.3 SWEEP SURFACE



Sketch the 3-Point Arc in the yz plane.
Sketch the Guide curve in the xy plane.
Method:
Pick: yz plane >> Sketch tool >>
Sketch.1 (3-Point Arc left) >> Exit Sketch
>> xy plane >> Sketch tool >> Isometric
View tool >> Sketch.2 >> xy plane >>

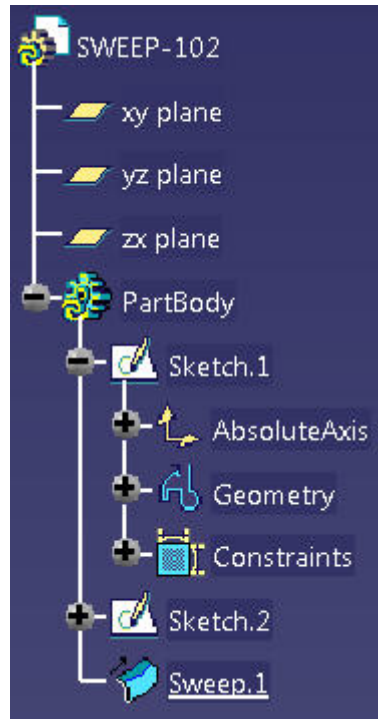
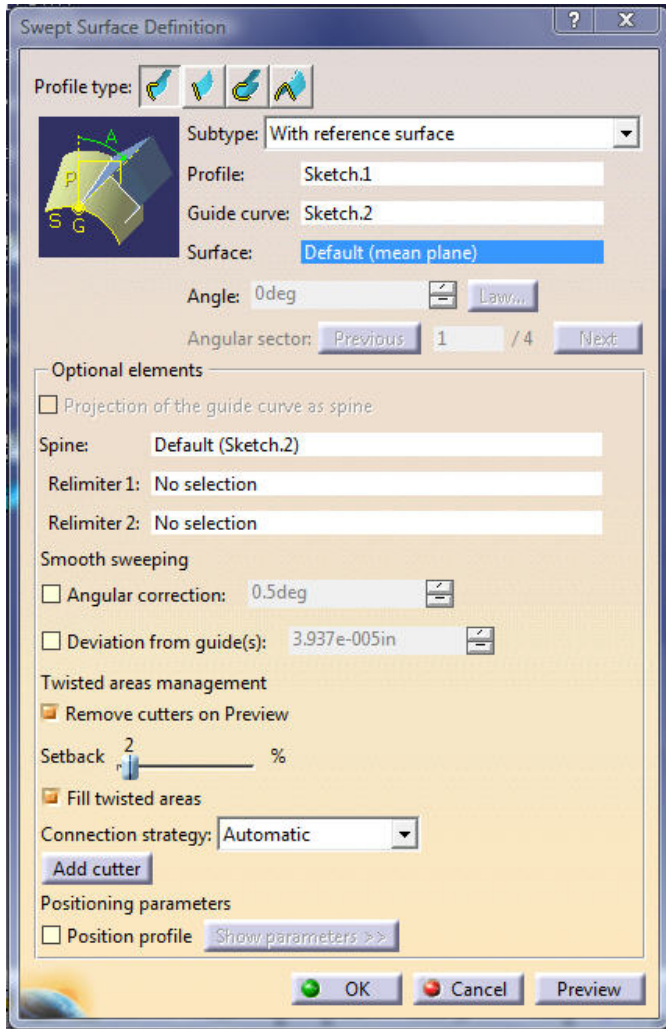


Start >> Wireframe and Surface Design.

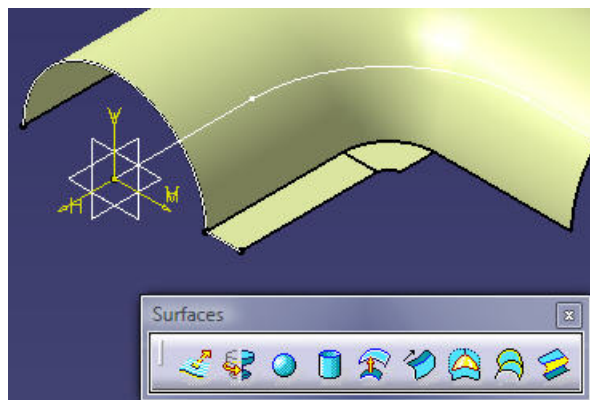
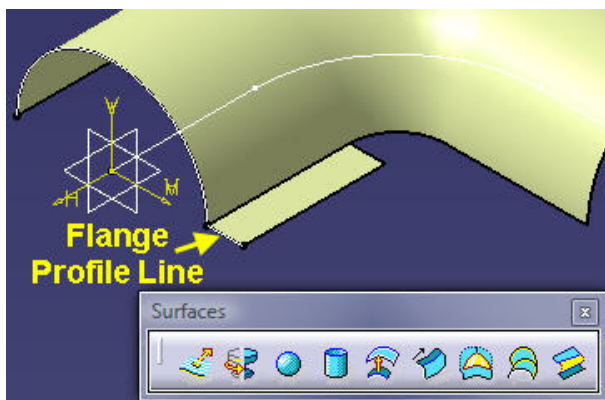
The "Swept Surface Definition" box below will open automatically.

Pick the: "Profile" box below and the Profile in the drawing above.

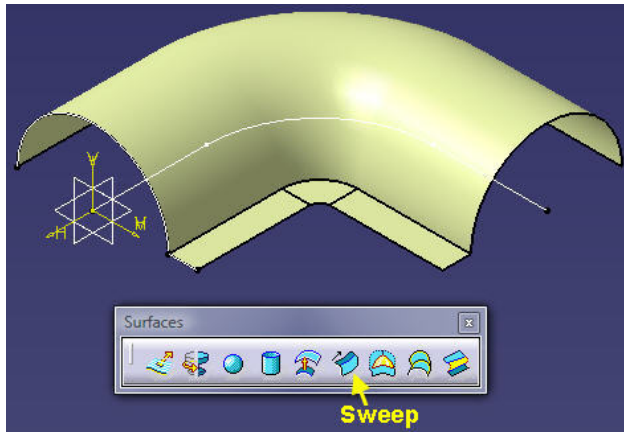
Pick the: "Guide Curve" box below and the Guide Curve in the drawing above.



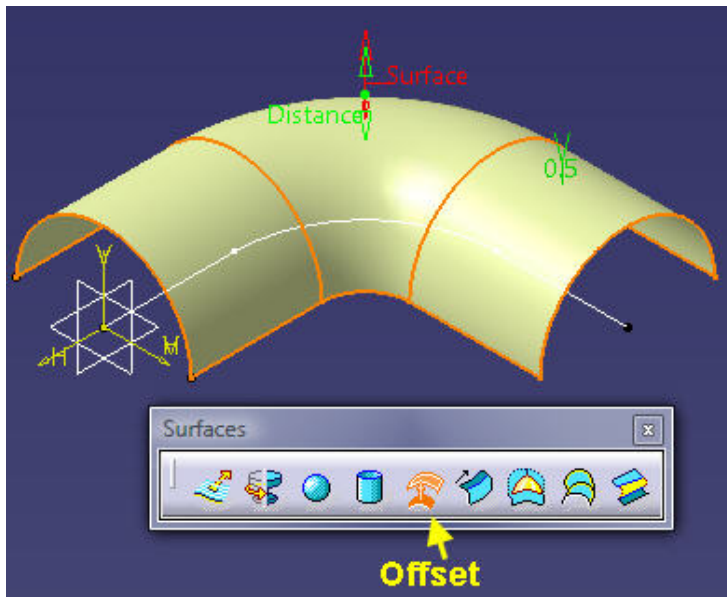
Sweep.1 has been added to the tree above right.



Sketch the flange profile above and use “Sweep” to create the first section of the flange. Repeat to create the remaining two sections of the flange.

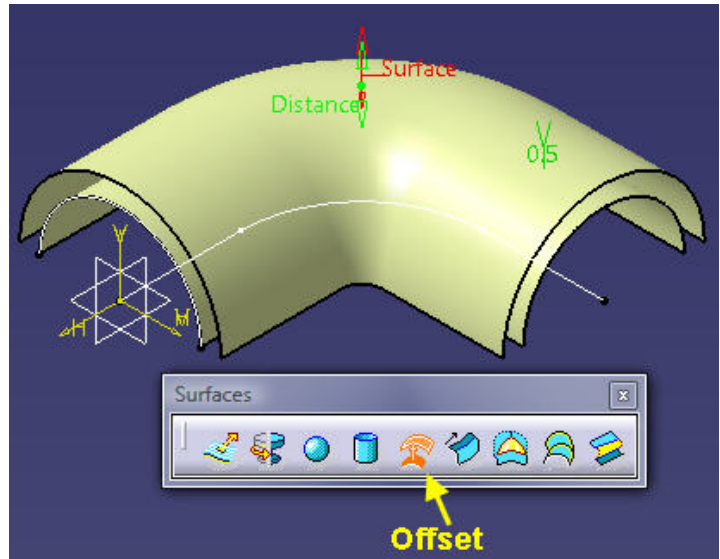
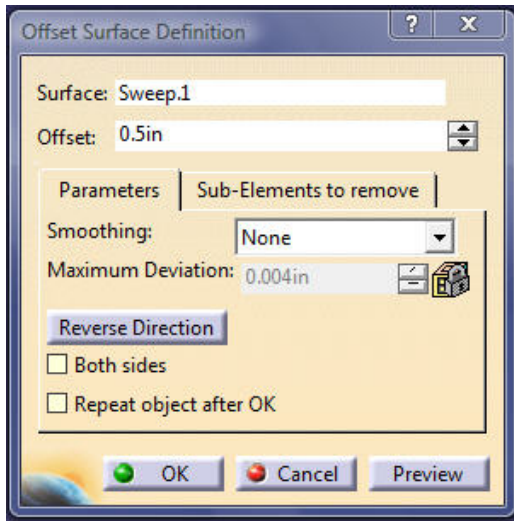


4.4 OFFSET SURFACE



Pick the: Offset tool >> Pick: Sweep.1 in the tree upper right.

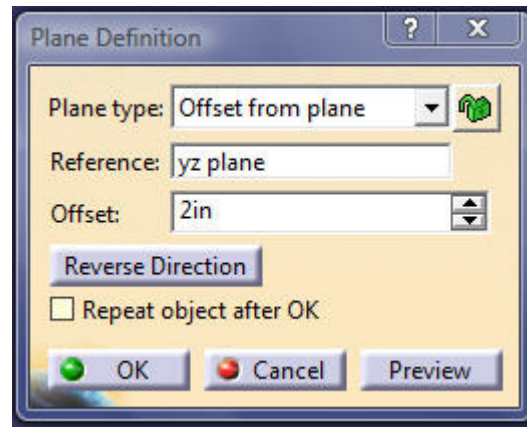
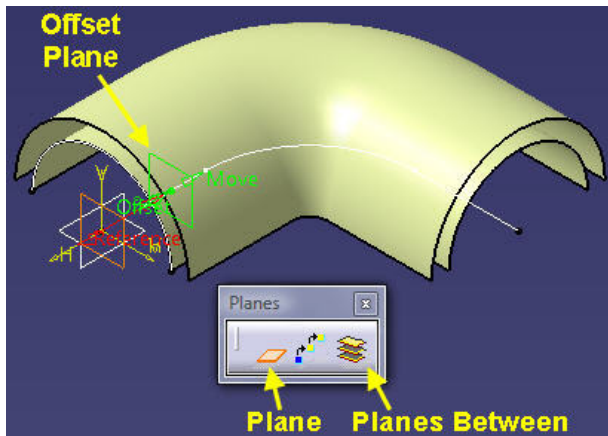
Offset.1 is added to the tree.



The "Offset Surface Definition" box above will open automatically.

The "Offset" surface has been created above.

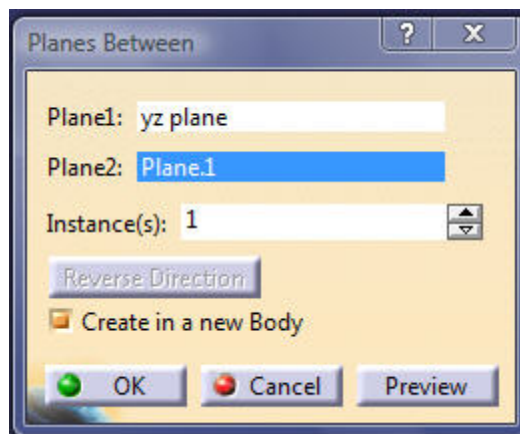
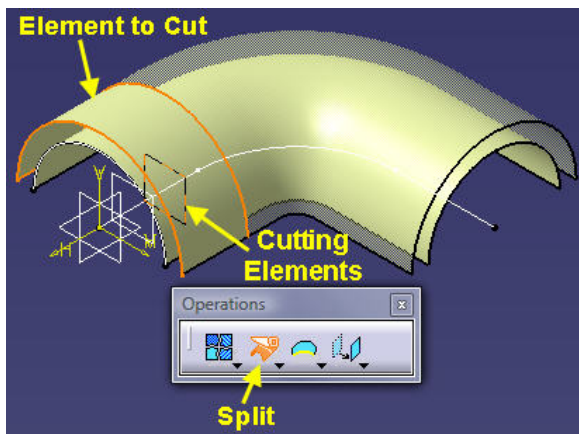
4.5 SPLIT SURFACE

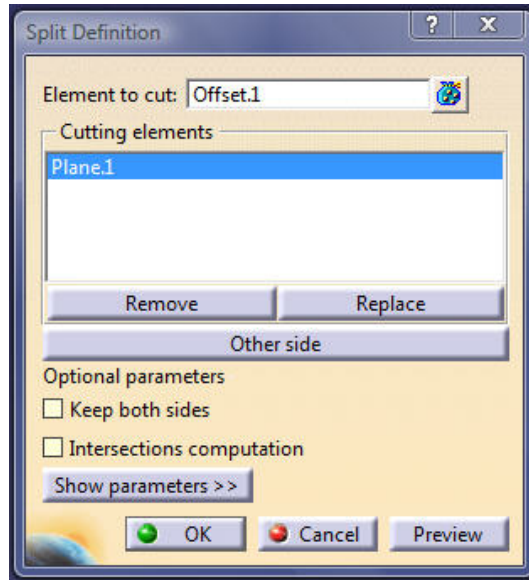
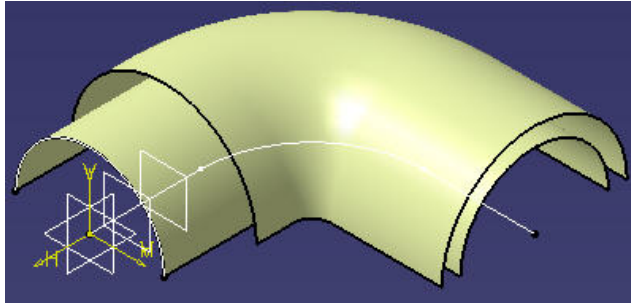


Pick the: "Plane" tool >> Offset >> 2in >> OK.

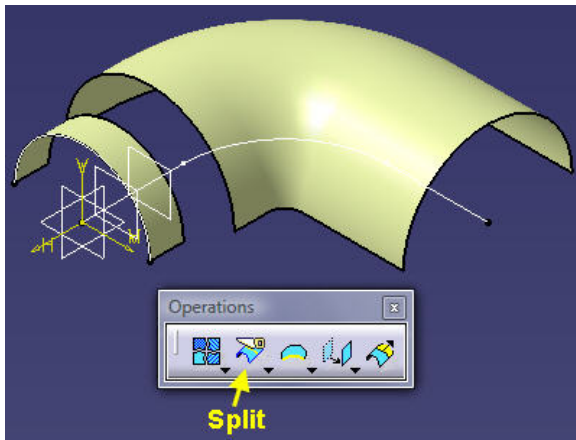
Pick the: "Planes Between" tool >> Plane1: >> yz plane >> Plane2: >> Plane.1 >>

Instance(s): >> 1 >> OK. The two planes become "Cutting Elements" in the Split operation.



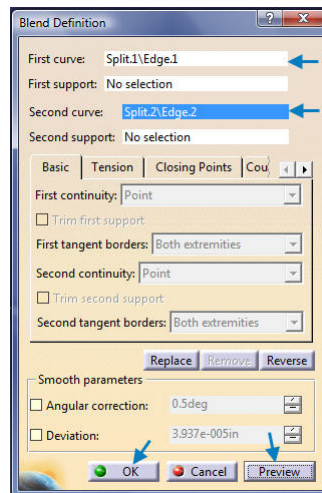
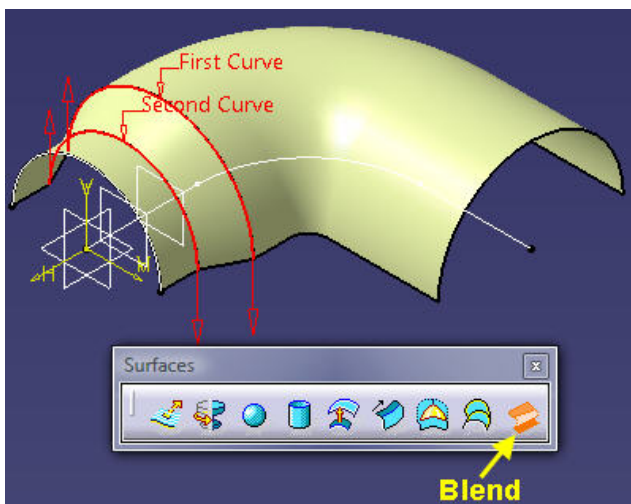


Pick the: "Split" tool >> Element to cut: >> Offset.1 >> Cutting elements >> Plane.1 >> OK.

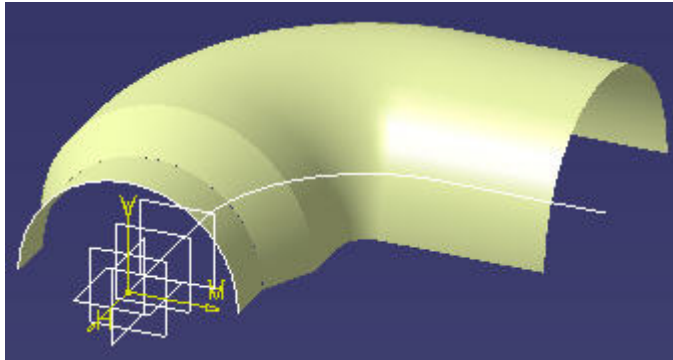


Pick the: "Split" tool >> Element to cut: >> Sweep.1 >> Cutting elements >> Plane.2 >> OK.

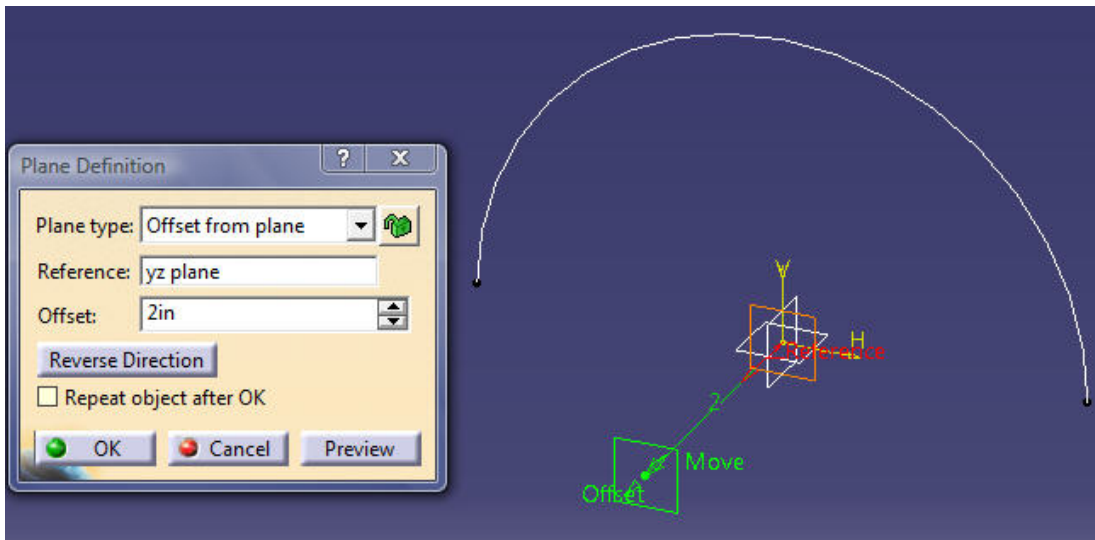
4.6 BLEND



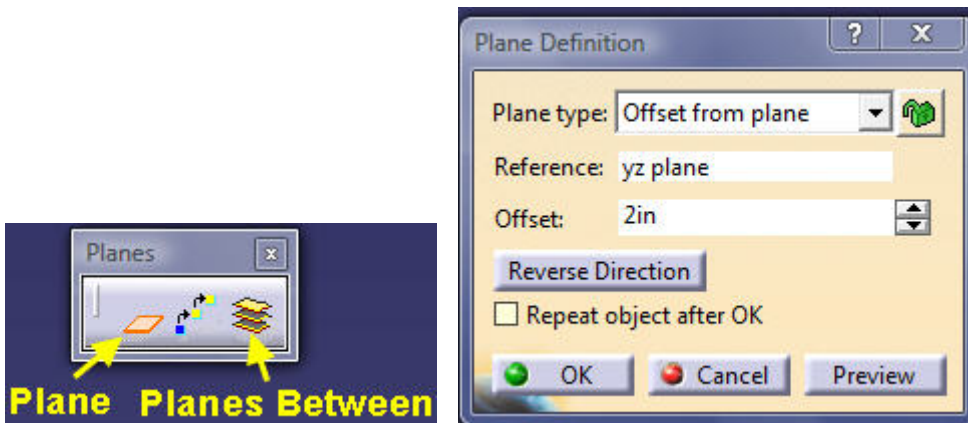
Above surface: File >> Save.



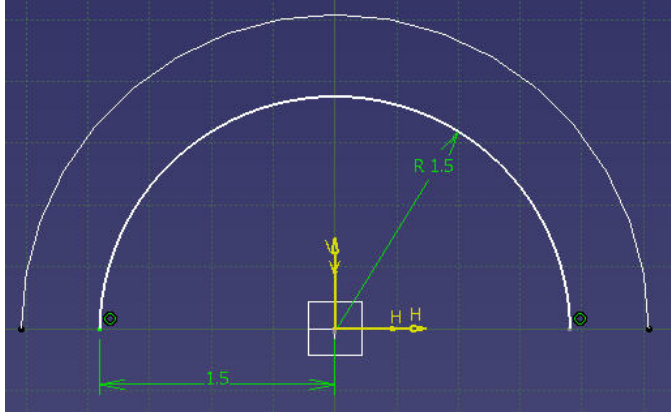
4.7 LOFT



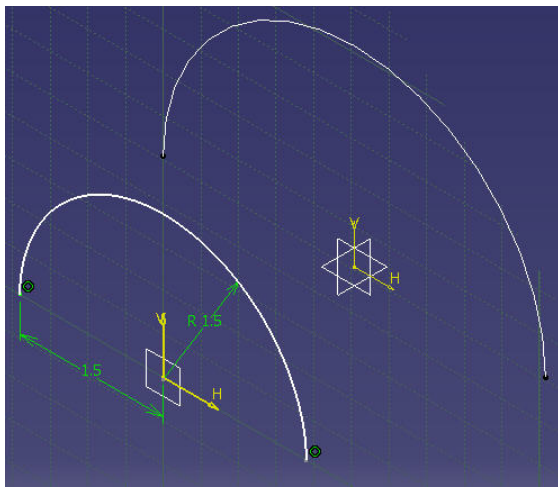
Sketch the 3-Point arc with a 2in radius in the yz plane.



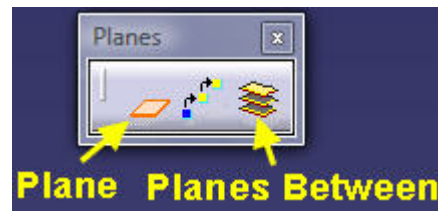
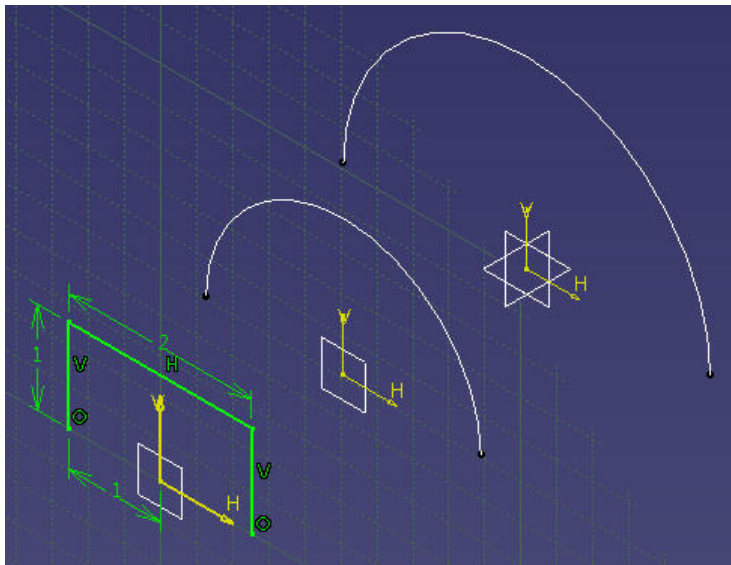
Pick the "Plane" tool >> "Offset from plane" >> Offset: >> 2in as shown above.



Sketch the second 3-Point arc with a 1.5in radius in the offset Plane.1 created above.

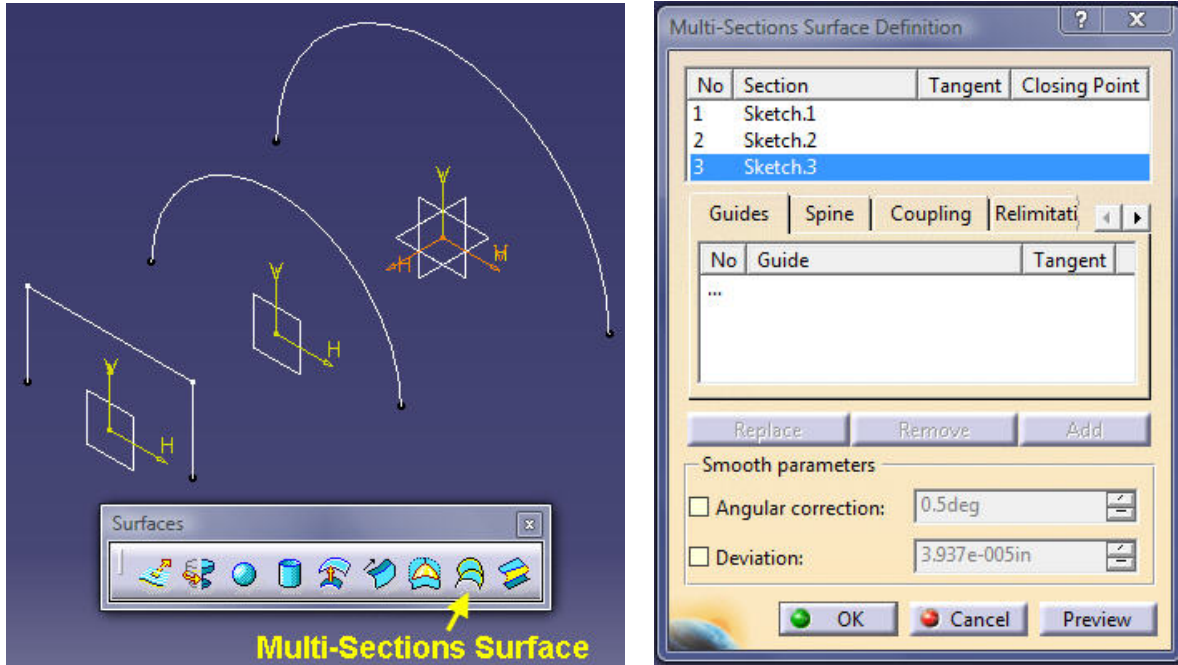


Pick the "Isometric View" tool and sketch in the 3D view for clarity.



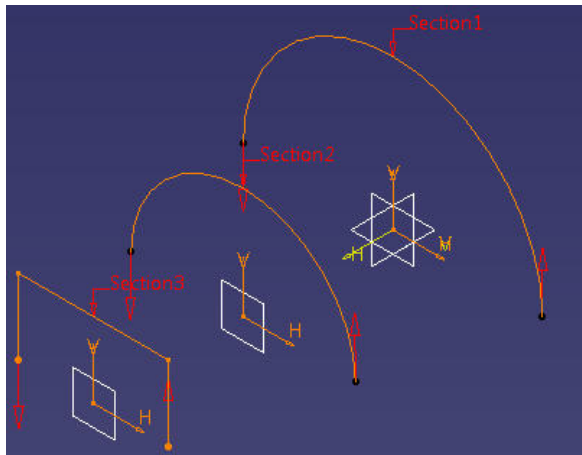
Pick the "Plane" tool >> "Offset from plane" >> Offset: >> 2in as shown above.

Sketch the 1in by 2in rectangle in the offset Plane.2.

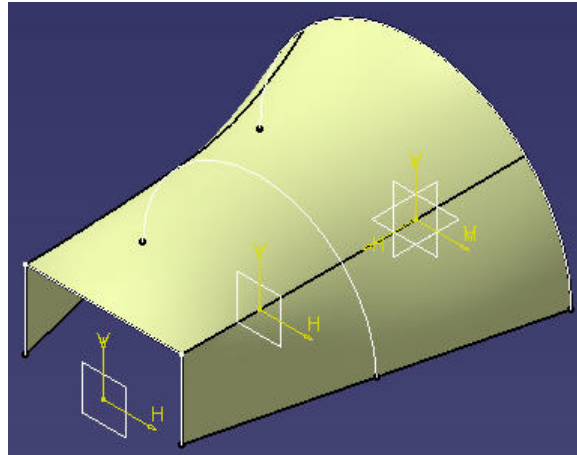


Multi-Sections Surface

Pick the: "Multi-Sections Surface" tool >> The Multi-Sections Surface Definition box will open.



Pick Sections: 1, 2, and 3.

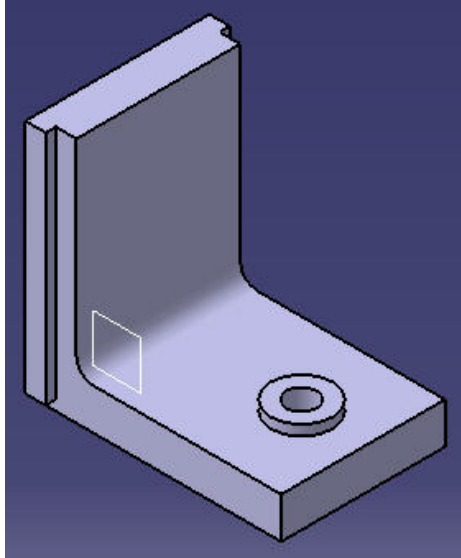


The completed "Loft".

5. FINITE ELEMENT ANALYSIS FEA

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

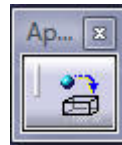
Make sure Num-Lock is off. Tools >> Options >> Units >> inches.



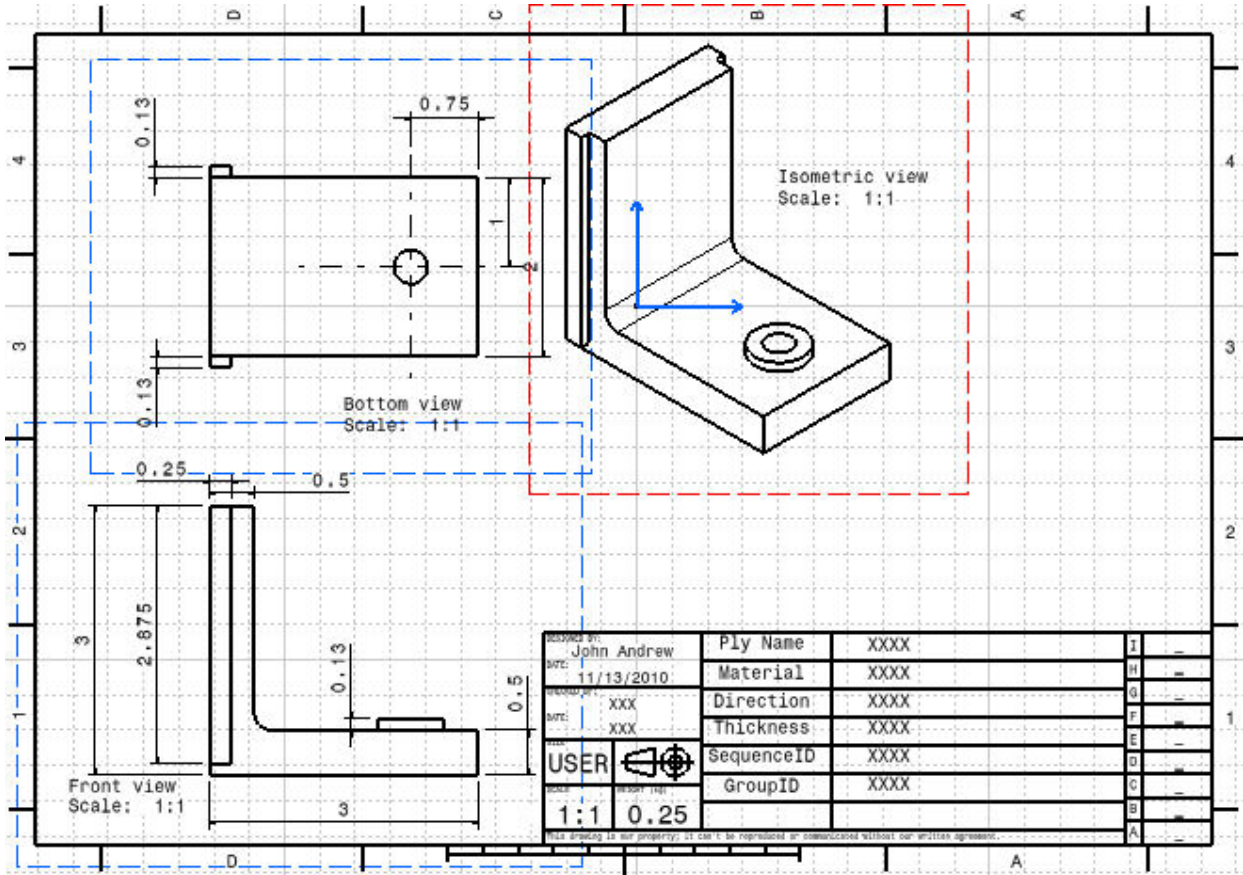
ANGLE BRACKET

The angle bracket has a lip on each vertical edge representing fillet welds.

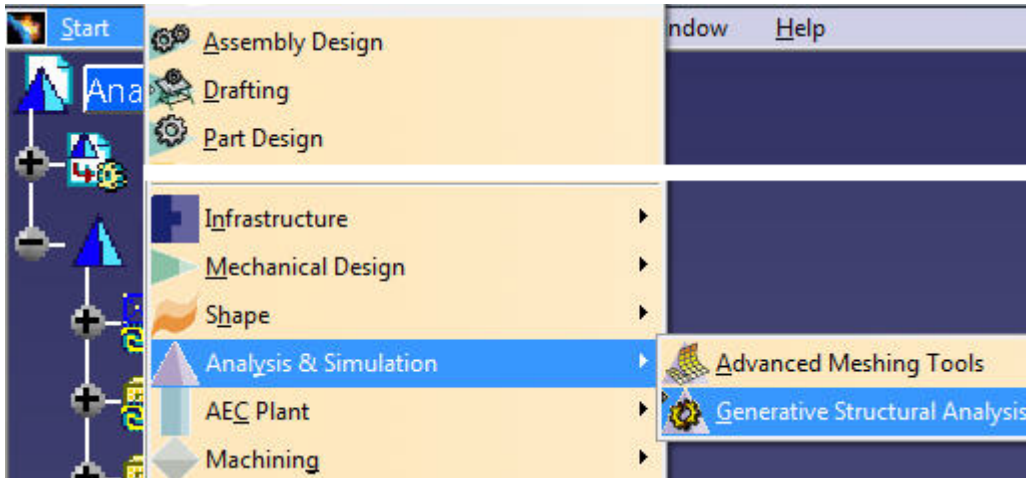
The ring around the hole is the area of pressure applied by the bolt head and washer.



Create the above angle bracket – dimensions below. Select: Apply Material >> Aluminum

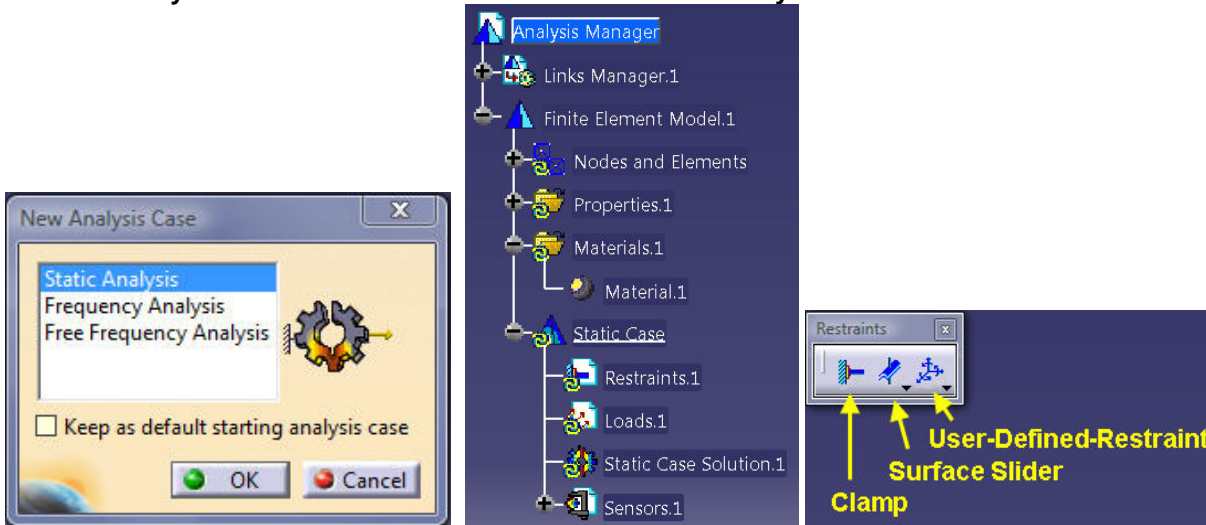


The dimensions of the Angle Bracket are given in the above Catia drawing.



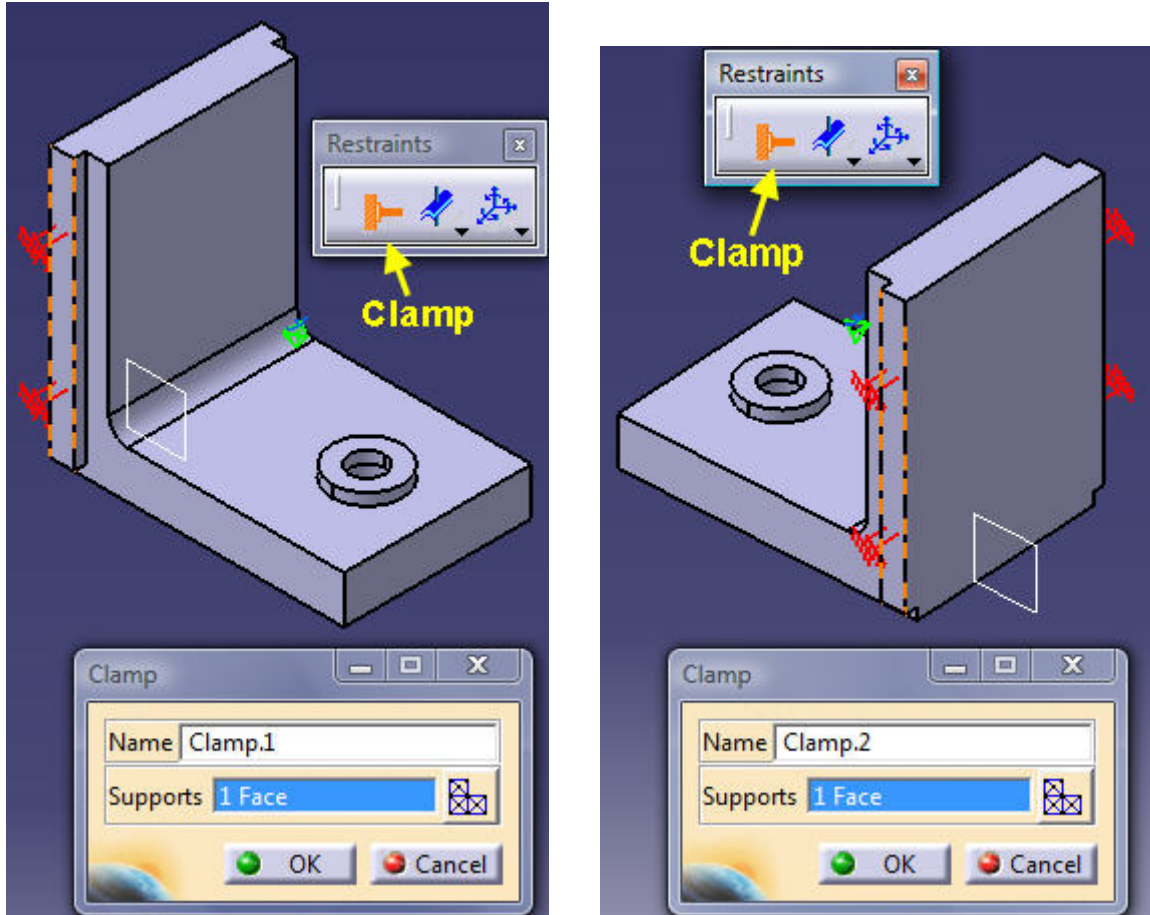
Use above drop-down menu:

Start >> Analysis & Simulation >> Generative Structural Analysis



"New Analysis Case" >> Static Analysis. Tree.

"Restraints" toolbar



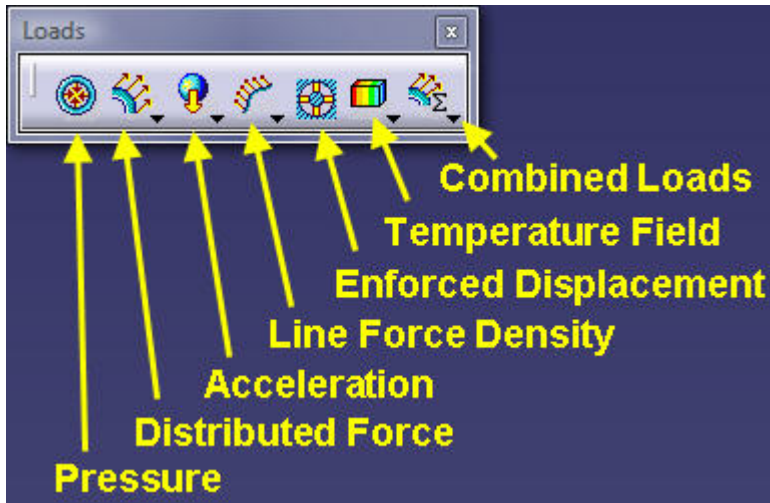
The selected area on the lip of the vertical leg of the angle bracket duplicates a fillet weld.

This weld area is “Clamped” above left and Right.

The clamps are added to the tree under Restraints.

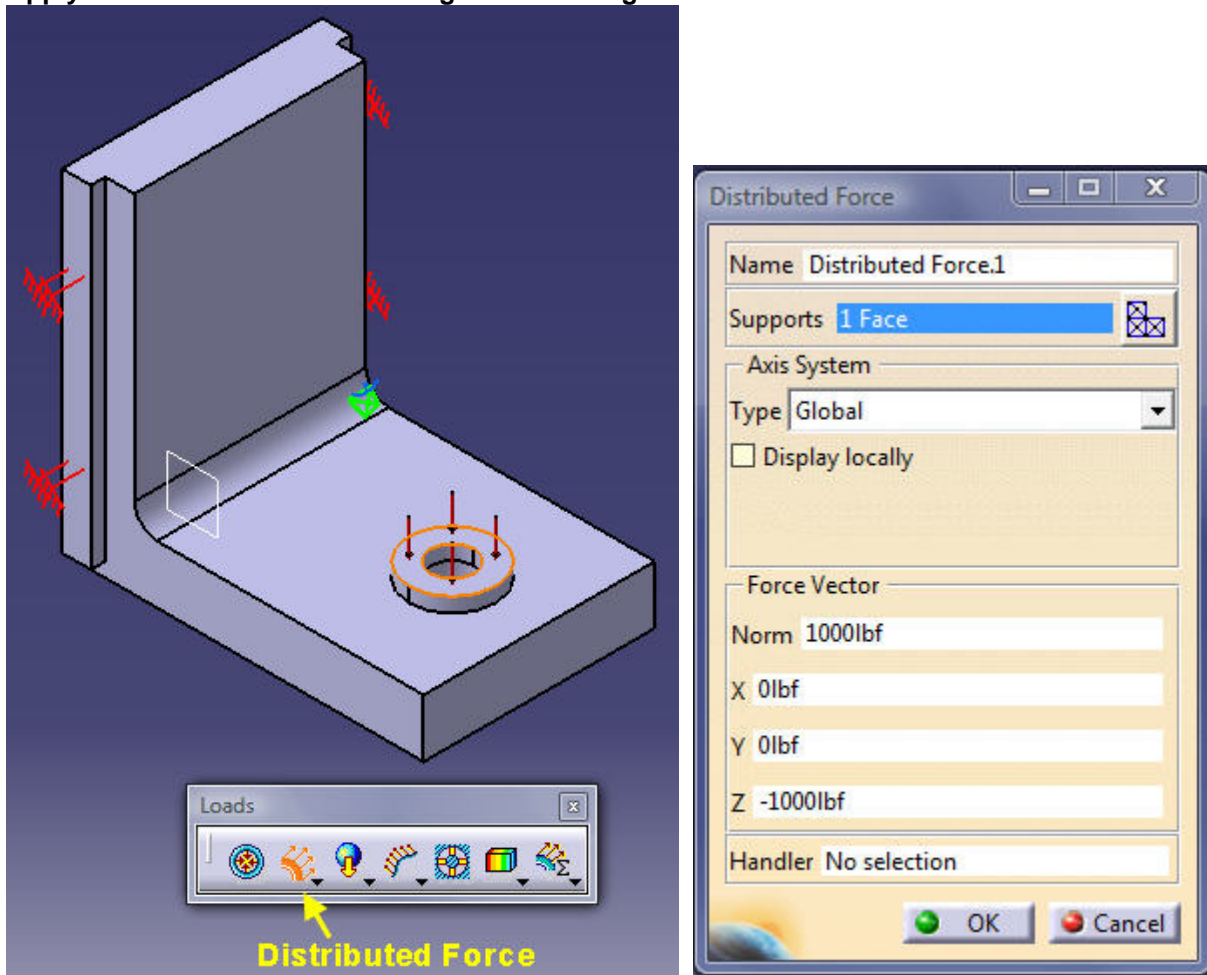
Also, note that the clamps are red.

They will be updated in the Analysis below.



The ring around the hole is the area of pressure applied by a bolt head with washer.

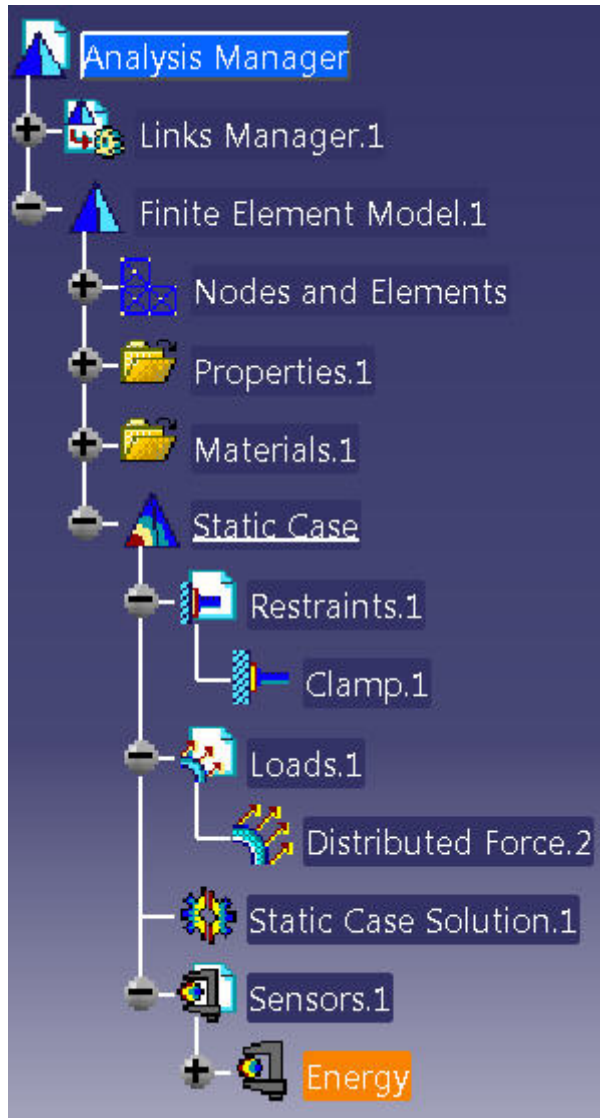
Apply a distributed force to the angle bracket ring around the hole.



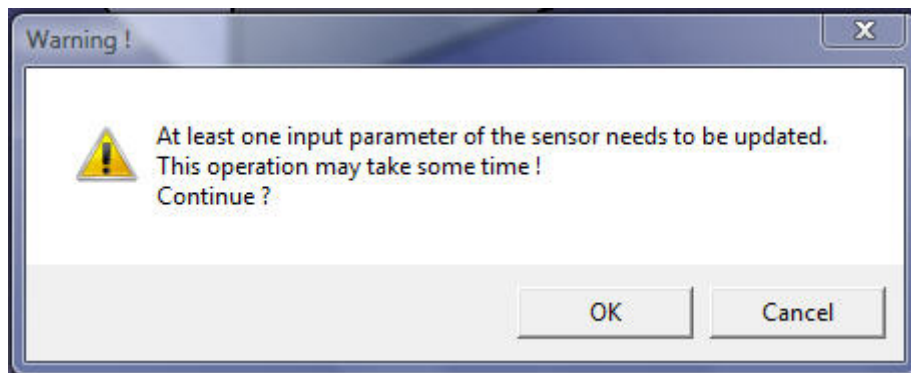
Pick: Distributed Force >> Point on ring area as shown above.

Update the Analysis:

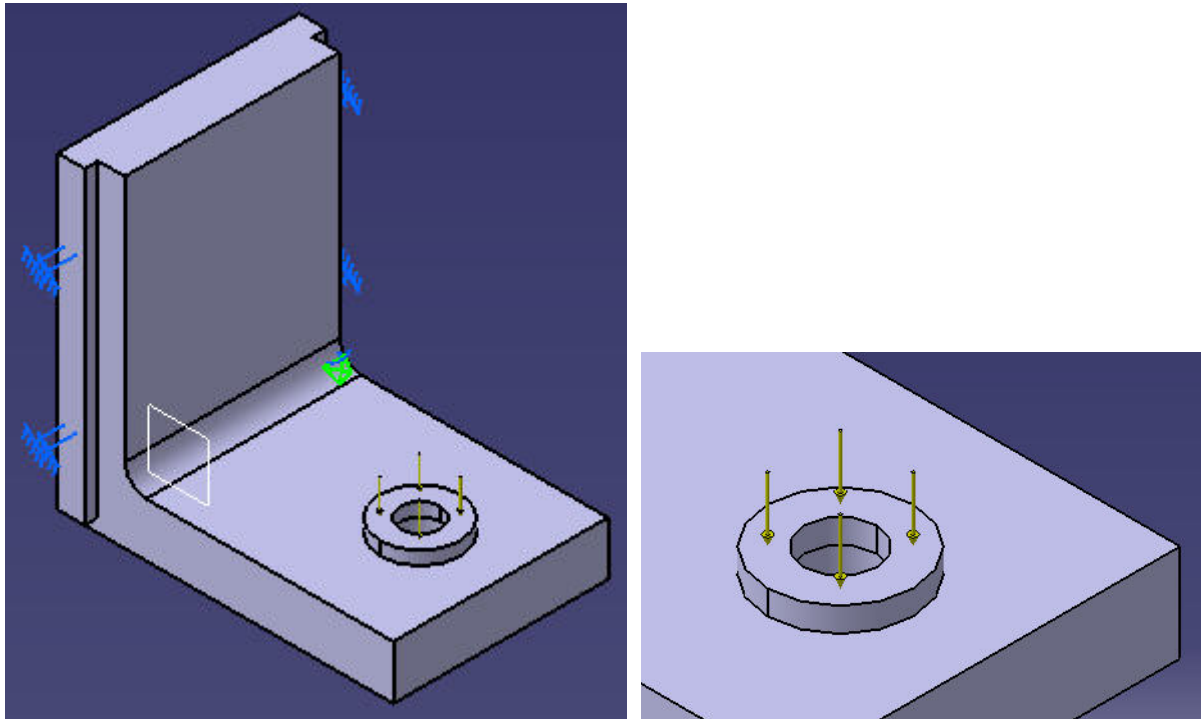
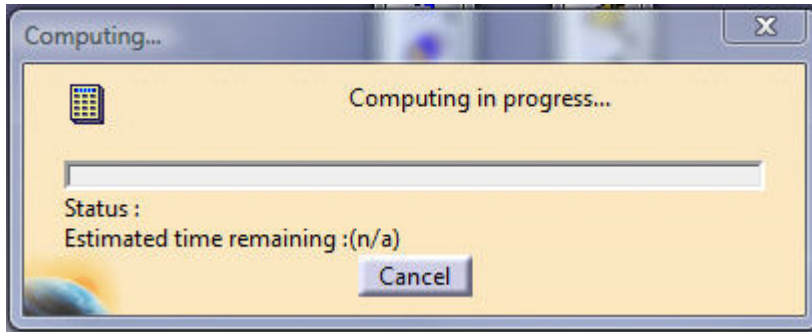
The boundary conditions set by the clamps and force are next updated.



Right click on the Energy tool in the tree, and select Update Sensor.



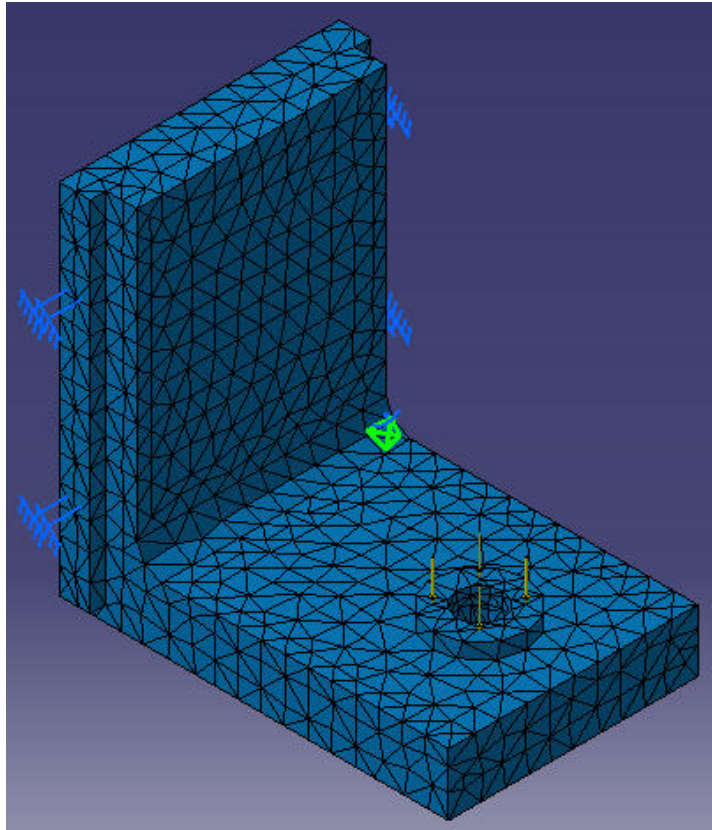
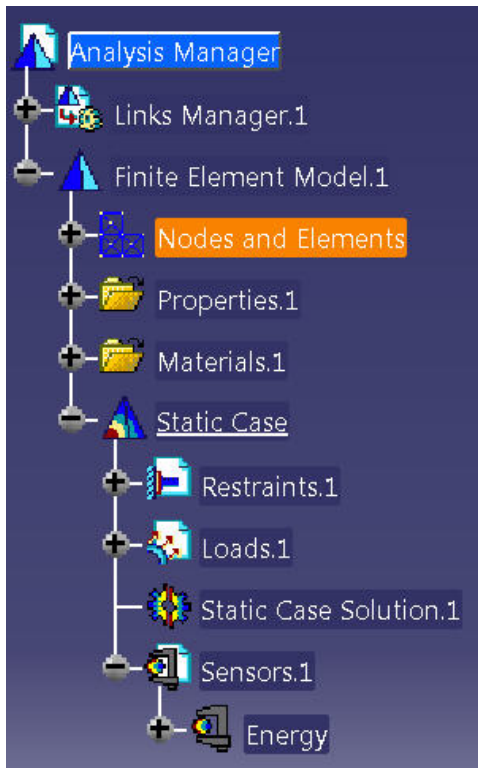
Click OK through the warning, and you will see computation boxes appear.



The clamps and the load no longer are red after the above update.

The green arrows show a uniform distributed force over the surface indicated.

File >> Save.

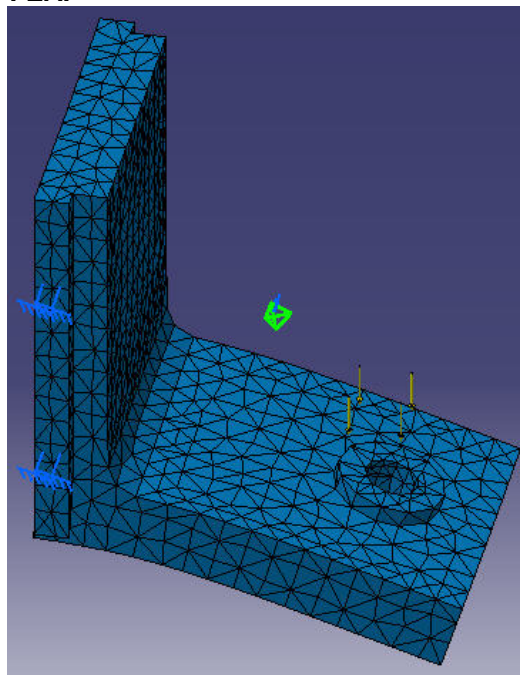


Right click on Nodes and Elements, then select Mesh Visualization.

The part is divided into many small tetrahedrons.

STRESS AND DEFLECTIONS

Now we will look at the results of the FEA.



First click the Deformation tool.

The mesh deformed, according to the amount of load or force.

Notice that the clamped welds have not moved.

Also there is a “Deformed Mesh entity under the Static Case Solution.

The mesh size can be changed.

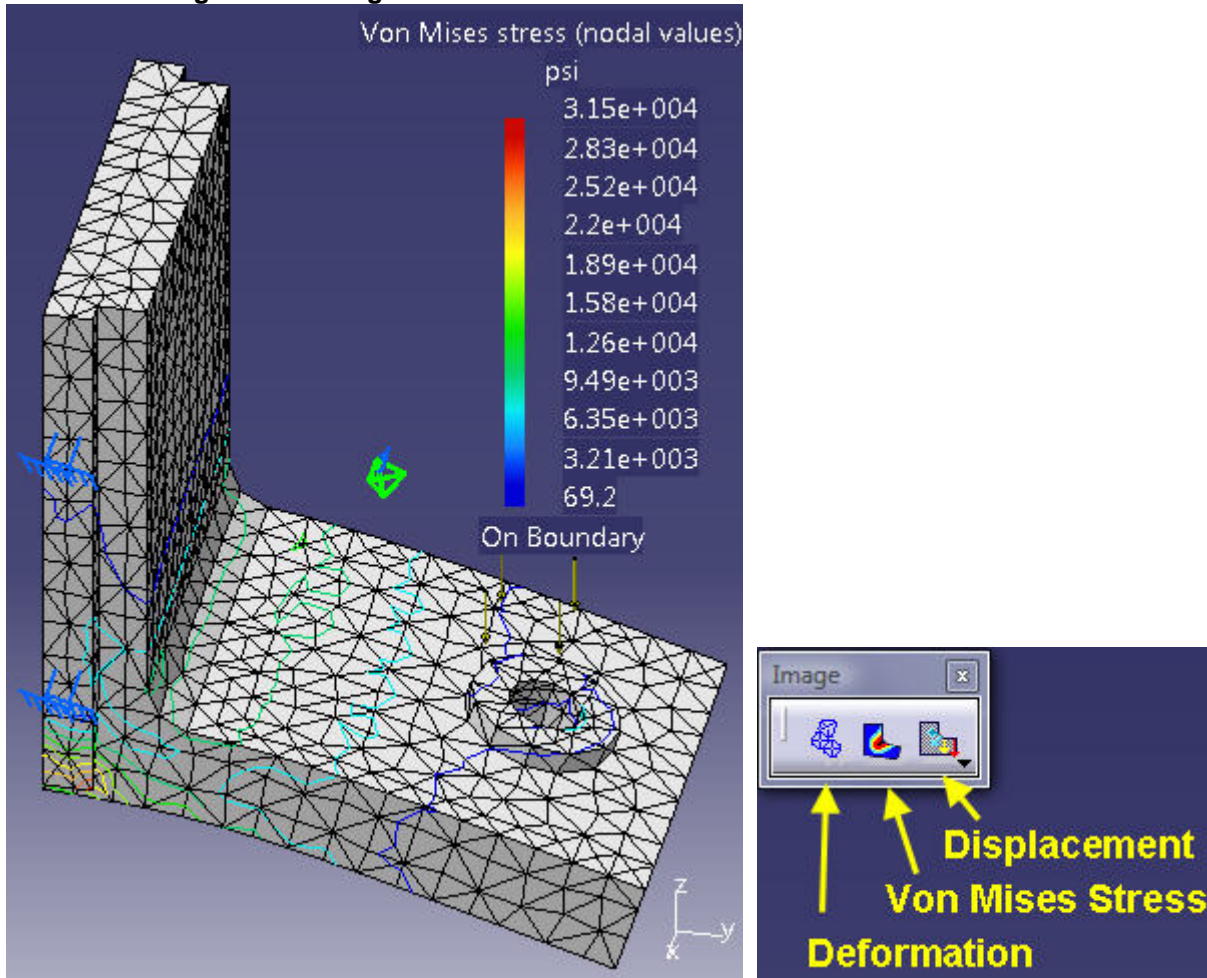
In the tree, double click on OCTREE Tetrahedron Mesh.

Change the size. Click OK.

Note that the analysis tree objects have gone out of date.

As before, right click on Nodes and Elements and select Mesh visualization.

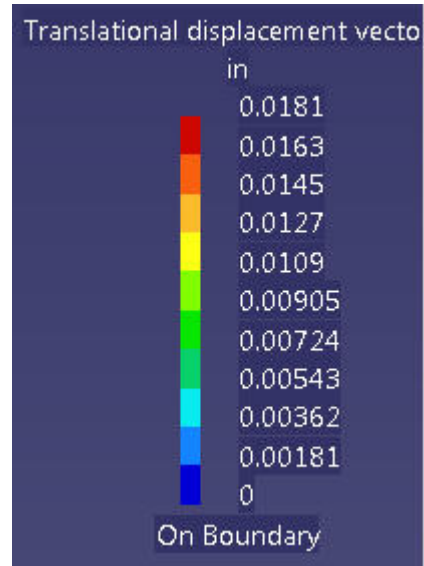
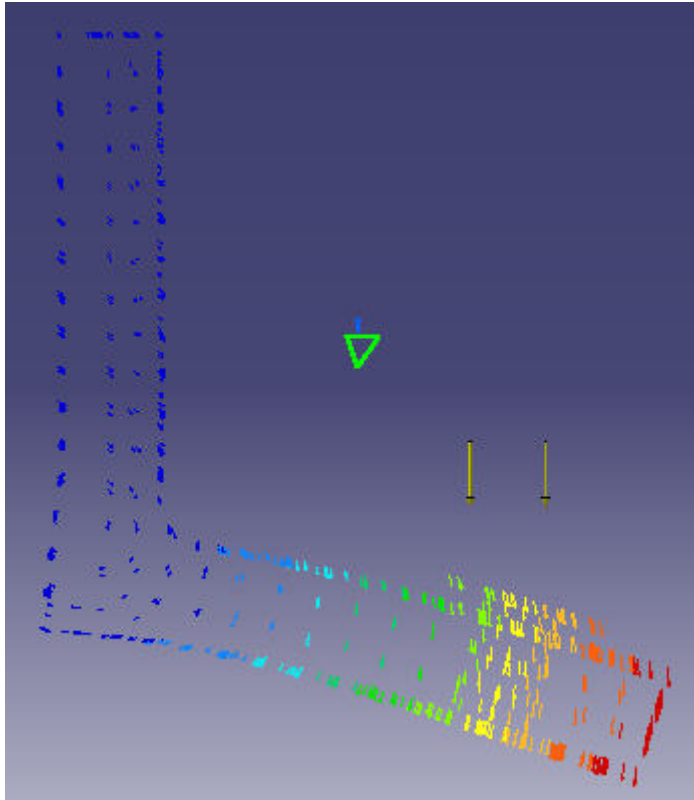
Click OK through the warning. You should now see a finer mesh.



Now click the Von Mises Stress tool.

This shows you a color coded stress plot across the mesh.

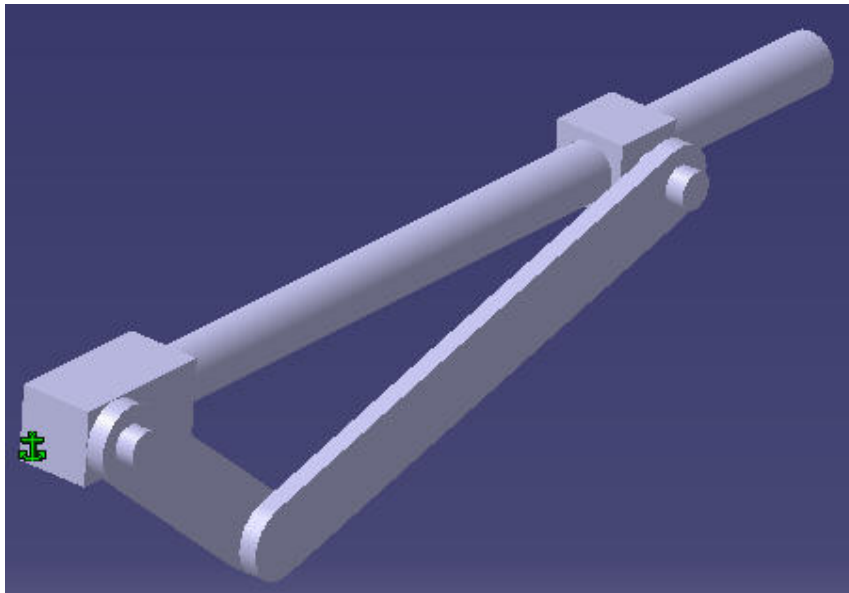
Note also that the previous image (deformed mesh) is now deactivated.



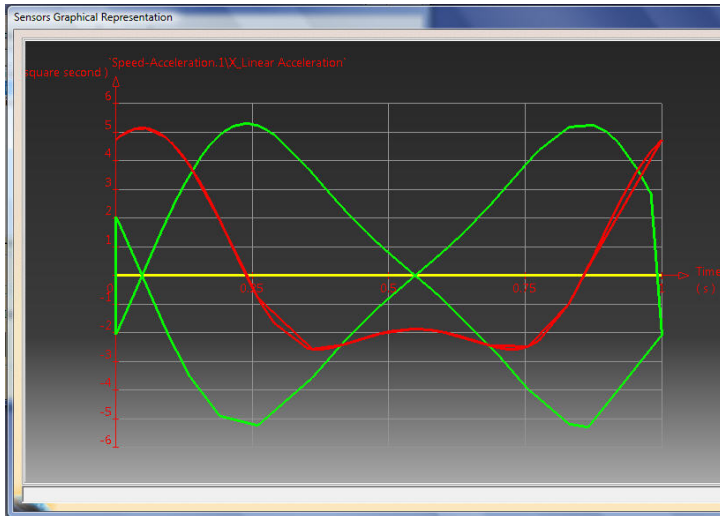
Click on the Displacement tool.

This shows displacement vectors, indicating the amount of displacement of the part due to the load.

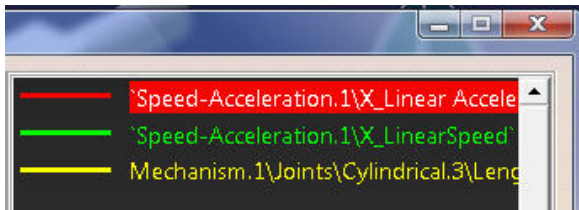
6. MECHANISM DYNAMIC ANALYSIS



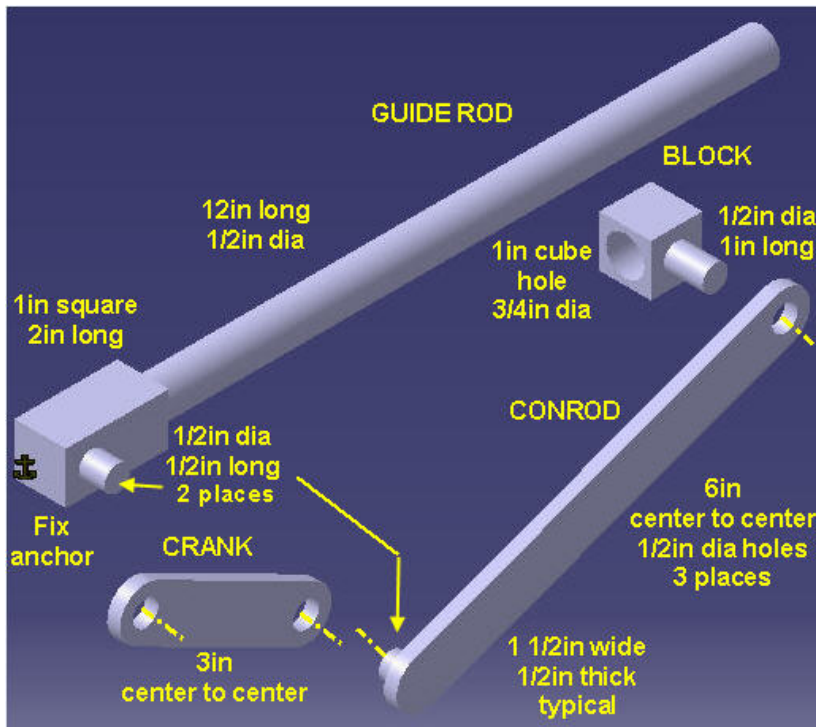
MECHANISM-A solid model created with Catia is pictured above.



The speed and acceleration VS time graph for the block sliding on the guide rod above is not part of this assignment.

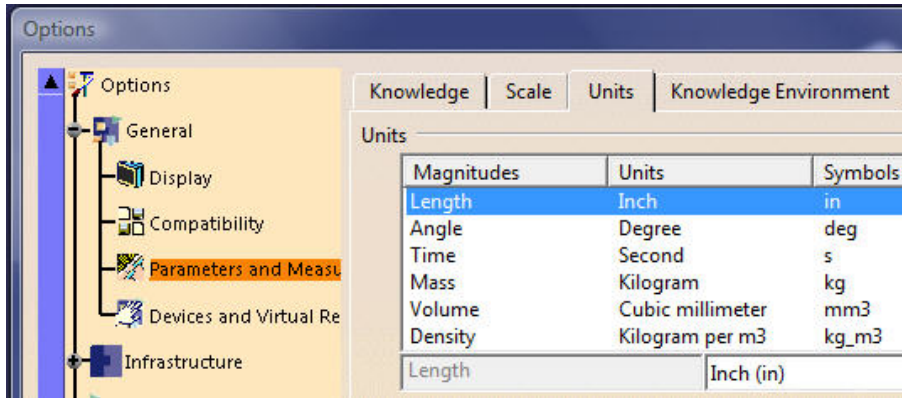


Mechanism velocity and acceleration legend.



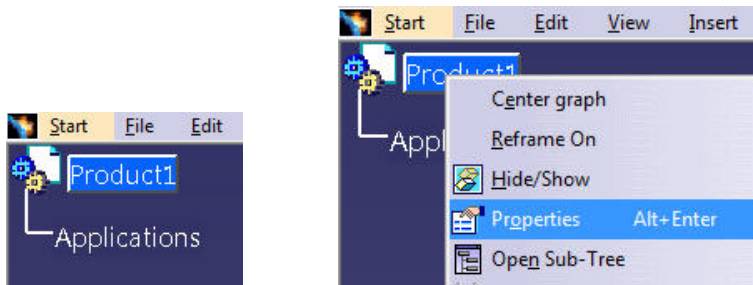
MECHANISM-A part dimensions.

ALL PART FILES IN AN ASSEMBLY MUST BE IN THE SAME FOLDER

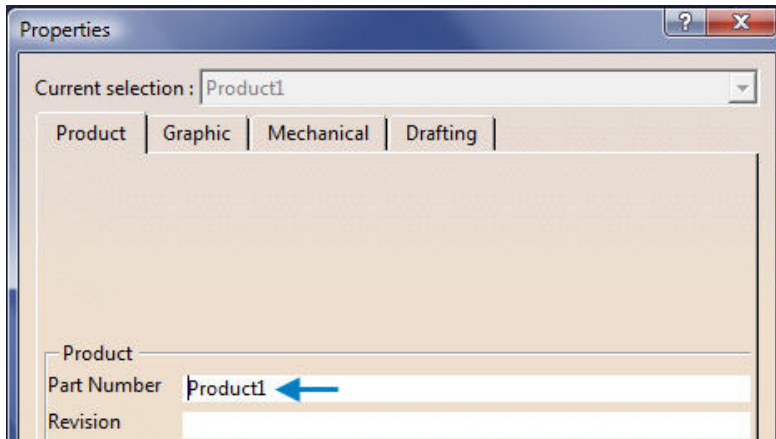


Set length to inches.

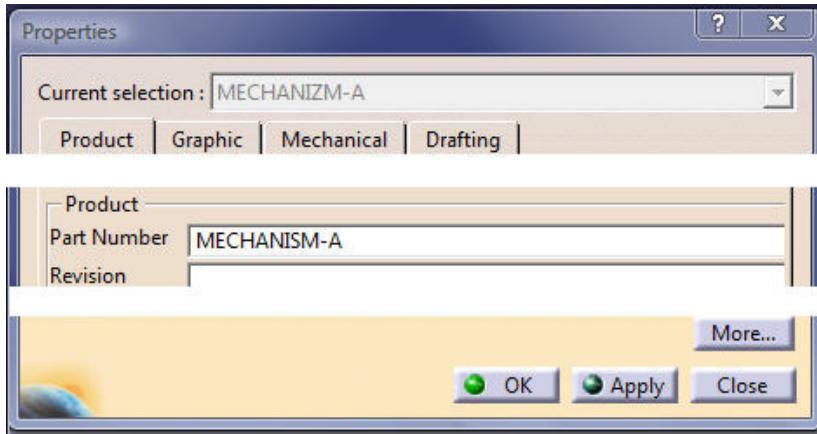
Tools >> Options >> Parameters and Measure >> Length >> Inch (in).



Start >> Assembly Design Right click on "Product1" >> Properties.



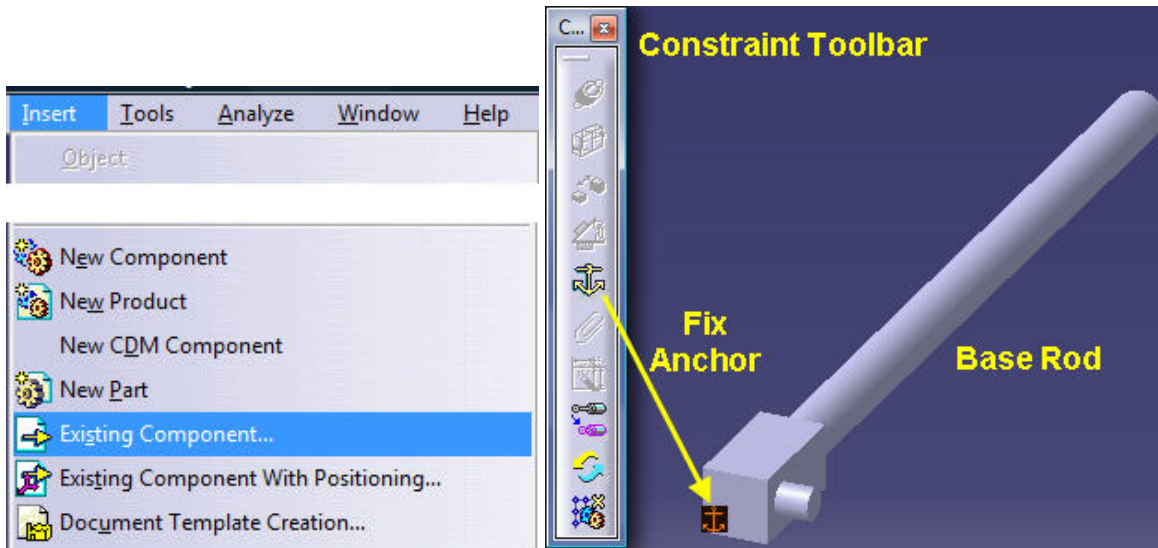
Edit "Product1" to read "MECHANISM-A" as shown below.



Part Number has been changed to “MECHANISM-A”.



Constraints Toolbar

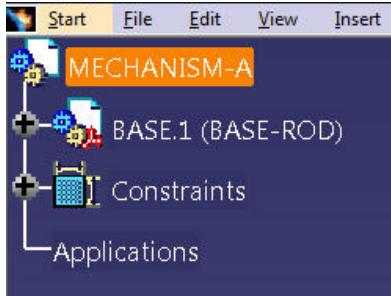


Insert >> Existing Component... >> Click on “MECHANISM-A” below >>

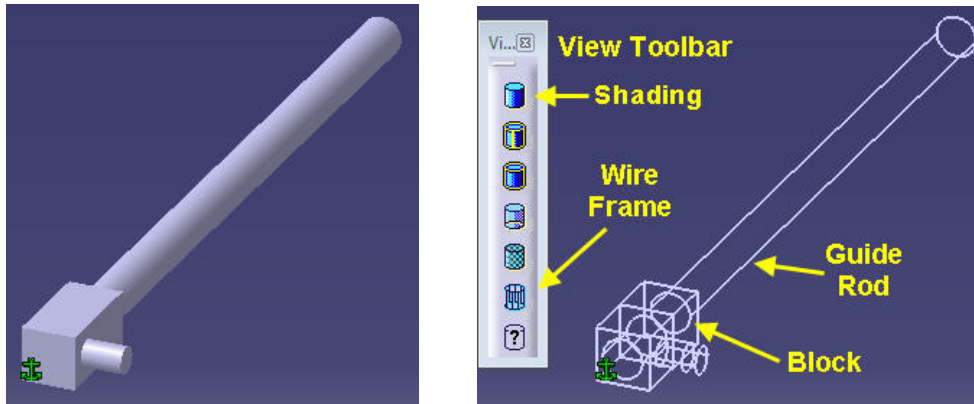
Browse Files >> Select the “BASE ROD” part.

The “BASE.1(BASE-ROD)” part has been added to the tree below.

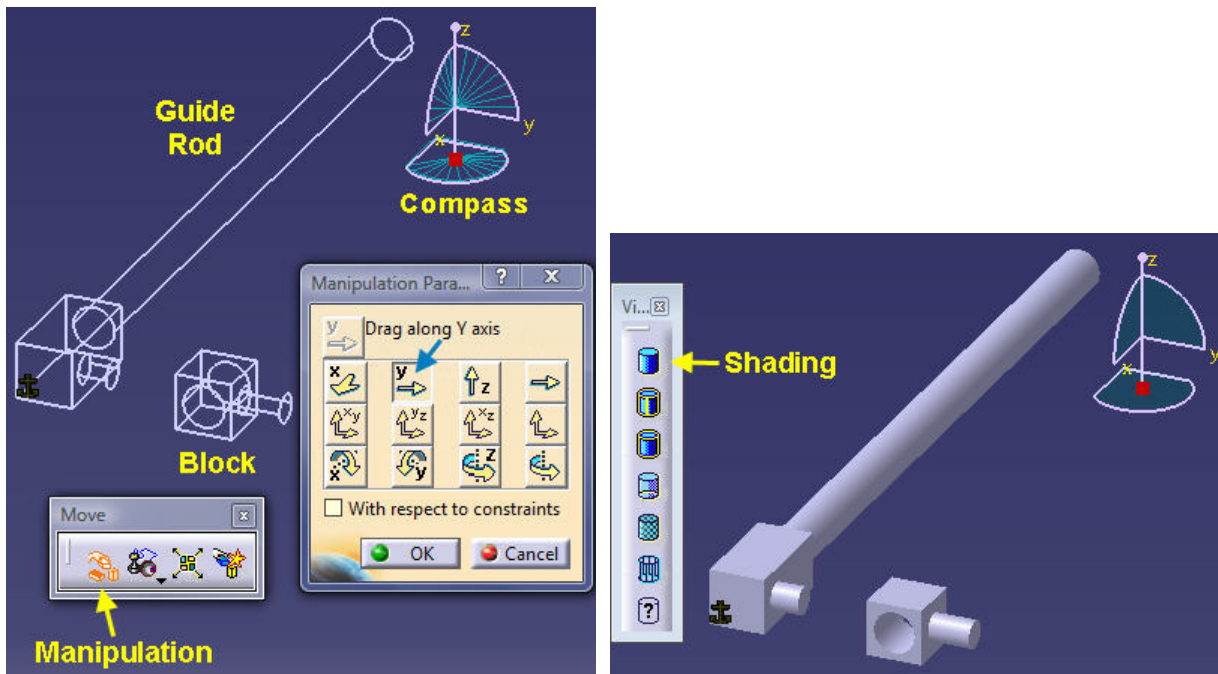
Pick the “Fix” anchor tool and place it on the Base rod.



Insert >> Existing Component... >> Click on "MECHANISM-A" above >>
Browse Files >> Select "BLOCK" part.

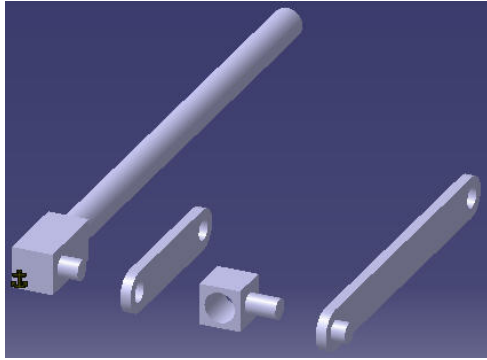


The "Block" part is inside the Guide Rod. On the View toolbar pick "Wire Frame" to see the "Block".

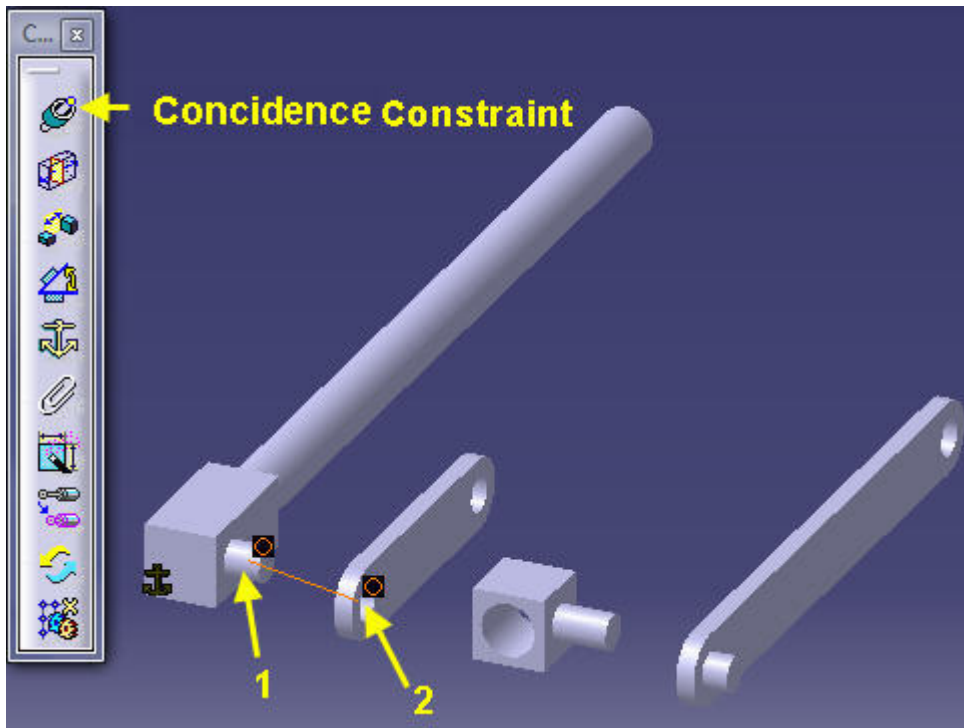


Pick the "Manipulation" tool on the "Move" toolbar.
Pick the "Y" direction arrow on the " Manipulation box".
Pick the Guide Rod and drag it away from the Block as shown above.

Insert >> Existing Component... >>
Click on "MECHANISM-A" above >>
Browse Files >> Select "CRANK" part.

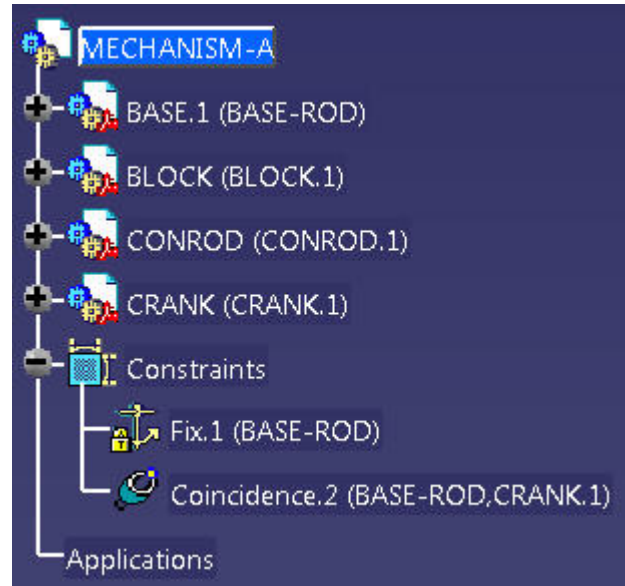
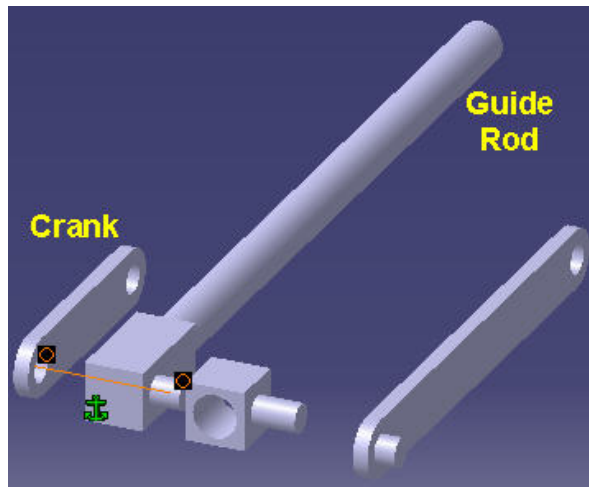


All of the Catia parts were created at the same origin.



On the "Constraint" toolbar pick the "Coincidence Constraint" tool.

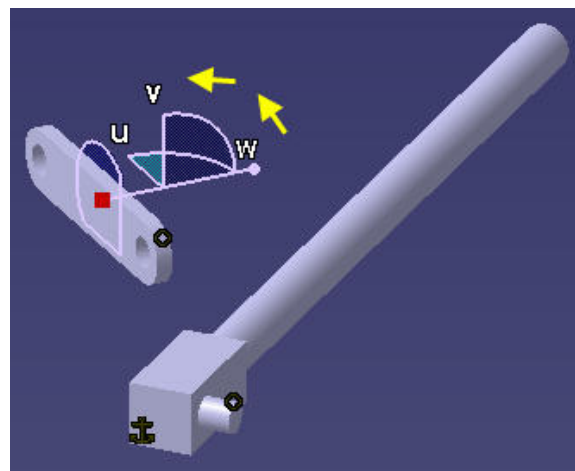
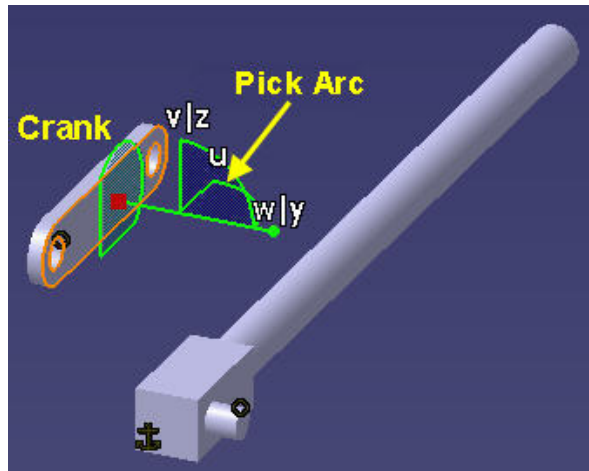
Pick-1 on Guide-Rod pivot pin surface >> Pick-2 in Crank hole as above.



Guide-Rod pivot pin and Crank hole axes are coincident.

See "Constraints" added to the Tree above.

View >> Compass turns the Compass on or off.



Pick the red ball in the Compass and drag it to the Crank as above left.

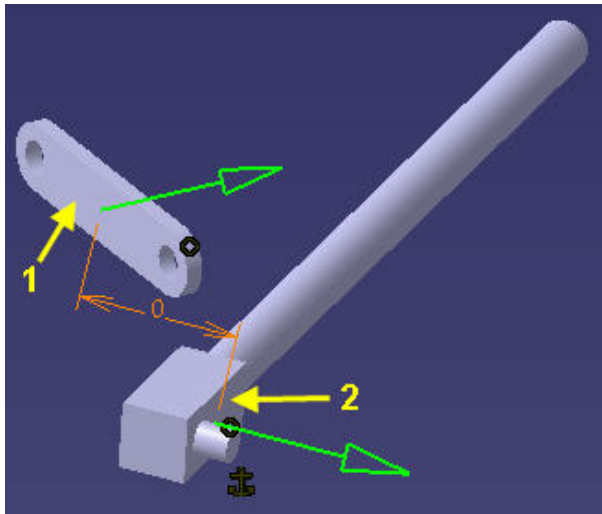
Pick Arc and release mouse button.

Hold Ctrl key and rotate the Crank as shown above right.



Pick the red ball in the Compass and drag it to the xyz Axes in the bottom right corner of the display.

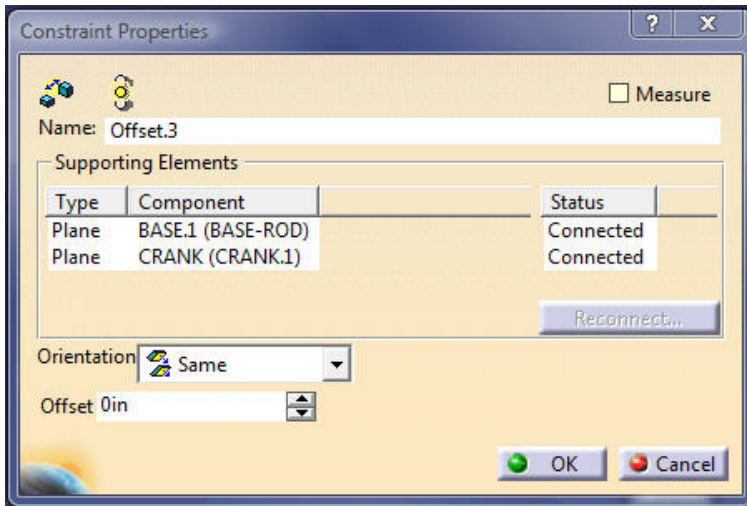
Release the mouse button and the Compass will return automatically to the top right corner.



Pick the "Offset Constraint" tool.
Pick-1 >> Pick-2 part flat surfaces.

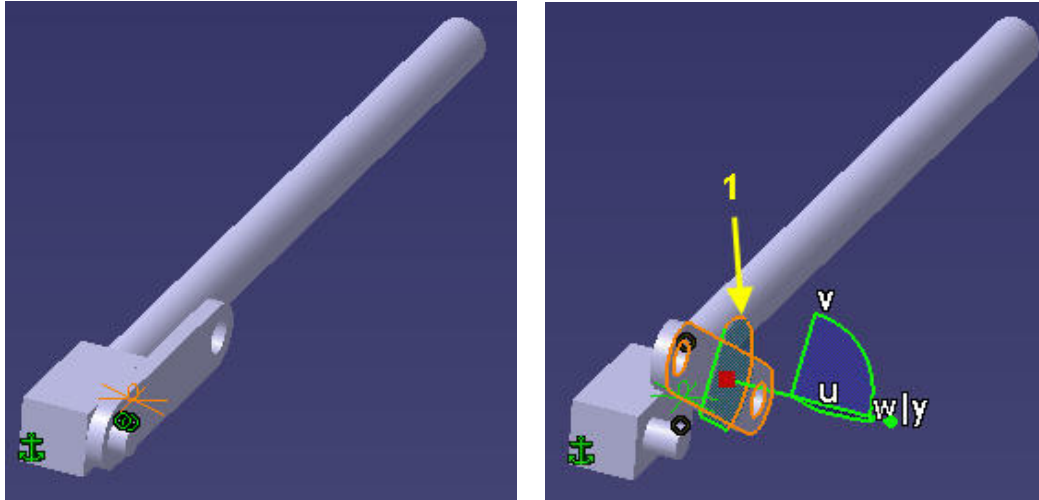


Click one arrow to reverse direction if arrows do not point as above.

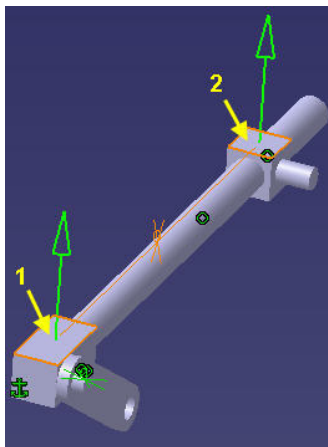


Edit Offset >> 0in as shown above.

Press keys: Ctrl + U to update the constraints.



The "Offset" constraint has been updated. Pick-1 arc >> Rotate the Crank.
Drag the Compass red ball to the Crank as above. Ctrl + U to update.



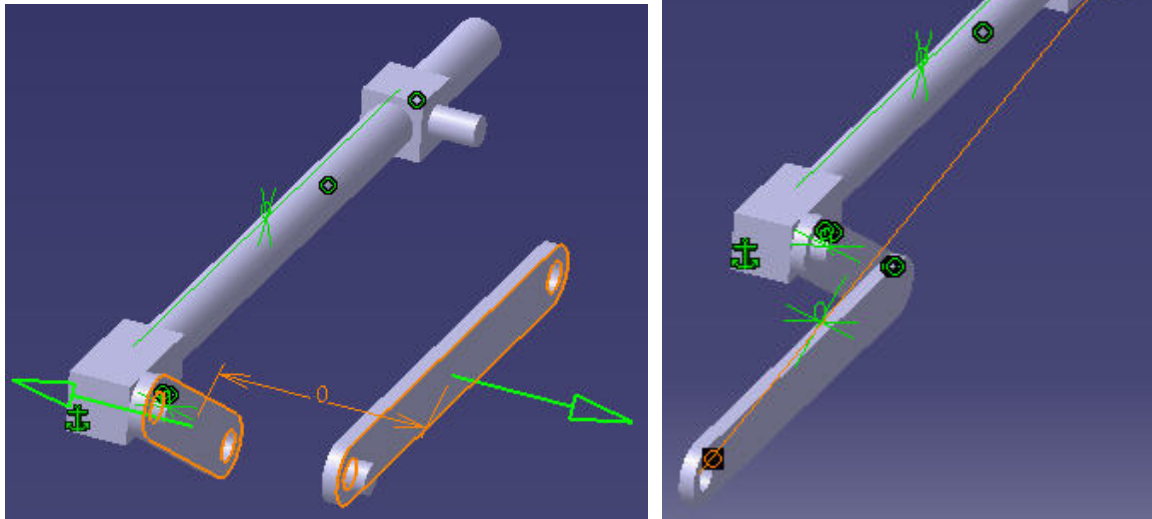
Pick the "Offset Constraint" tool.

Pick-1 >> Pick-2 part flat surfaces.

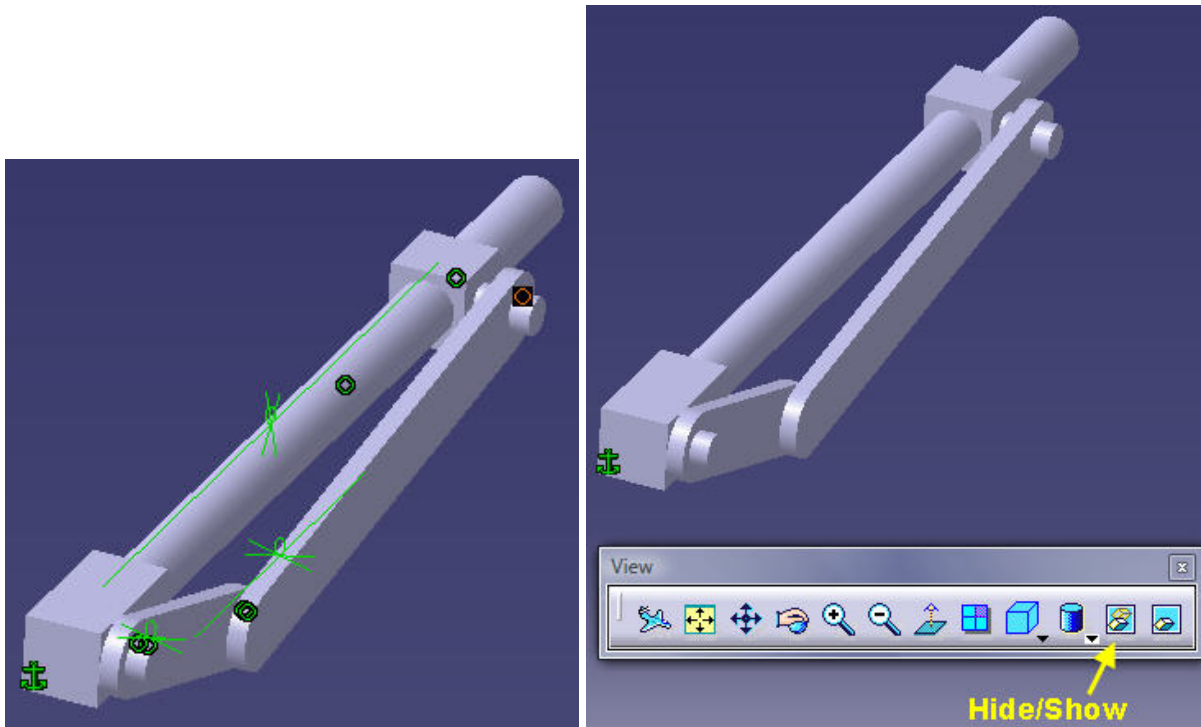
On the "Constraint" toolbar pick the "Coincidence Constraint" tool.

Pick on the Guide-Rod cylindrical surface >> Pick in the Block hole.

Ctrl + U to update.



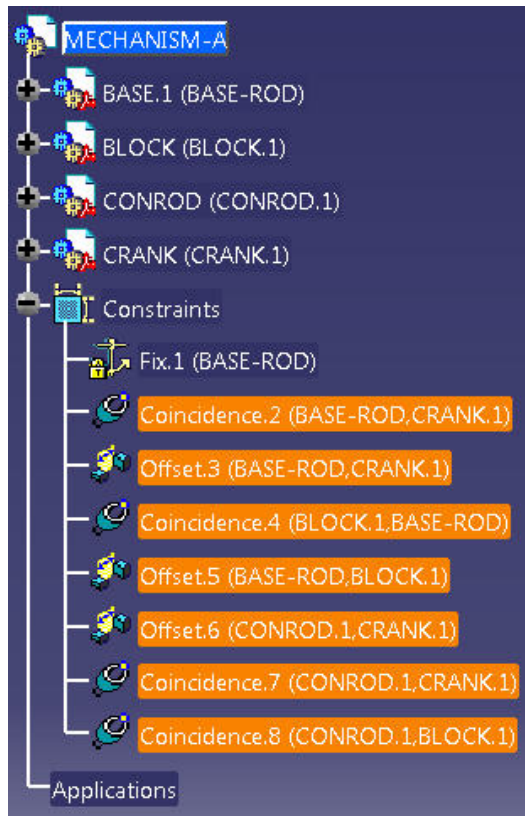
Use the “Coincidence Constraint” tool and the “Offset Constraint” tool to constrain the Crank and Conrod as shown above.



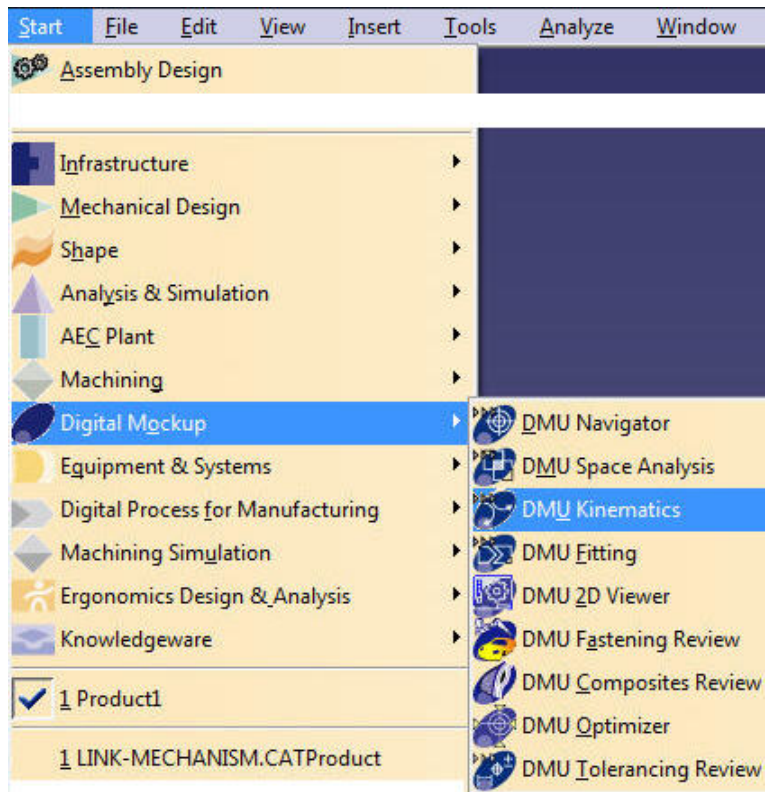
In the tree below pick the top constraint, Shift Key, and bottom constraint.

All constraint are selected this way.

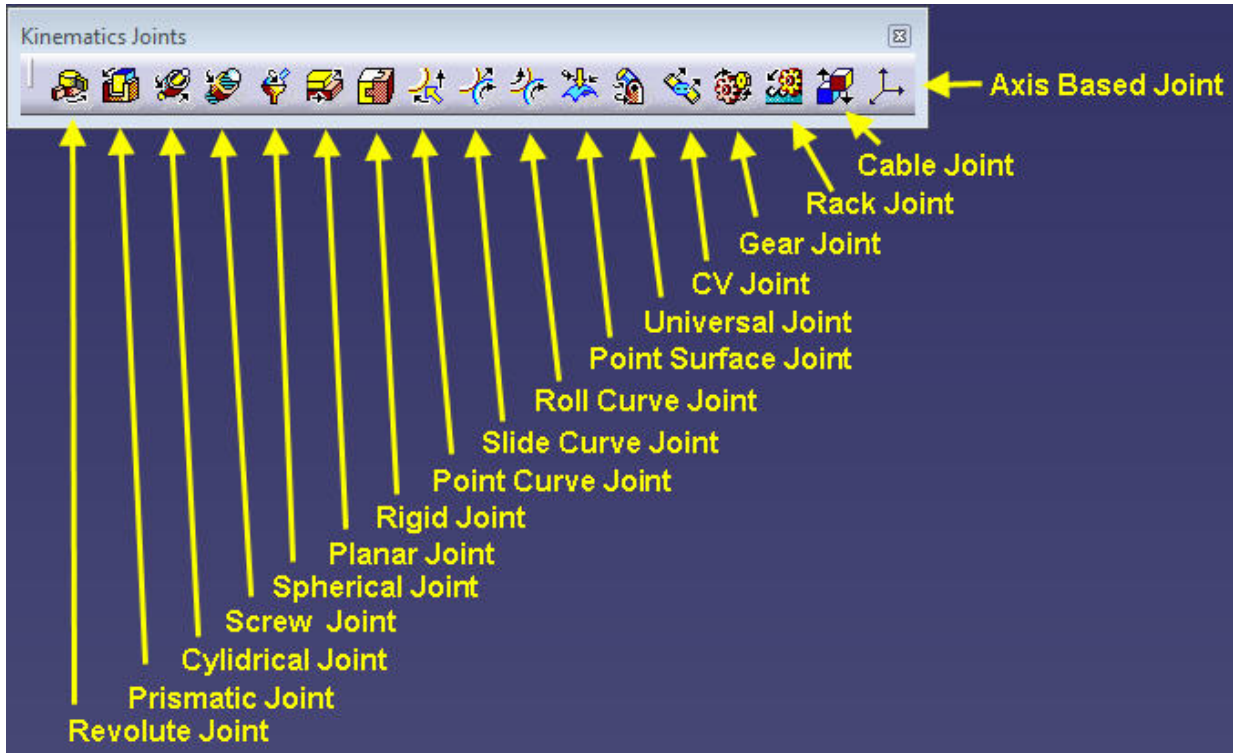
Pick “Hide/Show” to hide all constraints.



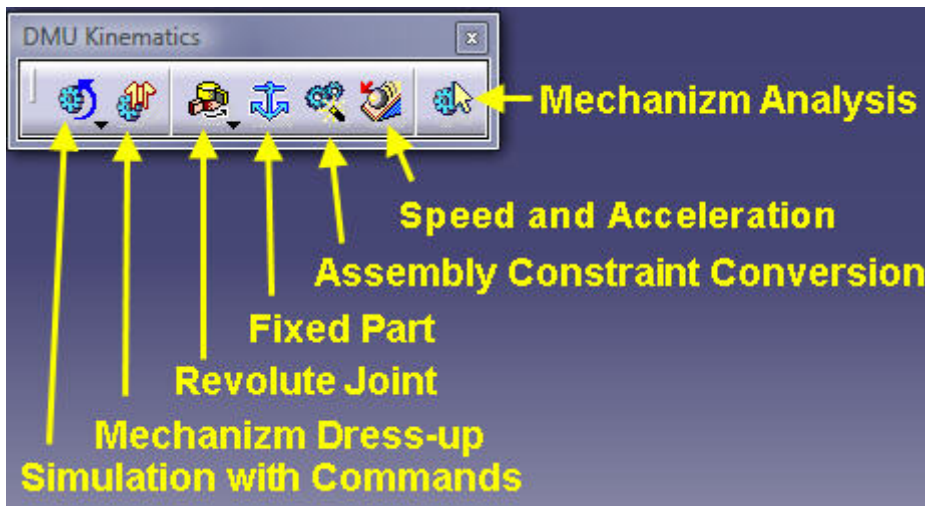
MECHANISM DYNAMIC ANALYSIS



Start >> Digital Mockup >> DMU Kinematics



“Revolute Joint” drop-down menu above on the “Kinematics Joints” toolbar shown below.



Pick the: “Assembly Constraint Conversion”.

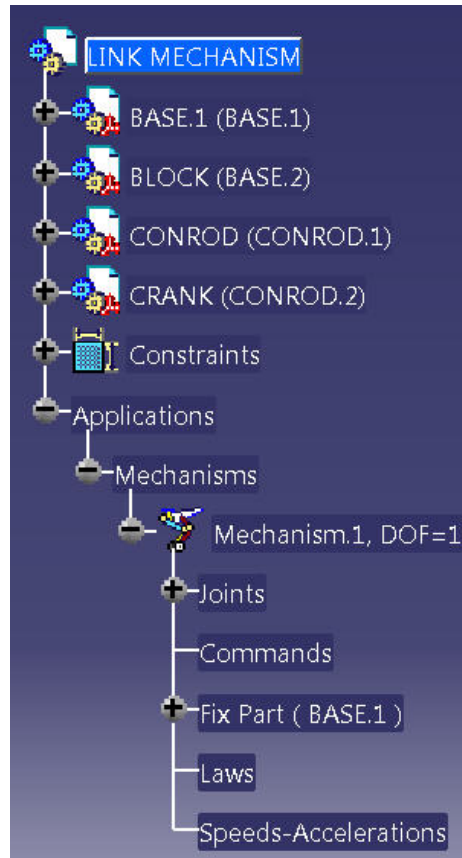
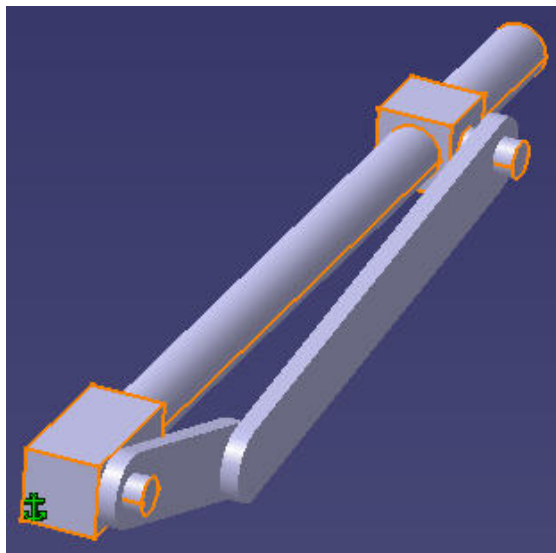


Pick “New Mechanism”

Select “Auto Create”

Make sure “Unresolved pairs” are 0 / 4

If not: close Catia, re-open the above assembly, and check all restraints.

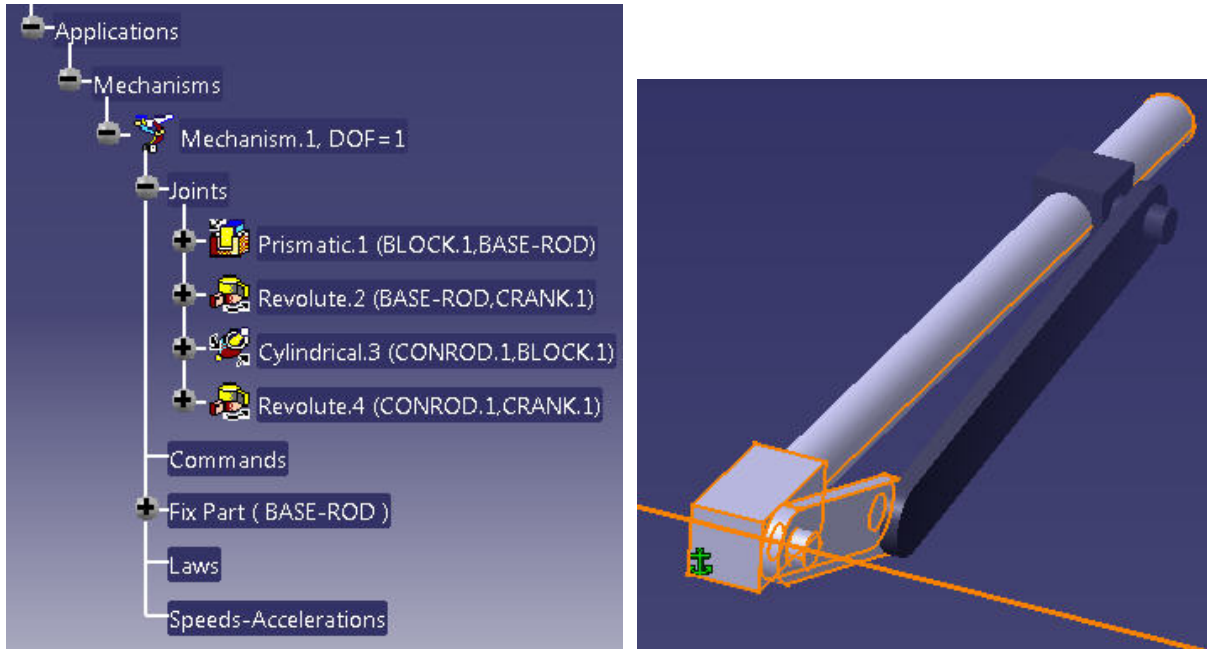


The highlight by Catia indicates the mechanism is operational.

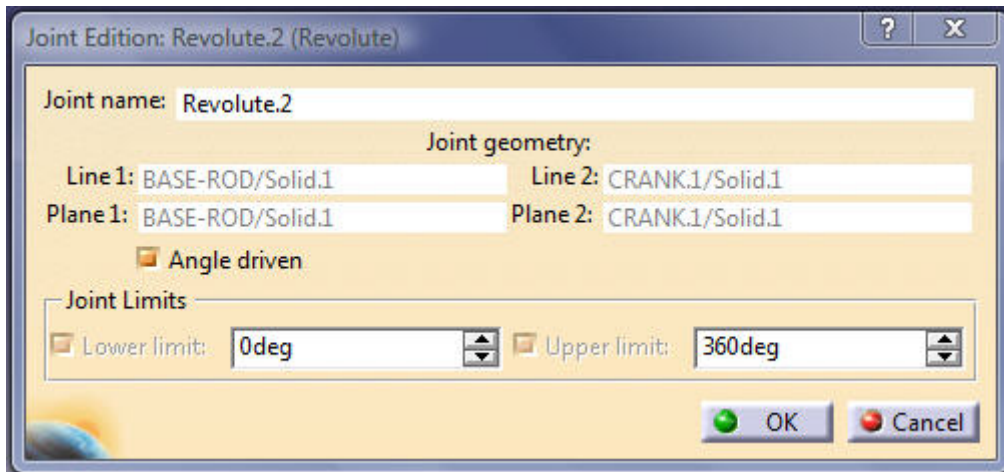
Degrees of Freedom in the tree equal 1 (DOF=1).

The one degree of freedom must be removed to convert to a mechanism.

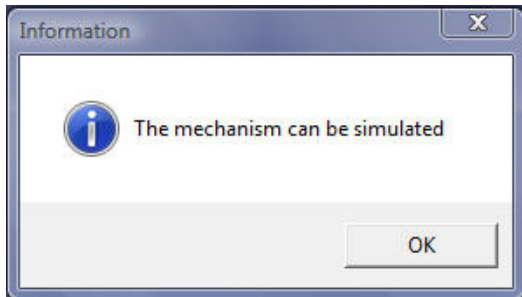
The Joint (BASE-ROD,CRANK.1) will be changed to “Angle Driven”.



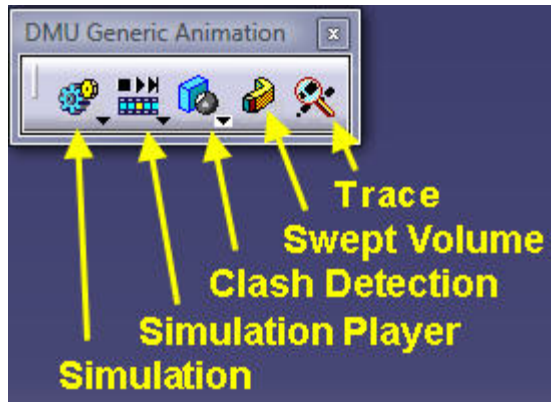
Double click on “Revolute.2(BASE-ROD,CRANK.1)
The “CRANK” has been converted into a “Revolute”.



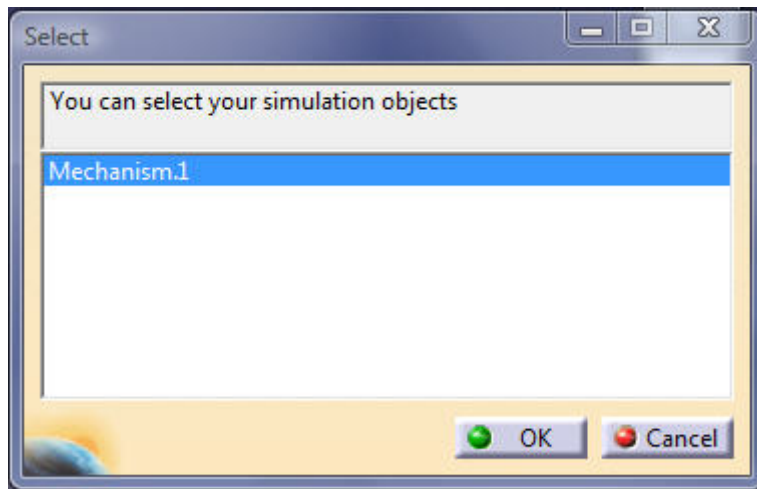
Pick “Angle Driven”.
Edit “Joint Limits” to 0deg and 360deg as shown above.



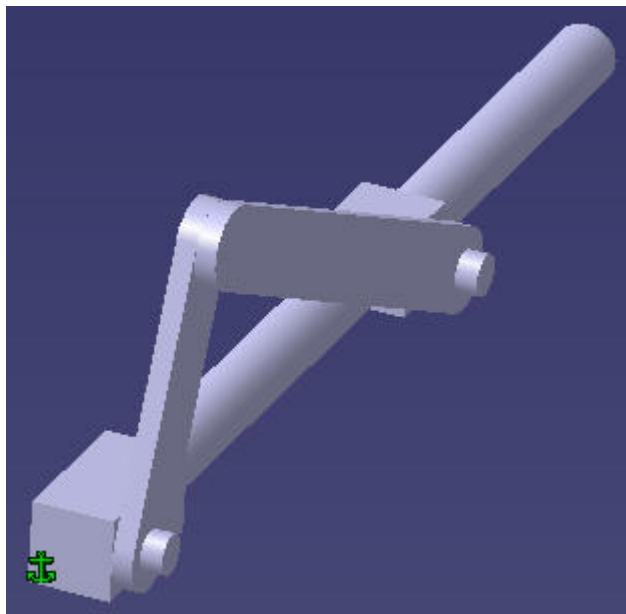
If the Information box above opens the mechanism will work.
If not: close Catia, re-open the above assembly, and check all restraints.



Select "Simulation" on the Digital-Mock-Up (DMU) Generic Animation toolbar.

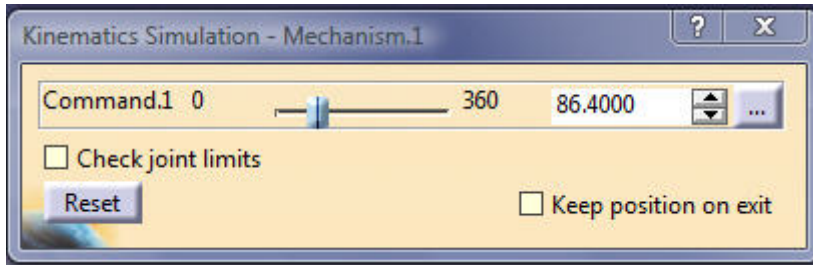


Pick, "Mechanism.1".

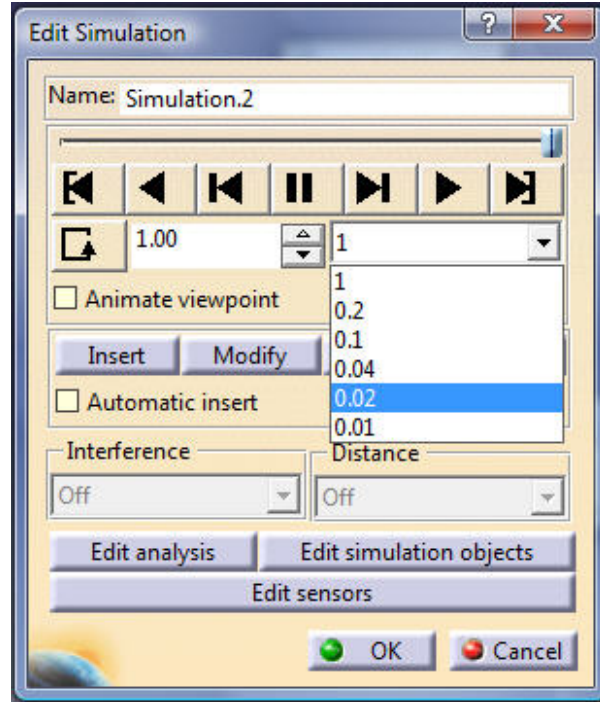
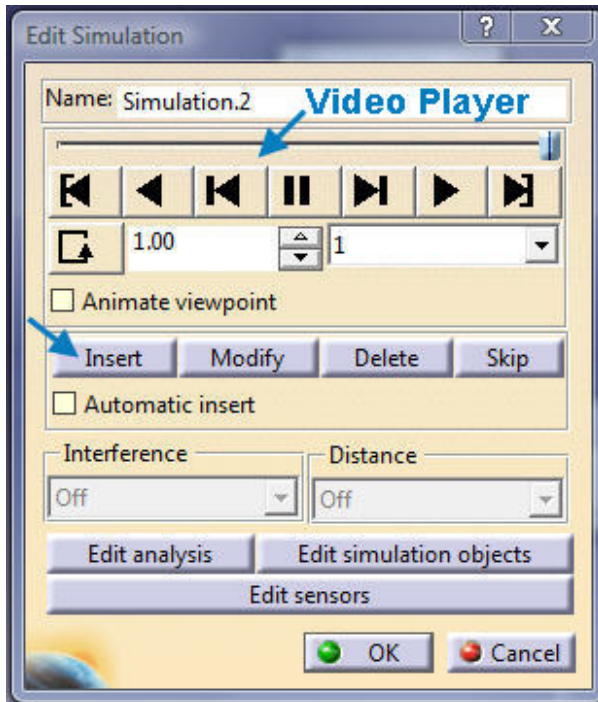


"Mechanism.1" is shown above.

The "Kinematics Simulation – Mechanism" box below will open next.



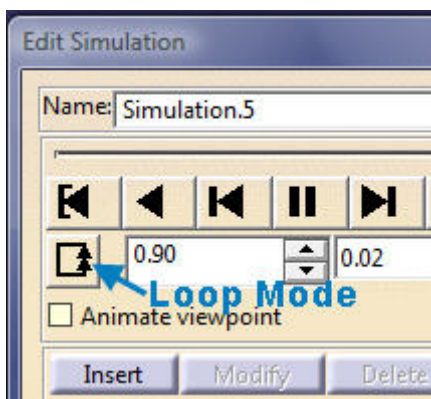
Slide the motion button above from left to right to rotate the Crank 360 degrees.



Pick "Insert" to activate the Video player buttons as above left.

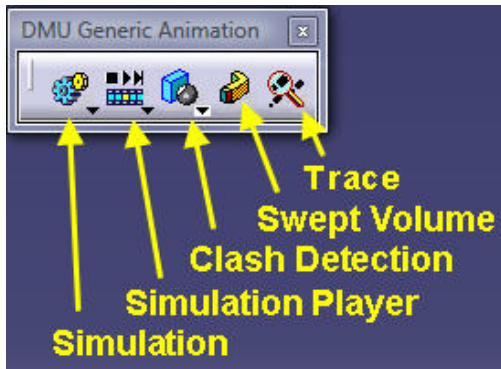
Change the interpolation step from 1 to 0.02 seconds as shown above right.

Pick the "Play" arrow and view the mechanism animation.



Double-click the "Loop Mode" button above for continuous motion in one direction.

Single-click the "Loop Mode" button for oscillating motion.



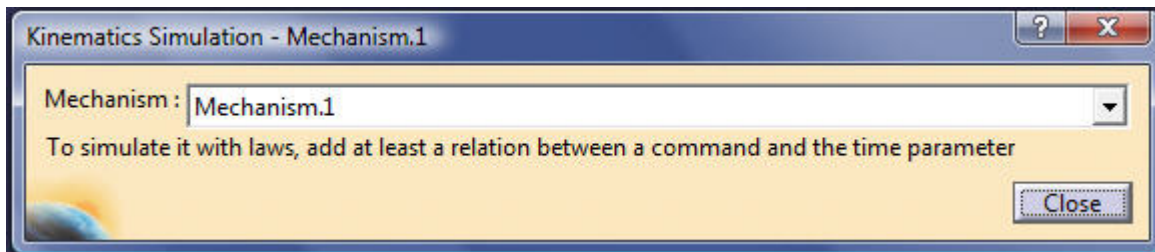
Select "Simulation Player" on the DMU Generic Animation toolbar.

SIMULATING MECHANISM DYNAMIC CONDITIONS

Time based physical laws.



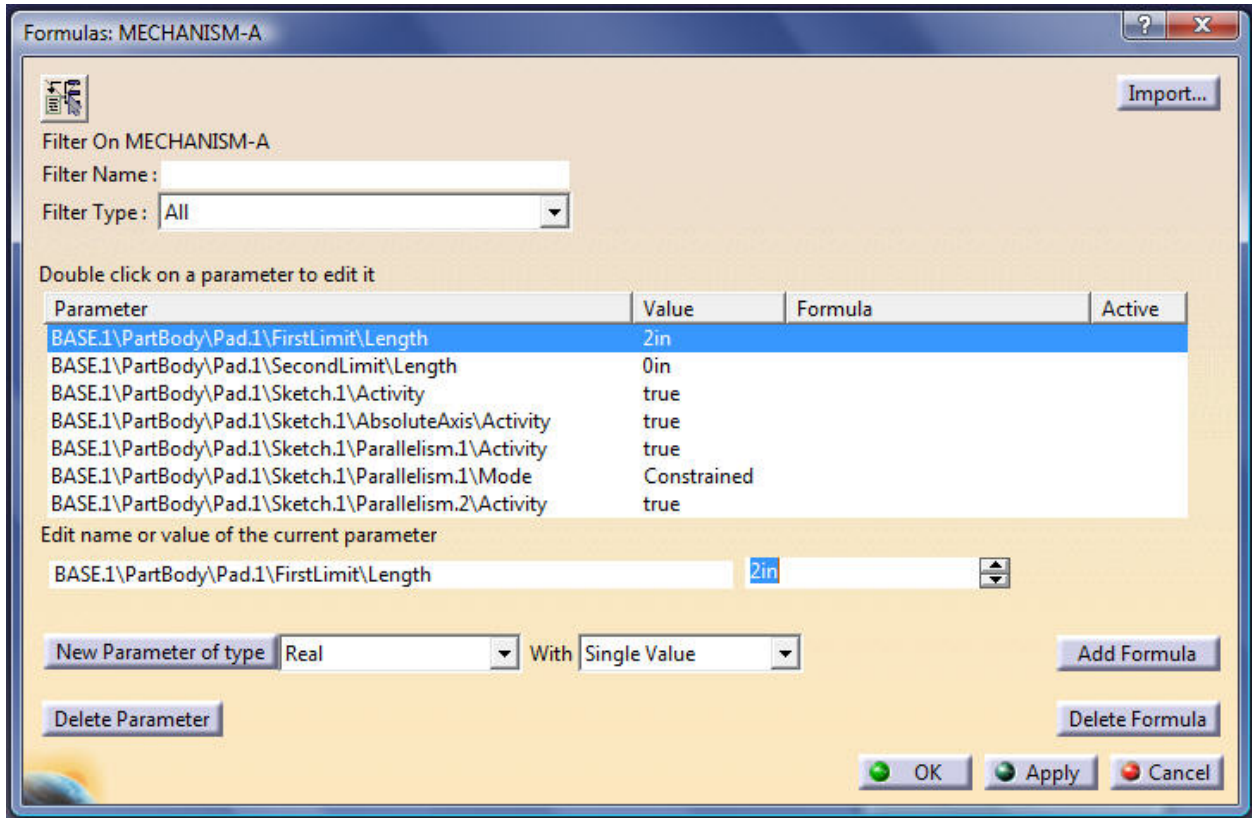
Pick the: "Simulation with Laws" tool.



The information box above will open indicating that a relation (or formula) is needed between a parameter (Crank angle) and time (seconds).

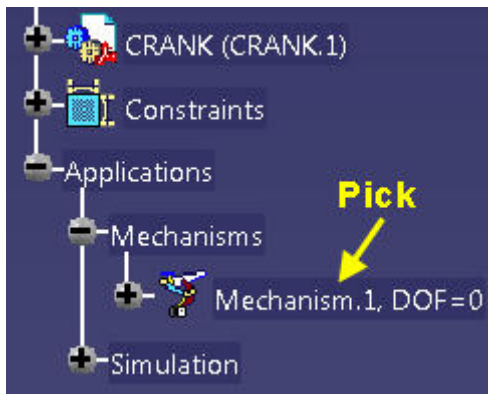


Pick the "Formula" tool in the "Knowledge" toolbar.



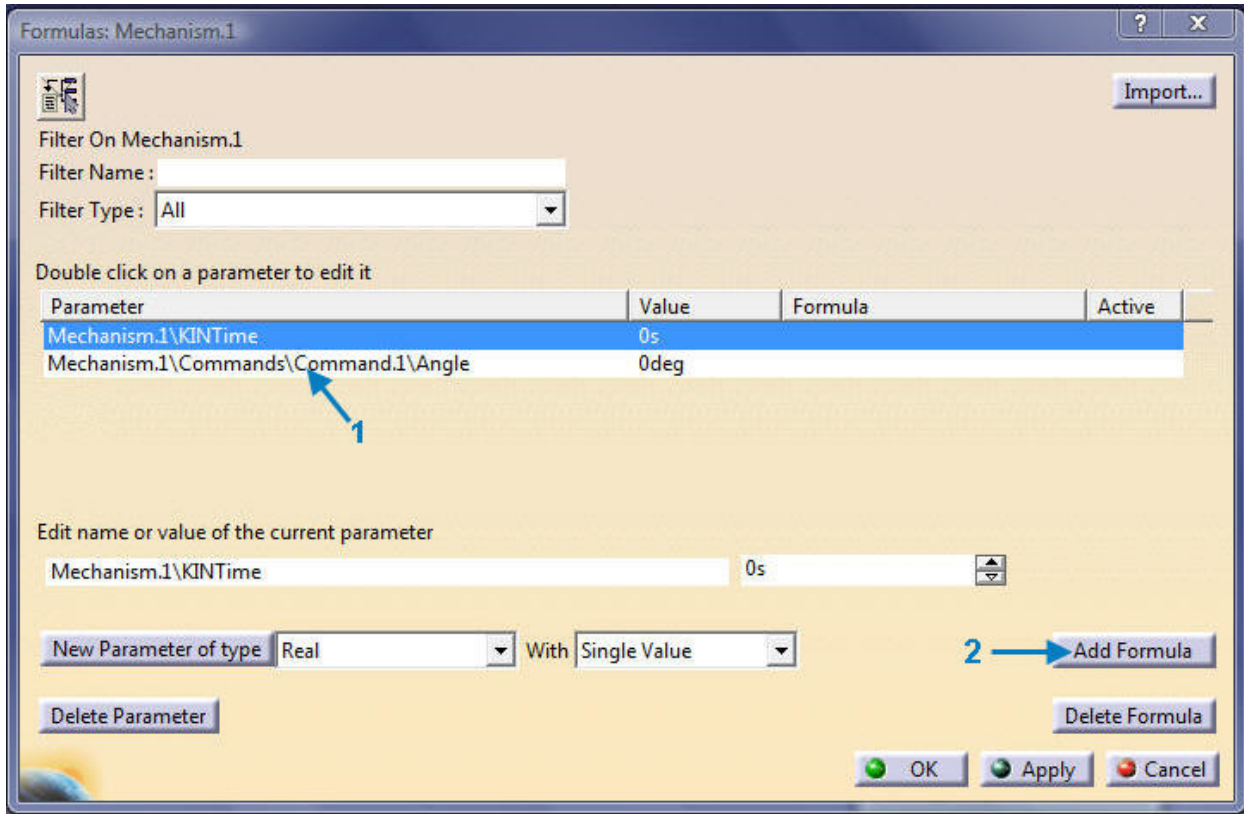
The “Formulas: MECHANISM-A” box will open.

The objective is to plot the Crank Mechanism: Position, Velocity, and Acceleration VS Time.

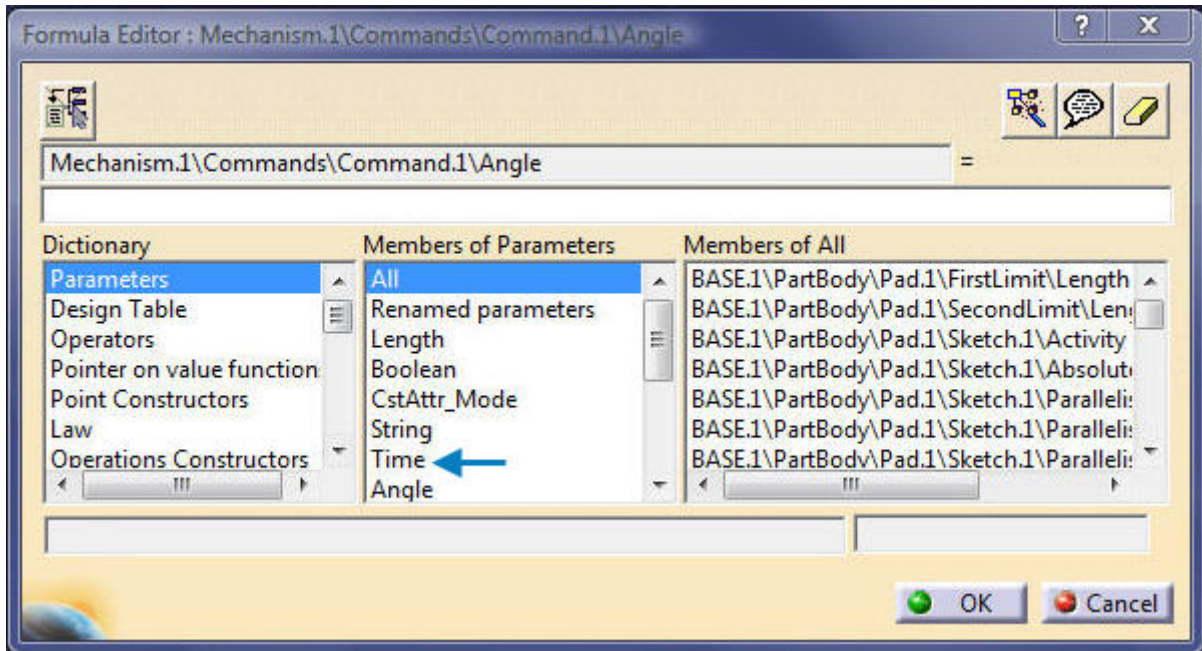


Pick “Mechanism.1,DOF=0” in the tree.

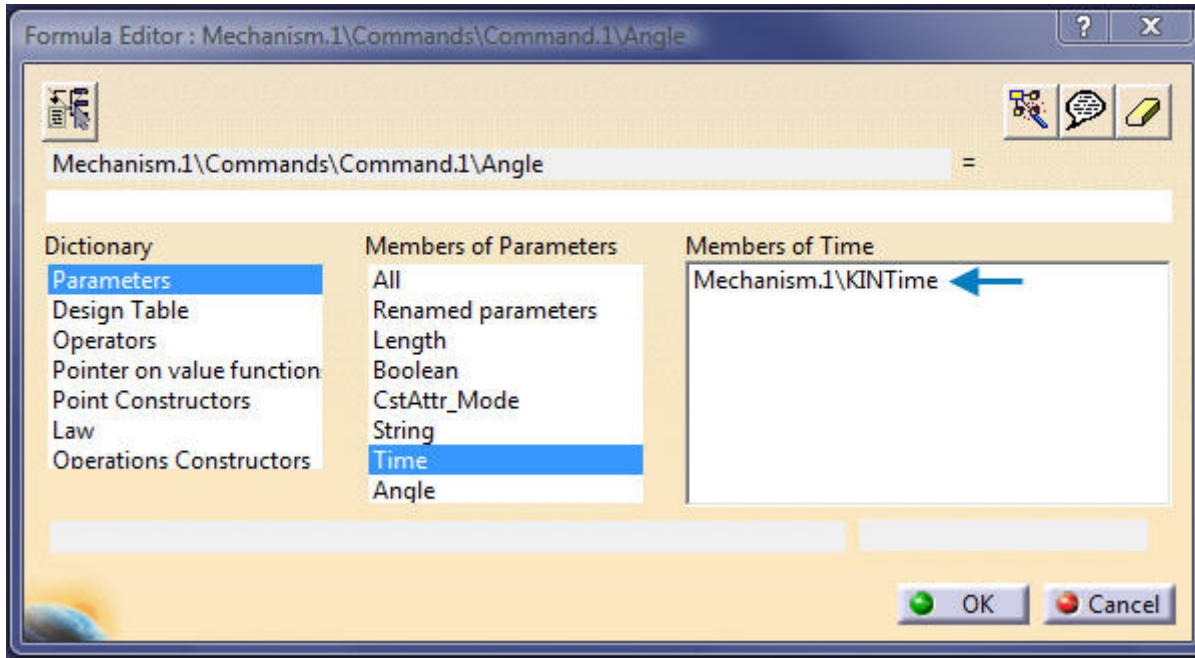
Only formulas associated with the Mechanism will now be displayed in the Formulas box.



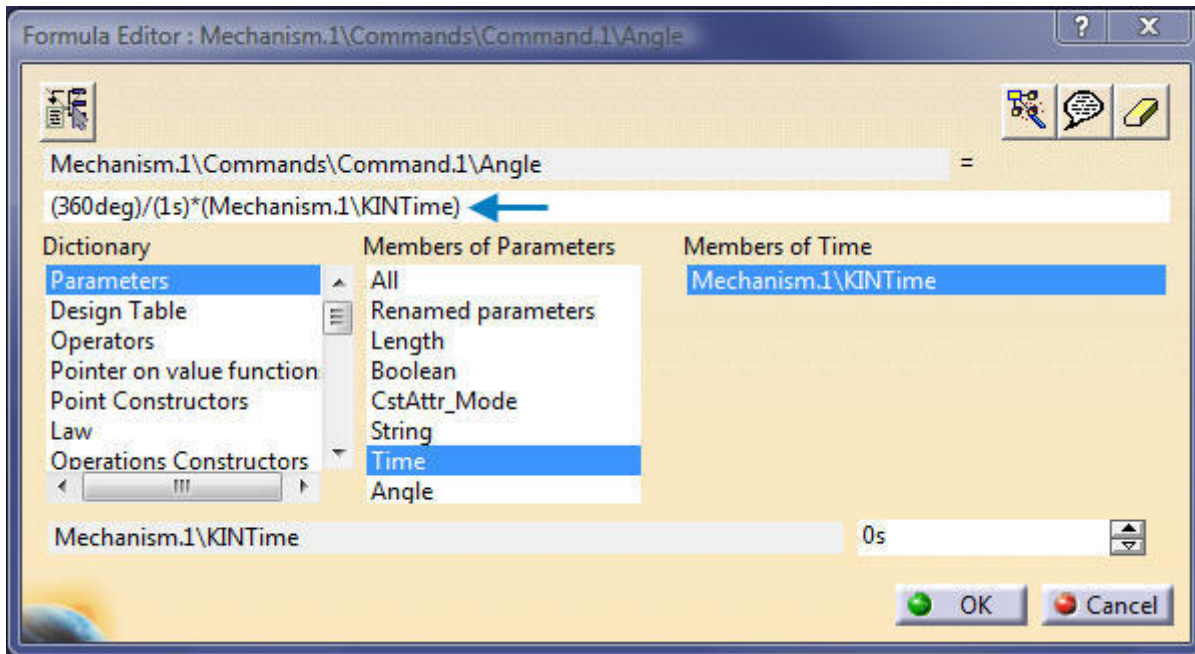
Pick-1 >> Pick-2 >> OK.



Pick: "Time".



With “Parameters” and “Time” selected double click: “Mechanism.1\KINTime”



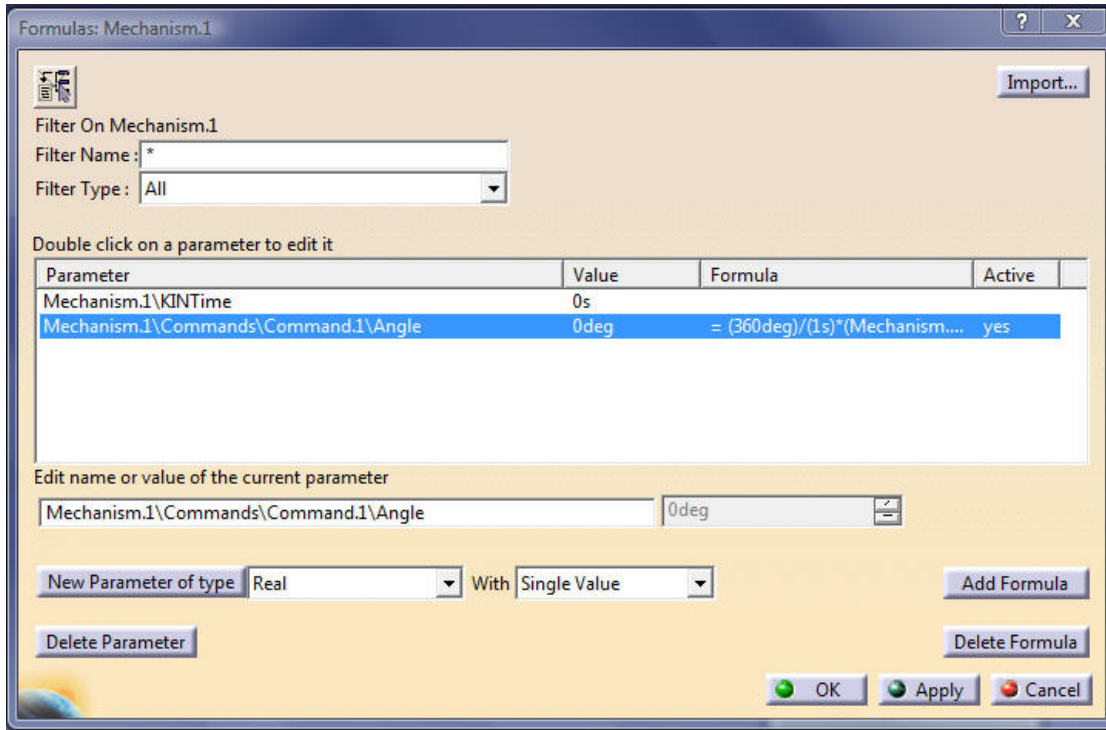
The Crank angular velocity is: $(360\text{deg})/(1\text{s})$

The time is: $(\text{Mechanism.1}\backslash\text{KINTime})$

Therefore the Crank Angle = Crank angular velocity x Time

Enter the right side of the equation: $(360\text{deg})/(1\text{s})*(\text{Mechanism.1}\backslash\text{KINTime})$.

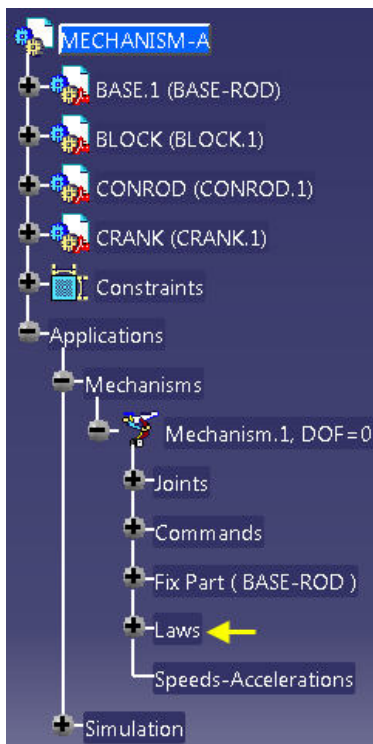
Pick: OK and the box below will open.



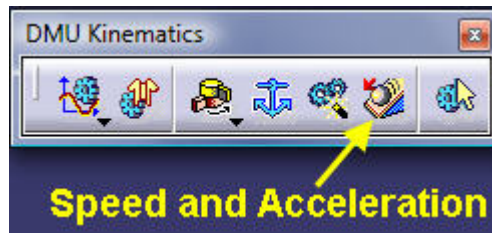
Check the units of distance.

The International system of Units (ISU) is the default.

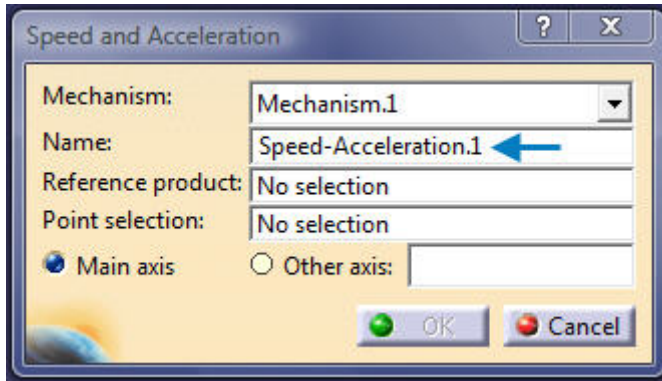
If the “Warning” box opens with the words: “Units are not homogeneous...” then change dimensions with: Tools >> Options >> Parameters and Measure >> Length >> Inch (in).



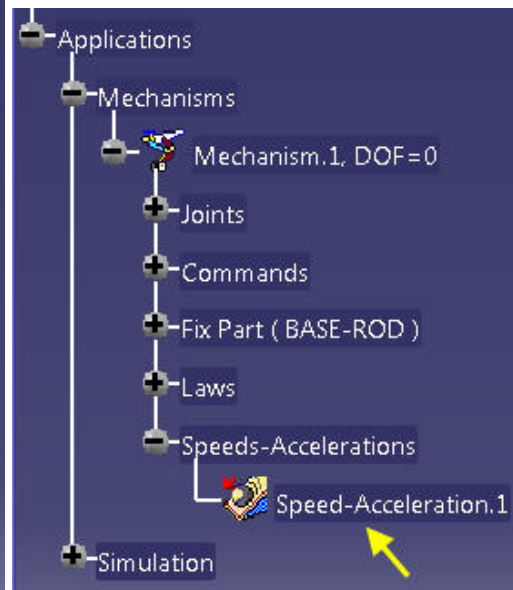
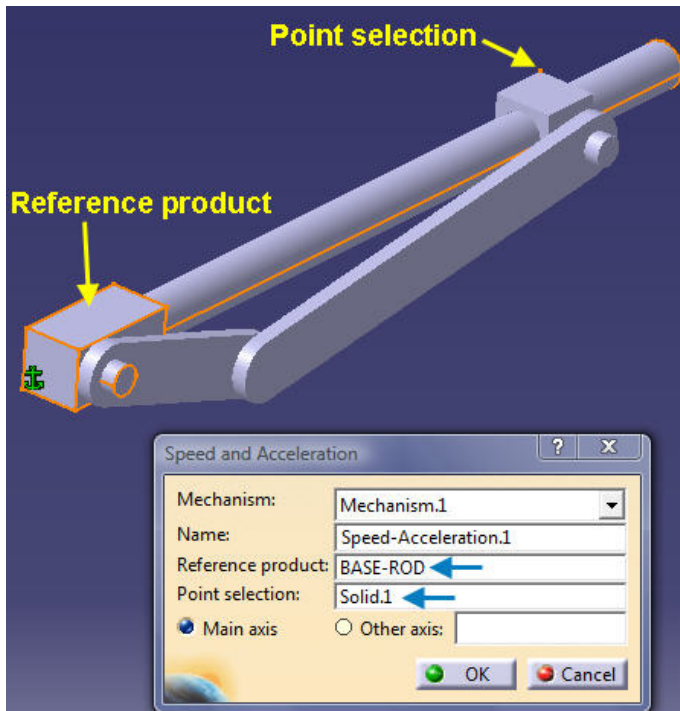
The “Law” branch has been added.



Pick: “Speed and Acceleration” on the DMU toolbar.



The "Speed and Acceleration" above box opens next.



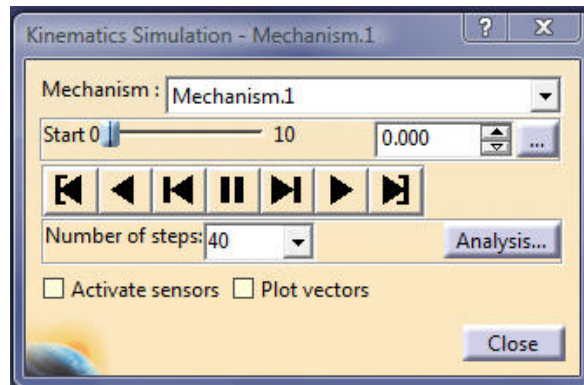
For "Reference Product" pick the BASE-ROD.

Note: Speef-Acceleration.1 is added.

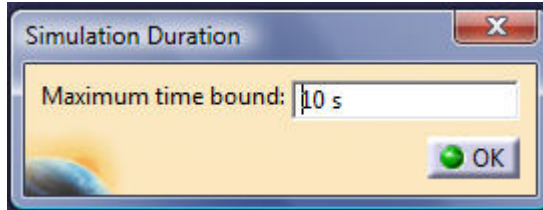
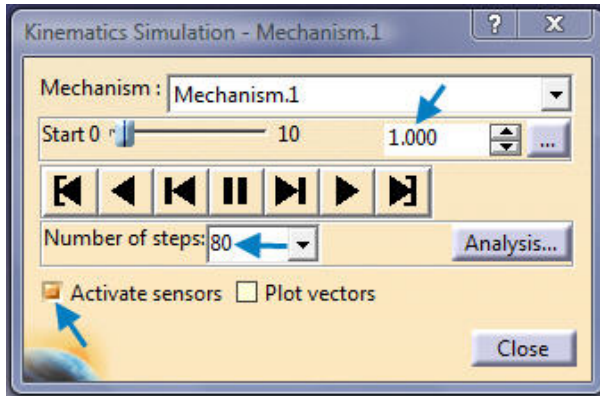
For "Point Selection" pick a point on the BLOCK. Catia changes the name from BLOCK to Solid.1.



Pick "Simulation with Laws".

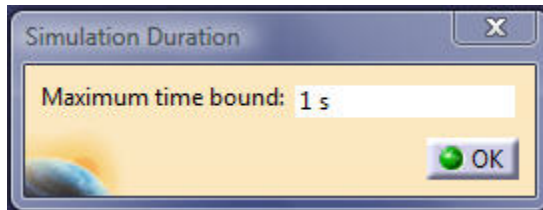
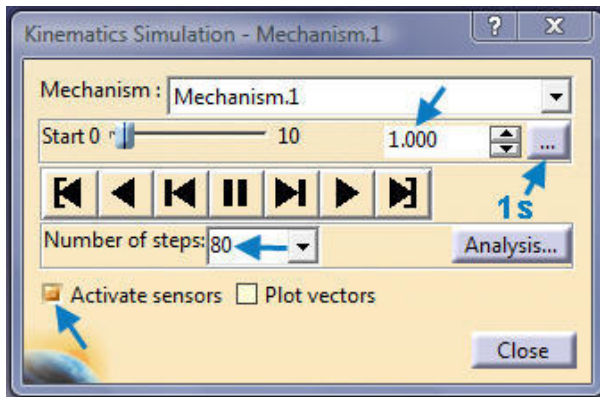


Default time is 10 seconds and steps are 40.



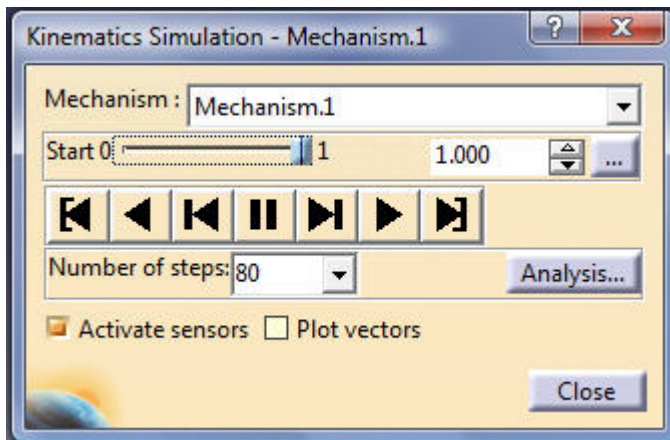
Pick “Activate sensors” and change steps to 80. Time is 10 seconds.

The “Sensor” box on page below will open.

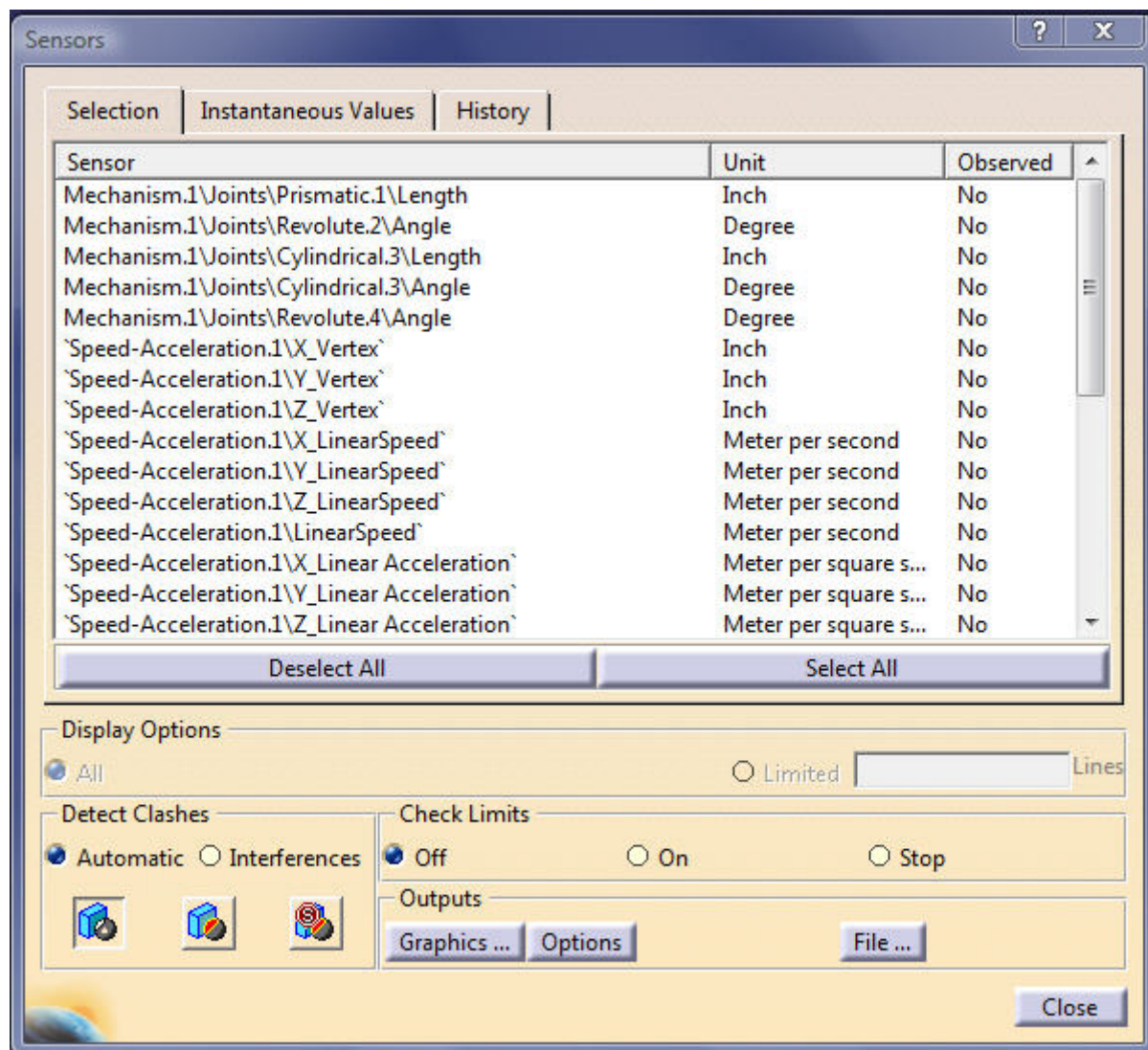


Pick the Browse button with 3 dots.

Change the time to 1 second.



The steps have been changed to 80 and time to 1.0 second.

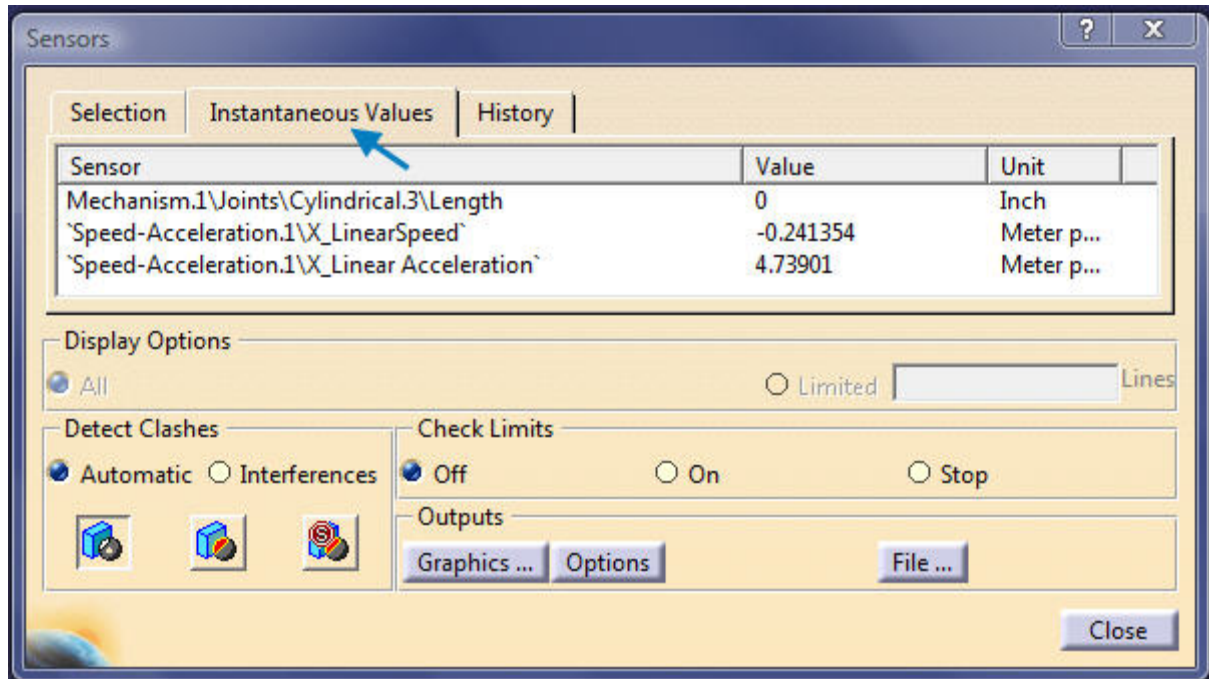


Pick:

Mechanism.1\Joints\Cylindrical.1\Length

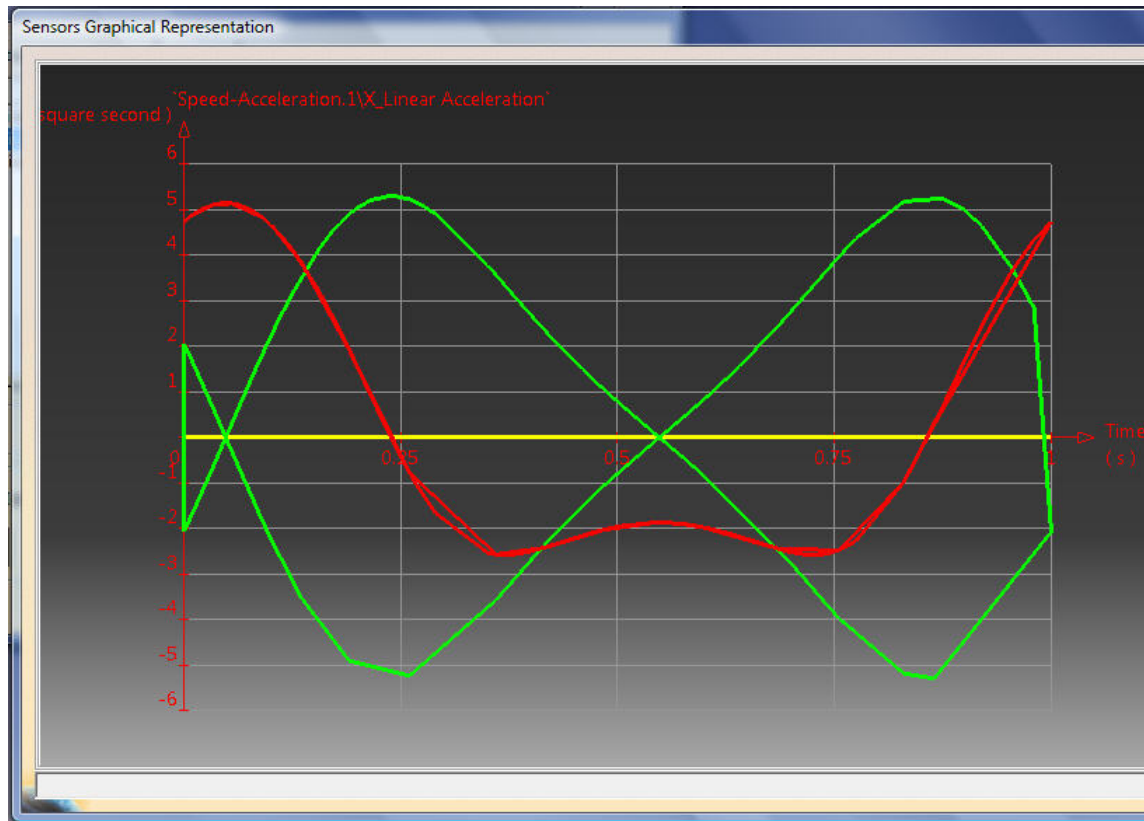
Speed-Acceleration.1\X_LinearSpeed

Speed-Acceleration.1\X_LinearAcceleration

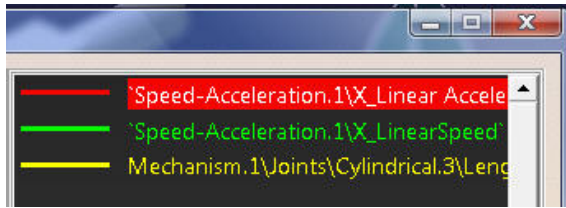


Pick the “Instantaneous Values” tab in the “Sensors” box.

DO NOT CLOSE THE SENSORS BOX. It will generate the plots below.



Speed and acceleration VS time graph for the block sliding on the rod will be plotted with Catia as shown above.



Legend.

END OF COURSE CONTENT