PDHonline Course M418 (6 PDH)

SolidWorks CAD Basics and Stress Analysis

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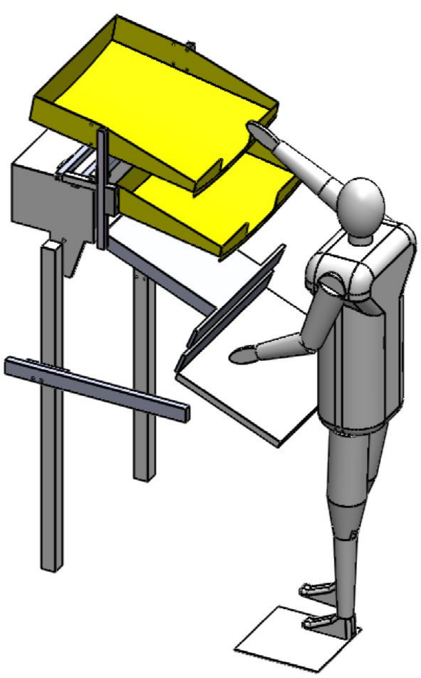
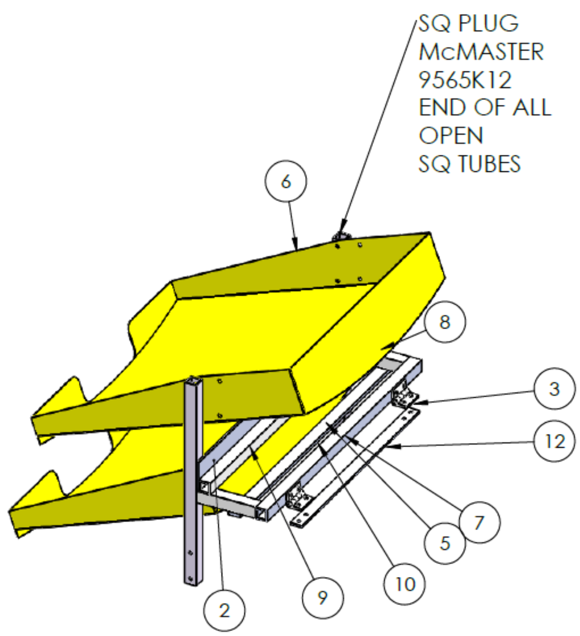
CONTENTS

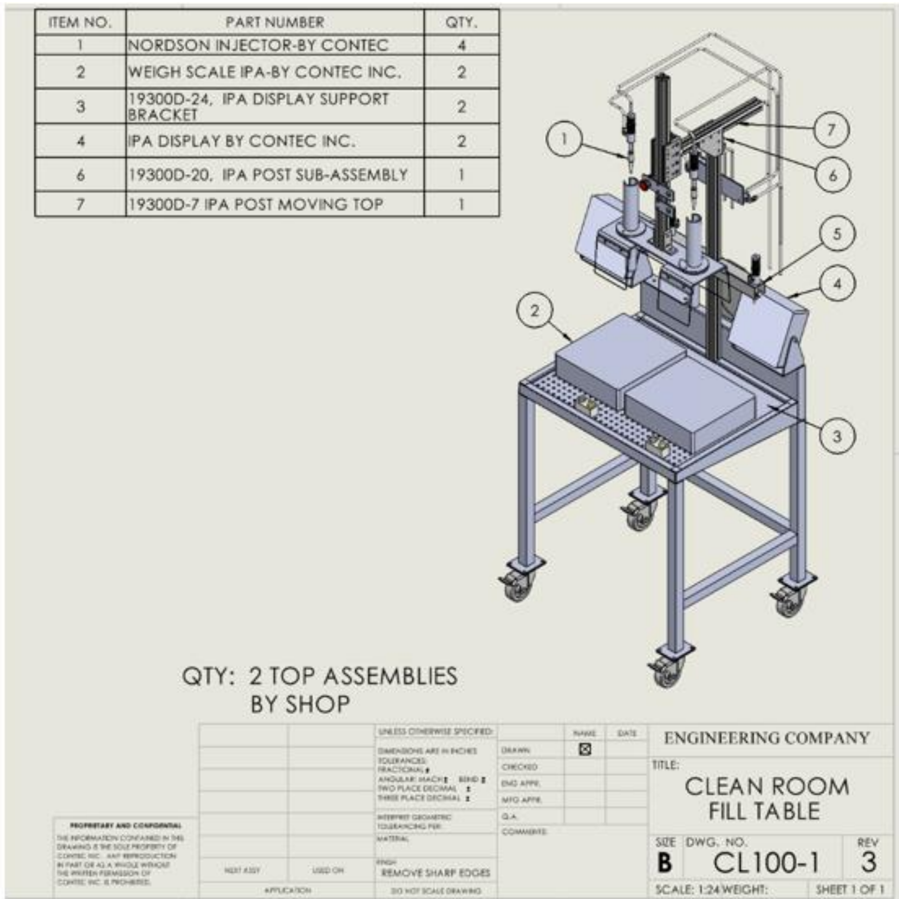
SolidWorks Parts, Assemblies, and Drawings



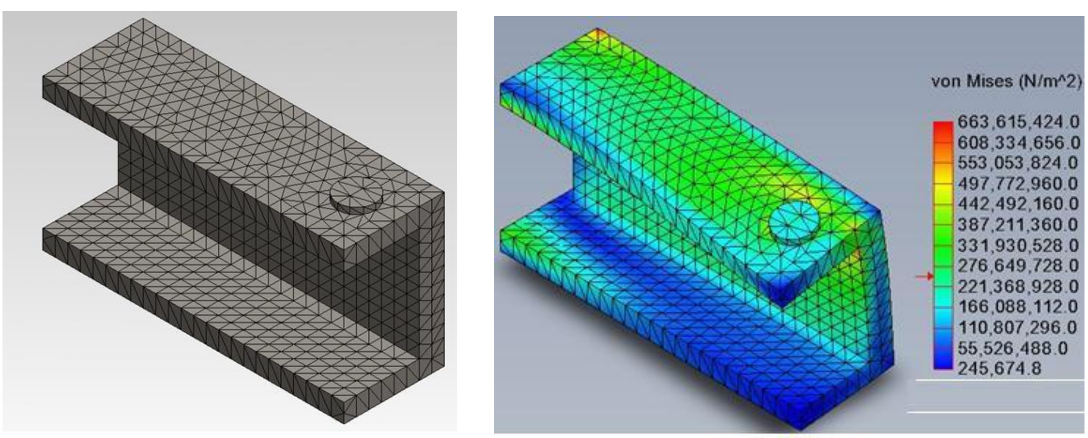
Shelves and Operator



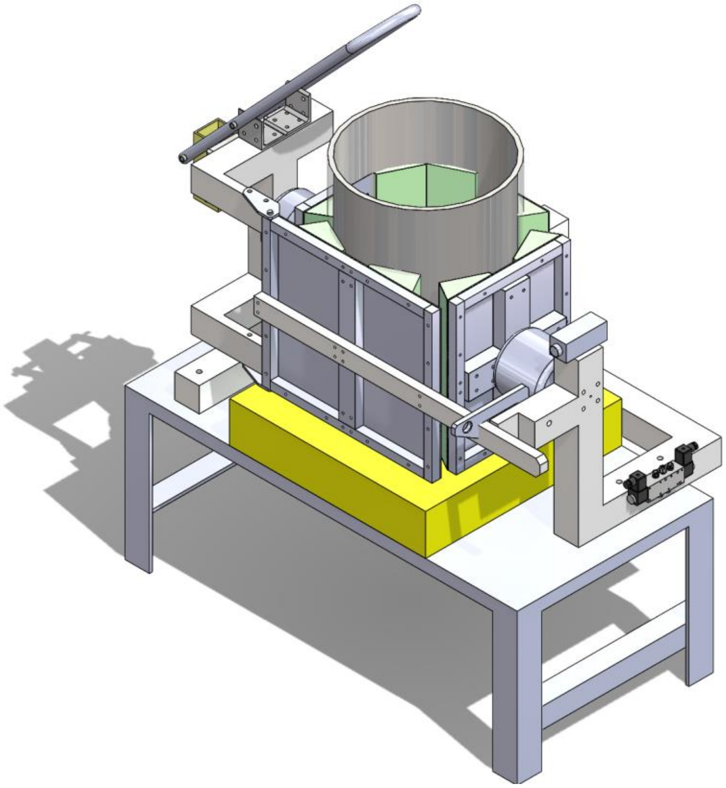
Shelves Assembly



Assembly Drawing with Bill of Materials



SolidWorks Finite Element Analysis (FEA) (add in)
Inch-pound and millimeter-kilogram units.



3D Manufacturing Assembly

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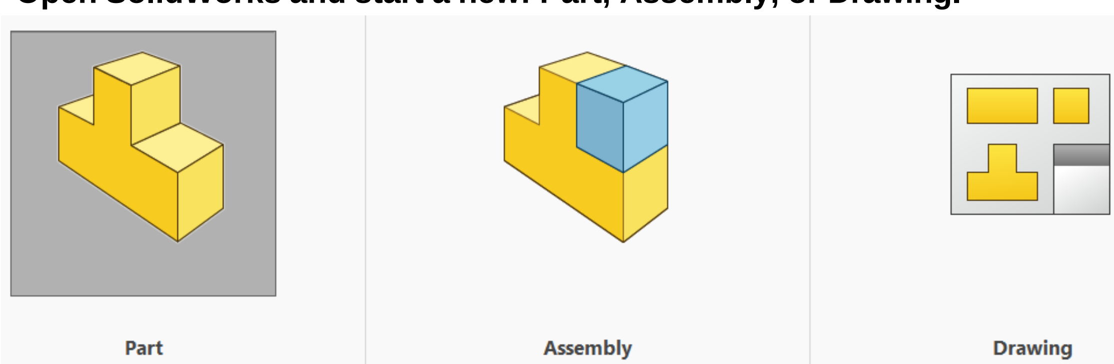
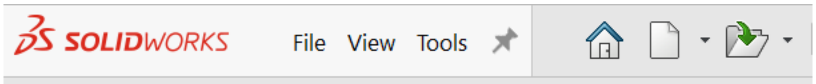
Application of this information to a specific project should be reviewed by a registered architect and/or professional engineer/surveyor. Anyone making use of the information set forth herein does so at their risk and assumes any and all resulting liability arising therefrom.

1 START PART

Open SolidWorks and start a new: Part, Assembly, or Drawing.

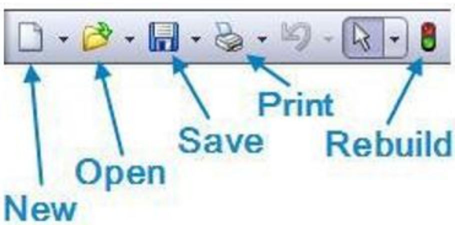


Select > New > Part > OK

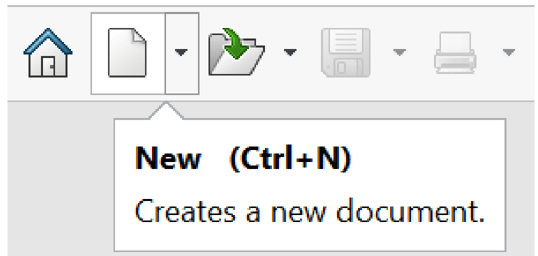


SolidWorks 2022

SolidWorks 2012



SolidWorks 2012

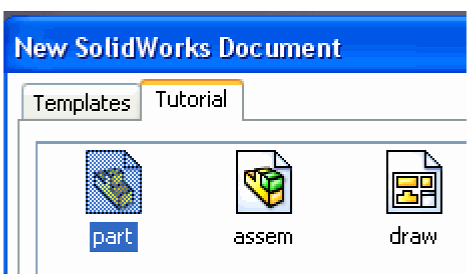


SolidWorks 2022

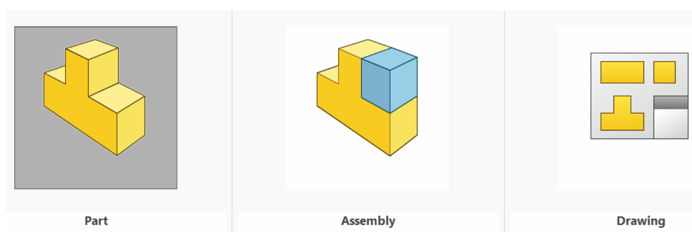
Click on the drop-down menu > New

"New SolidWorks Document" below will open.

Part, Assembly, or Drawing

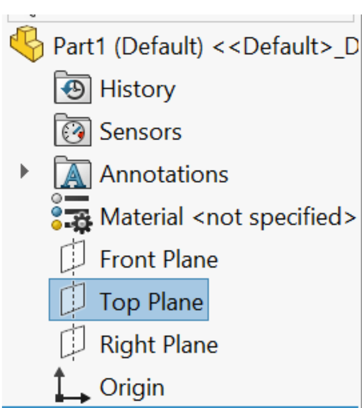


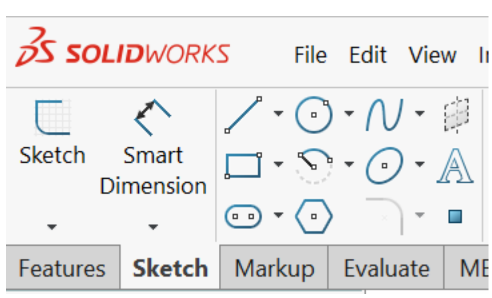
SolidWorks 2012



SolidWorks 2022

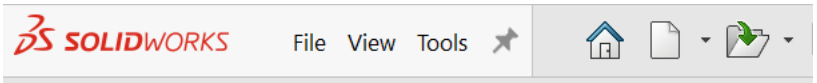
Select > Part

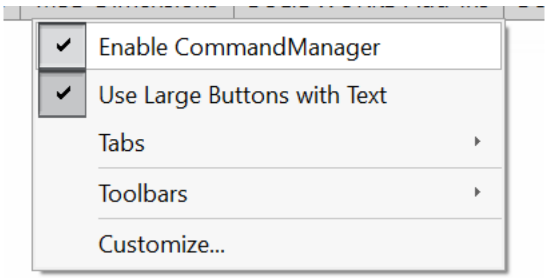




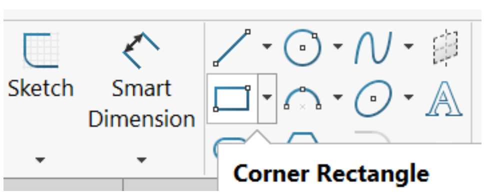
Select > Top Plane > Perpendicular > Sketch > Rectangle

If the > Features & Sketch & Markup & Evaluate menus do not open>



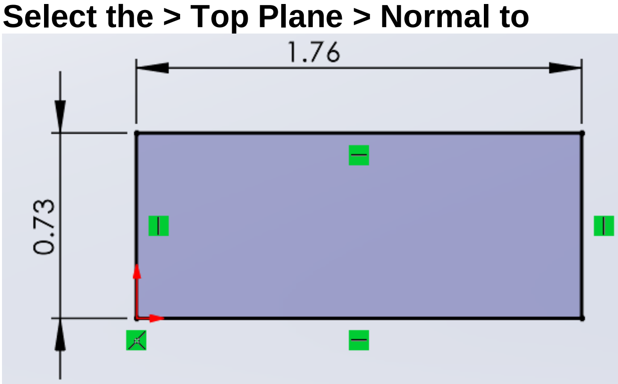


Right click under the SOLIDWORKS banner above and select > Enable Command manager.



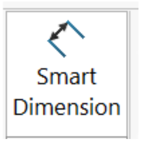
Select the > Corner Rectangle tool.

Select the > Top Plane > Normal to

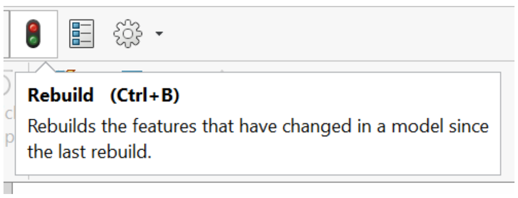


Start sketch at the Origin (two red arrows)

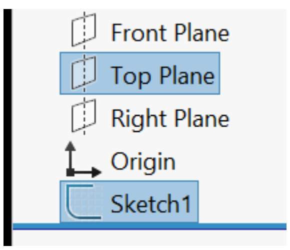
Sketch a rectangle starting at the origin.



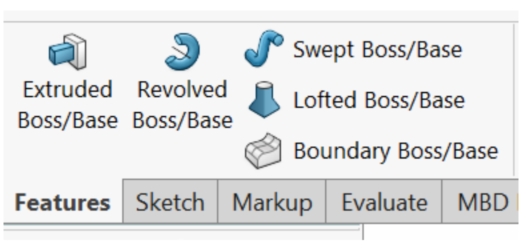
Select > Smart Dimension > Pick one dimension > Type > 1.00
Pick the other dimension Type > 2.00



When the sketch is dimensioned > **Pick the figure 8 red & black (Rebuild) tool.**

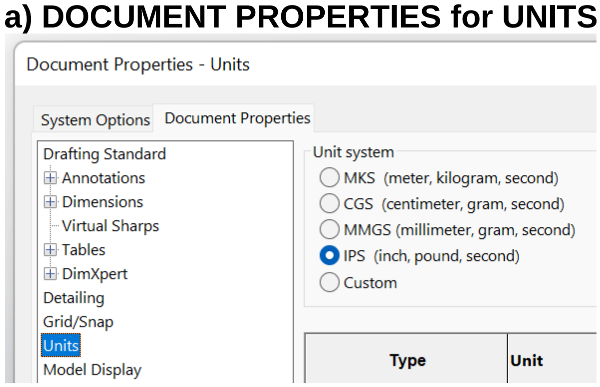


Select > Sketch > Top Plane



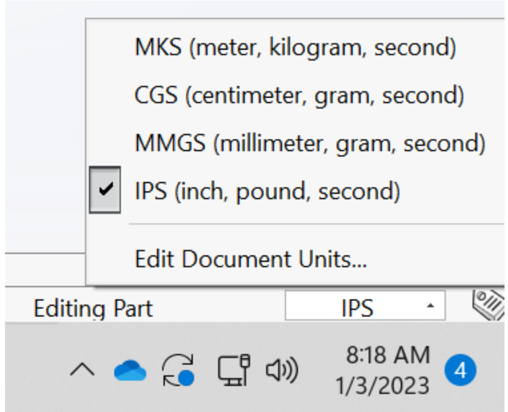
Select > Features > Extruded Boss/Base

a) DOCUMENT PROPERTIES for UNITS

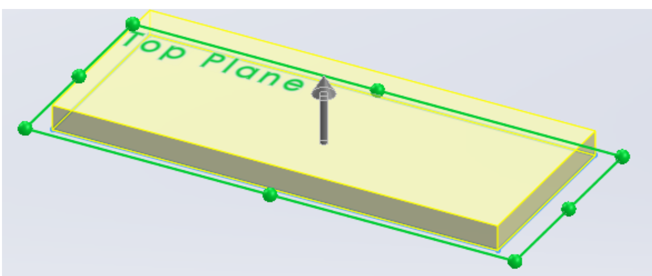


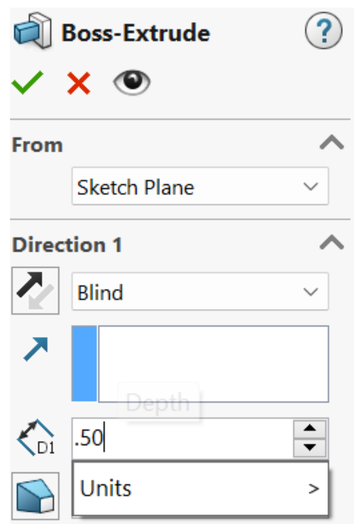
To Select US or Metric dimensions > Tools Options > Document Properties > Units.

b) DOCUMENT PROPERTIES for UNITS

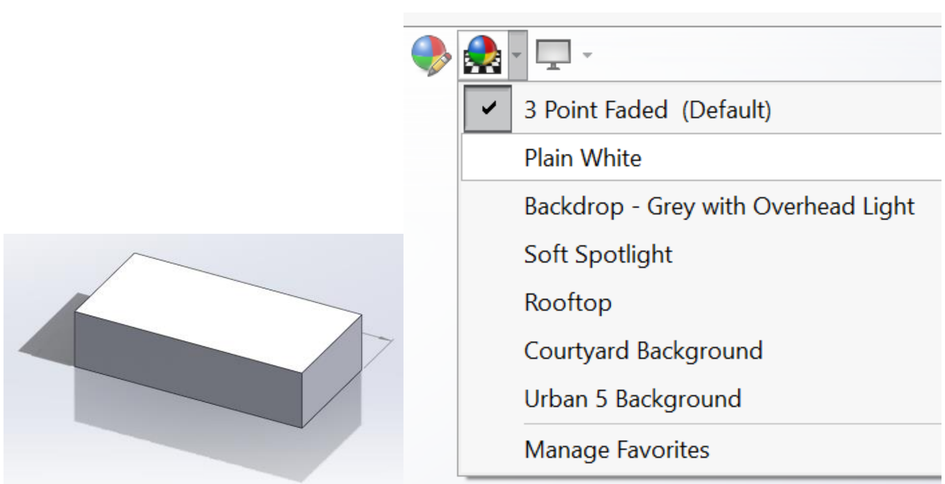


Select > IPS (Display bottom right).

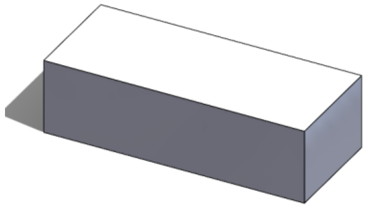


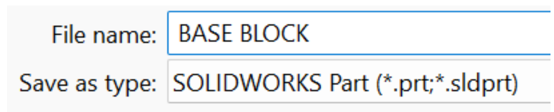


(Boss-Extrude) menu will open > **Type > .50 > OK.**

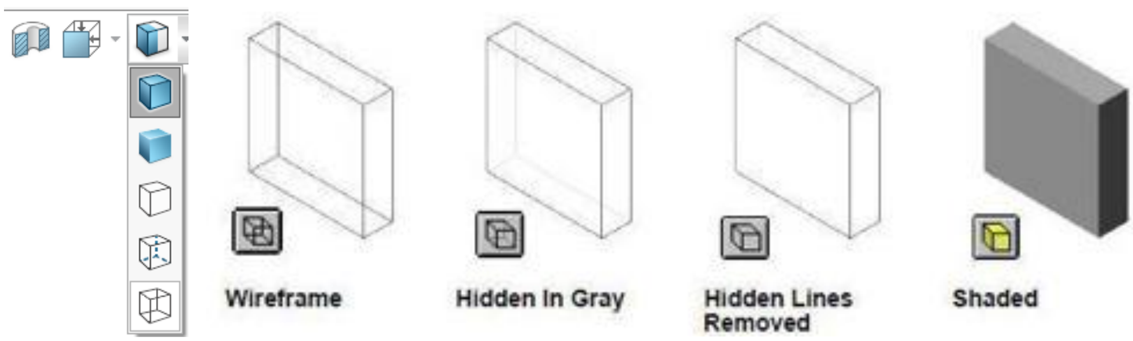


Select (Apply Scene) menu.> Select (Plain White).





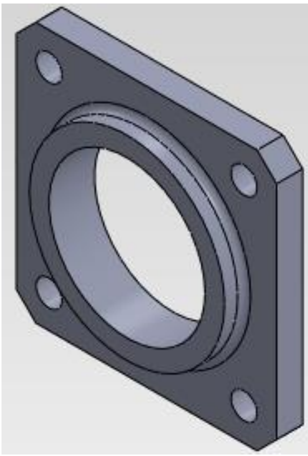
Select > File > Save As > BASE BLOCK
A Part 3D model has been created and saved.



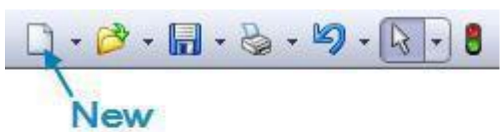
Display Style

Click the "Display Mode" icon to obtain the part or assembly modes above.

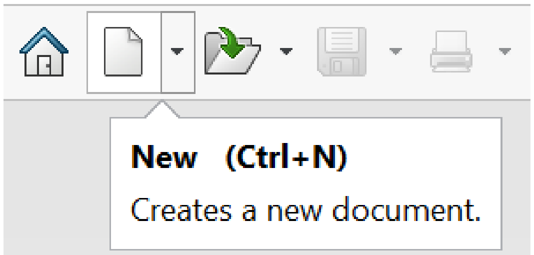
SQUARE FLANGE



Follow the steps below to create the, "FLANGE BRACKET" solid model shown above.

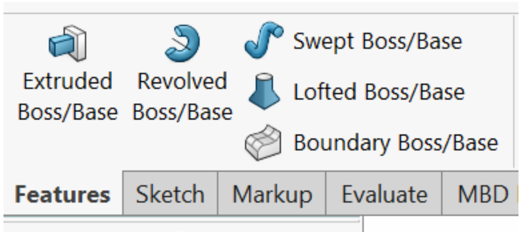


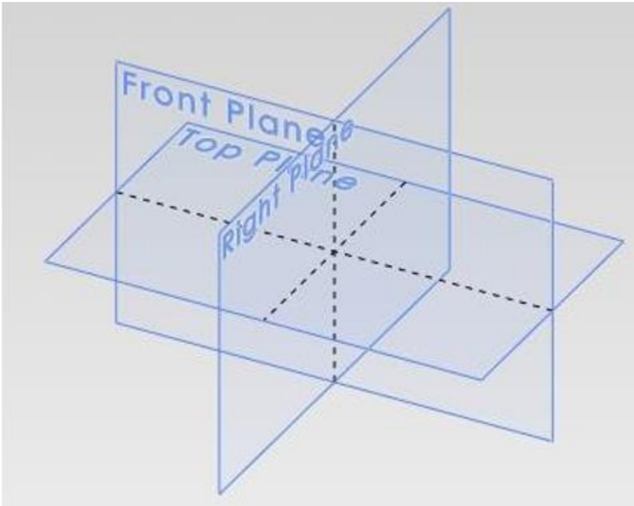
Start the 3 Dimensional Model by clicking on the "New" icon or pick drop down menu: Insert > New.



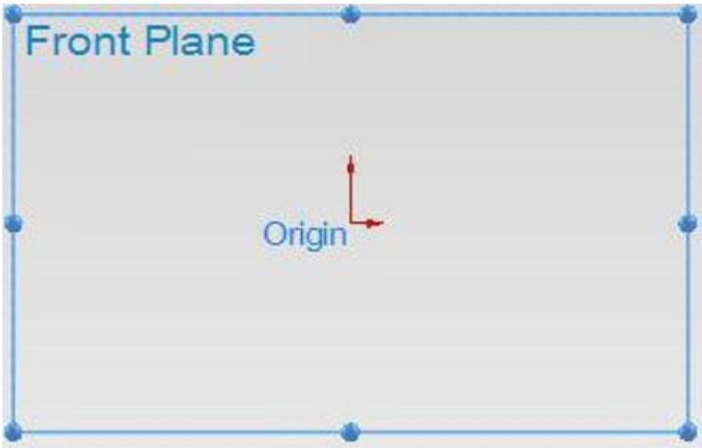
Click left mouse button: New > Part > OK. The "Feature Tree" below will now open.

A two-dimensional sketch must be created on a selected plane or surface before the desired solid model can be created.

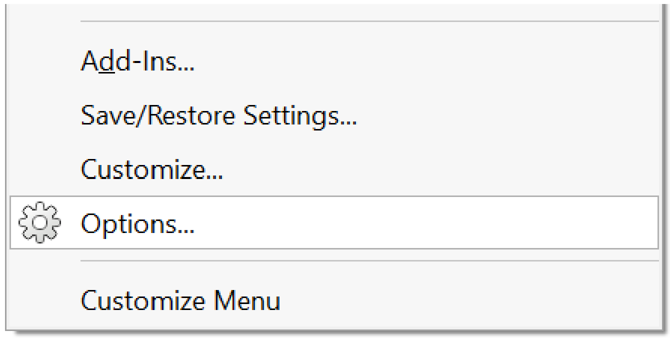




Select > Features > Extruded Boss/Base

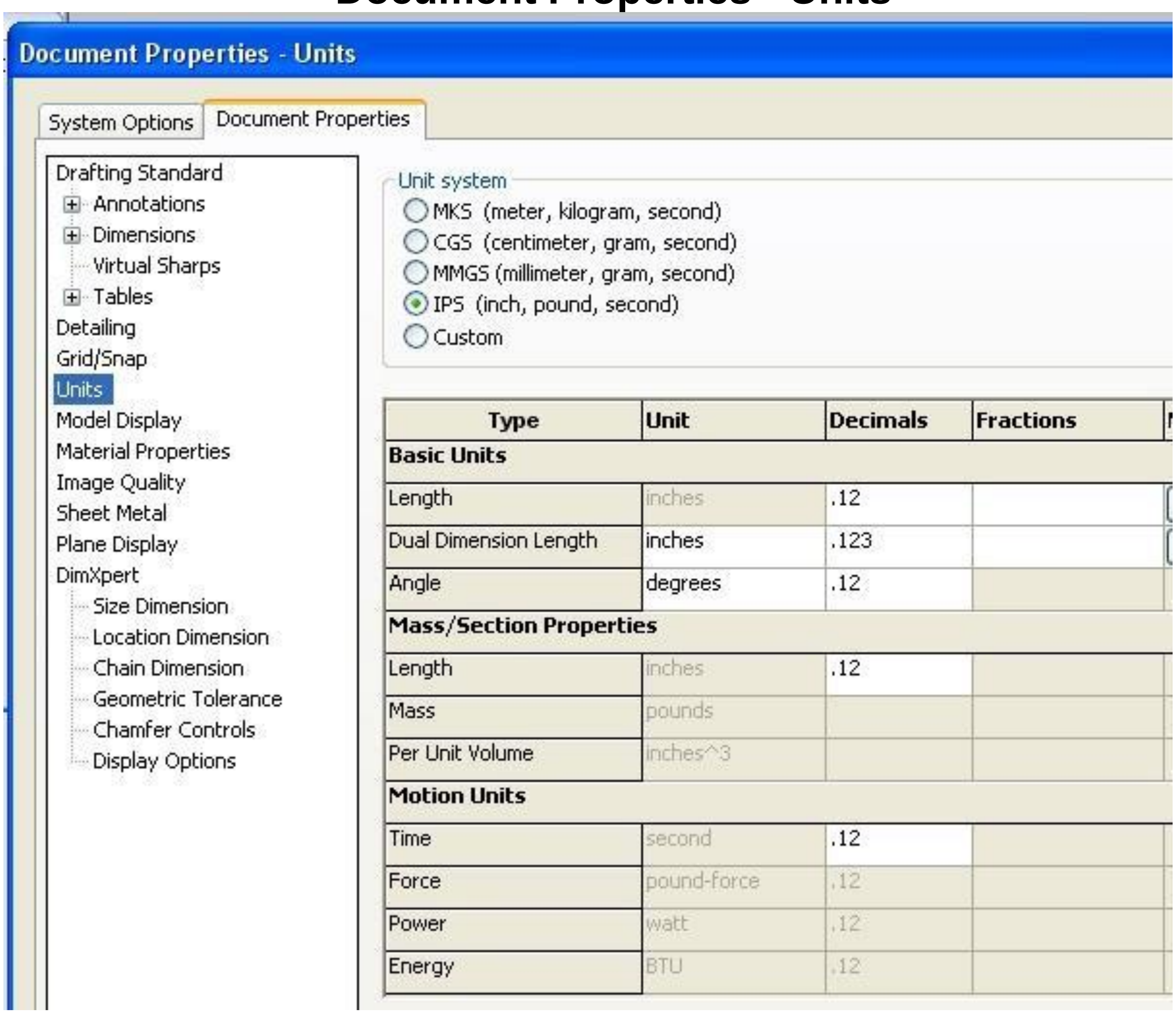


"Front Plane" Make a profile sketch on the selected Front Plane
The Origin, x, and y directions are shown in the chosen front plane.



Tools > Options

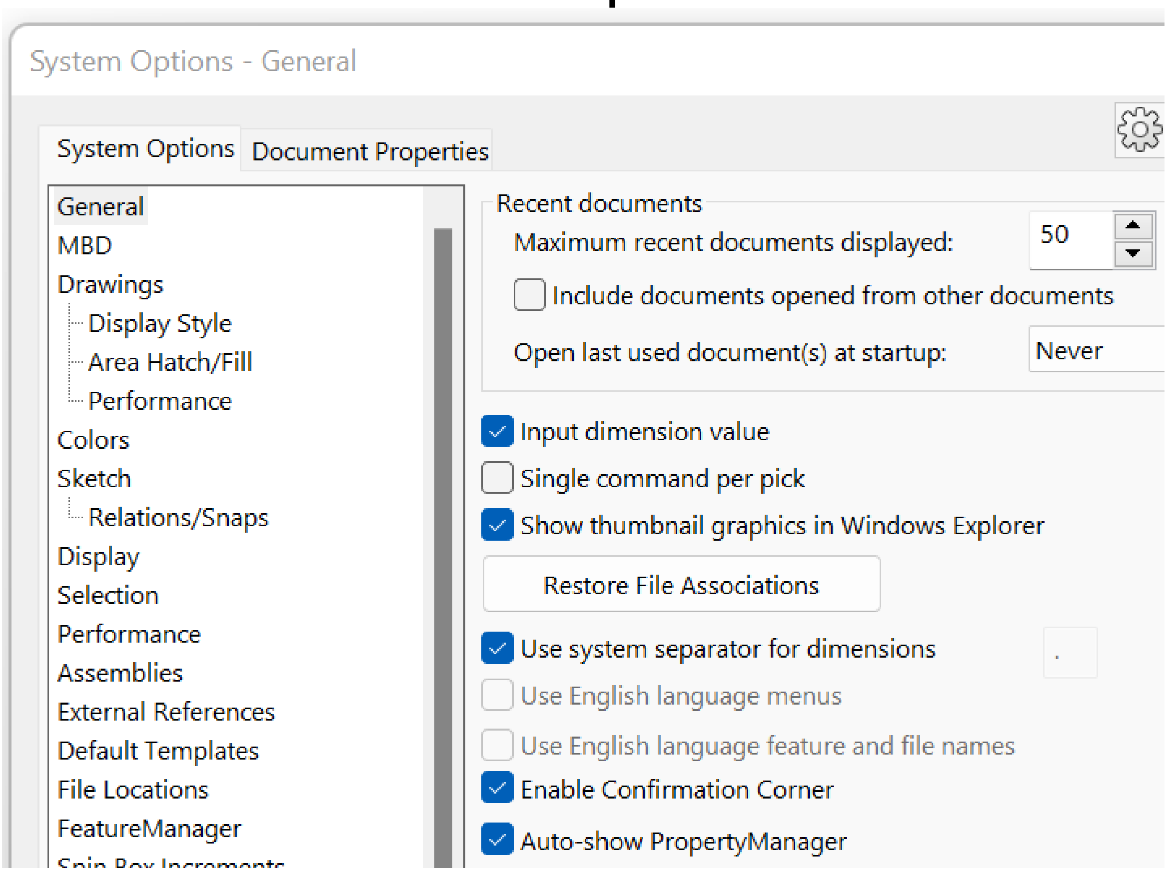
2012
Document Properties - Units



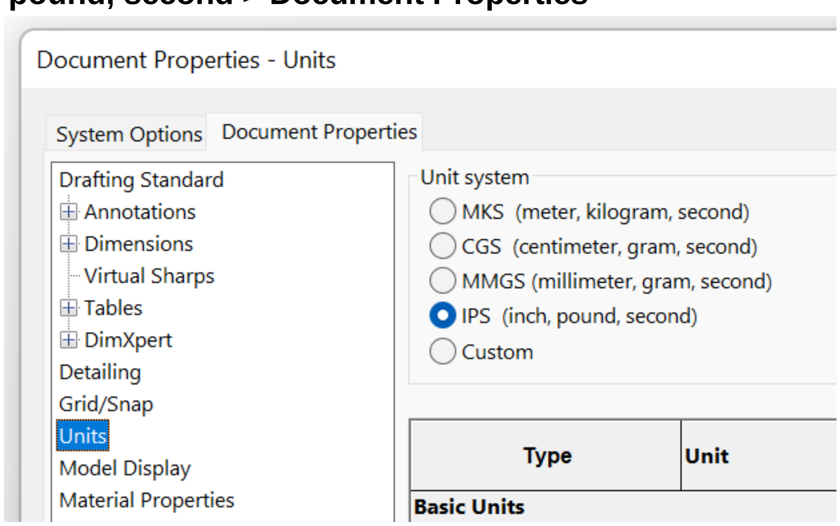
Open the "Document Properties" box shown above to change units of measurement.

2022

Document Properties - Units

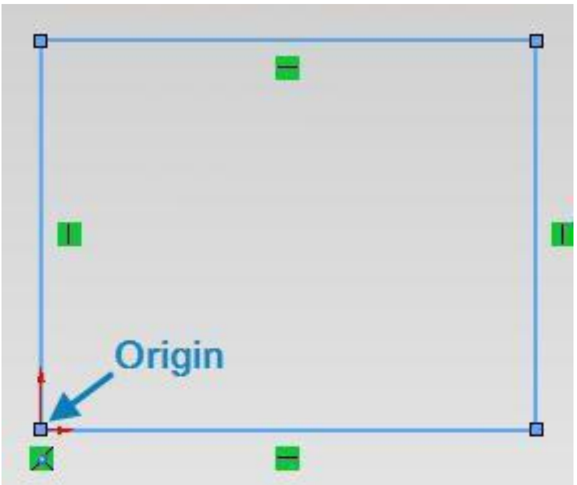


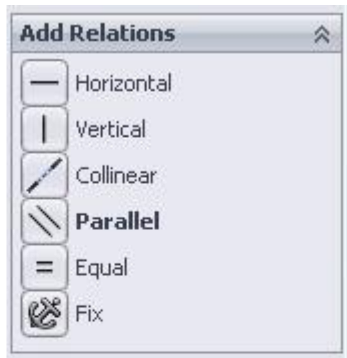
Pick drop down menu: Tools > Option > Document Properties > Units > IPS (inch, pound, second > Document Properties



Select > MMGS (millimeter, gram, second) is also available.

Pick: “Sketch” tab > Pick the “Rectangle” tool >



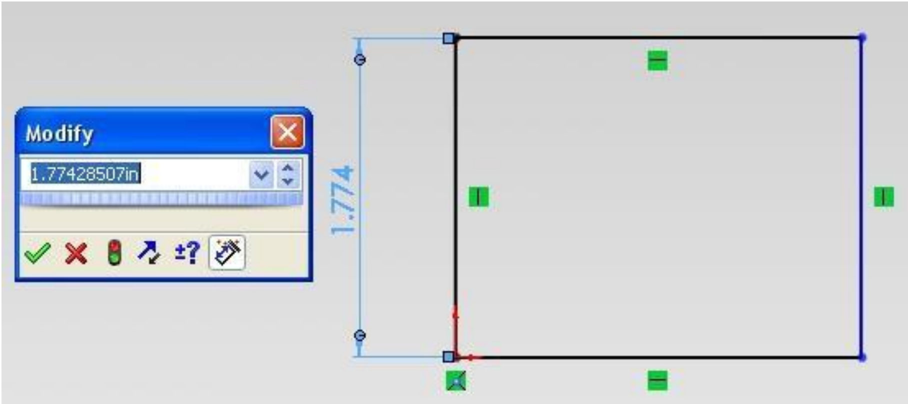


Right click the “Origin” > Drag mouse pointer to a temporary top right corner > Click. Click the green check mark (OK) to complete the rectangle command.

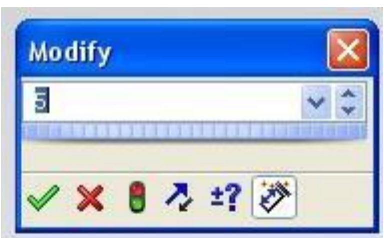
<u>Horizontal, Vertical and other geometric relations between lines are added automatically by SolidWorks.</u>

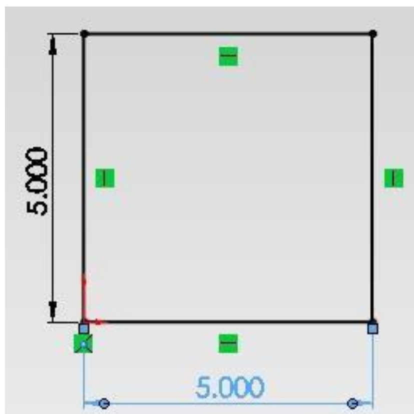
Or manually using drop down menu: Insert > Relations.

Pick “Smart Dimension” tool > Pick the left side of the rectangle >

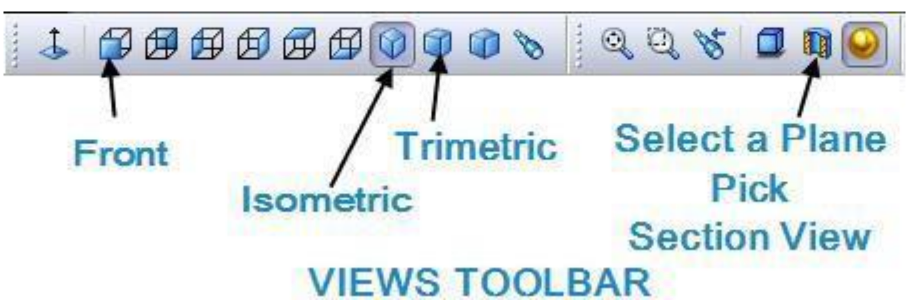


Drag dimension away from the rectangle and pick to place the dimension as above. Modify the dimension > type 5 > Click check mark to complete the command.



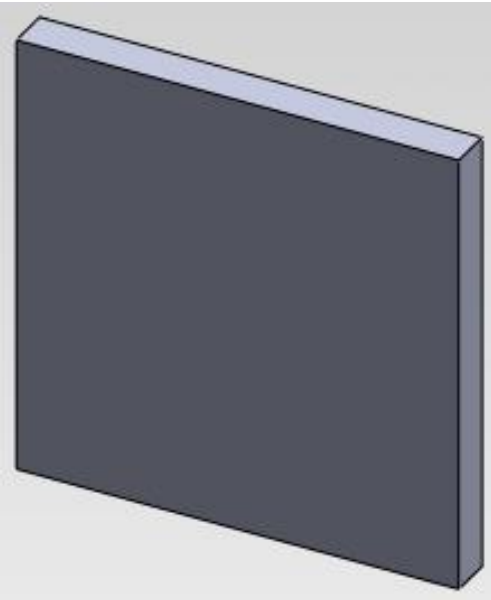


Dimension a side normal to the first dimension.



Modify the dimension > type 5 > Click check mark to complete (OK). Click "Exit Sketch".

Pick the "Isometric View" icon in the "Views" toolbar above.

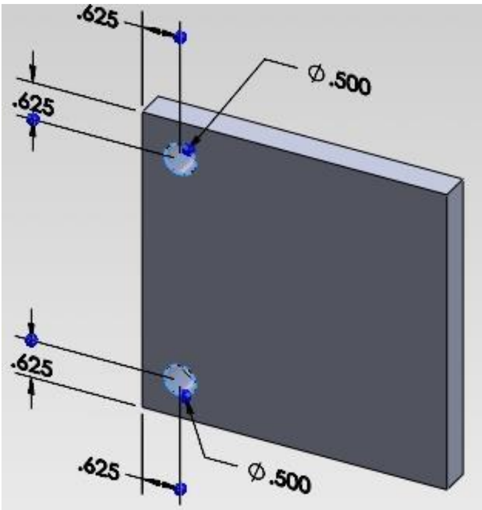


<u>**TYPE f to fit the object in the display**</u>

Pick "Extruded Boss/Base" > Blind > D1 thickness > Type D1 dimension > 0.50in >

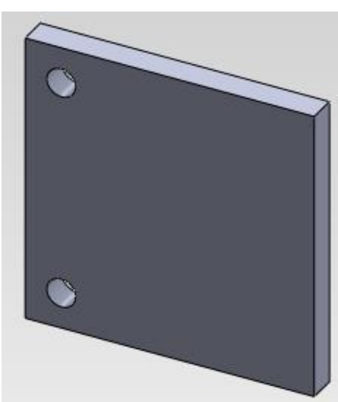
Click the above green check mark (OK) to complete the extrude boss command.

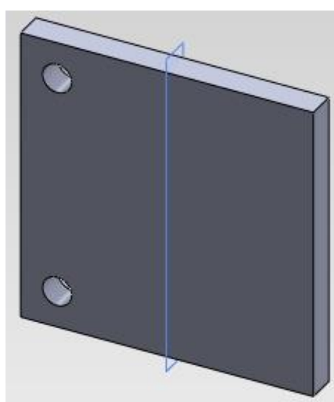
Note: The "Boss/Extrude" dialog box allows extrusion in both directions perpendicular to the profile sketch plane. See "Direction 1" and "Direction 2" above and in section 10 - First Assembly
- Pipe Elbow below.



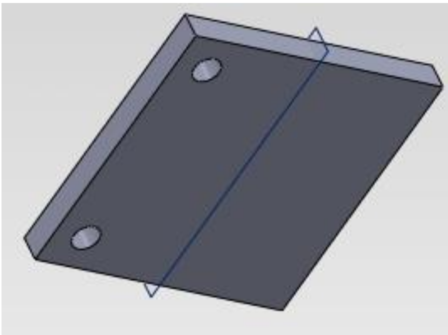
Model the two holes above.
Pick the front surface of the part above > Select the "Features" tab > Pick "Extruded Cut".

Pick" Sketch tab > Circle tool > Sketch the 2 holes shown above > Smart Dimension tool > add
0.500 inch diameter and the above hole location dimensions > Click: Exit Sketch.



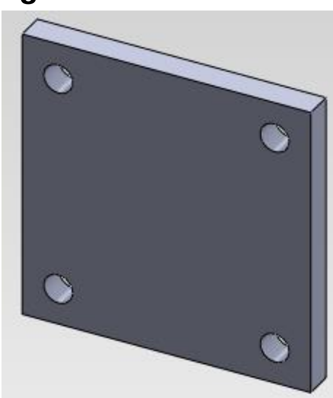


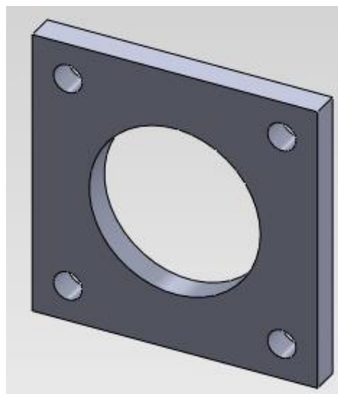
REFERENCE PLANES
Insert > Reference Geometry > Plane > Pick the right side surface >

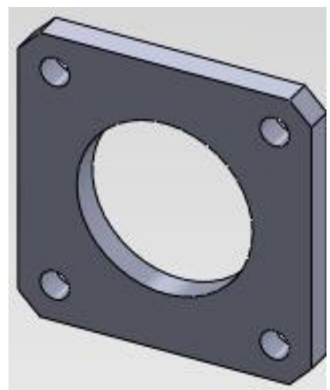


Rotate the part by holding the mouse wheel down and drag horizontally across the part > Pick the left side surface > the above "Mid-Plane" is created by SolidWorks.

Insert > Pattern/Mirror > Pick the two left holes to mirror > the two holes on the right side are created below.

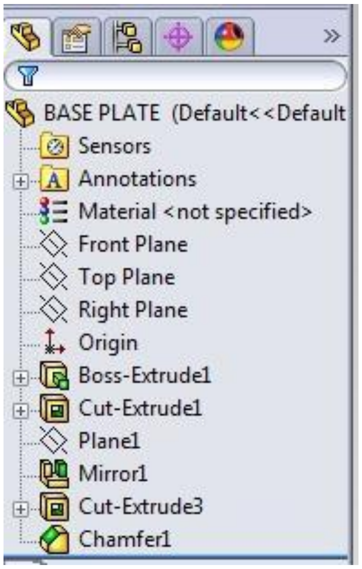


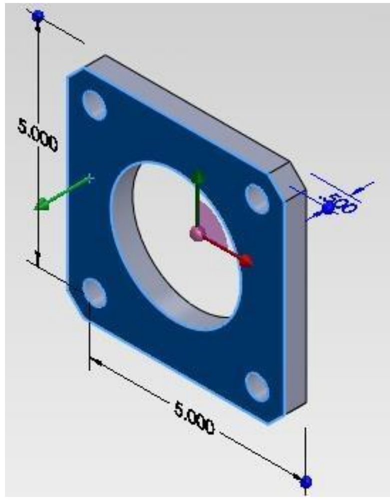


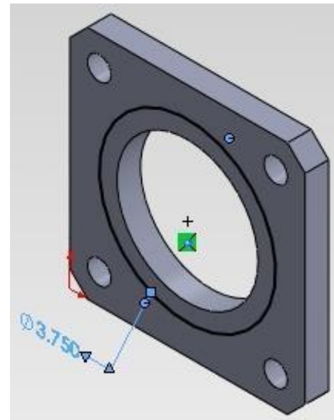


Cut-Extrude > Sketch the large center hole on the front surface of the part > Click Smart Dimension > Dimension the center hole 3.00 inch diameter and add the hole location dimensions > Exit Sketch.

Insert > Cut > Extrude > Thru all > Click check mark.
Insert > Features > Chamfer > Pick each of the 4 corners > OK
Or click the "Fillet" icon drop down menu > Chamfer > Pick each of the 4 corners >

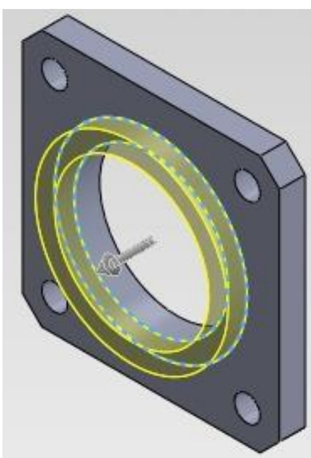
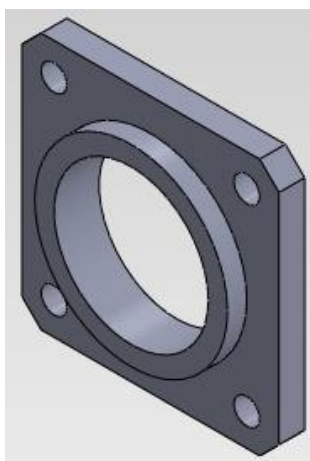
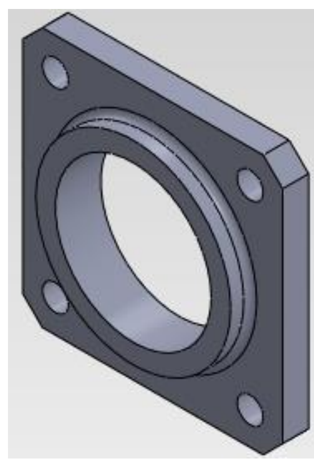






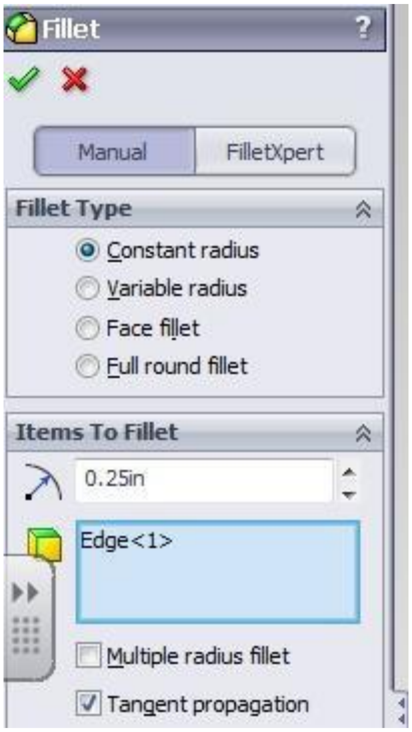
The "Features Tree" above lists all operations performed on the part (or assembly) model. Double-click on an icon to modify that feature in the part.

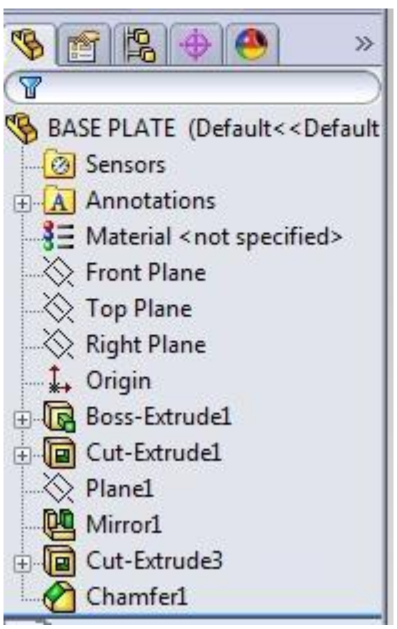
Pick the front surface > Sketch > Circle tool > Smart Dimension > 3.00 diameter >

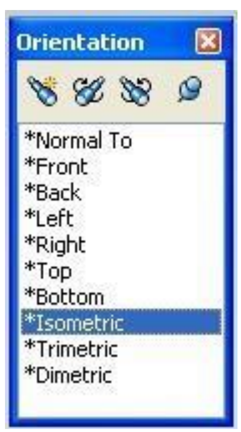




Tools > Sketch Tools > Convert entities > Click on edge of the 2.75 inch diameter circle > Click the green check > Exit Sketch.

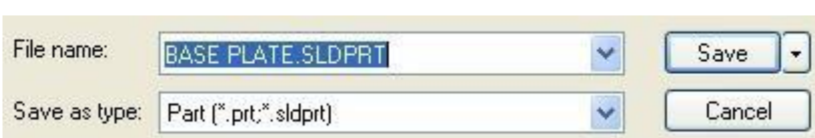
Insert Boss/Base > Extrude > Pick the ring > Click the green check. Click ring base > "Fillet" icon or Insert > Features > Fillet/Round







Click "Isometric View" in the "Orientation" dialog box above.

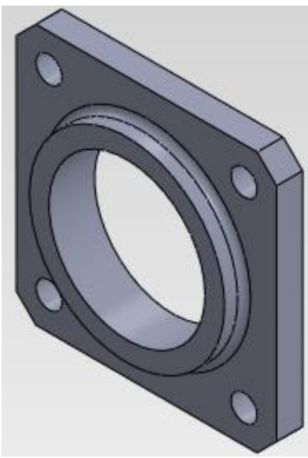


Save As > BASE PLATE > ".SLDPRT" is added by SolidWorks.

PARAMETRIC CAD

The rectangular plate solid model with five holes has been fully dimensioned and saved.

It is possible to re-open this part in SolidWorks, double click on its surface, and change one or all of its dimensions.



SolidWorks software utilizes a design feature called parametric computer aided design, a method of linking dimensions and variables to geometry in such a way that when the values change, the part changes as well.

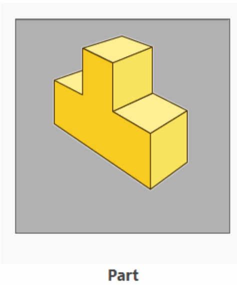
A parameter is a variable to which other variables are related, and these other variables can be obtained by means of parametric equations.

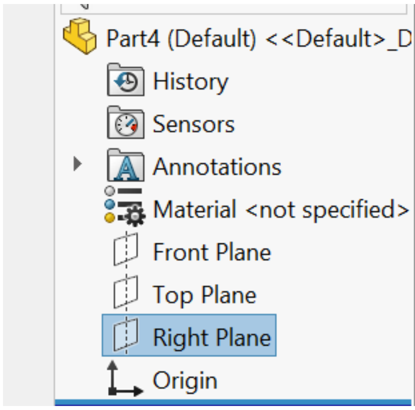
In this manner, design modifications and creation of a family of parts can be performed in remarkably quick time compared with the redrawing required by traditional CAD.

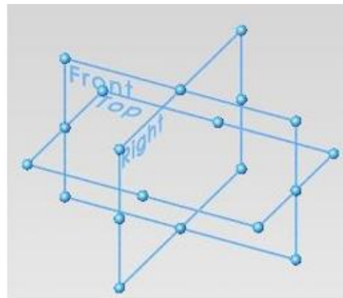
In the past five years, PTC's success has prompted major CAD players to offer similar functions.

Parametric modification can be accomplished with a spreadsheet, script, or by manually changing dimension text in the digital model.

2 ROUND PART

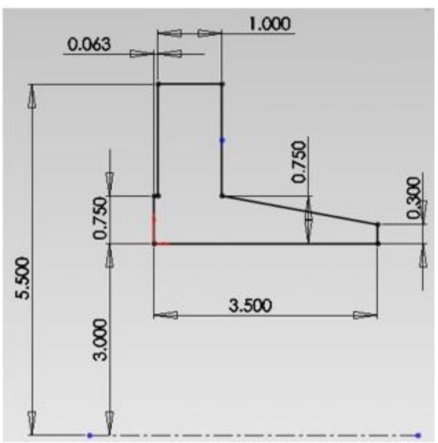




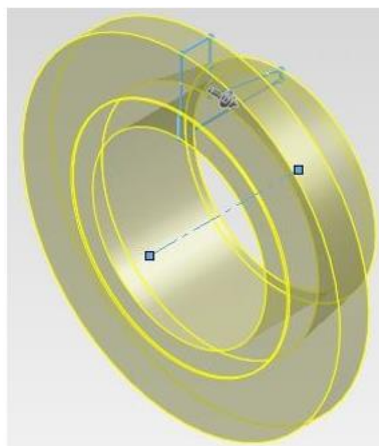


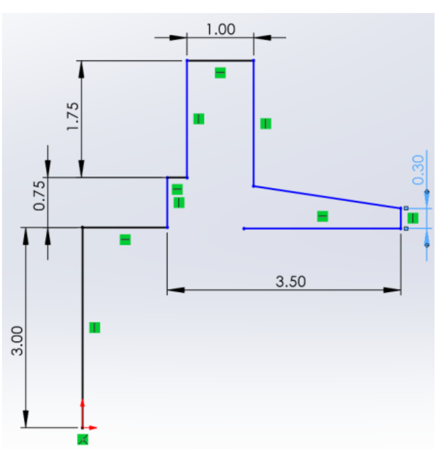
Select > Part > Right Plane

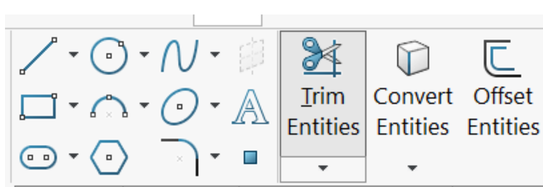
The two-dimensional sketch below is created on the (Right Plane).



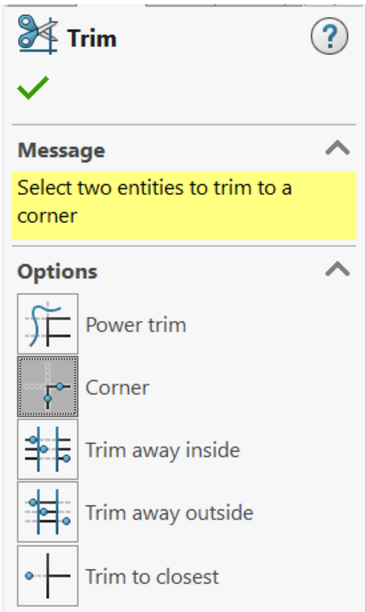
Completed Sketch

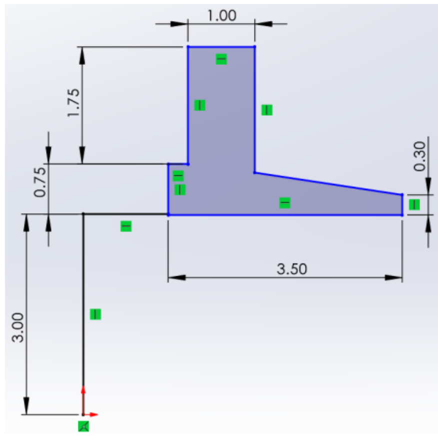

Revolved > Boss/Base





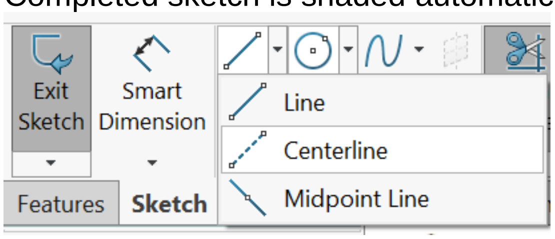
Complete the sketch with the (Trim) tool.



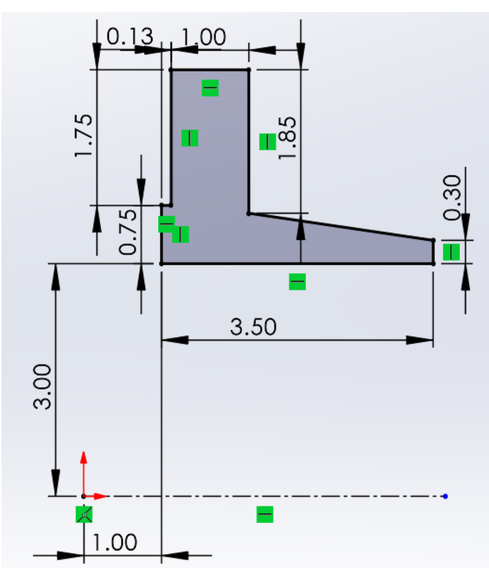


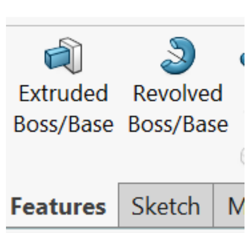
Select > Corner
Pick bottom horizontal line > Pick left vertical line.

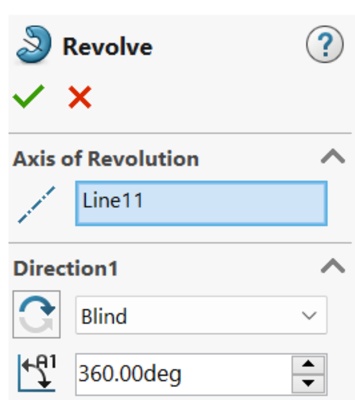
Completed sketch is shaded automatically.



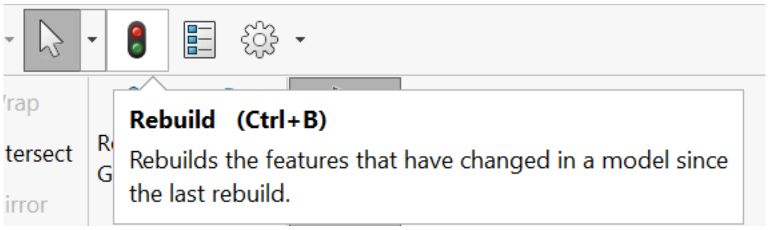
Select > Line > Centerline



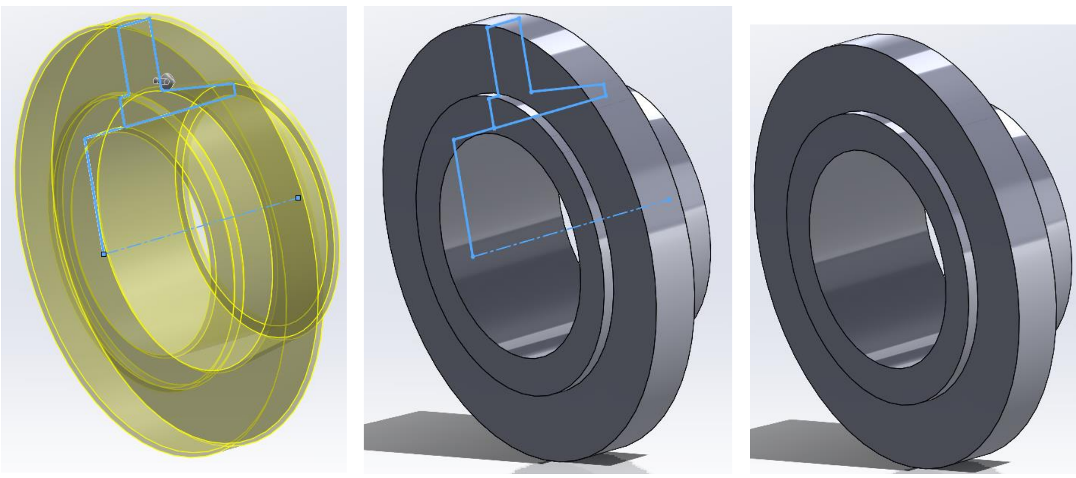




Sketch > Centerline > OK > Rebuild > Feature > Revolved Part Boss/Base



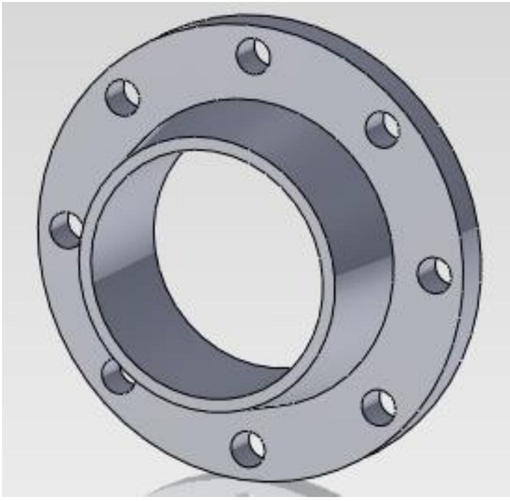
Select > Rebuild

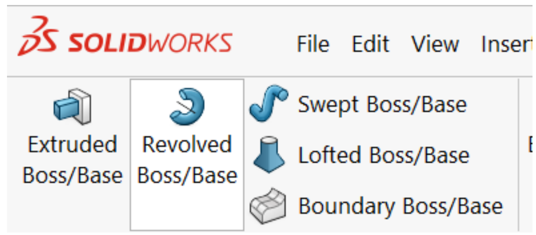


A two-dimensional sketch must be created on a selected plane or surface before the desired solid model can be created.

3 CIRCULAR PATTERN OF HOLES

Left, click the "Sketch" tab shown above to obtain the sketch tools.





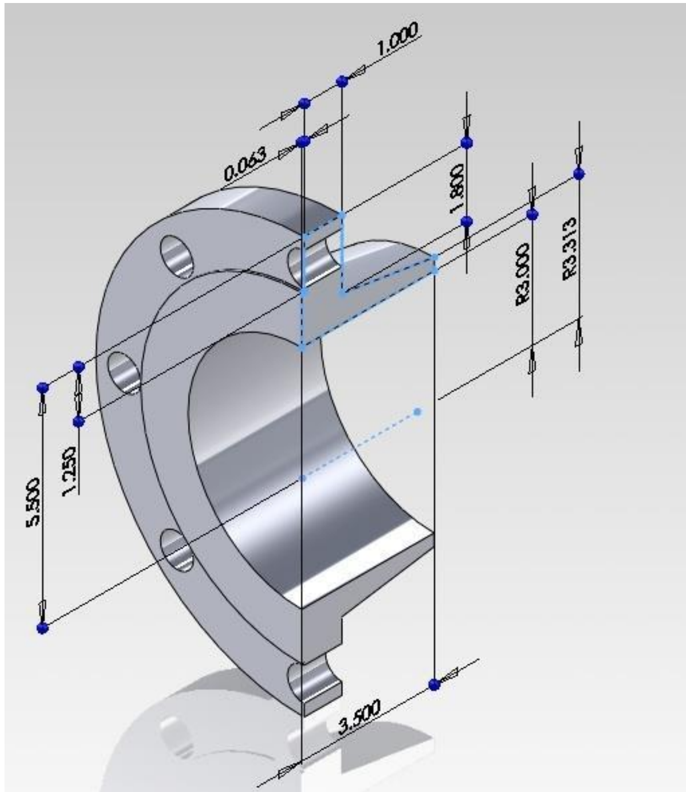
Follow the steps below to create the above "FLANGE" revolved shape solid model. Start the 3-Dimensional Model by clicking on the "New" icon > OK

Right click > "Right Plane" under the "Features" tab shown above. Origin and x and y directions are shown in the Right Plane.

"Sketch" tab > Pick the "Line" tool shown above.

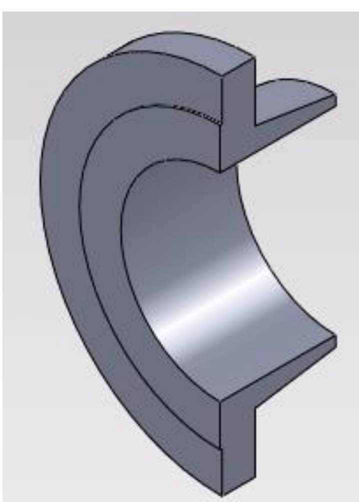
CREATE A CIRCULAR PATTERN OF HOLES

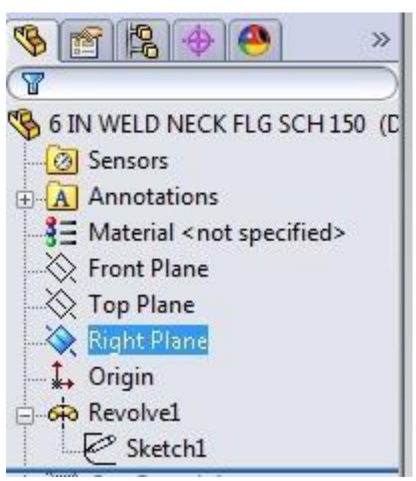
A first hole must be created in a part before a circular pattern of identical holes can be made.

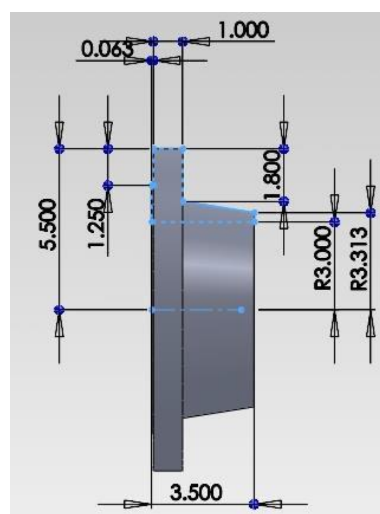


To obtain a part sectioned view pick an existing plane in the Features Tree > "Right Plane" or create a plane relative to an existing plane or surface by clicking: Insert >Reference Geometry > Plane > Pick an existing plane > Create a new plane at the desired section location > OK > Pick the "Section" icon below.

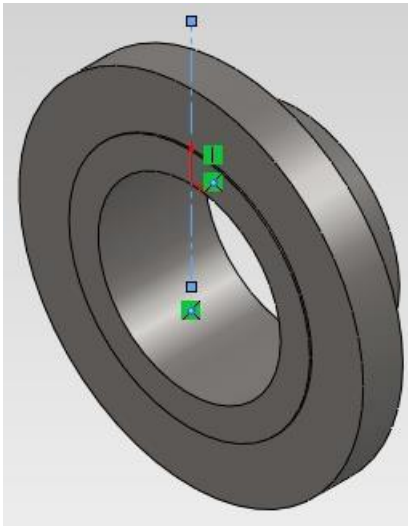
Click "Right Plane" in the Features Tree > Click the "Section" tool icon > Click the "Reverse View".





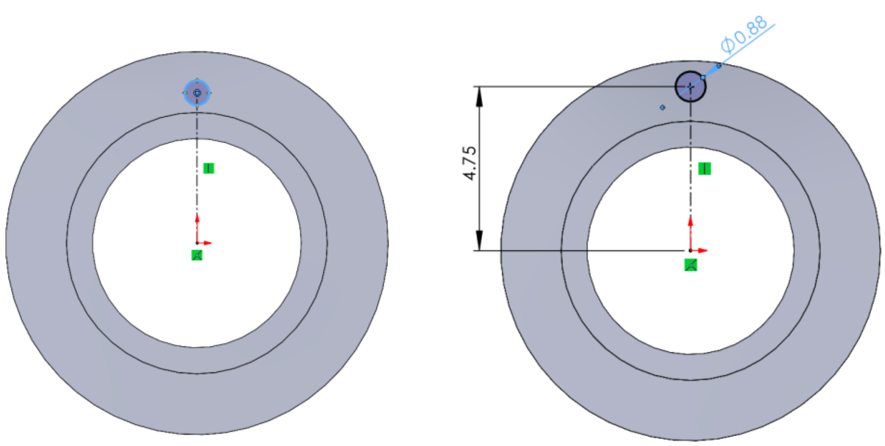


1 Click Isometric icon, then click Shaded view mode.
2 Click Right Plane in the Feature Manager design tree.
3 Click Section View on the View toolbar, or click View, Display, Section View.

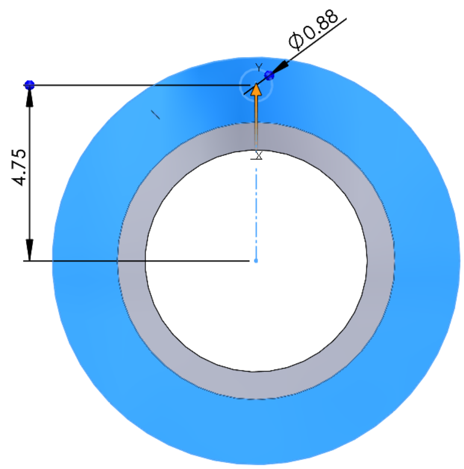


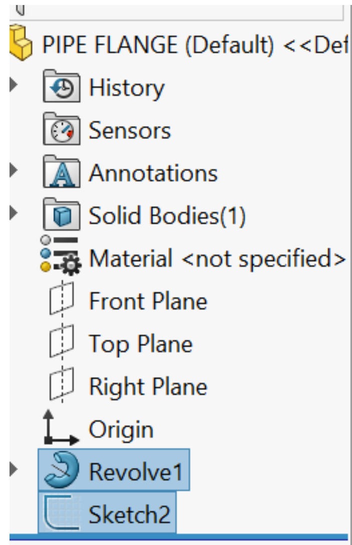
Pick the flange front surface > Sketch >
Pick: “Line” drop down menu > Pick: “Center line” icon > Pick flange center hole center point > Drag up > Pick top end point of this centerline > Existing Relations above are > Vertical & Coicident1 > Add Relations > Vertical > OK.

“Cut” the first bolt hole.

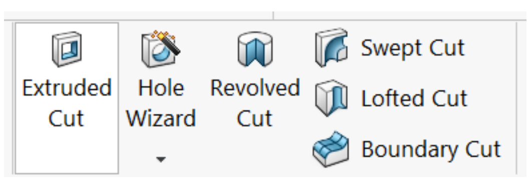


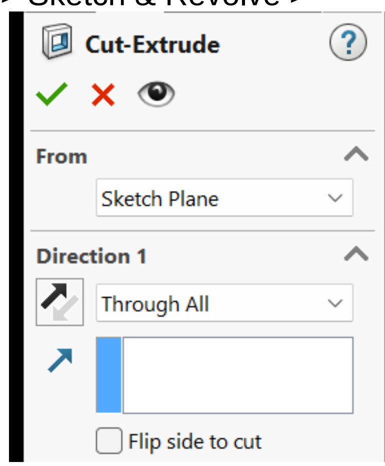
Select Flange surface > Sketch > Centerline > dimension (4.75) > Circle diameter (0.875)



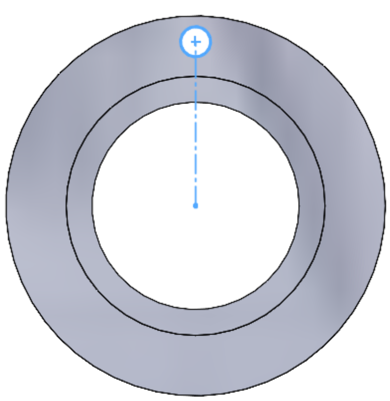


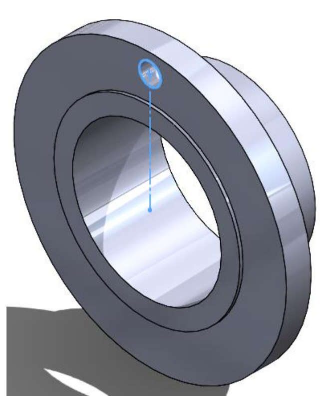
Select > Rebuild > Select > Flange > Select > Sketch & Revolve >

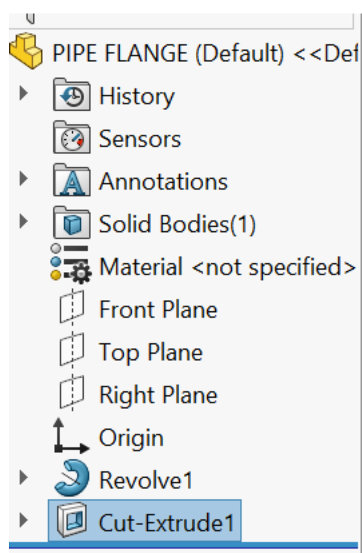




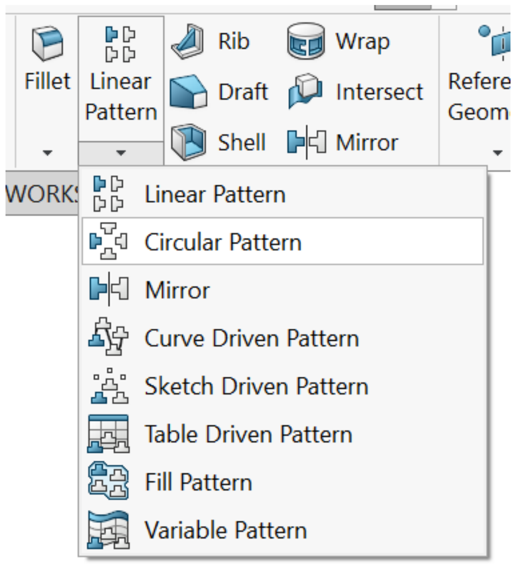
Select > Rebuild > Features > Extruded Cut > Drop-Down-Menu (Through All) > OK



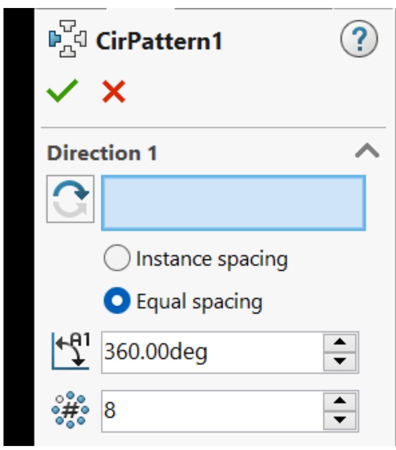


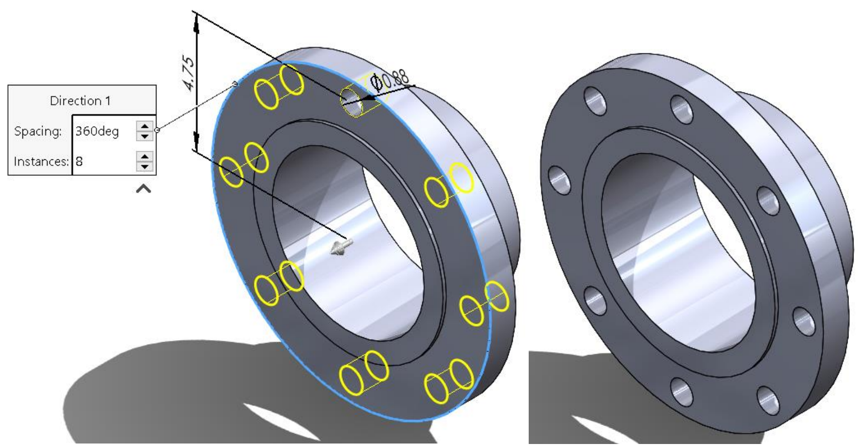


First Hole in Flange is Created (Cut Extrude1)



Select > Linear Pattern Drop-Down Menu > Circular Pattern



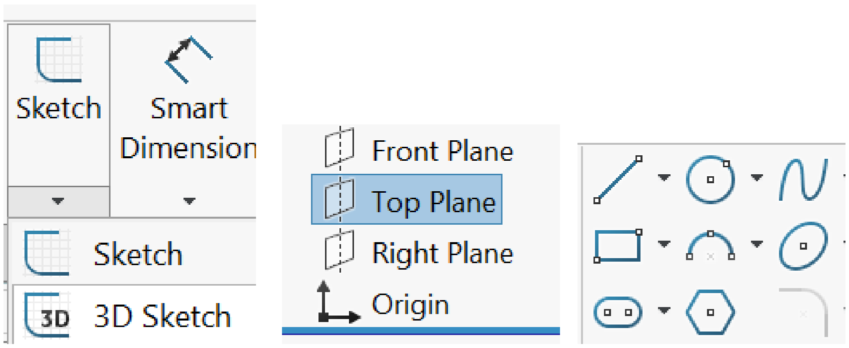


Equal Spacing > 8 (Holes) > OK **Circular Pattern of Holes are Created**

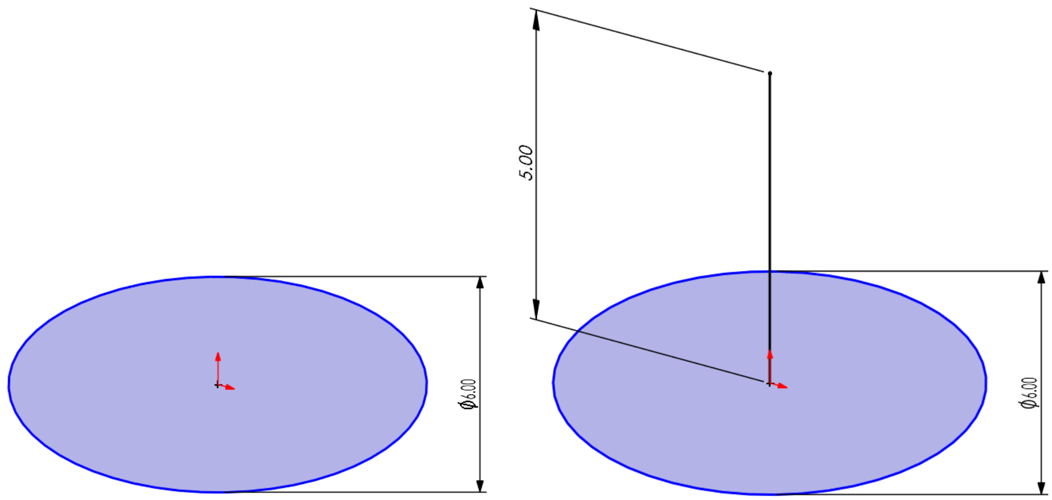
Pick: "Smart Dimension" icon > Dimension the hole 7/8-inch diameter and 4.750 radius > Exit Sketch.
Pick the bolt diameter circle > Insert > Cut > Extrude > OK

LOFT and SWEEP

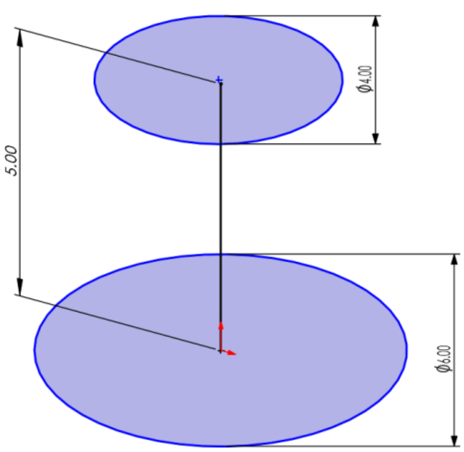
LOFTED BOSS BASE

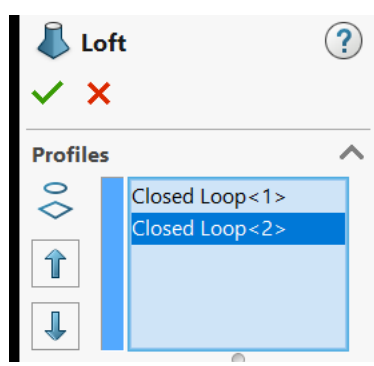


Start Part > Sketch > 3D Sketch > Top Plane >

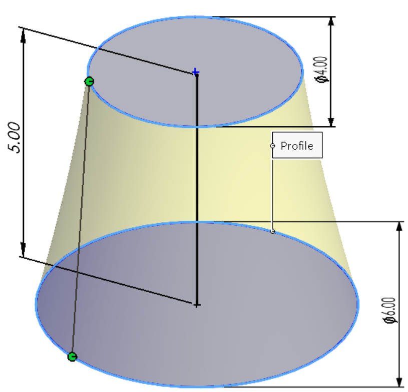


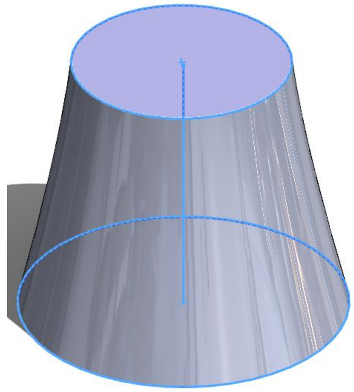
Sketch > 3D Sketch > 6” Diameter > Tab key for plane > Sketch vertical line > 4”

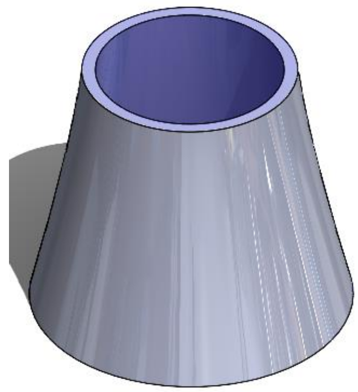


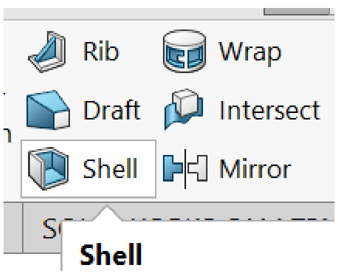


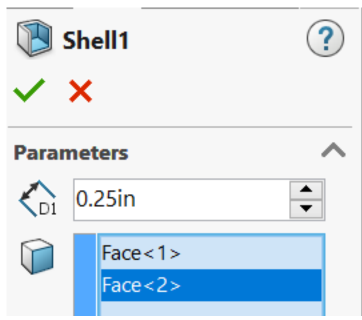
Sketch top circle > 4” diameter > Feature Lofted Boss/Base > Loft > OK

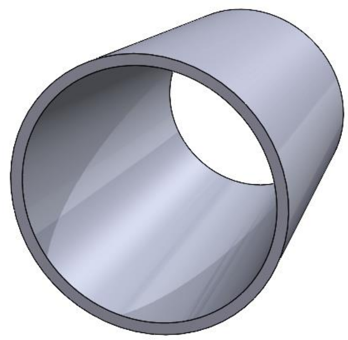












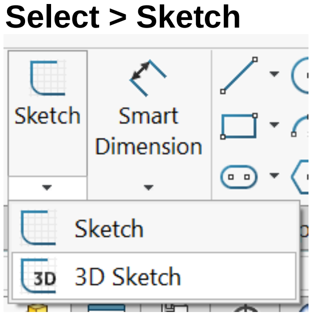
Circle tool 3.000 radius > Circle tool 2.000 radius > OK

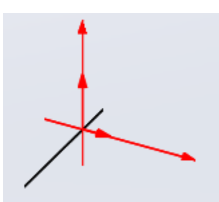
Insert > Boss/Base > Loft > Pick 3.000 radius circle profile > Pick 2.000 radius circle profile > Ctrl + Q to exit sketch. Click on Front Plane > Insert > Reference Geometry >

Follow the steps below to create the above channel bracket and perform a finite element analysis to determine the stress distribution and deflections due to applied loads.

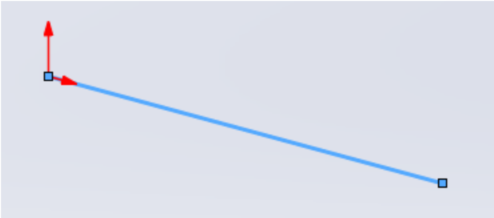
SWEEP & LOFT with 3D SKETCH

Select > Sketch

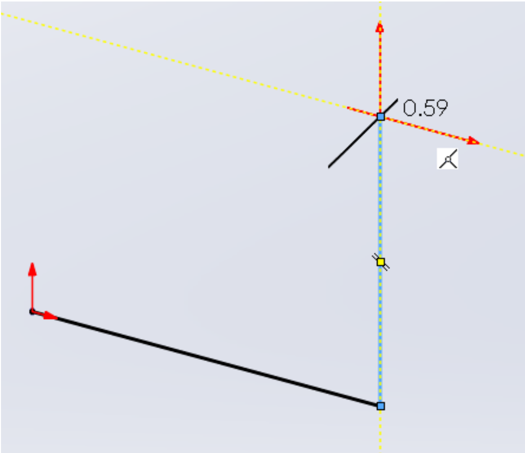




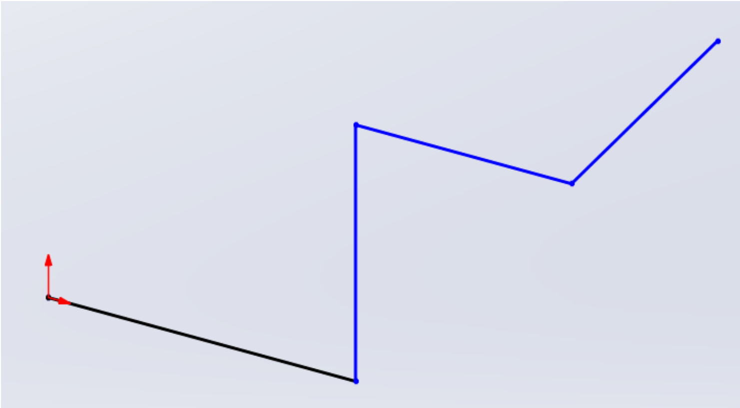
Drop down menu > 3D Sketch > Line (Start a line at Origin).



Sketch horizontal line.

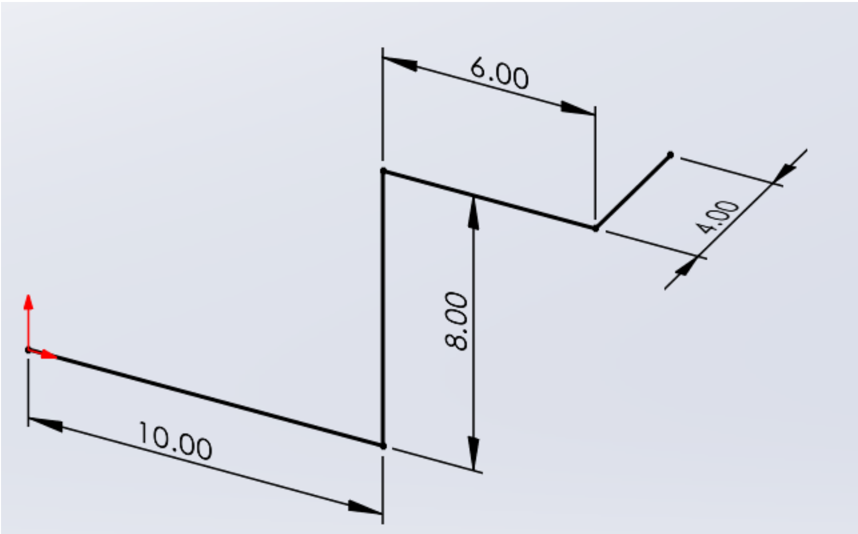


Vertical line.

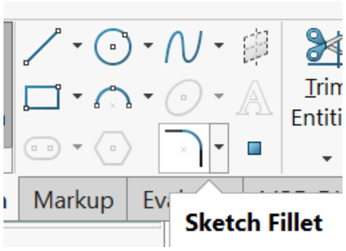


(Tab) to change line direction.

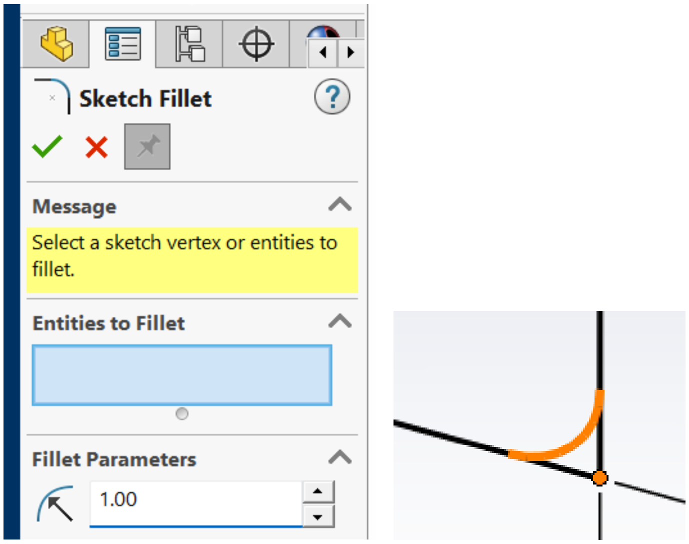
Horizontal (Y) direction followed by horizontal (Z) direction.



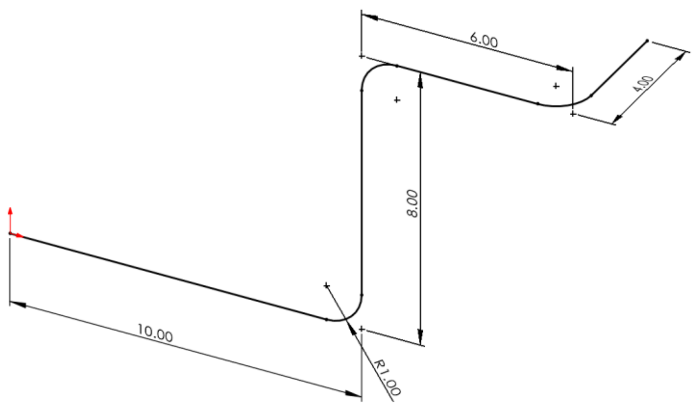
Place dimensions.



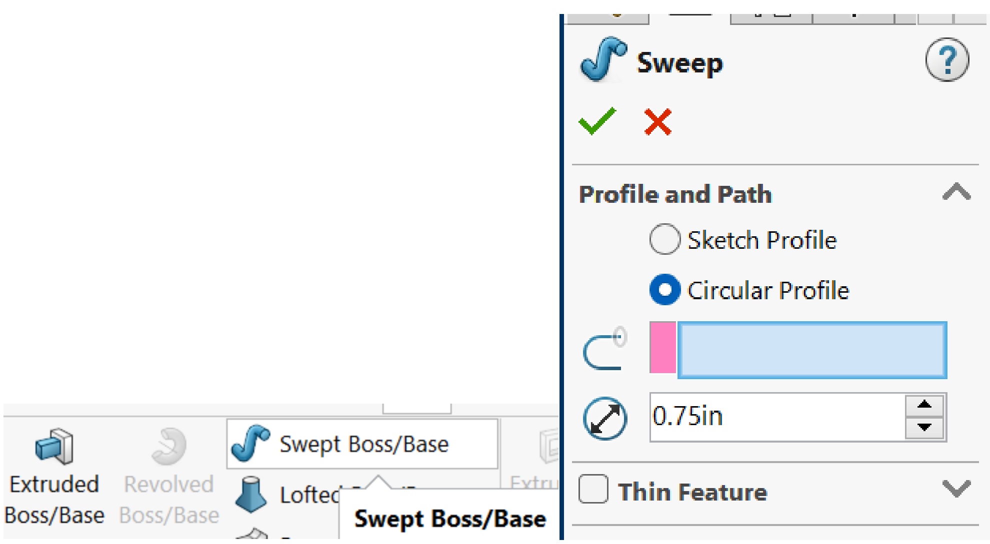
Select > (Fillet) tool.



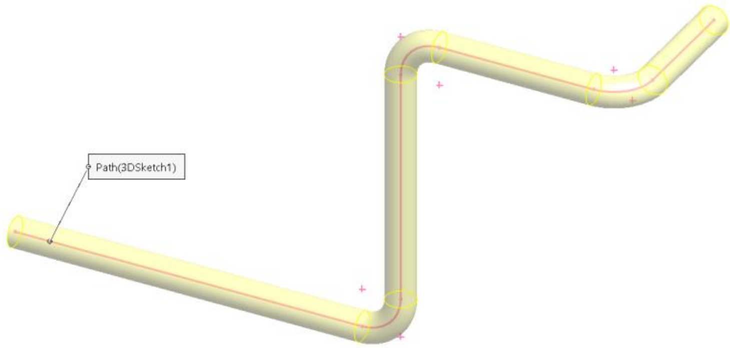
Enter Fillet radius > (1.00) > Pick each corner.



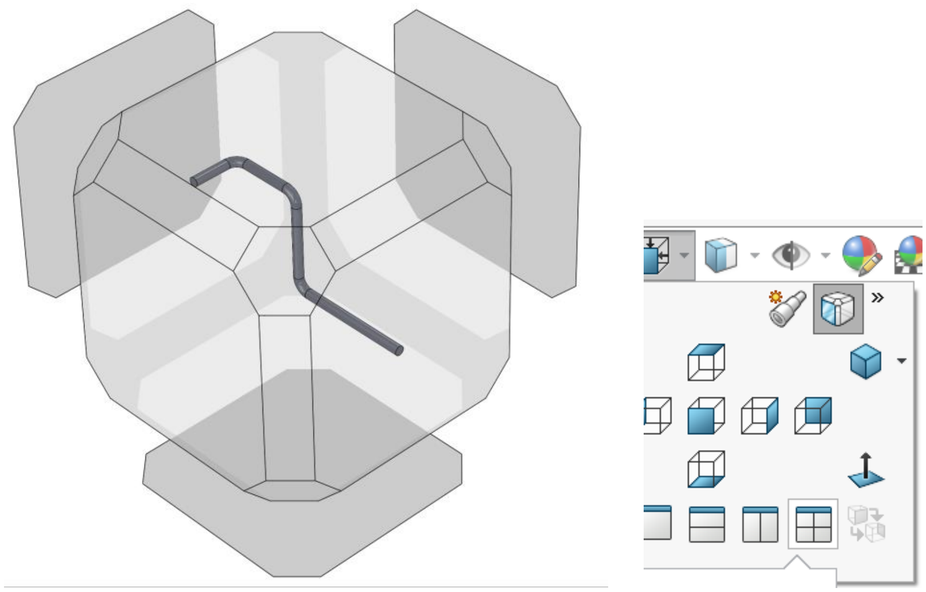
Place Fillet Radius at a corner.



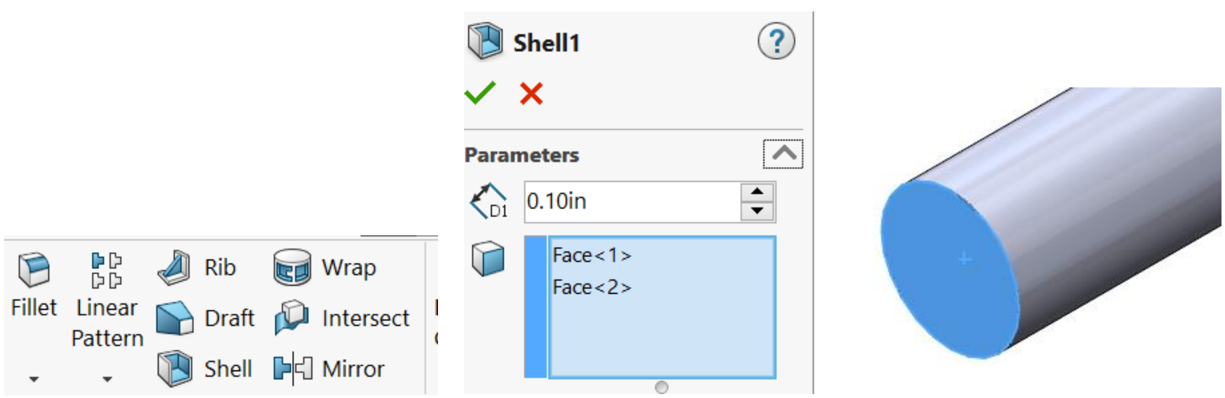
Select > (Sweep Boss Base) > Circular Profile > 0.75 in diameter.



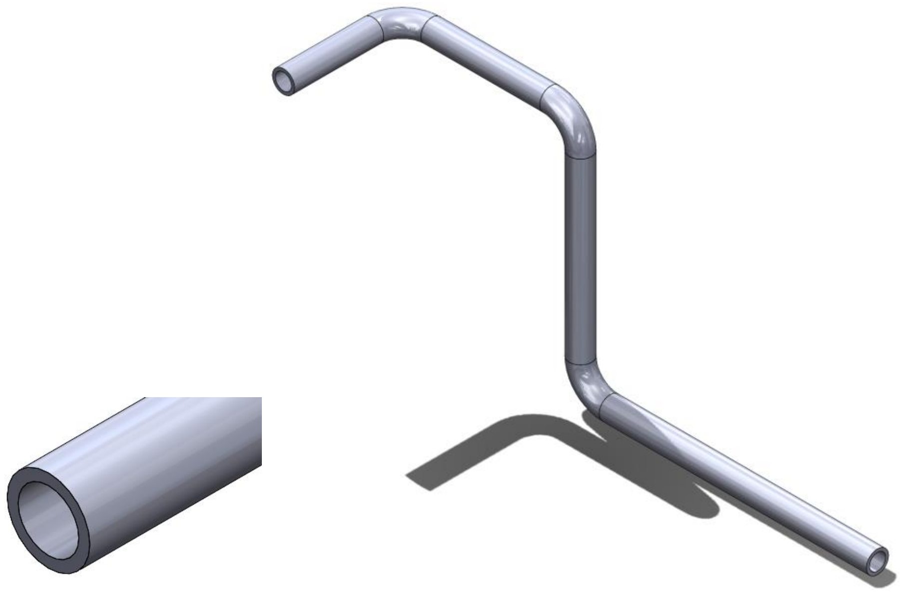
Select the sketched Sweep Line > 3D rod is created > OK.



Select (View Orientation) tool > Pick one corner.



Select (Shell) tool > Enter shell thickness (.25) > Pick each end of rod > OK.



Tube is created.

6 BOTTOM-UP ASSEMBLIES

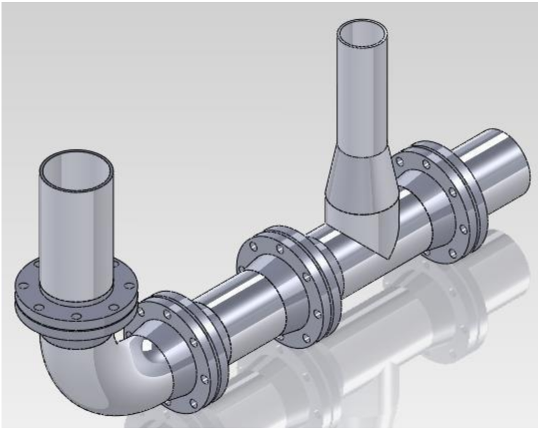
Bottom-up is the traditional method used by CAD operators. Each part is modeled and saved.

Next the individual parts are inserted into an assembly using geometric relations to position them in a subassembly or top assembly.

Insert saved parts and sub-assemblies into SolidWorks then “mate” adjacent parts or sub- assemblies together in a final assembly.

Any changes to a part will need to be done by editing it individually.

This technique is practical to model parts already designed and fabricated, like purchased parts and components (nuts, bolts, bearings, motors, pulleys, etc.), in general, parts that are imported, and which do not change their shape and dimensions.



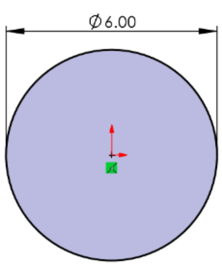
TOP-DOWN ASSEMBLIES

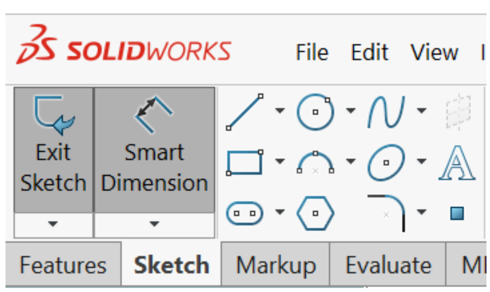
Top-down assemblies were created from parts modeled "inside" the assembly, being related to "driving" entities inside the assembly which control the shape, features, dimensions and position of those parts, in a way that changes introduced to the "driving" entities "drive" the configuration of all the "in-context" modeled parts and therefore the entire assembly.

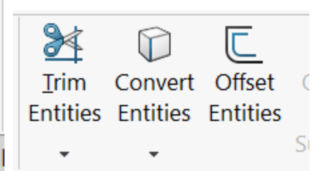
Top-down modeling makes possible the creation of parametric assemblies systems, which cannot be done using the Bottom-up technique alone.

Creating a properly structured Top-down assembly requires more analysis and work that the creation of a Bottom-up model, however, the advantage of top-down modeling for people doing product design is that very little work (and time) will be required when design changes occur, since all parts and components will automatically update to new shapes, dimensions, position, etc. as new input parameters are entered into the "driving" entities at the assembly level.

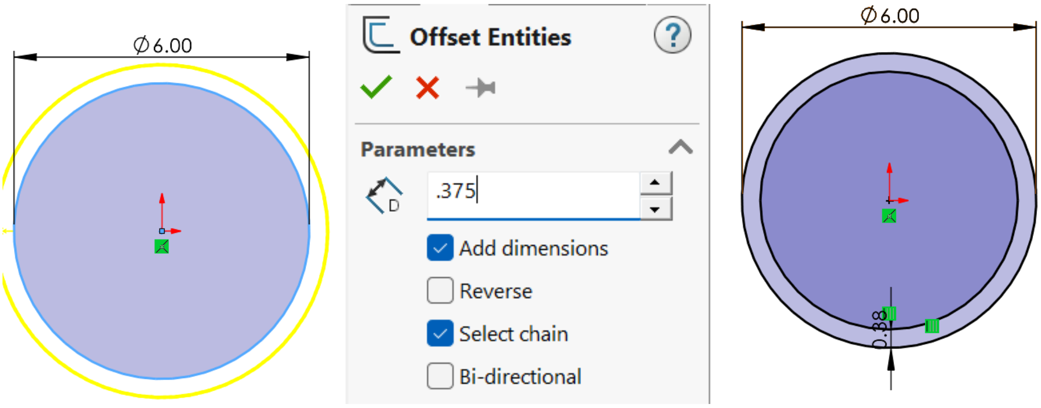
8 EXTRUDE



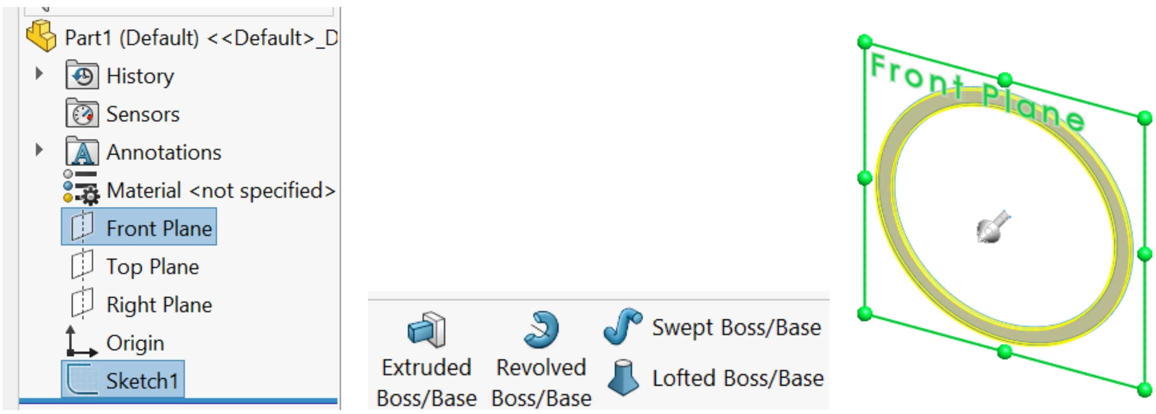




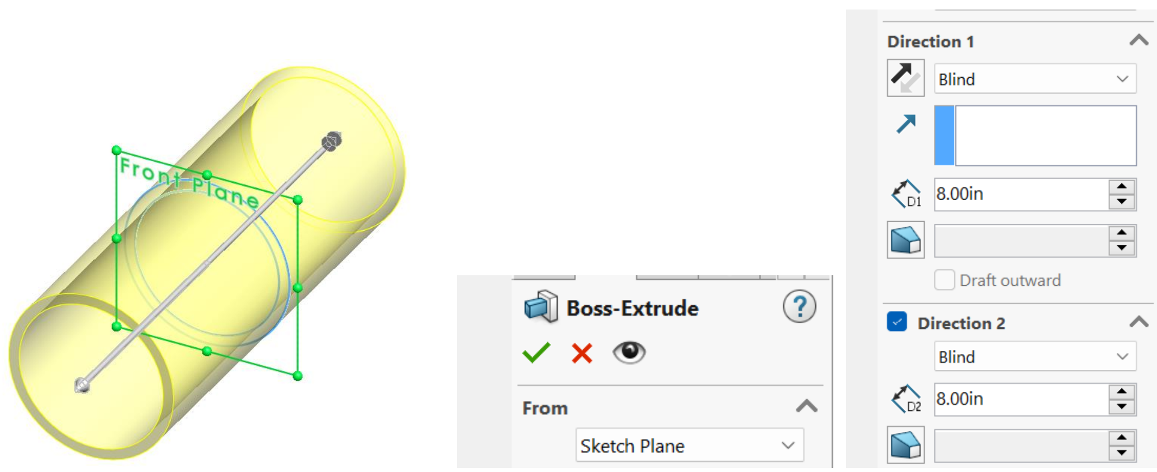
New > Part > OK > Pick Front Plane > Sketch > Circle tool > 6.00inch diameter circle > OK



Select > Offset > .375 > Pick circle > Pick inside circle

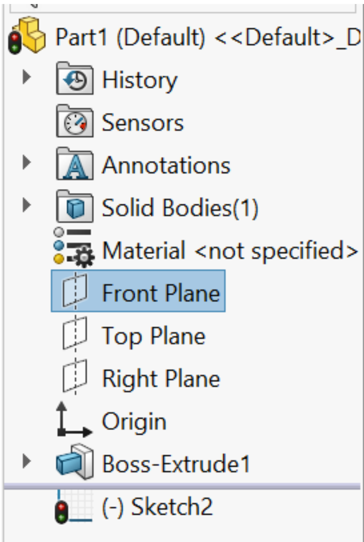


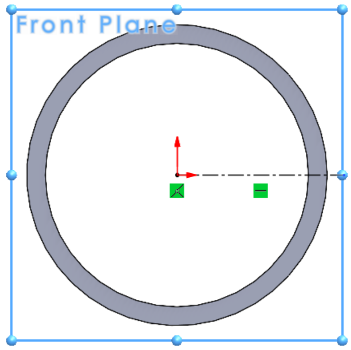
Rebuild > Sketch > Front Plane >

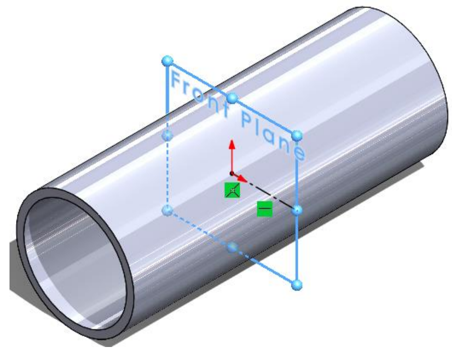


Select > Extruded Boss/Base > Blind > Direction-1 > 8” > Direction-2 > 8” > OK

The pipe extruded in two directions is shown above.





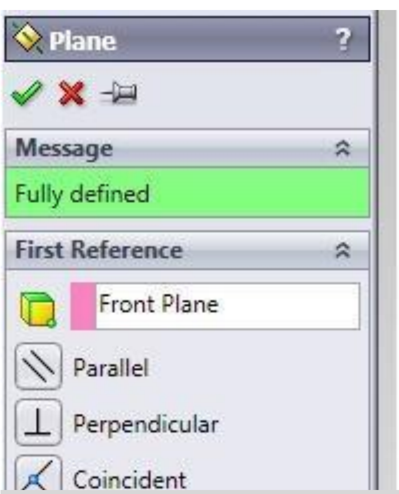


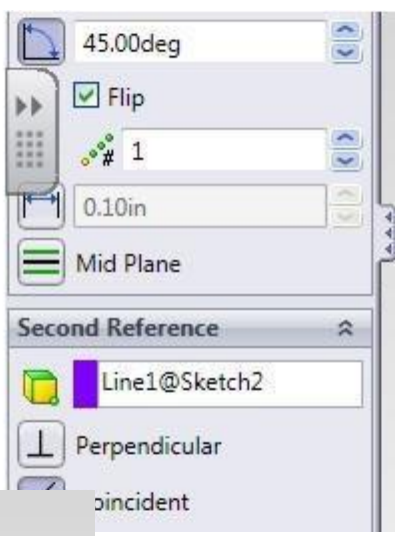
Pick the front plane in the Features Tree > Sketch >

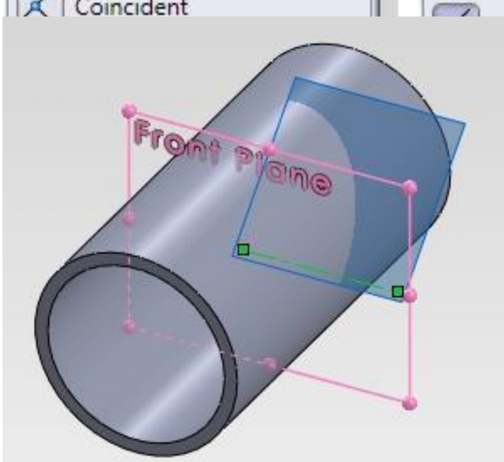
Pick the Origin and drag right to create the horizontal line above >

Exit Sketch > Rebuild

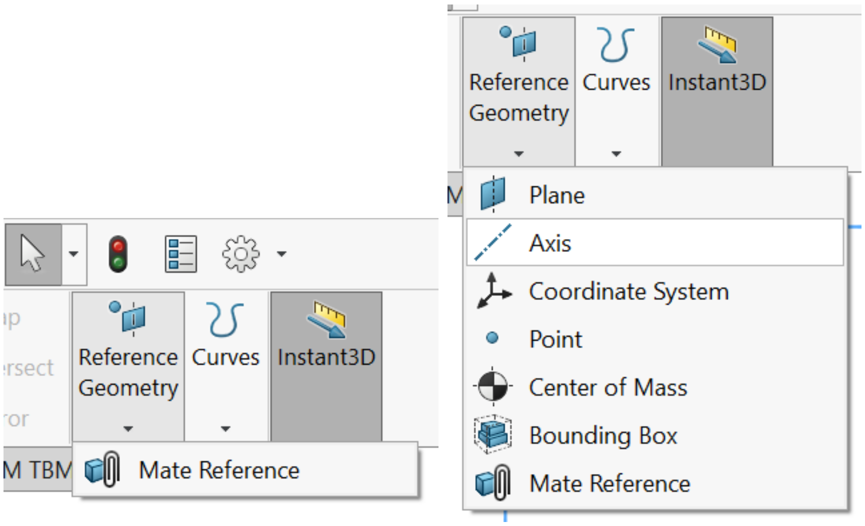
REFERANCE PLANE



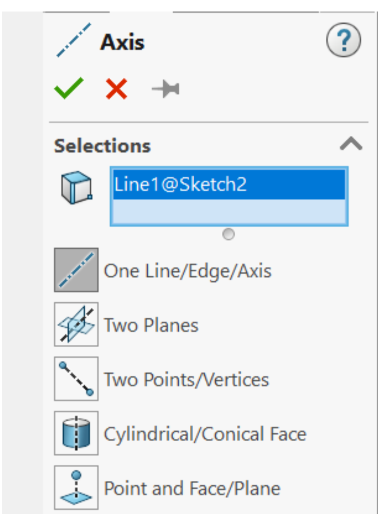


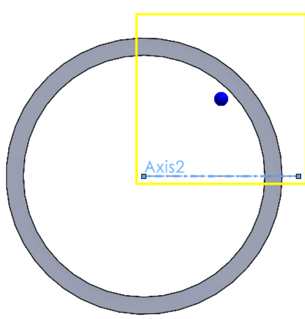


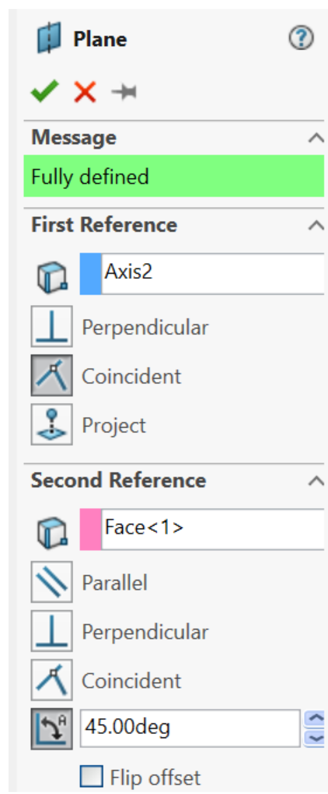
Feature > Reference Plane >



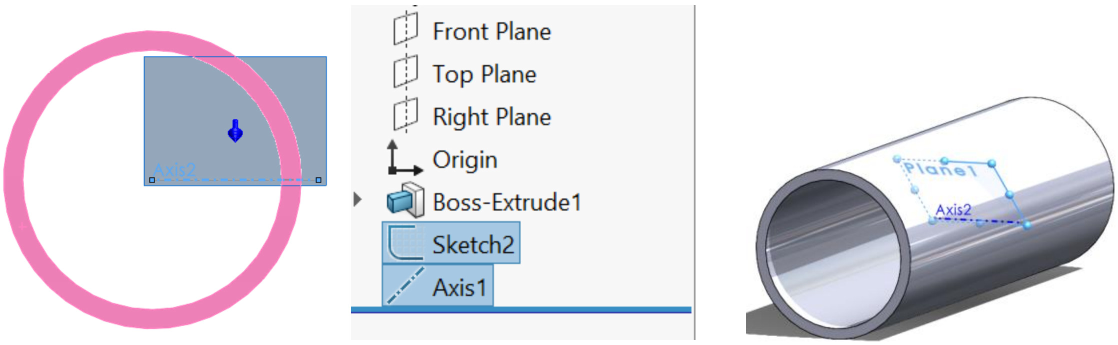
Pick the front plane > Select> Reference Geometry > Axis > Plane First Reference > Front Plane



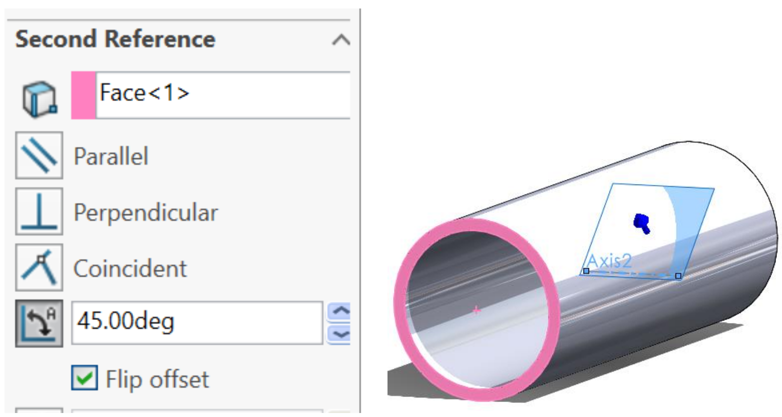




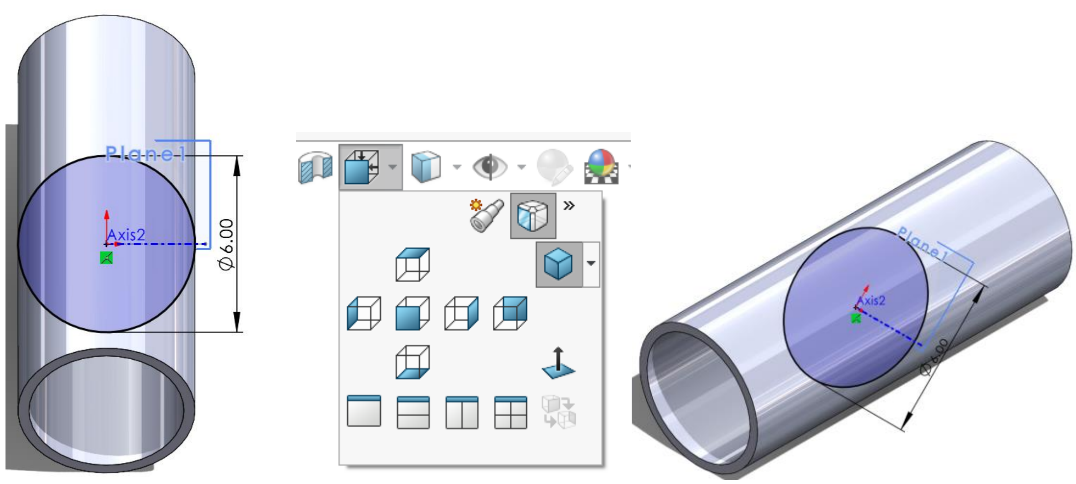
Select > Sketch > Axis > OK **Reference Geometry > Plane >**



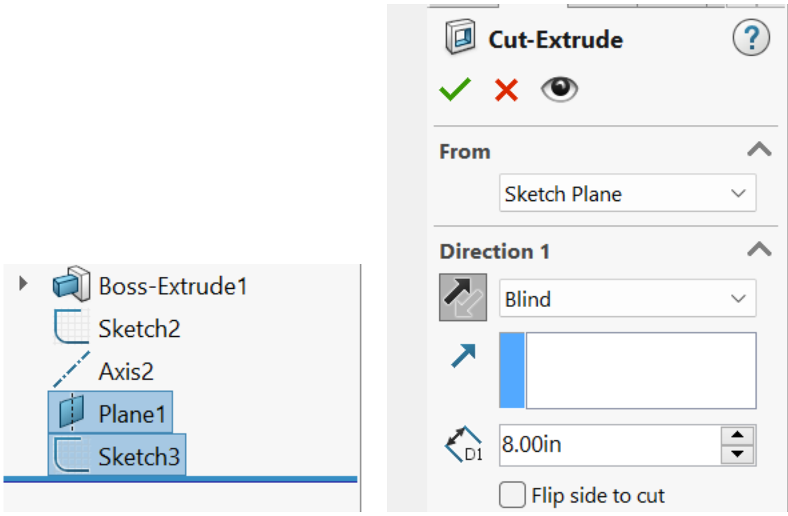
Second Reference > Face<1> Pipe End (Red) > 45.00 deg > OK



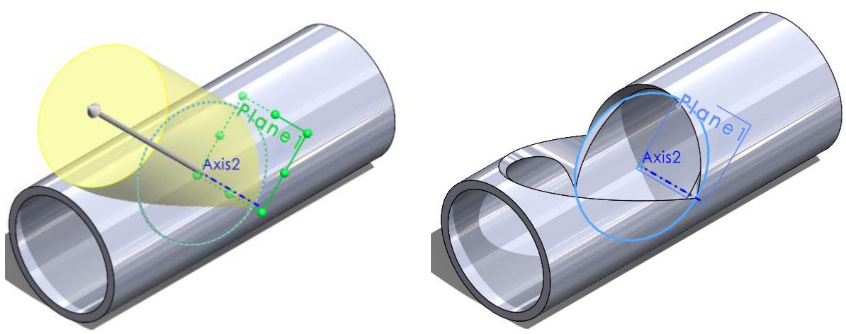
Plane > Edit > Flip Offset > OK



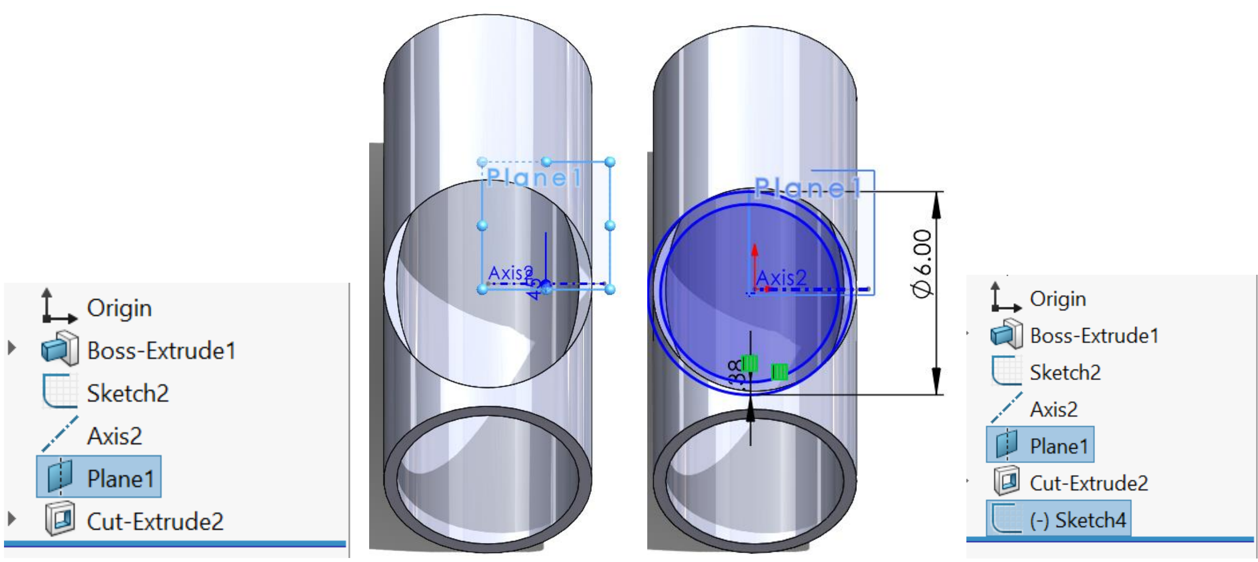
Sketch circle on plane1 > 6.00 Dia. > Offset > 0.375 > Rebuild



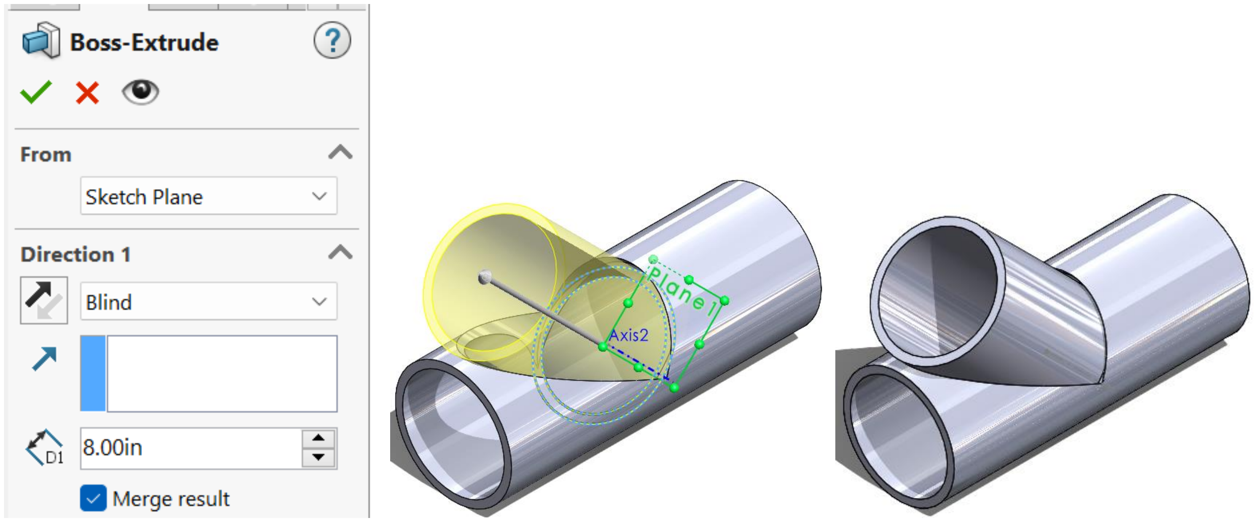
Select > Plane-1 & Sketch 3 > Features > Extruded Cut > 8.00



Extrude Cut is completed.



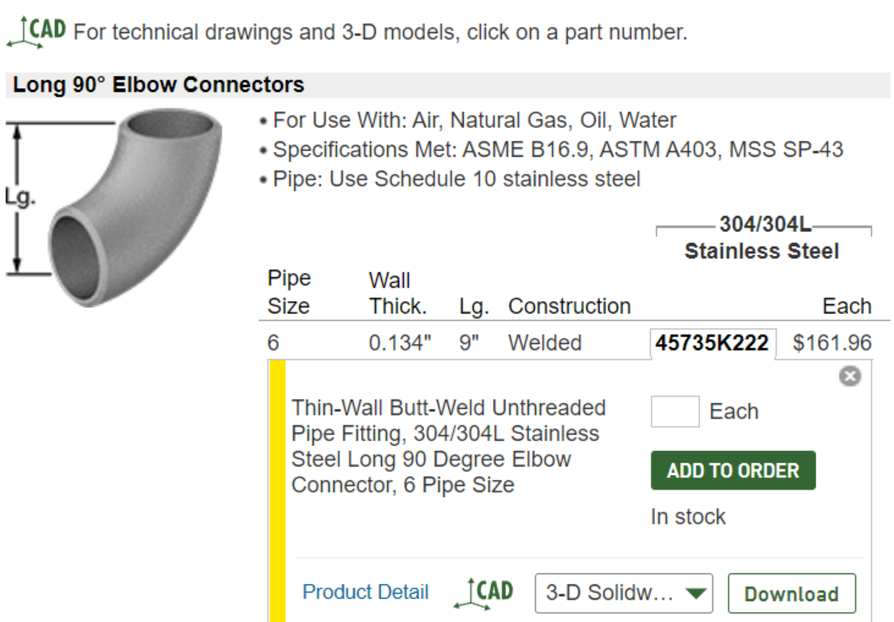
Select > Plane-1 Sketch > Circle > OK
Dimension > 6.00 Offset > .375 > Rebuild
Select > Sketch & Plane-1 >



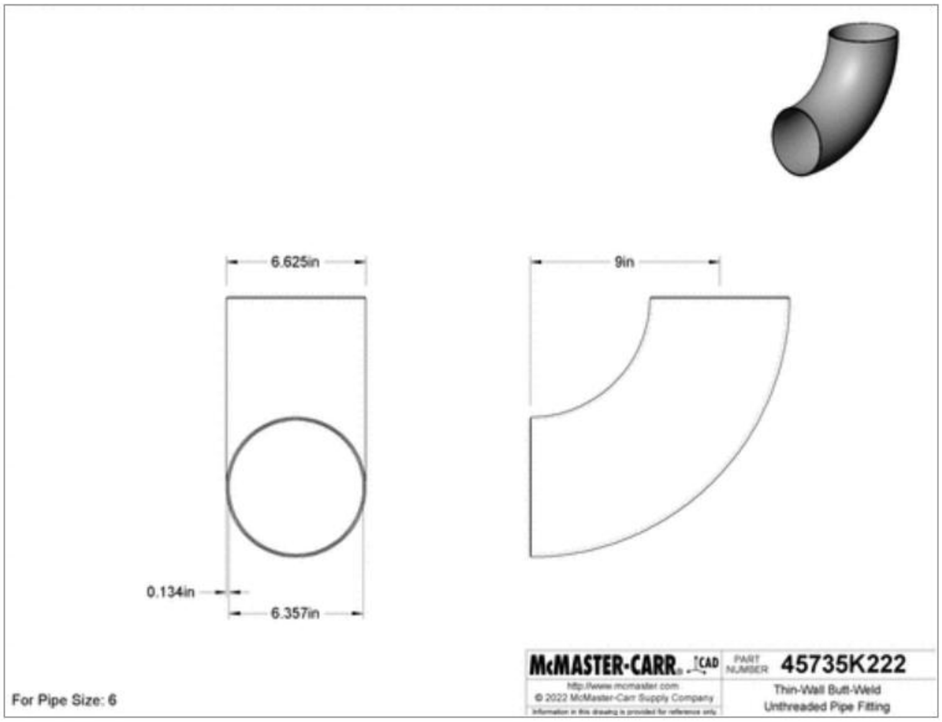
Select > Boss extrude > Blind > 8.00in > OK
Select > Plane-1 > Hide
Select > Axis-2 > Hide

PURCHASED EQUIPMENT

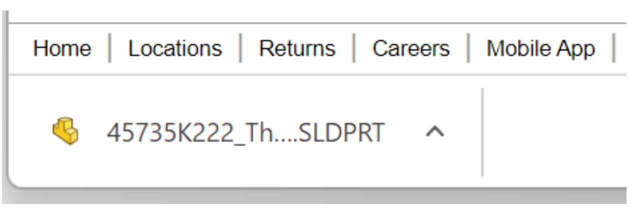
McMaster-Carr web site has 3D SolidWorks models of thousands of equipment items.
www.mcmaster.com > Pipe Fittings > Scroll down to (Butt Welded Fittings)
Open SolidWorks



Select > Download > Drawing will open in SolidWorks



Product Detail in McMaster-Carr.



MACHINE GUARD with GAS SPRINGS

GAS SPRING

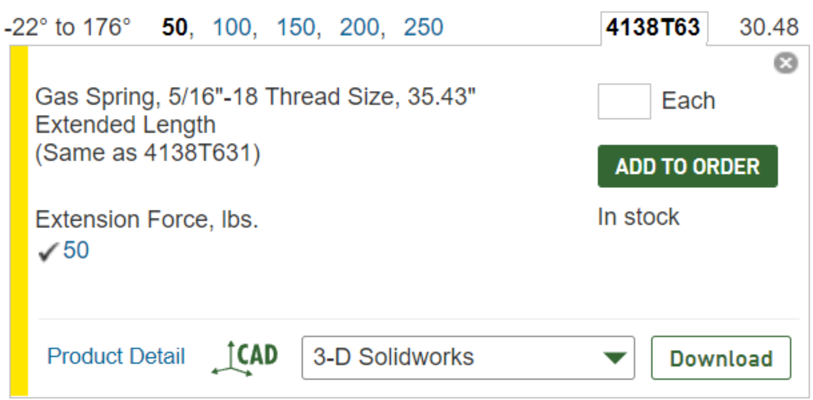
GUARD CLOSED

GUARD OPEN

Open SolidWorks

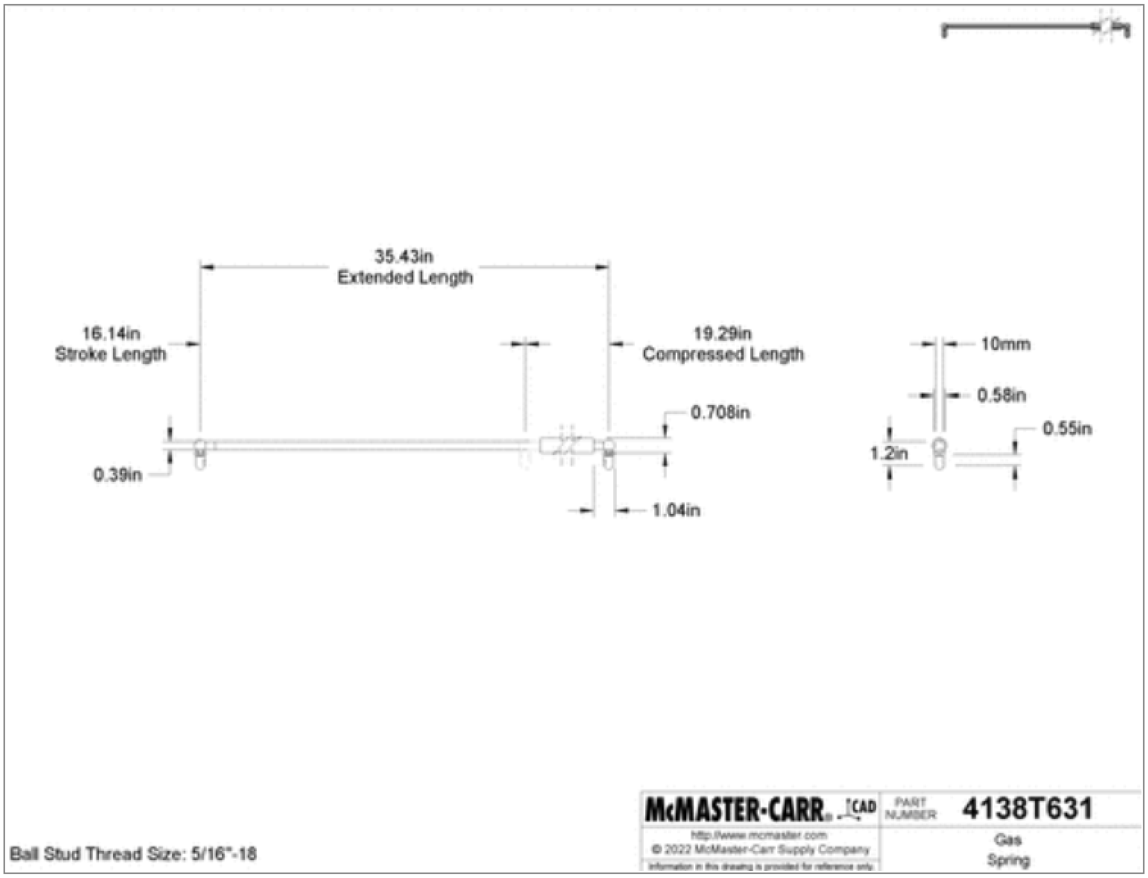
Go to > www.macmaster.com

McMaster-Carr 4138T631
Select > 4138T631_Gas Spring 50 LB FORCE – BODY



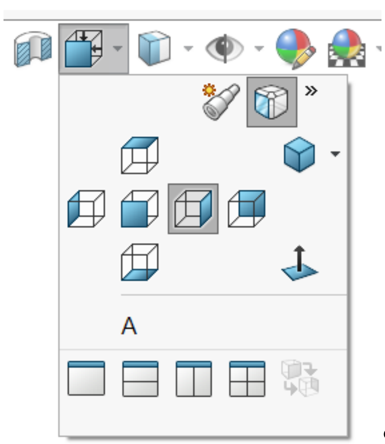
Select > 22” to 176” > 4138T63 > Part description and cost.

The Gas Spring 3D model is one part.
It will not retract or extend.

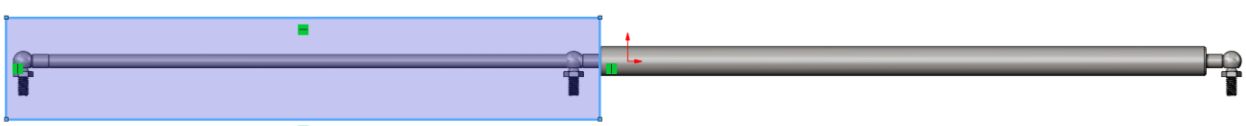


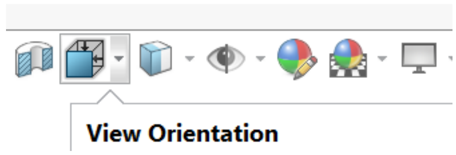
Part Detail in McMaster-Carr.

Select > Gas Spring > Download > The part will open in SolidWorks

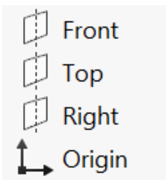


Select > Side View.

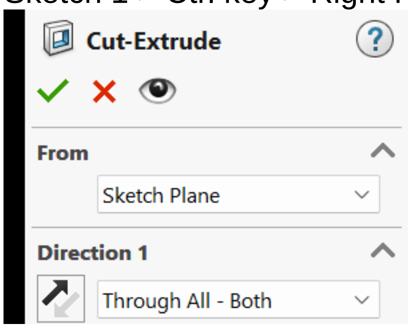




Select > View Orientation

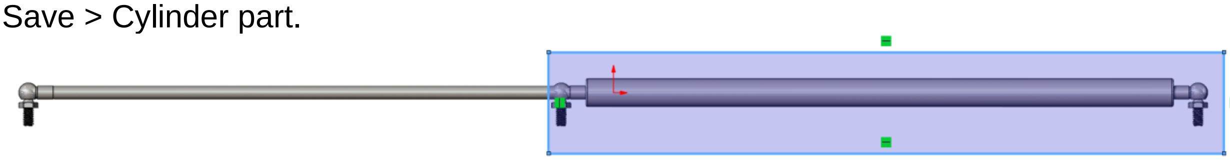


Pick > Right Plane > Sketch Rectangle > OK > Rebuild > Sketch 1 > Ctrl key > Right Plane >



Features > Extruded Cut > Through All – Both > OK

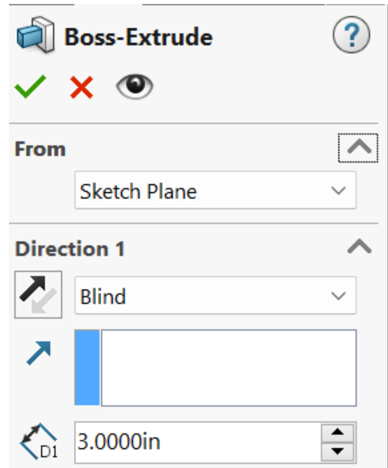
Save > Cylinder part.



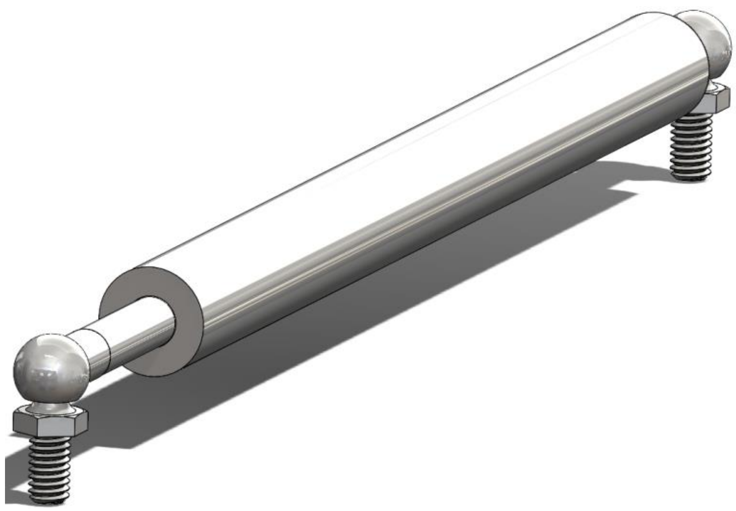
Features > Extrude Cut > Through All – Both > OK

Save > Cylinder Rod part.

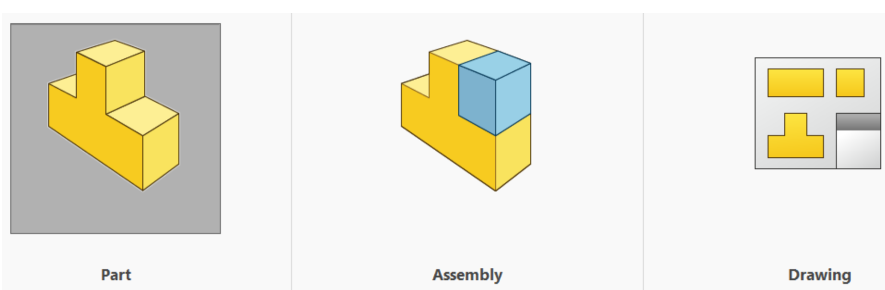
Select Rod cut > Sketch > Circle (Diameter equals rod diameter) > OK > Rebuild.



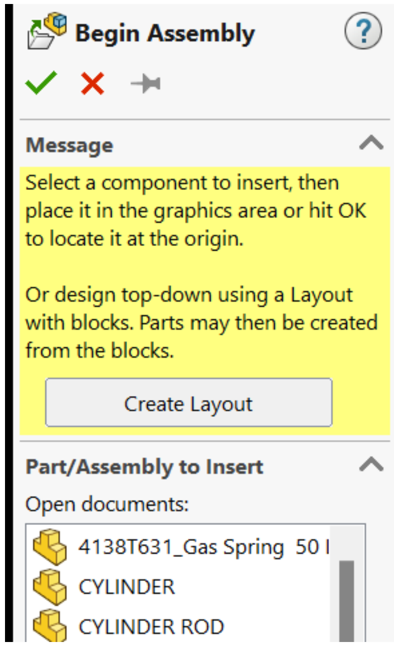
Select > Circle Sketch on rod end > Ctrl key > Pick rod end
Features > Extrude Boss/Base > Blind > 3.00 in > OK
Rod extension is added.



Assemble > Cylinder and Rod. **First object in an assemble will be anchored.**

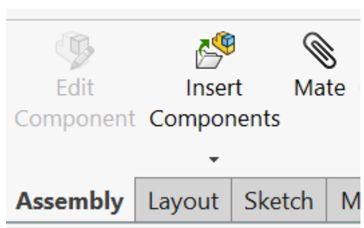


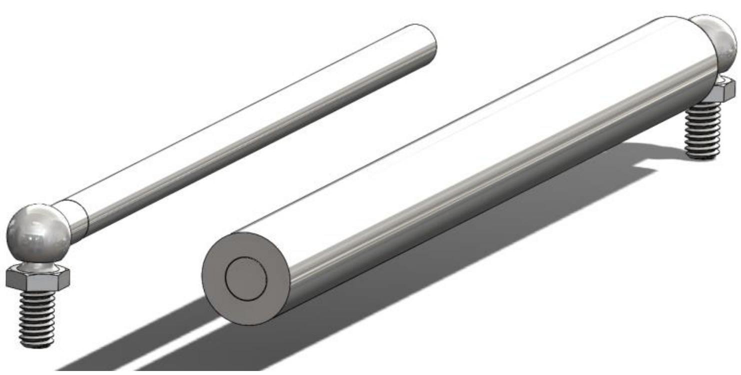
Assembly > OK



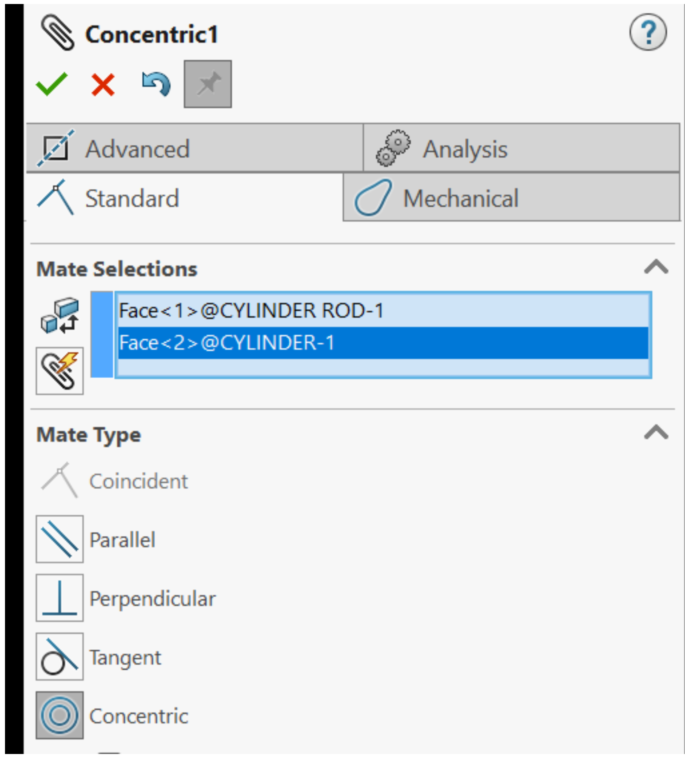
Select > Cylinder > Drag into drawing area > **First object will be anchored**.

Select > Rod > Drag into drawing area.

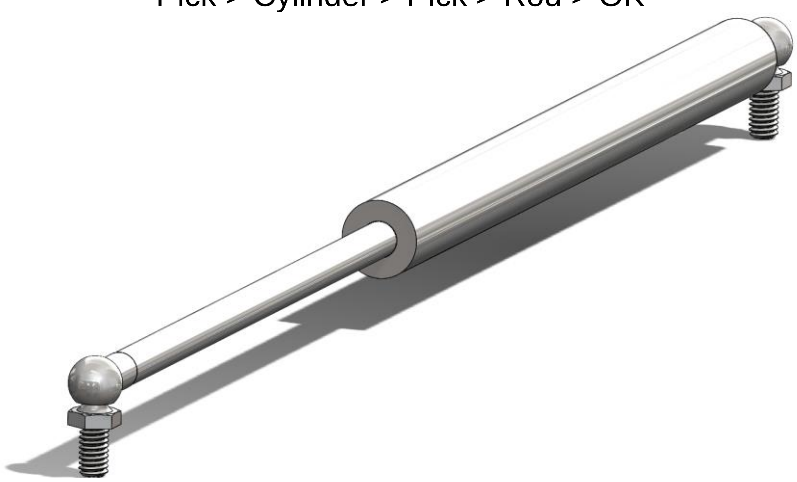




Select > Assembly > Mate > Concentric >

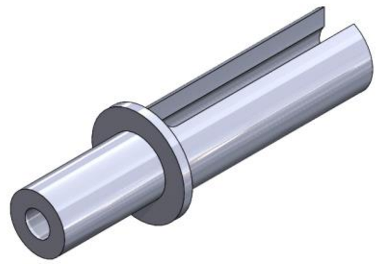


Pick > Cylinder > Pick > Rod > OK

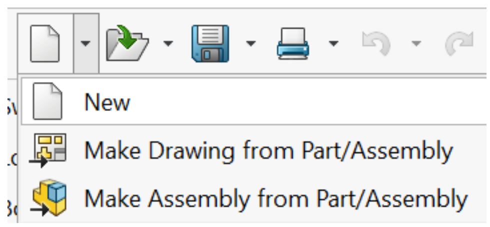


Rod will slide in cylinder.

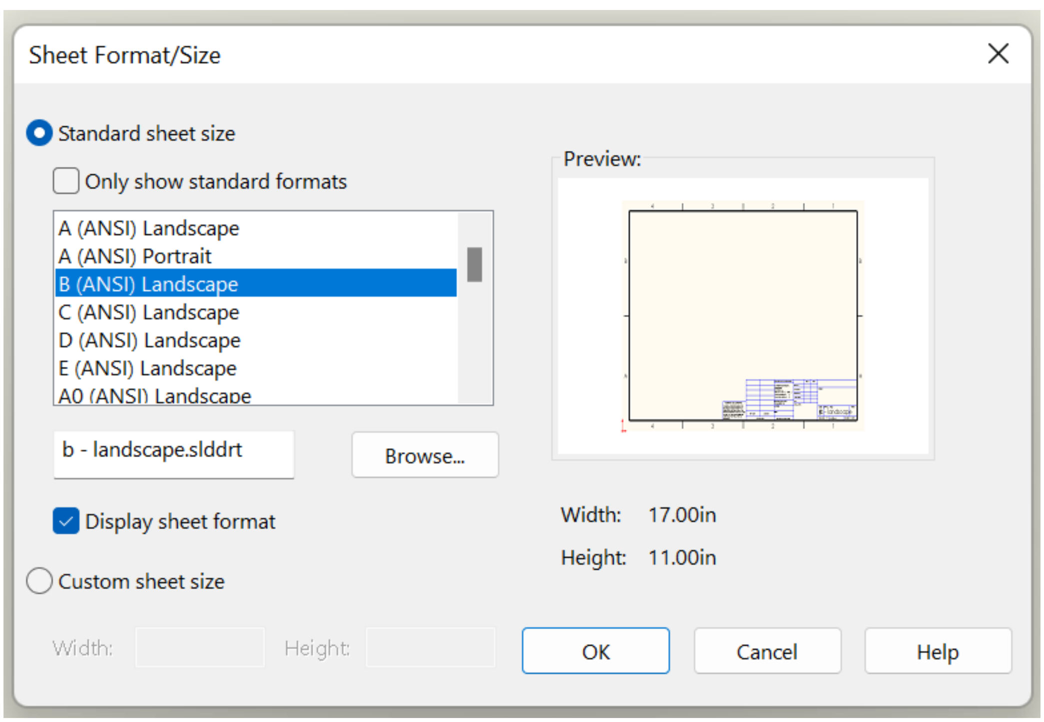
12 DRAWING



3D PART

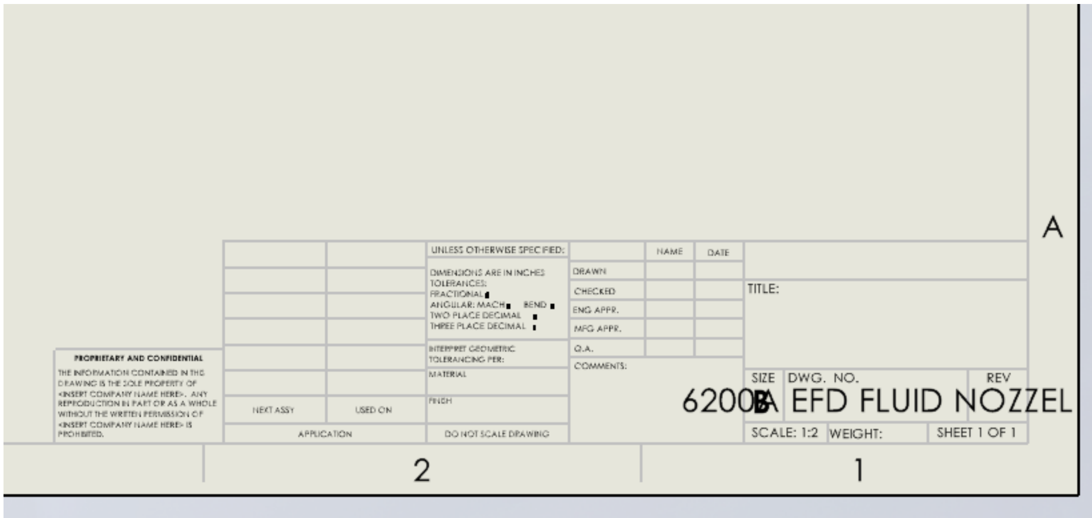


Select > New > Drawing

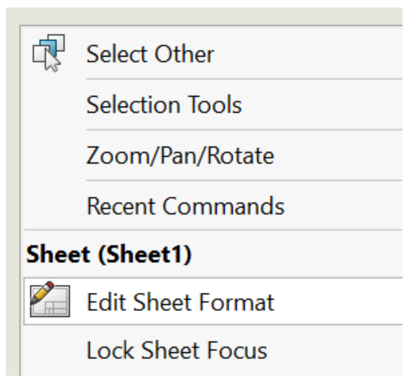


Uncheck > Only show standard formats

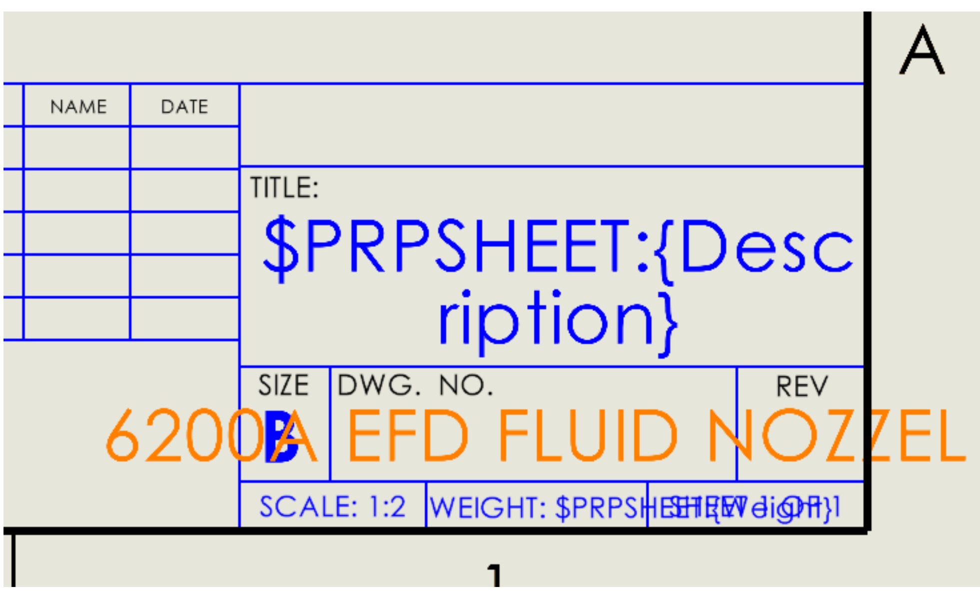
Select B(ANSI) Landscape (11" X 14")
OK

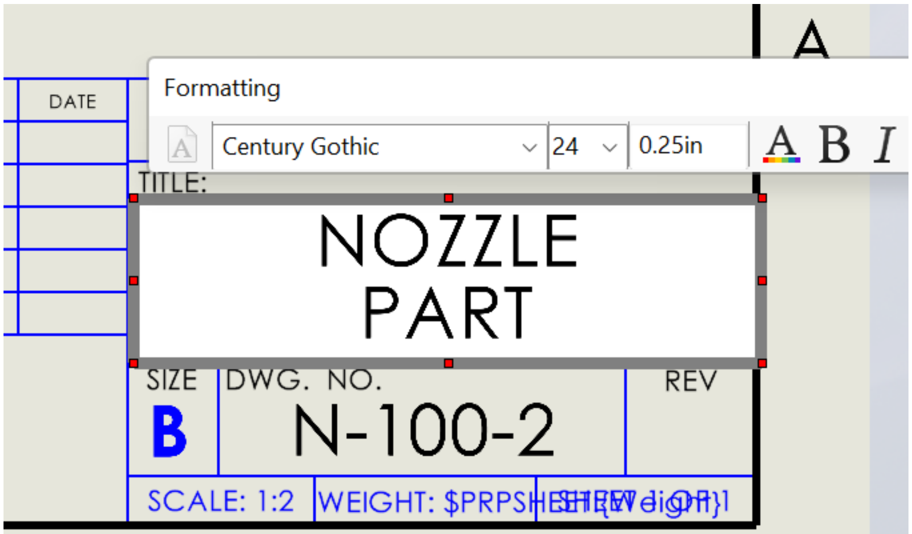


Edit Drawing Title

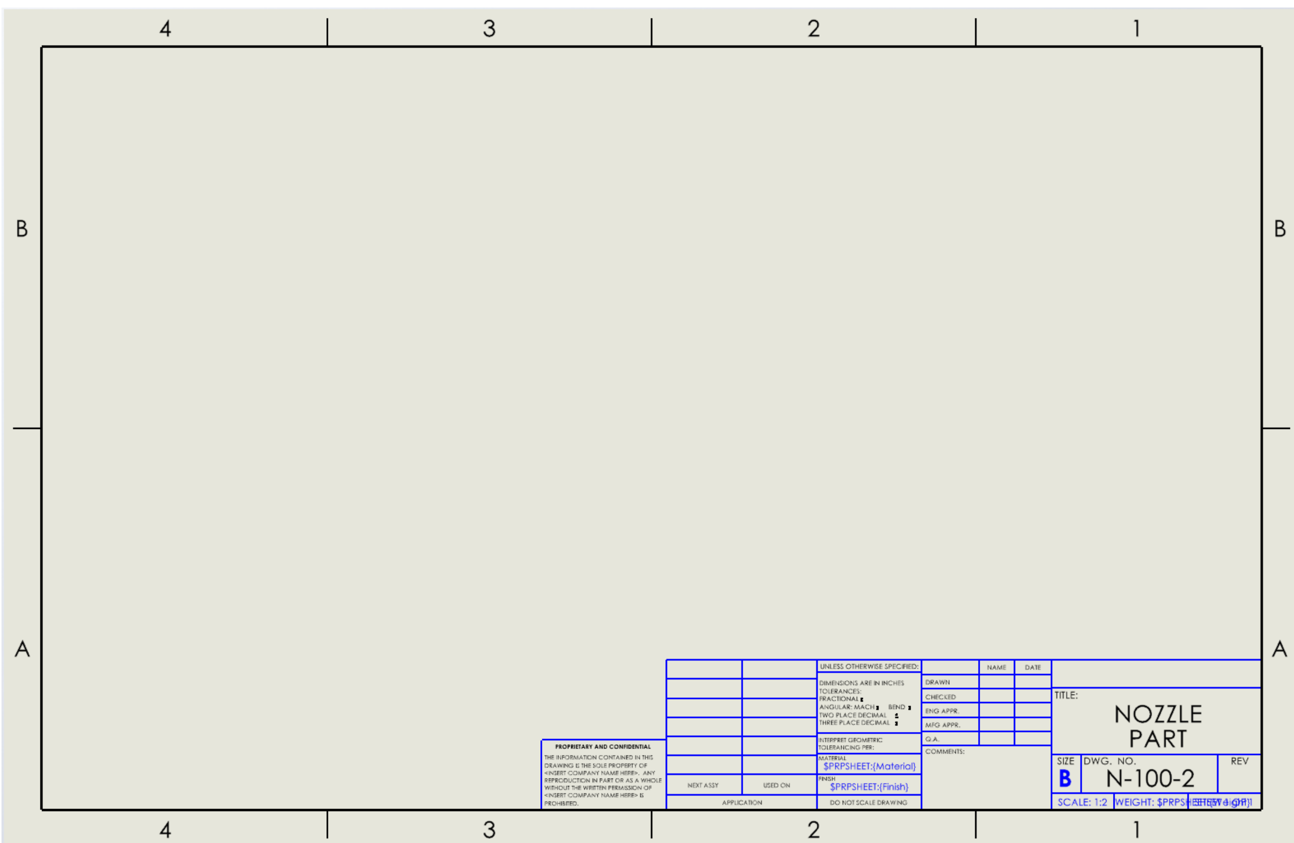


Right click on drawing > Edit Sheet Format.



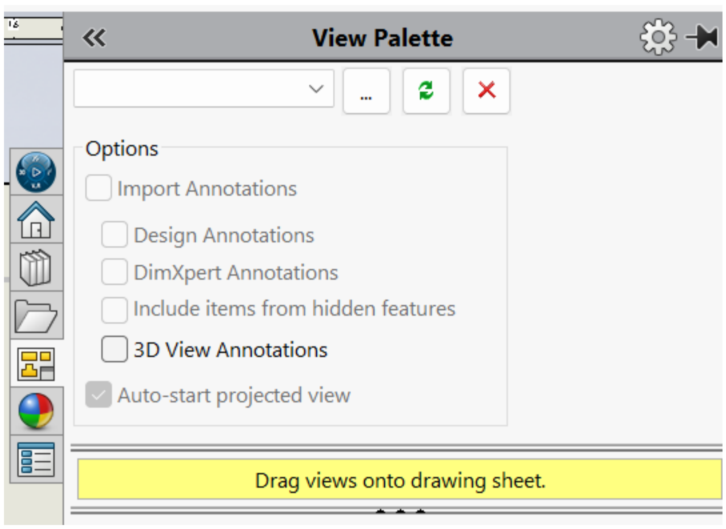


Type drawing title and drawing number.

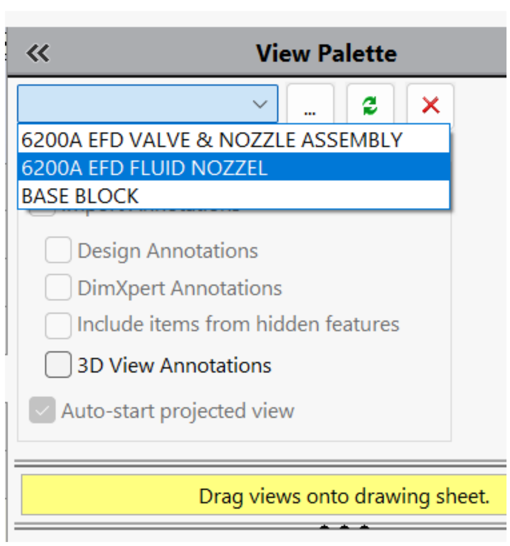


Empty drawing sheet

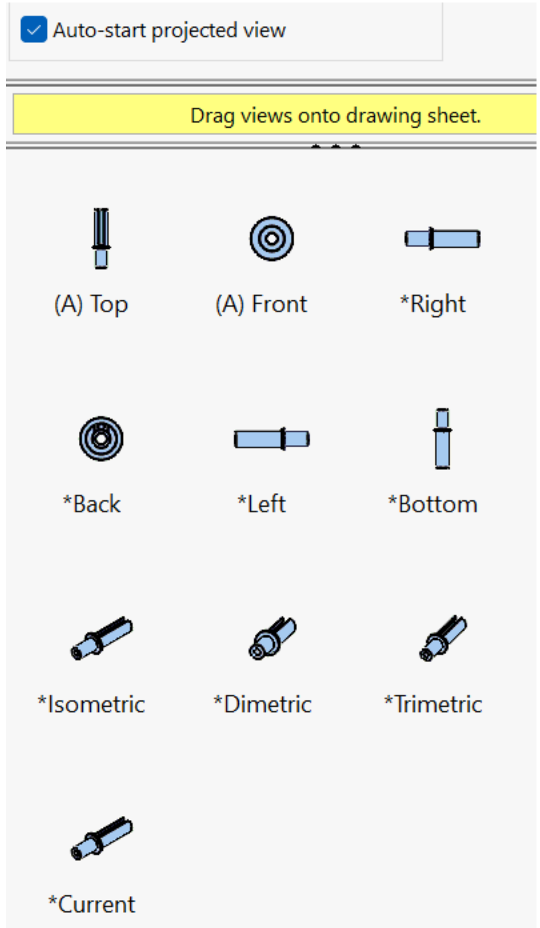
Select > View Pallet (Display – right – top)



Select Drop-down list of parts and assemblies open in SolidWrks.

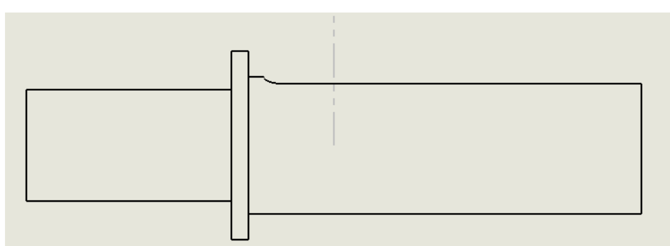


Select > 6200A EFD FLUID NOZZLE.

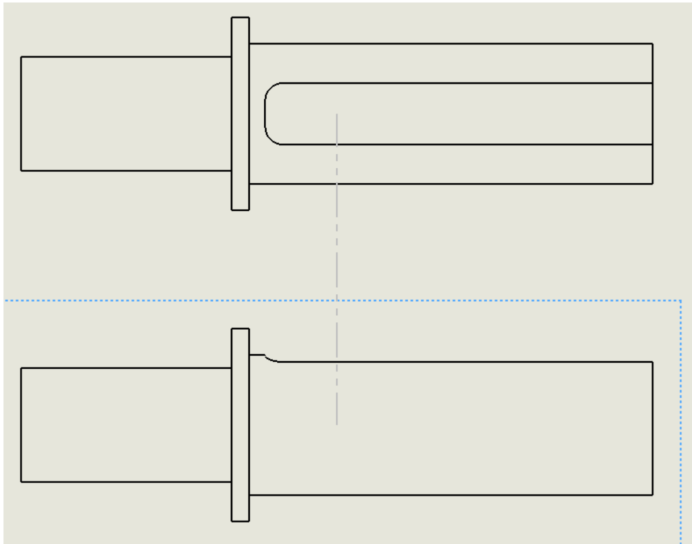


Drag views onto drawing sheet menu.

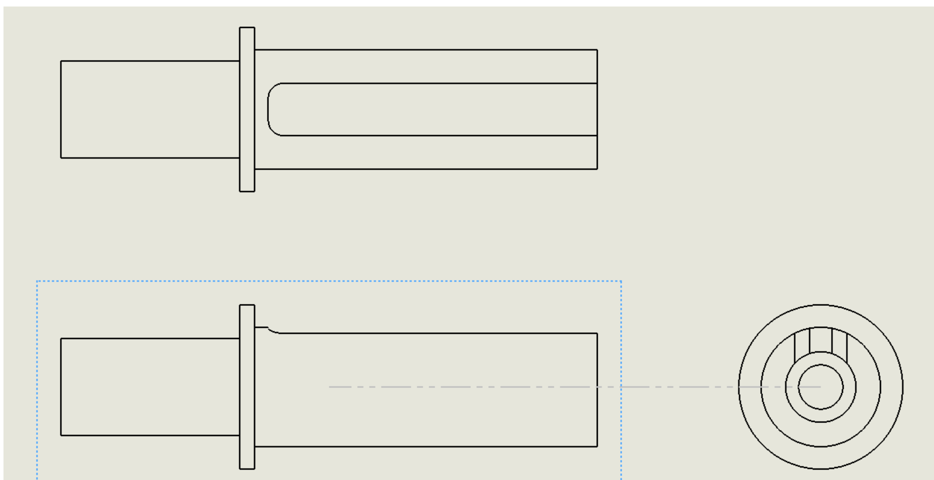
Select > Right



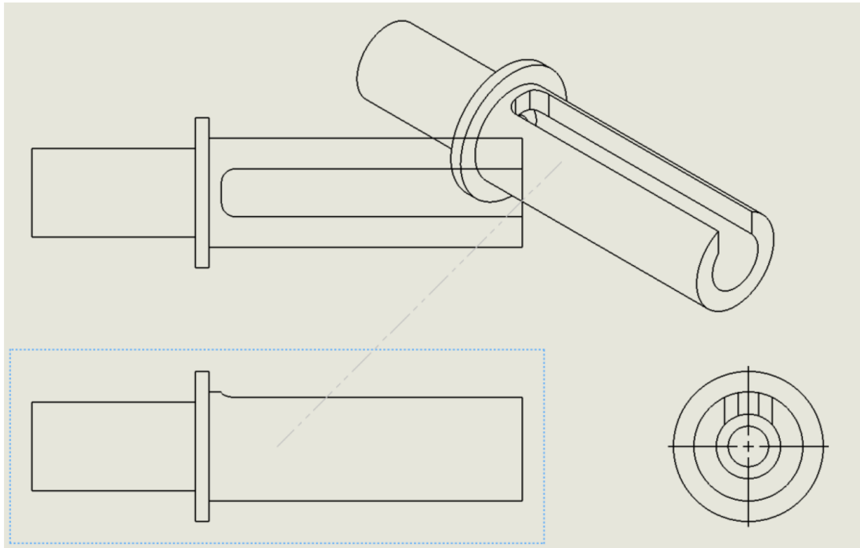
Place Right View in drawing.



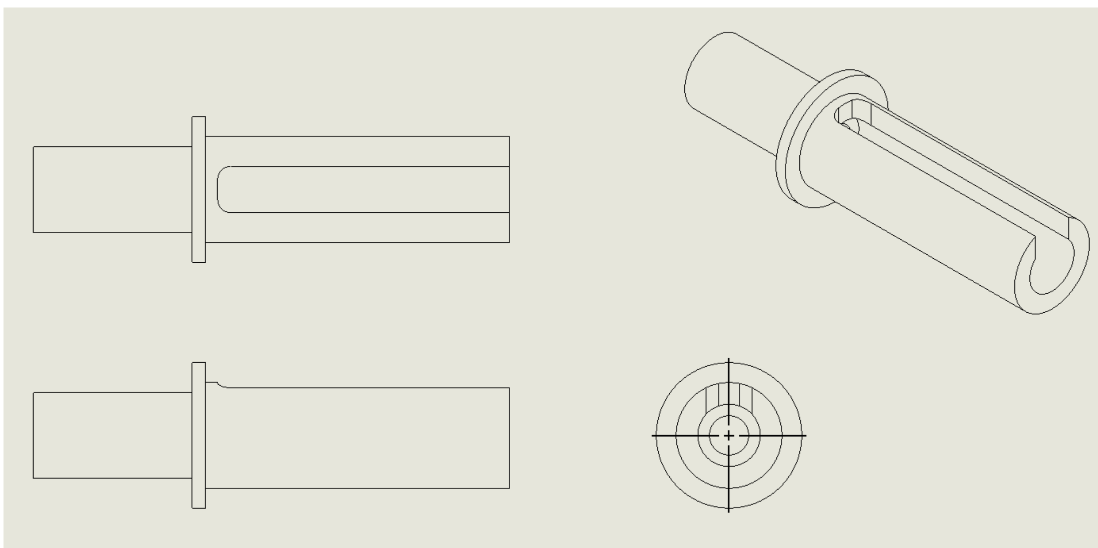
Select > Right View > Drag up > Place Top View.



Select > Right View > Drag right > Place End View.

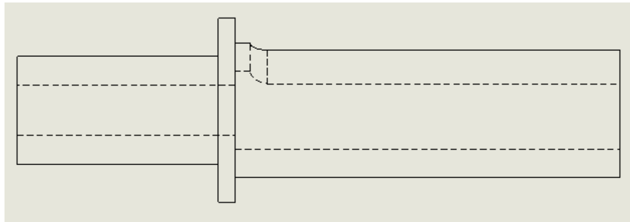


Select > Right View > Drag up at an angle for 3D View

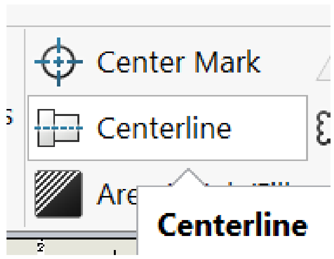


Place 3D View

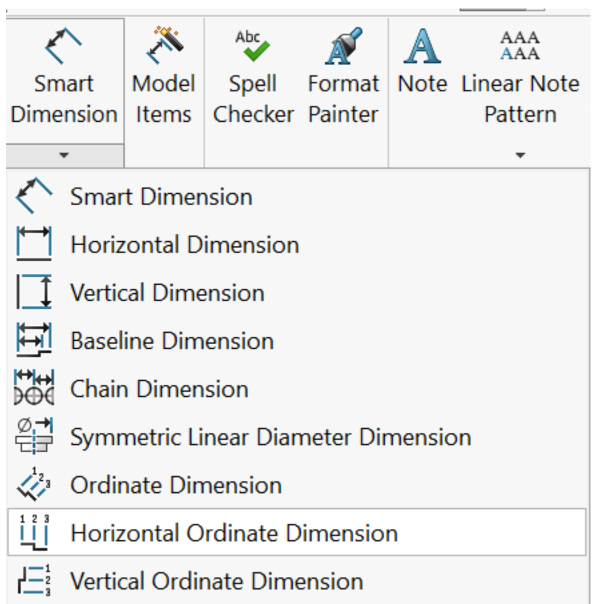
Select > Hidden Lines Visible



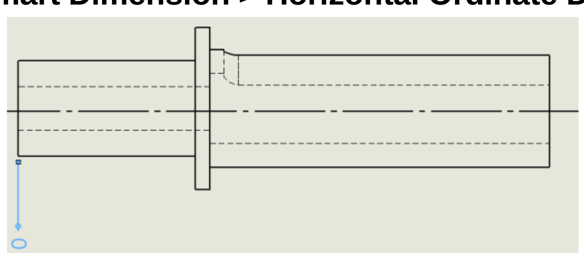
Pick > Right View



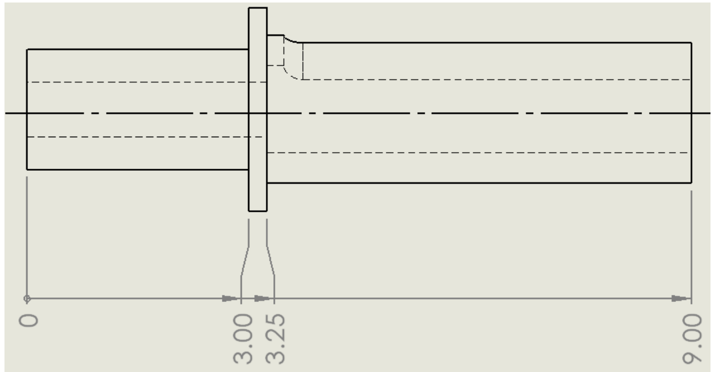
Annotations > Centerline > Pick two sides of part.
Pick end of centerline > Drag to extend.



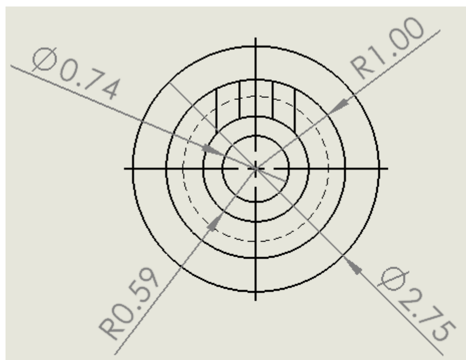
Select > Smart Dimension > Horizontal Ordinate Dimensions



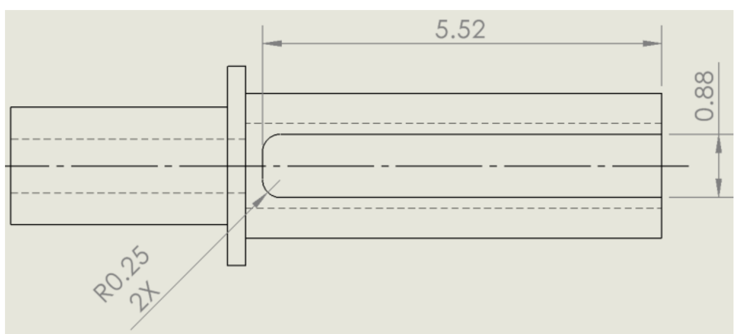
Pick left end of part > Drag down zero dimension.at left end.



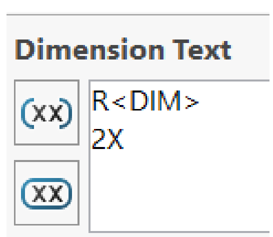
Pick lines to be dimensioned.



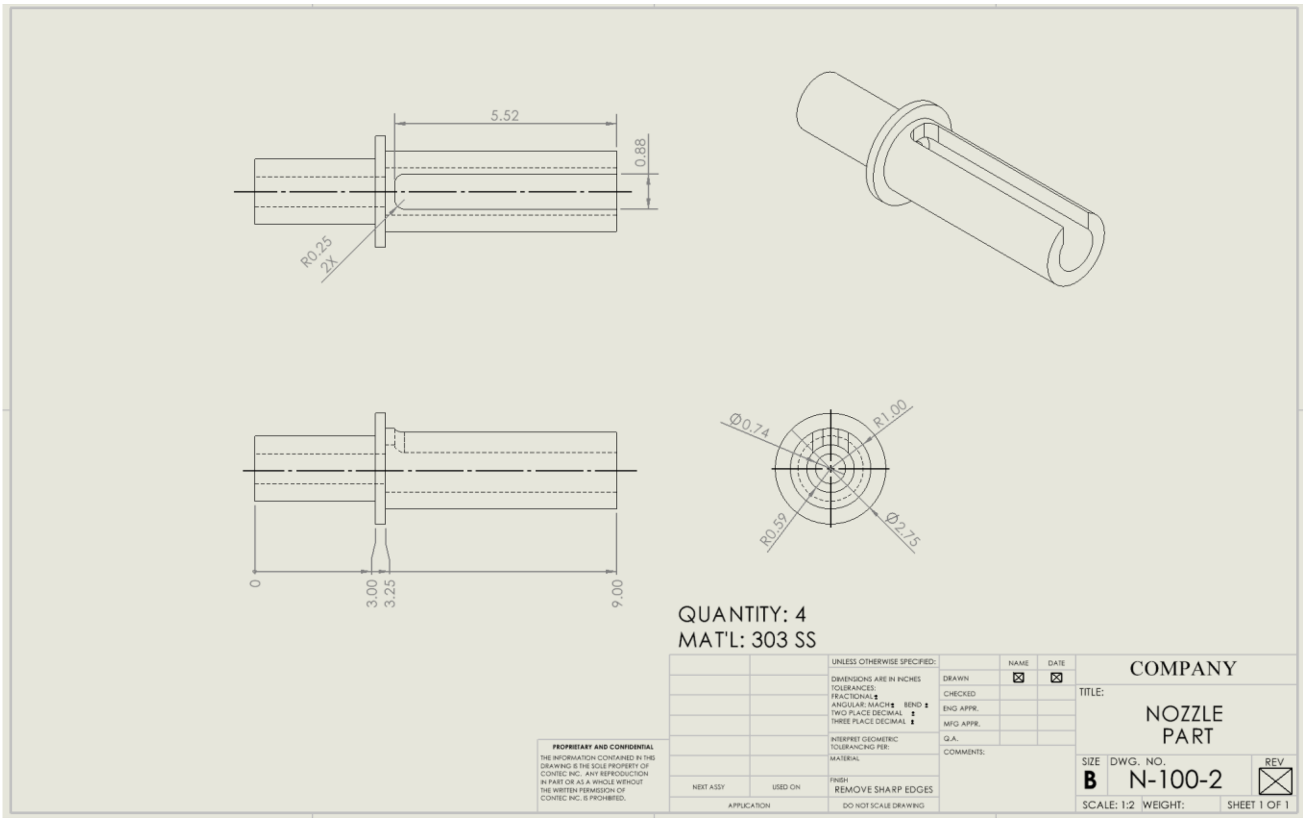
Smart Dimension > Pick circles.



Smart Dimension > Place dimensions.



Edit Dimensions with > Dimension text.

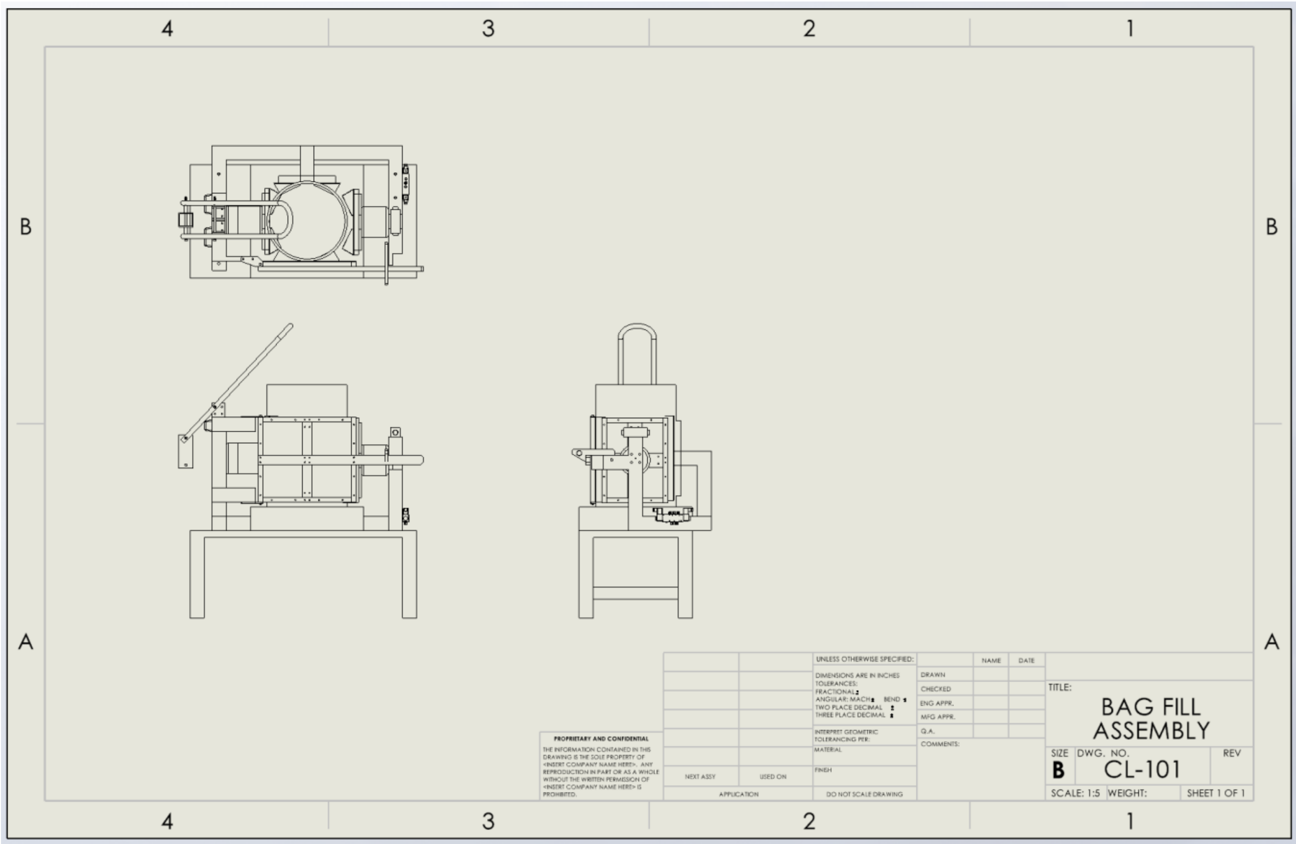


Completed Drawing with dimensions, Quantity and Material

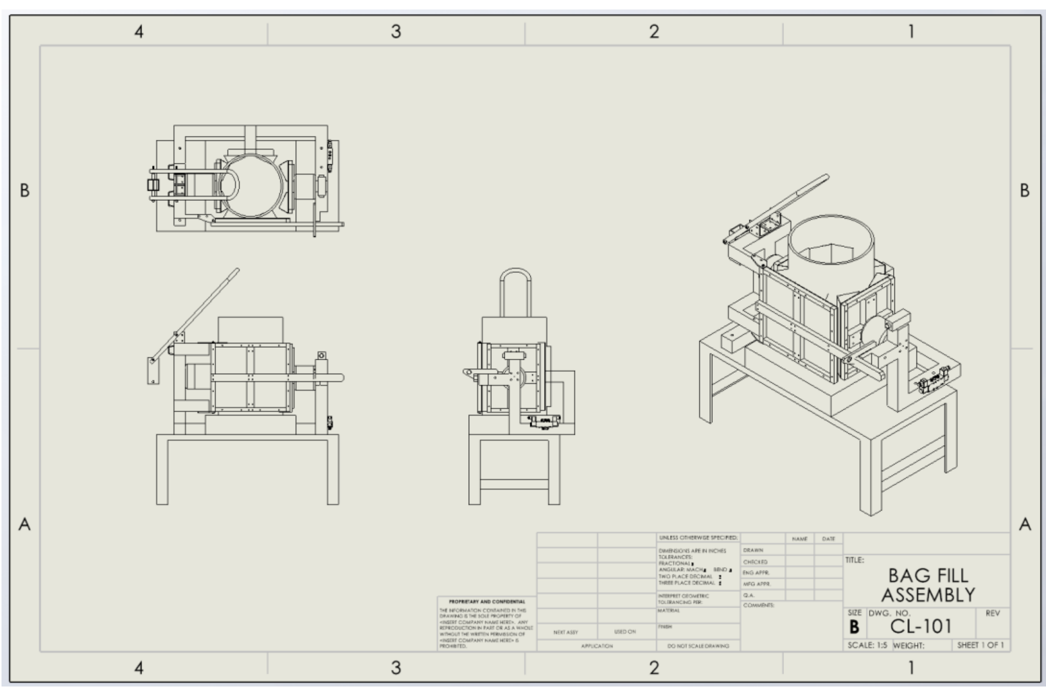
13 BILL OF MATERIALS

ITEM NO.	PART NUMBER	QTY.
1	28500 AIR BAG FRAME	1
2	28500 AIR BAG AND WALL	2
3	285800-B WEIGH SCALE	1
4	AIR BAG WALL PLATE	1
5	28500AIR SPRING EXTENDED 4_3 INCHES	1

Assembly Drawing



B size drawing with standard 3-view drawing of the assembly



Three View Drawing with Isometric View

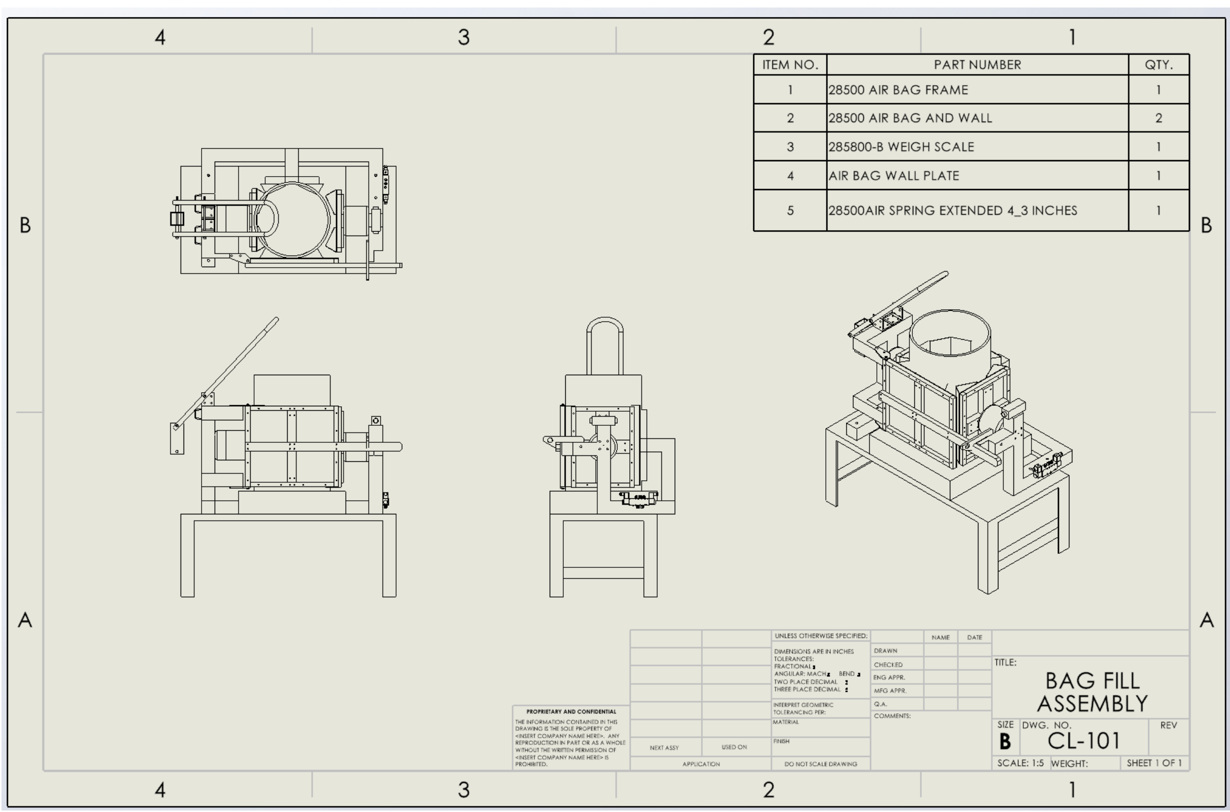
Select > Isometric View

Select > Isometric View > Insert > Tables > Bill of Materials > OK

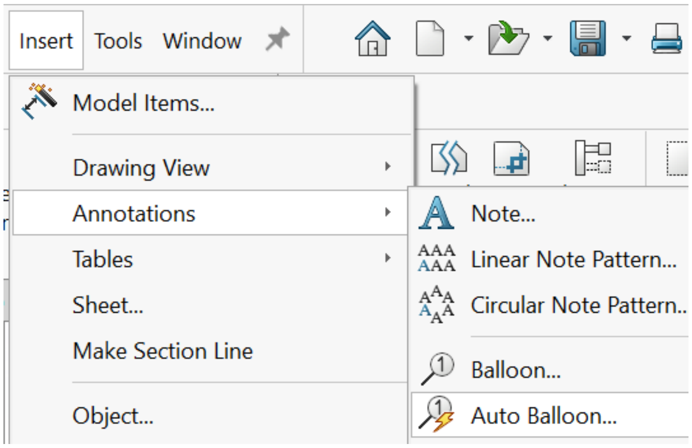
Select > C column > Delete Key

ITEM NO.	PART NUMBER	QTY.
1	28500 AIR BAG FRAME	1
2	28500 AIR BAG AND WALL	2
3	285800-B WEIGH SCALE	1
4	AIR BAG WALL PLATE	1
5	28500AIR SPRING EXTENDED 4_3 INCHES	1

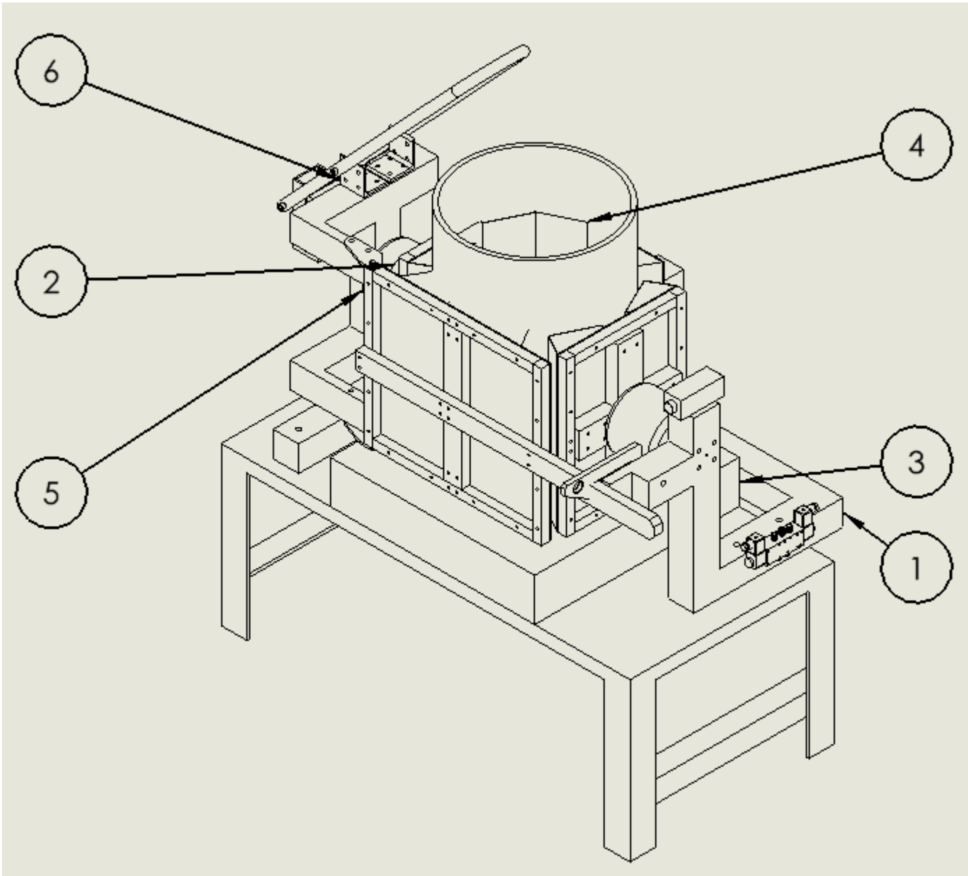
Drag > Bill of Materials to Drawing Location



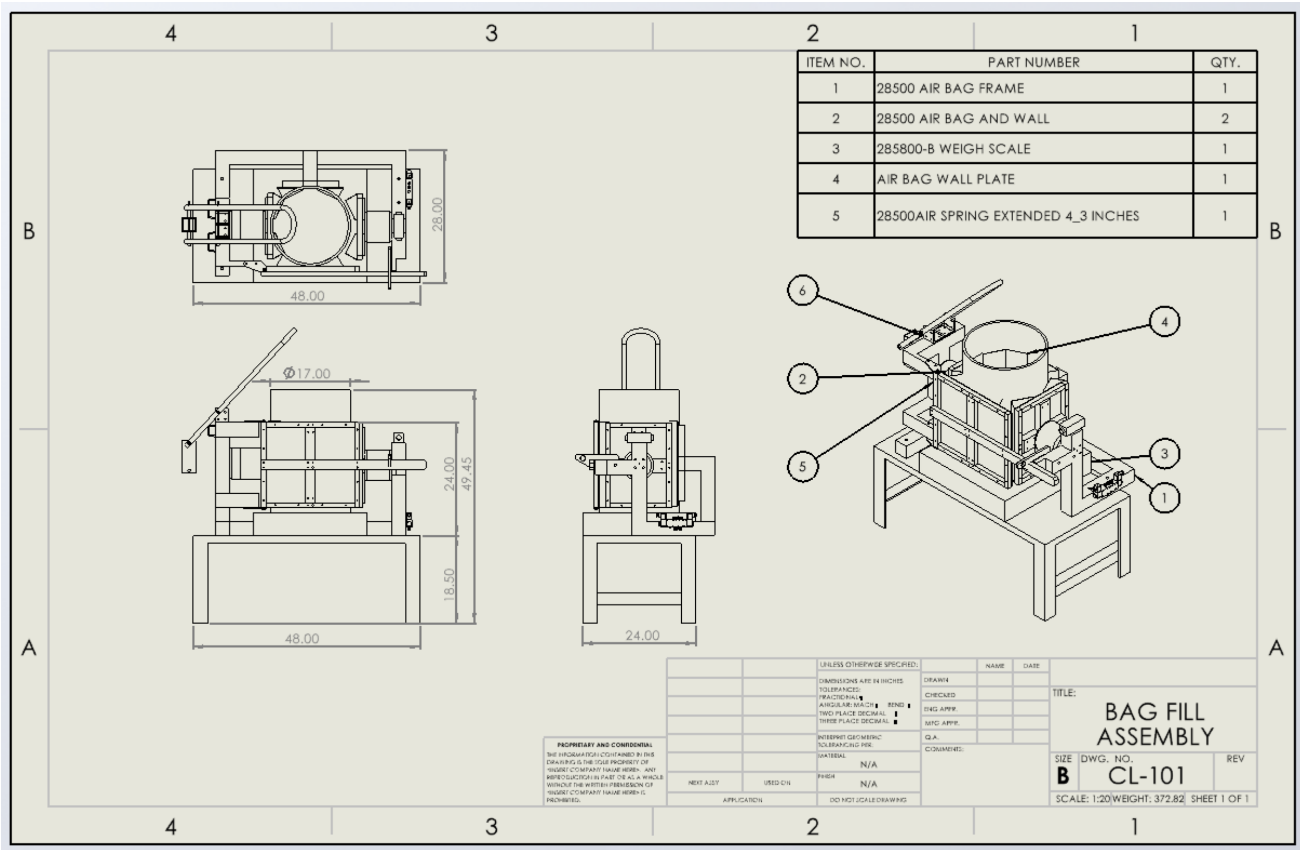
Select > Isomeric View



Select > Insert > Auto Balloon > OK



Isometric View with Numbered Balloons



Completed Drawing

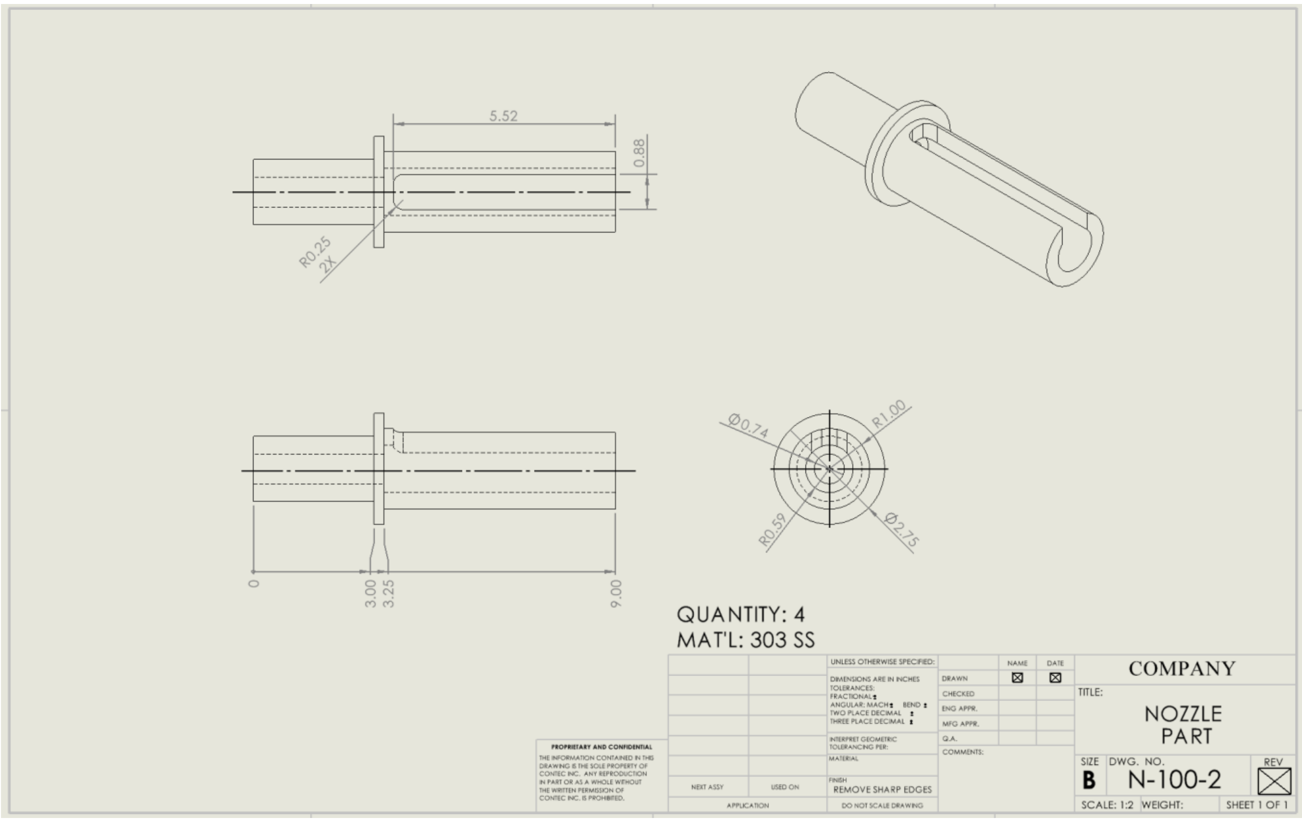
The bill of materials in the top right corner is created automatically from the 3-dimensional assembly model.

Drawings are created from part and assembly models in drafting views in a drawing document. Part numbers in balloons are created automatically.

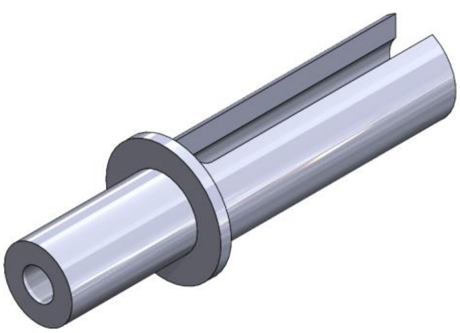
Any dimension can be revised in any part on the drawing and the part model will "Rebuild" to match. Click on the Rebuild icon to activate the dimension changes.

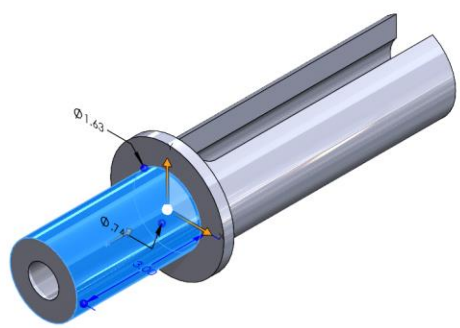
<u>**Associativity between parts, assemblies, and drawings assures that changes made to one document or view are automatically made to all other documents and views.**</u>

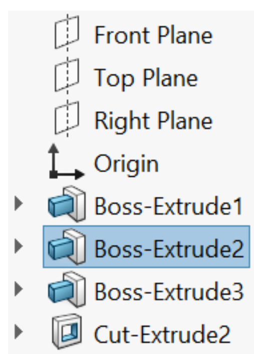
14 REVISE DIMENSIONS



Part Drawing

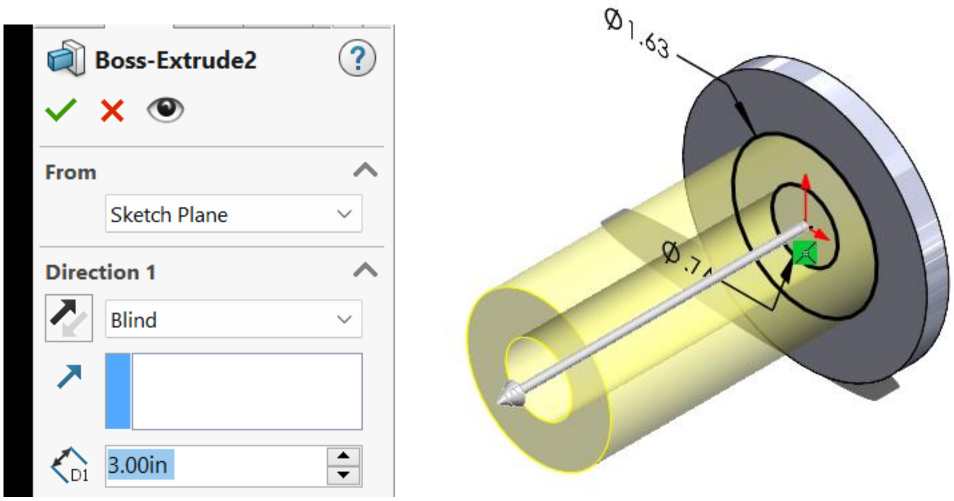




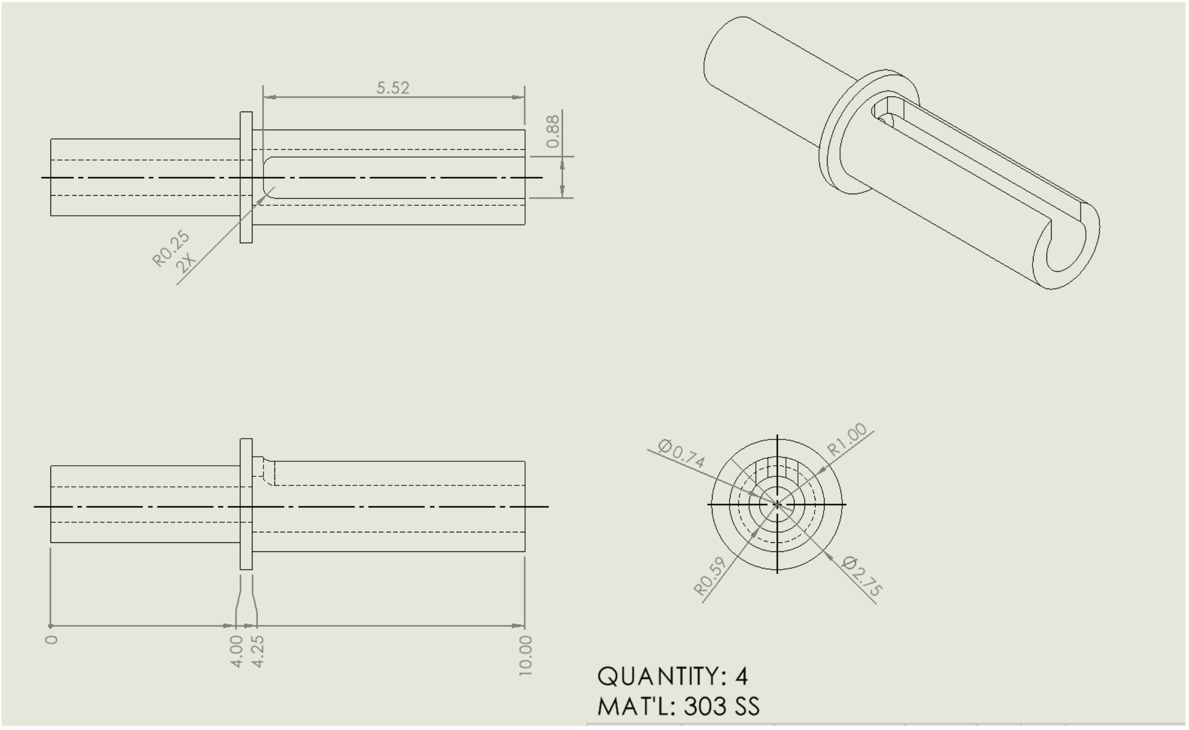


Part 3D Model Select > Left end of part Boss Extrude2

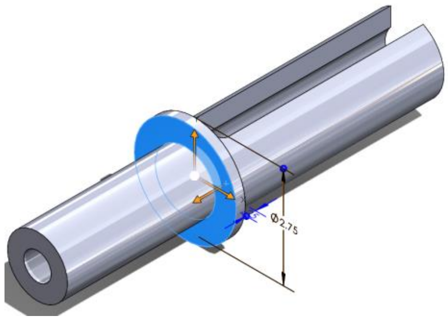
Right click > Boss Extrude2

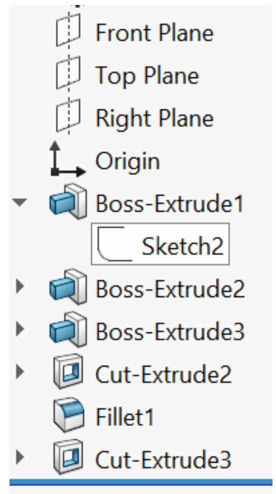


Type > 4.00 in place of 3.00.> OK

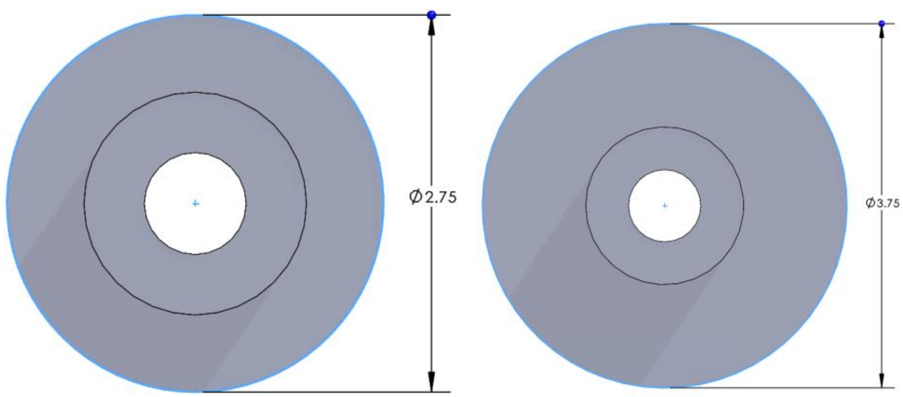


3.00 Dimension is revised to 4.00





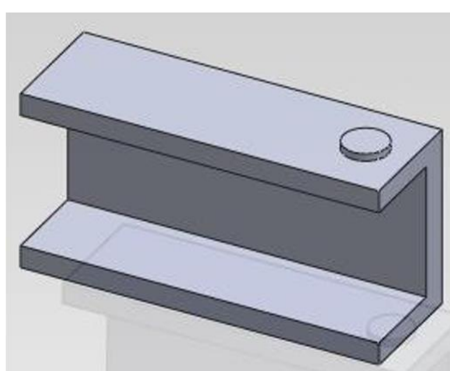
Select 3D model flange Select > Sketch2 > Edit Sketch > Normal To



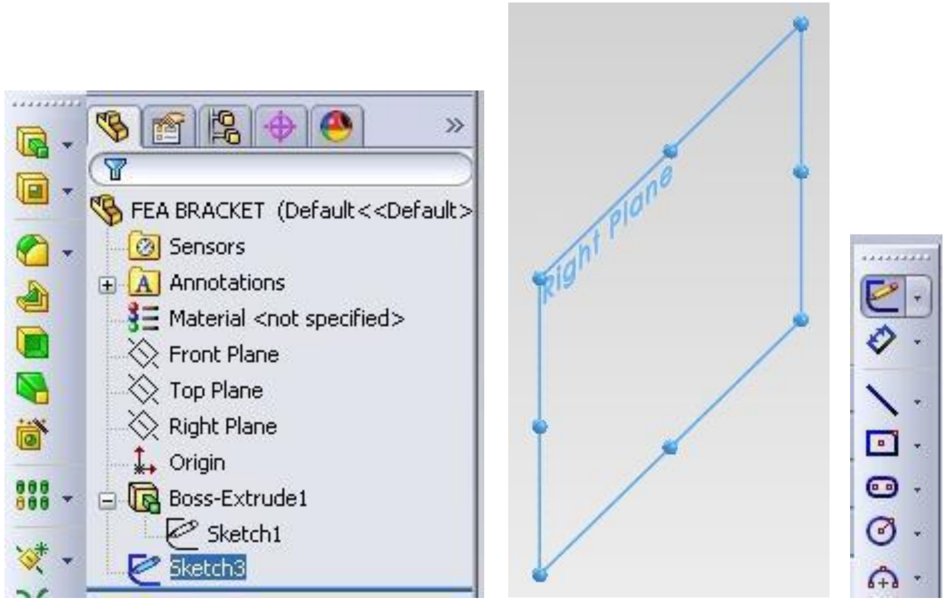
Flange dimension is changed from 2.75” to 3.75”

16 FINITE ELEMENT ANALYSIS (FEA)

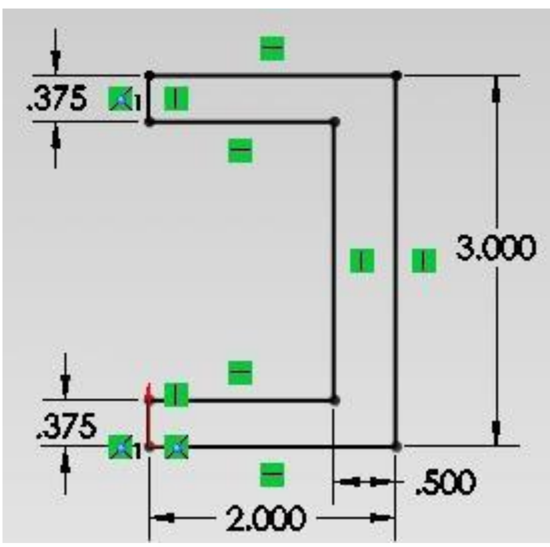
SolidWorks CAD software includes finite element analysis applied to: stress, deflection, fluid flow, and temperature distributions.

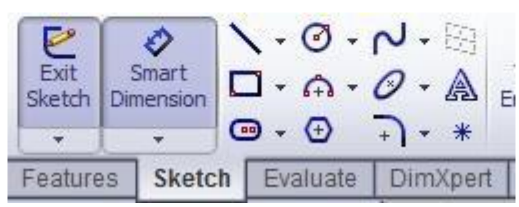


Open SolidWorks and build a part – Example.



Pick the: Right Plane icon > Sketch icon >

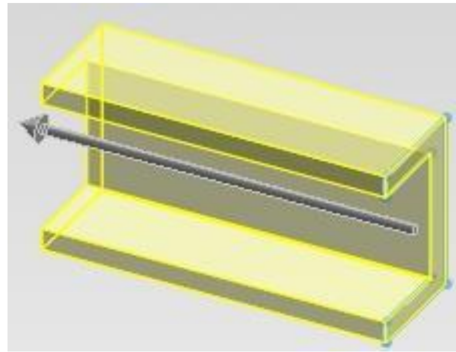




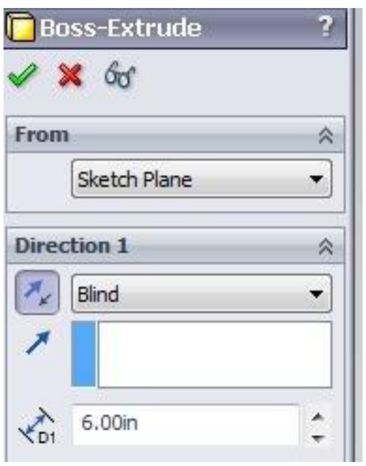
Sketch the above channel shape.

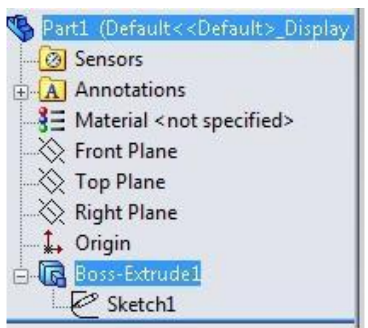
Pick > Sketch > Line tool > Pick the bottom left corner as shown above > Sketch the channel profile one straight line at a time.

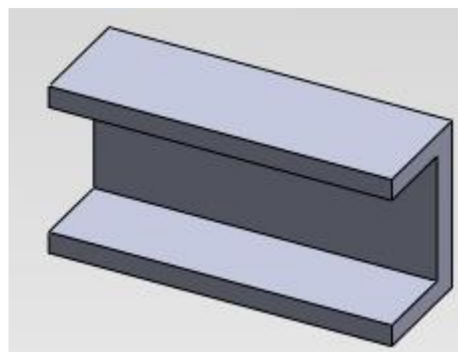
Smart dim > .375 > .500 > 2.000 > 3.000 > Exit Sketch > OK



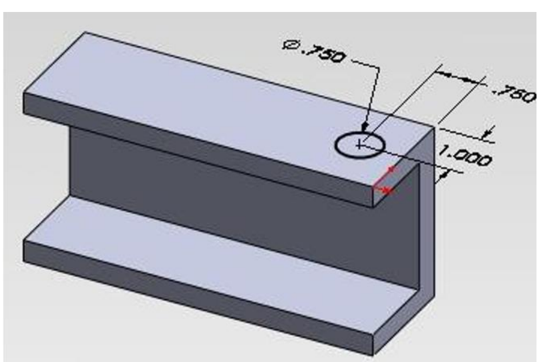
Add thickness (6.00 inches) to the rectangle by extruding it.

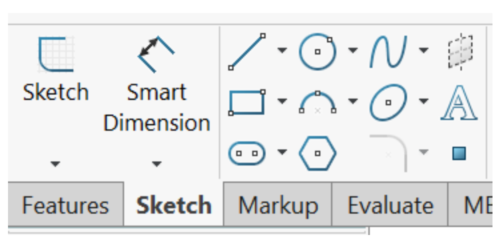






Select the "Boss-Extrude" icon > Blind > 6.000 > OK   File >Save As > CHANNEL BRACKET Create a round "Load Zone" .750-inch diameter on the top surface of the channel.

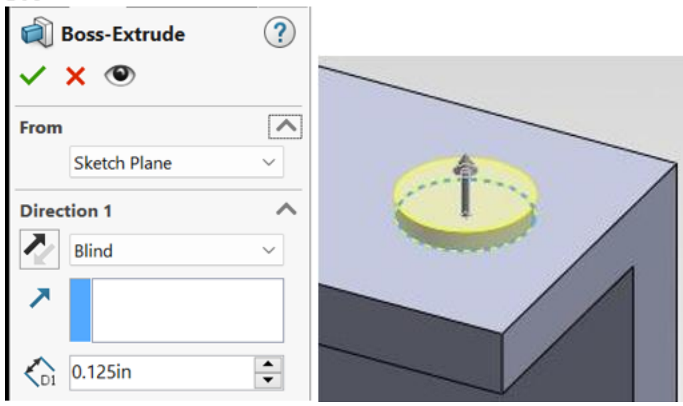




Pick the top surface of the channel > Sketch > Circle tool > Sketch the circle > With "Smart Dimension" Add the dimensions shown above.

Extrude the .750-inch diameter circle.

Pick: Extruded Boss/Base > Pick the .750 inch diameter circle > Blind > 0.125 inch > OK



Circular "Load Zone" .750 inch diameter.

Completed Part

Open the add-on Finite Element Analysis software two ways:

1. **Drop down menu: Tools > SimulationXpress. Next add a "Fixture" or anchor > Pick left end surface as shown below > OK**

2. **Or: Office Products > SolidWorks > Simulation (wait a moment for the FEA add-on to open)**

Pick: "Options" drop down menu > System of units > English inch-pound-second (IPS) or ISO

Boundary Conditions: When a component is isolated for analysis, the way in which that component is attached to another must be simulated with boundary conditions. In this case, we have chosen a fixed restraint, which means that every point on the back face of the bracket is prevented from moving in any direction.

While this seems to be a reasonable assumption, it may not be entirely accurate.

If screws are used to attach the bracket to a wall, then the top screws may stretch enough to allow the top of the bracket to separate from the wall.

Also, the wall itself may deflect slightly.
The choice of proper boundary conditions to simulate actual constraints is often one of the most important decisions to be made for an analysis.

Analysis Type: In a static analysis, we assume that that loads are applied slowly.

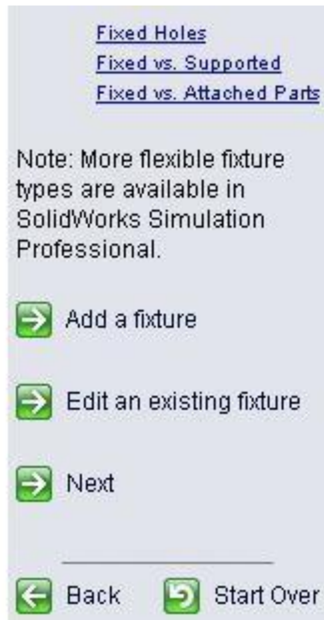
If loads are applied almost instantaneously, then dynamic effects need to be considered.

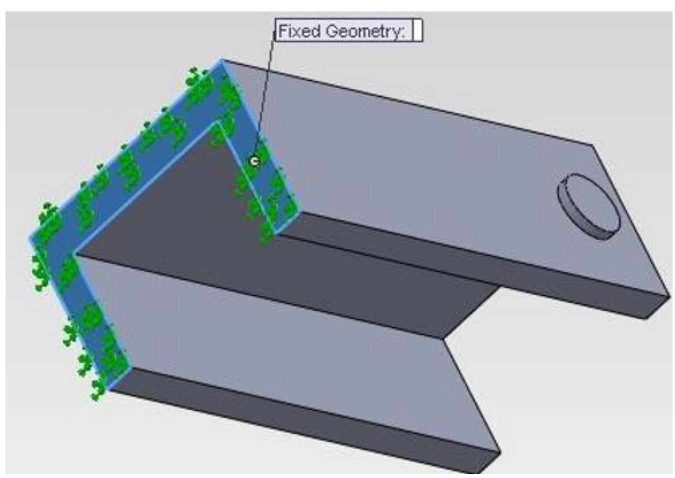
A linear static analysis assumes that the response of the structure is linear – for

example, a 20-lb load produces stresses and deflections that are exactly twice that of a 10-lb load.

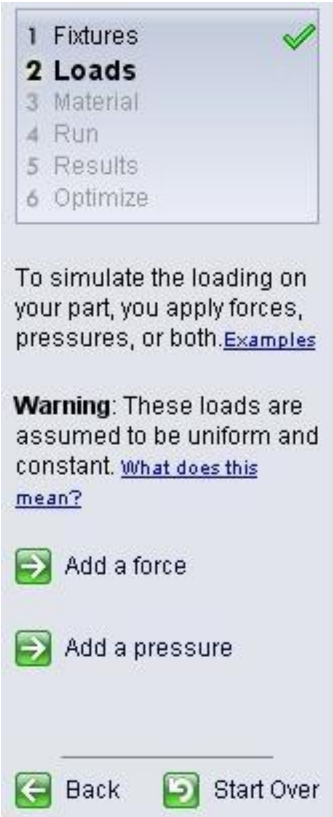
However, if the deflections are relatively large, then the stiffness of the part changes as the part deflects.

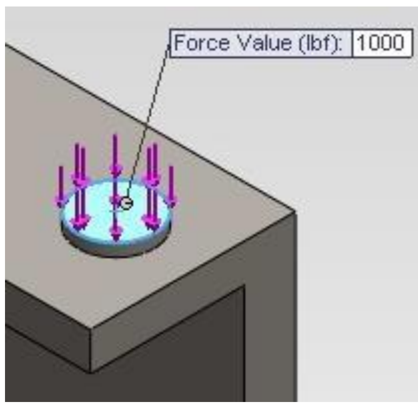
In that case, a large-deflection analysis, in which the load is applied incrementally, and the stiffness re-calculated at every step, may be required.



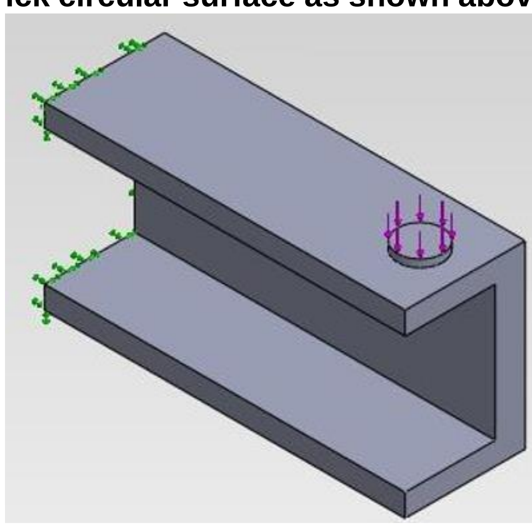


Add a fixture > Pick the channel left end surface as shown above > Next

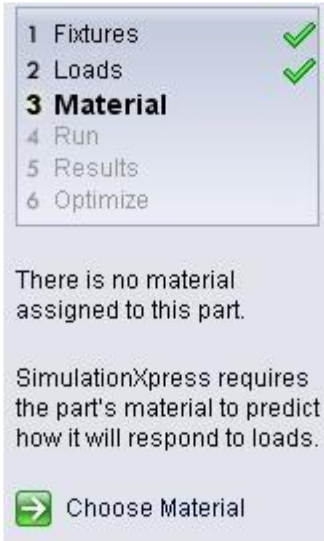




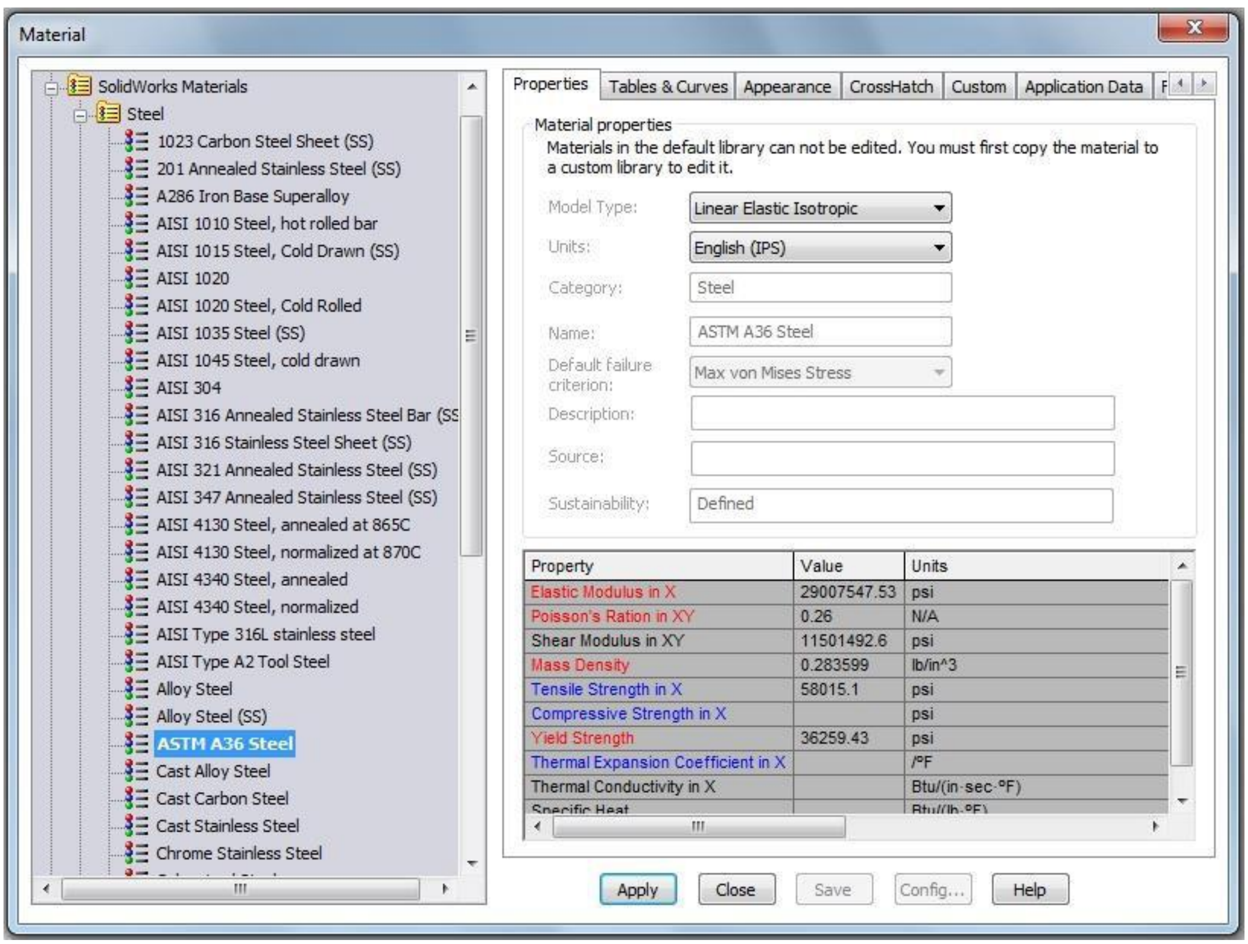
Next > Add a Force > Pick circular surface as shown above > OK > Next



The channel is now fixed at the left end and a 1000 lb load is applied to the Load Zone.

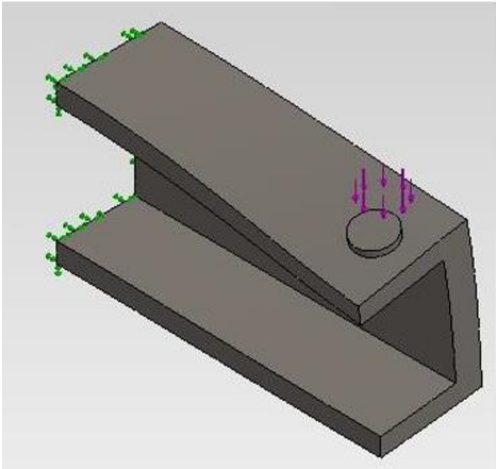


Choose Material > ASTM A36 > Apply > Close



Pick "ASTM A36 Steel" > Apply > Close > Next

Pick "Run" > Run Simulation >

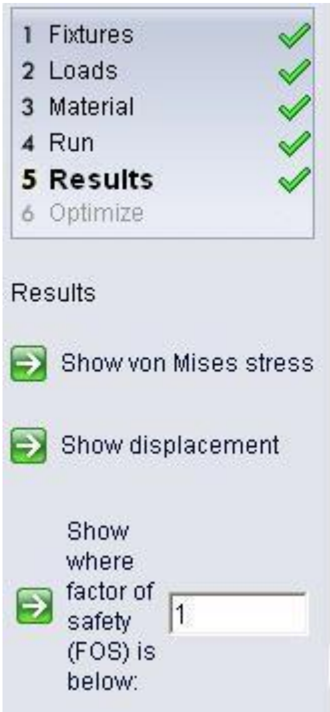


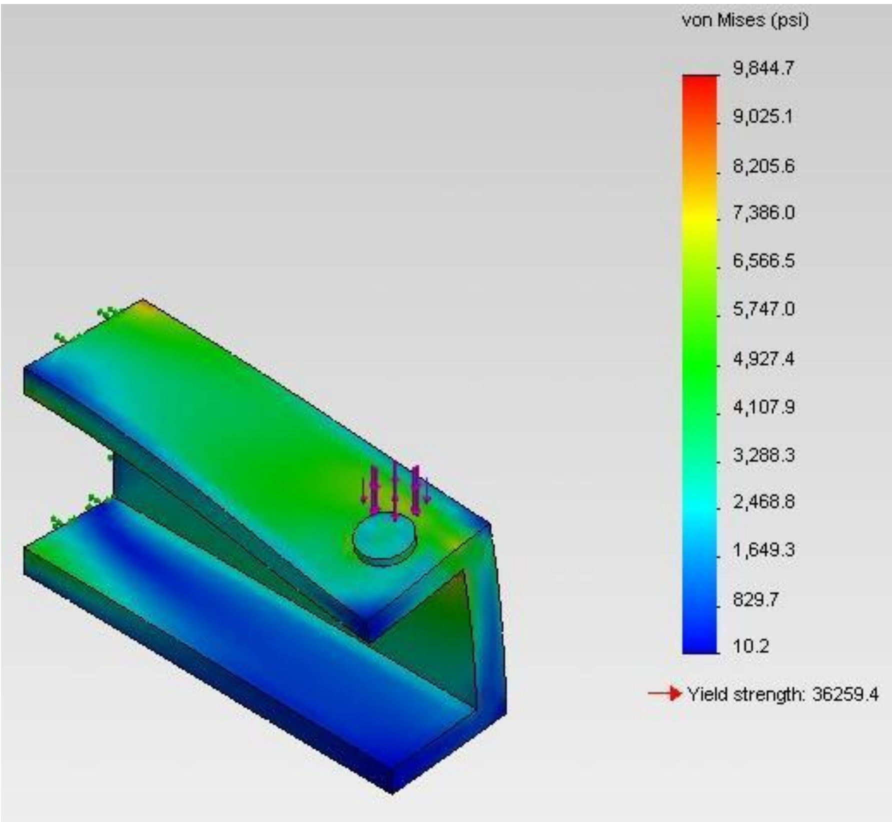
Run Simulation

Pick “Results” > Play > Stop animation > view

results below. Does the part deform as you

expected? > Yes, continue >





Show VonMises stress distribution > Show Displacement >

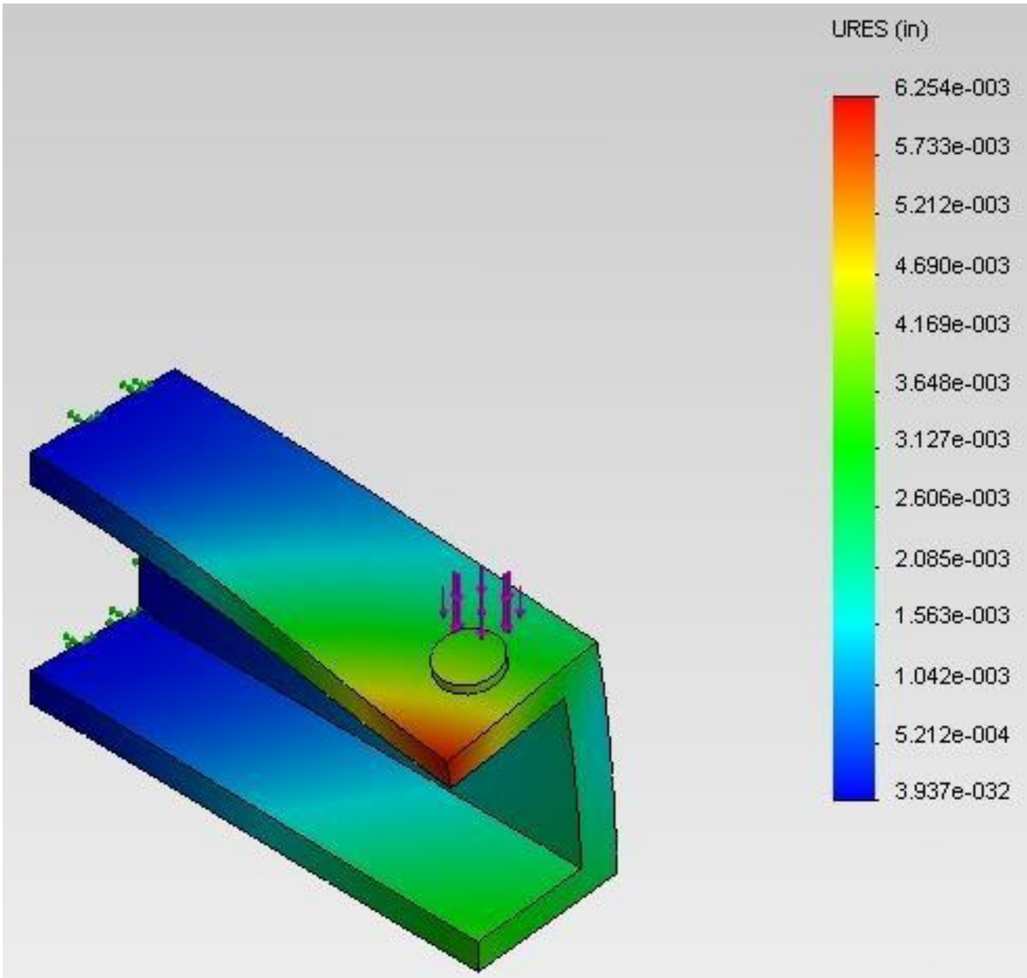
View "VonMises" resultant stresses.
Mesh Size: A finer mesh, with more elements, will generally produce more accurate results at the expense of longer processing time. For simple parts and a relatively fast computer, the longer processing time is not significant.

However, for complex analyses (such as non-linear and time dependent analyses), mesh size can significantly impact processing time.

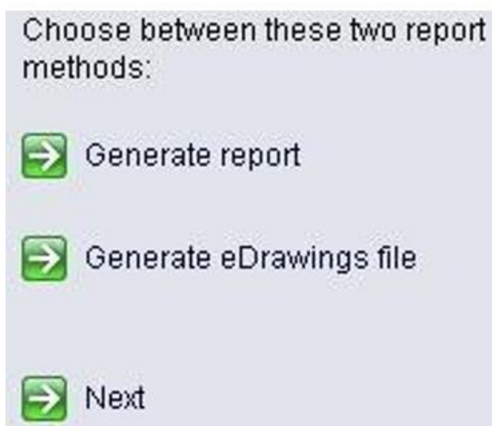
How many elements are needed for accuracy? Sometimes it is necessary to experiment with different meshes until the results converge to a solution. In other cases, the mesh can be refined to create more elements in a local area where stresses are greatest.

Element Type: There are many element types, such as plates, shells, truss members, beam elements, and solid elements. SolidWorks Simulation allows for solid elements to be created from solids, or shell elements to be created from either surfaces or solid mid-surfaces.

Although solid elements are typically chosen when a solid model is available, solid elements are not always the best choice for many applications. Often, a few beam or shell elements will provide more accurate results than hundreds of solid elements.

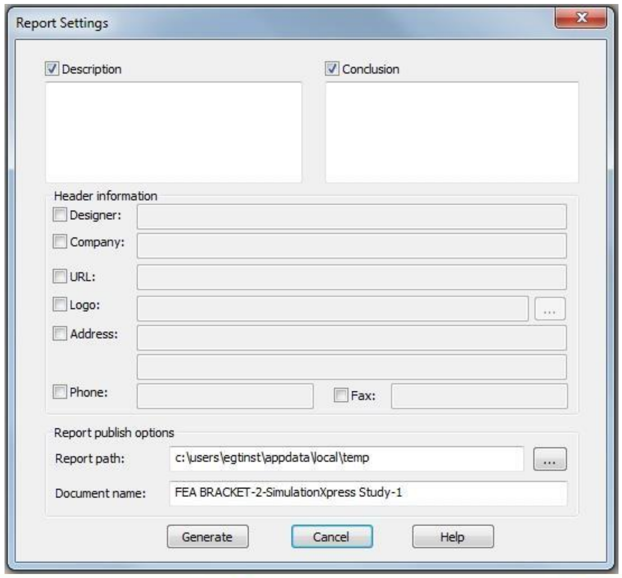


View Results



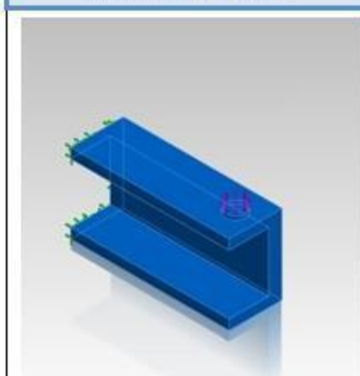
Generate Report

FEA BRACKET-2-SimulationXpress Study.analysis.eprt



Boss-Extrude2 	Solid Body	Mass:4.48235 lb Volume:15.8052 in^3 Density:0.283599 lb/in^3 Weight:4.47931 lbf	G:\A57-SOLID WORKS\SOLIDWORKS FEA\FEA BRACKET-2.SLDPRT Feb 28 19:08:50 2012

Material Properties

Model Reference	Properties		Components
	Name: Model type: Default failure criterion: Yield strength: Tensile strength:	ASTM A36 Steel Linear Elastic Isotropic Max von Mises Stress 36259.4 psi 58015.1 psi	SolidBody 1(Boss-Extrude2)(FEA BRACKET-2)

Mesh Information

Mesh type	Solid Mesh
Mesher Used:	Standard mesh
Automatic Transition:	Off
Include Mesh Auto Loops:	Off
Jacobian points	4 Points
Element Size	0.251027 in
Tolerance	0.0125513 in
Mesh Quality	High

Mesh Information - Details

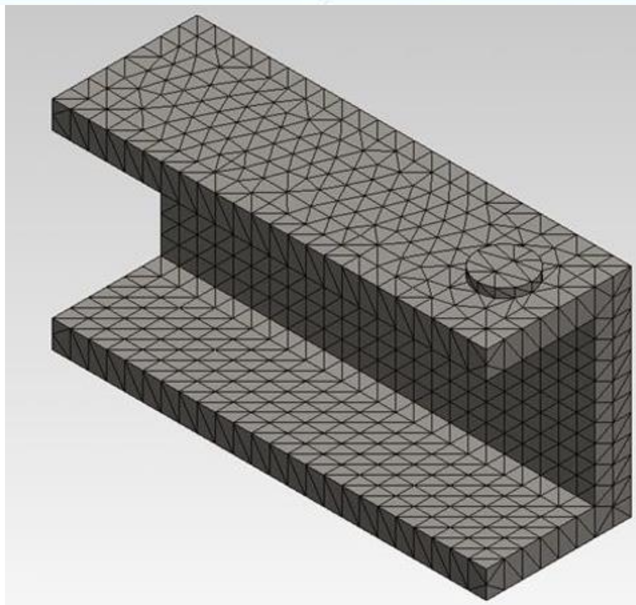
Total Nodes	12584
Total Elements	7397
Maximum Aspect Ratio	4.3633
% of elements with Aspect Ratio < 3	99.7
% of elements with Aspect Ratio > 10	0
% of distorted elements(Jacobian)	0
Time to complete mesh(hh;mm;ss):	00:00:02
Computer name:	ET-EGT-423-INST

Mesh Information

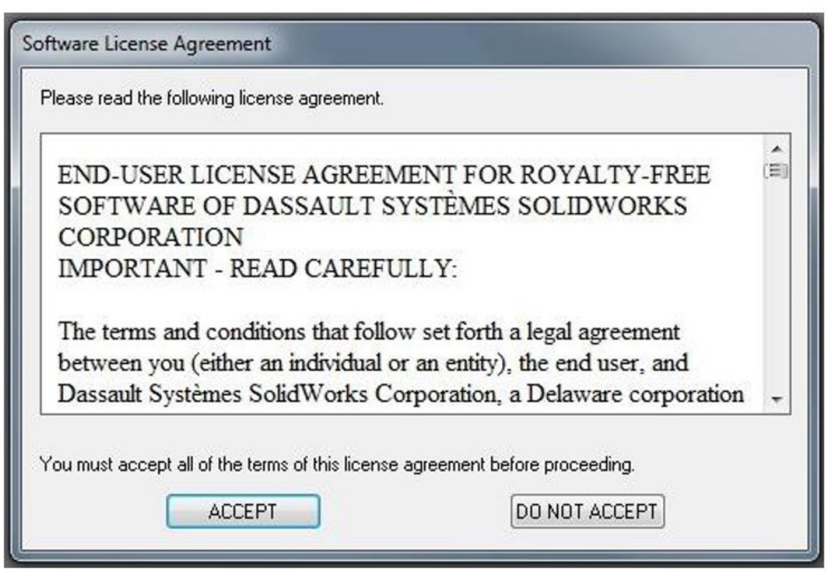
Mesh type	Solid Mesh
Mesher Used:	Standard mesh
Automatic Transition:	Off
Include Mesh Auto Loops:	Off
Jacobian points	4 Points
Element Size	0.251027 in
Tolerance	0.0125513 in
Mesh Quality	High

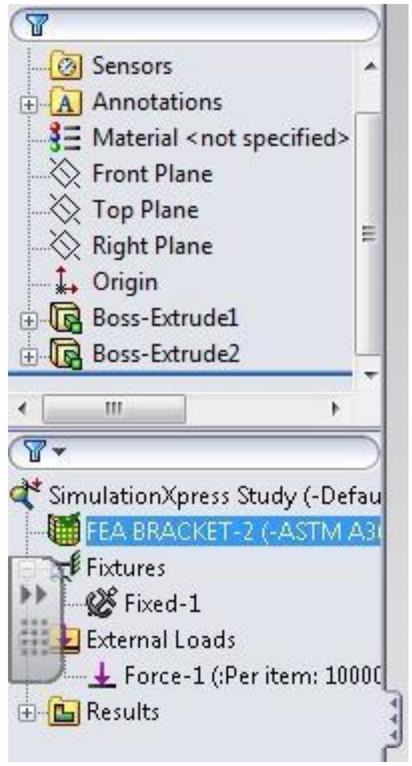
Mesh Information - Details

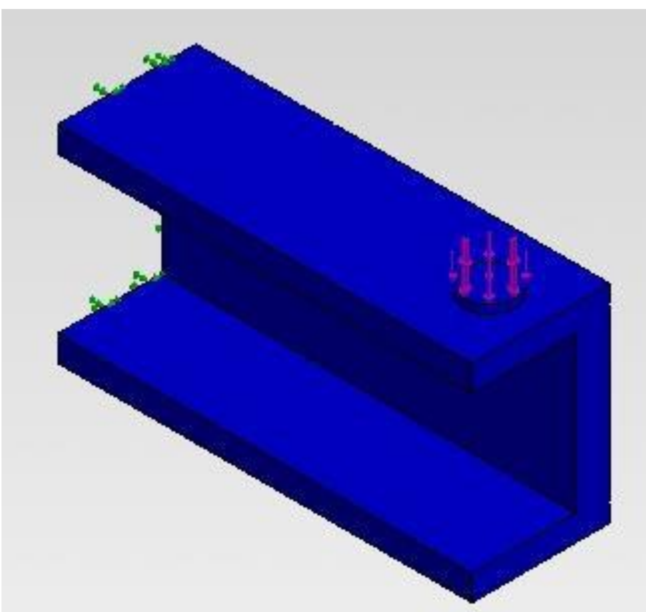
Total Nodes	12506
Total Elements	7339
Maximum Aspect Ratio	3.2659
% of elements with Aspect Ratio < 3	99.9
% of elements with Aspect Ratio > 10	0
% of distorted elements(Jacobian)	0
Time to complete mesh(hh;mm;ss):	00:00:05
Computer name:	ET-EGT-432-INST

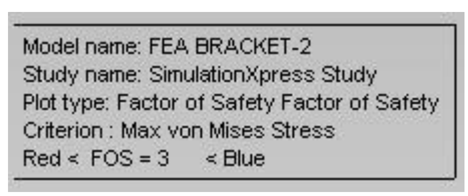


Generate eDrawing File >

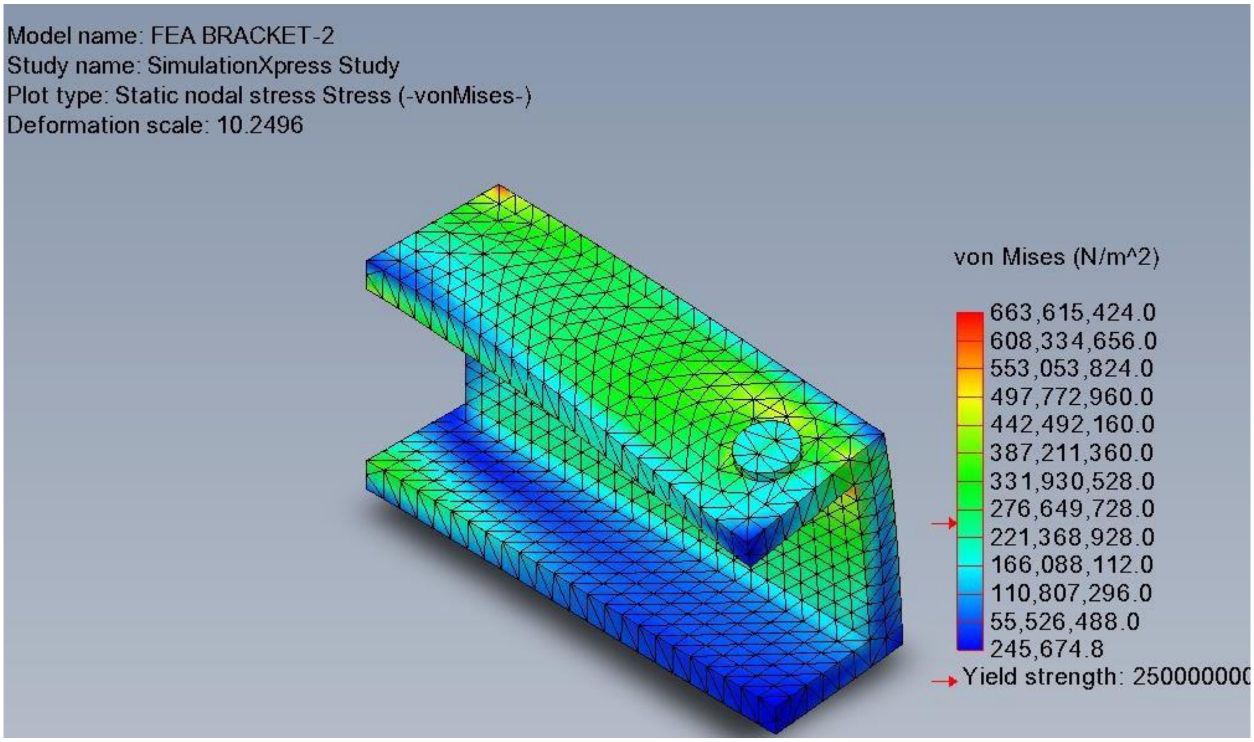








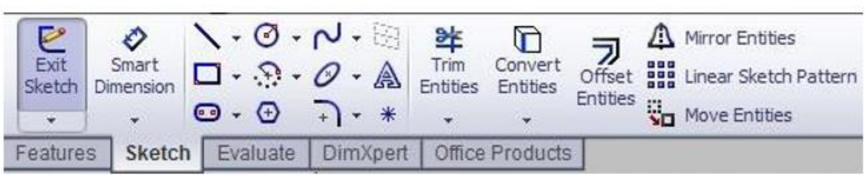
The factor of safety in the blue area is greater than 3.00.



SOLIDWORKS MENUS

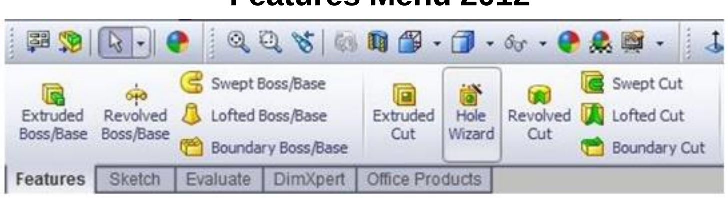
Sketch Menu 2023

Sketch Menu 2012

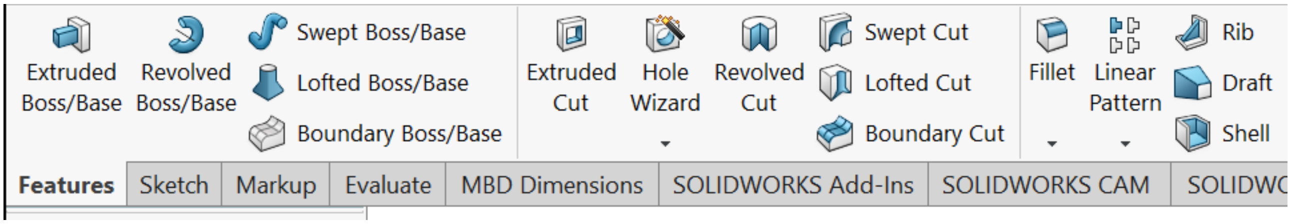


Start each part by clicking the "Sketch" tab to open the tools shown above used to create a two-dimensional profile.

Features Menu 2012



Features Menu 2023



Convert a sketch into a three-dimensional solid model by clicking the "Features" tab to open the tools shown above. The above are "Sketch Features".

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WEB LINKS

SolidWorks web site: www.solidworks.com
3D ContentCentral online at: (http://www.3dcontentcentral.com/default.aspx) is a free source of SolidWorks part and assembly models.
SoldWorks in Ten Minutes video:
http://www.youtube.com/watch?v=pFy8iijJSHM&feature=related Getting Started with SoldWorks video:
http://www.youtube.com/watch?v=cmC2MLRetko&feature=related
Large Assembly layout and motion
http://www.youtube.com/watch?v=uMnd69- aueM&feature=related

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END OF COURS