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Mechanical Engg

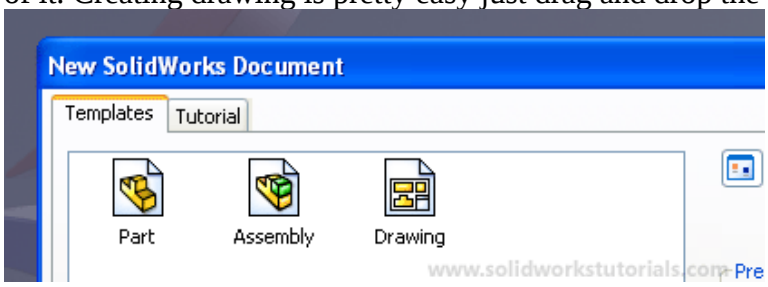
Vedharanyam-614 810

A manual to mechanical designers

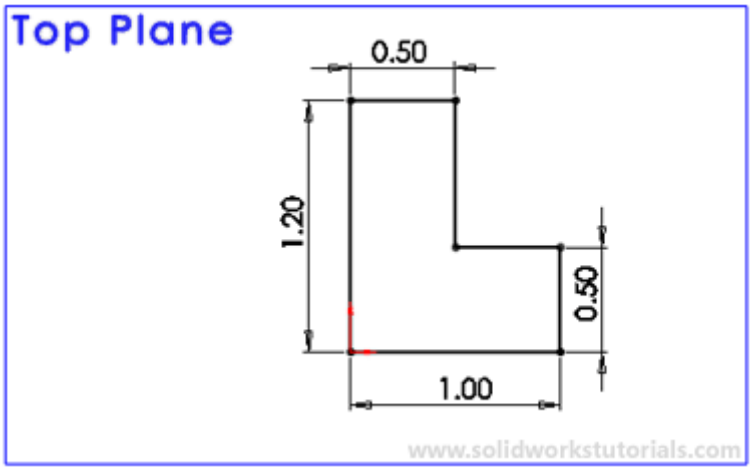
How Solid works Works?

Solid works Overview

Solid works main idea is user to create drawing directly in 3D or solid form. From this solid user can assemble it directly on their workstation checking clashes and functionality of it. Creating drawing is pretty easy just drag and drop the solid to drawing block.

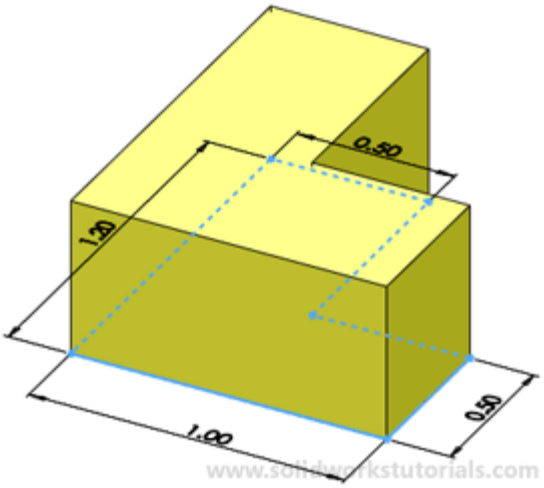


Part

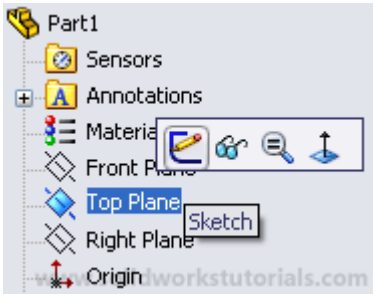


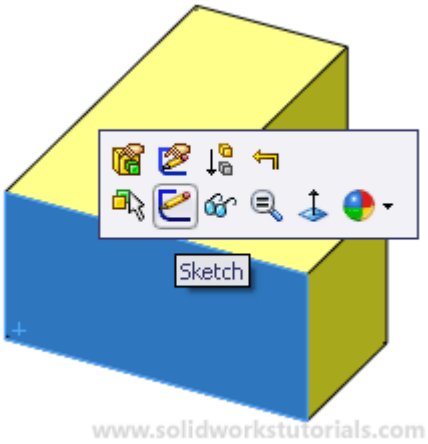
Part is created by sketch.

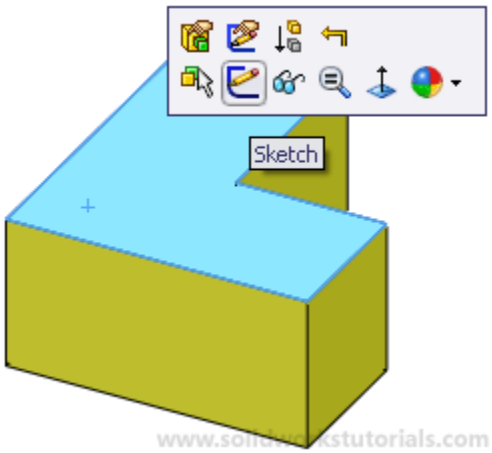
Sketch is the base to define your part, form and features.



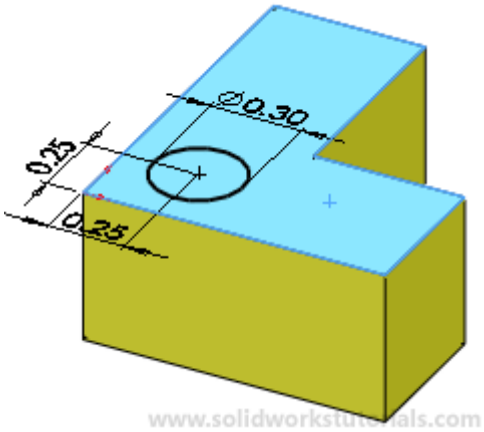
Before you start creating sketches you must select plane or face where the sketch will be place on.





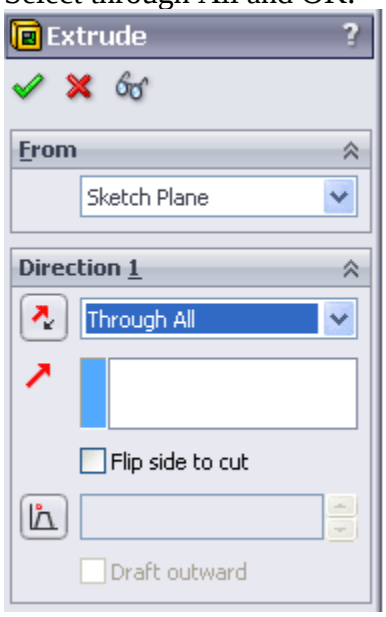


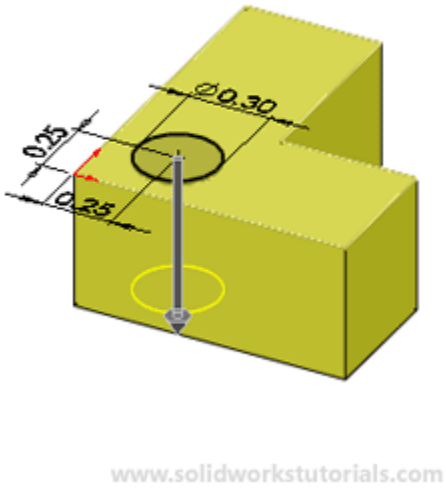
After select plane or face the sketch will be, sketch on it!

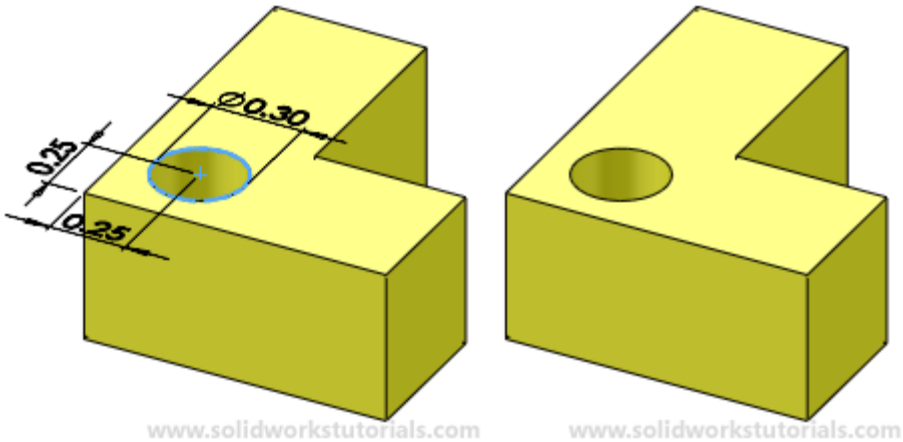


When you done with sketch, adding features it is your next step. Select Feature>Extruded Cut

Select through All and OK.



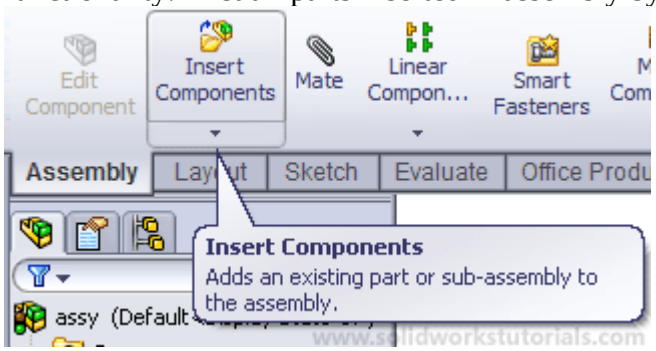




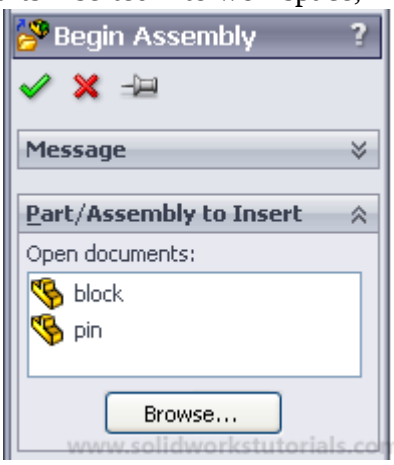
Learn [how to create this part step by step solid works tutorial here]

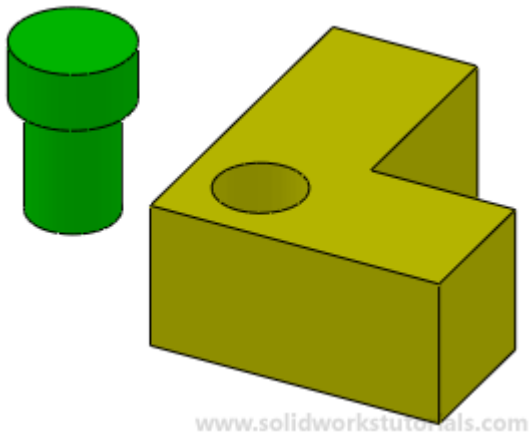
Assembly

Assembly is how all parts works together in assembly, checking for clashes and it functionality. First all parts inserted in assembly by Insert Component tool.

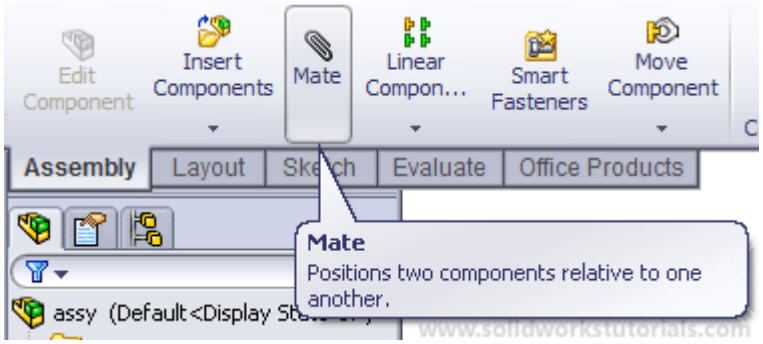


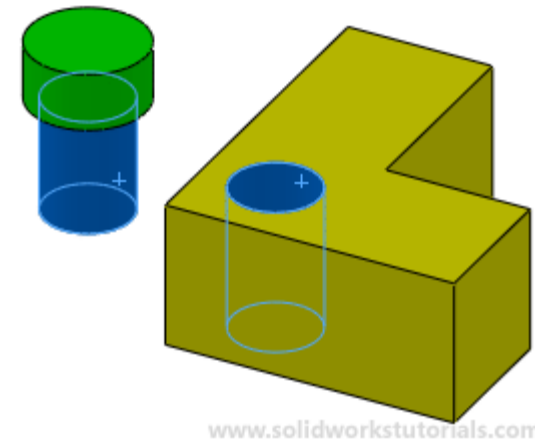
When all parts inserted into workspace, Mate is command to define how parts mate with each other.

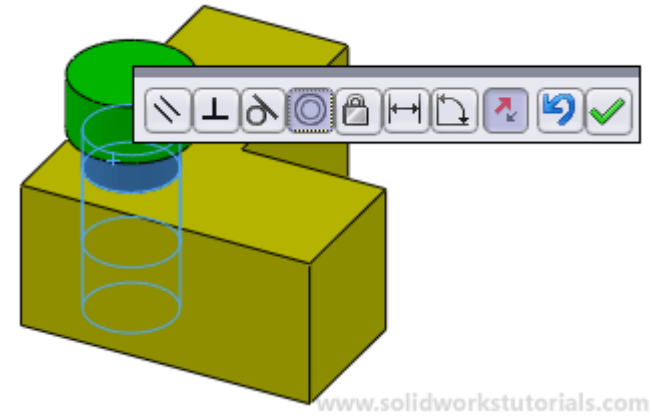




Let's mate this block and pin together, click Mate and select pin face and hole face, OK.

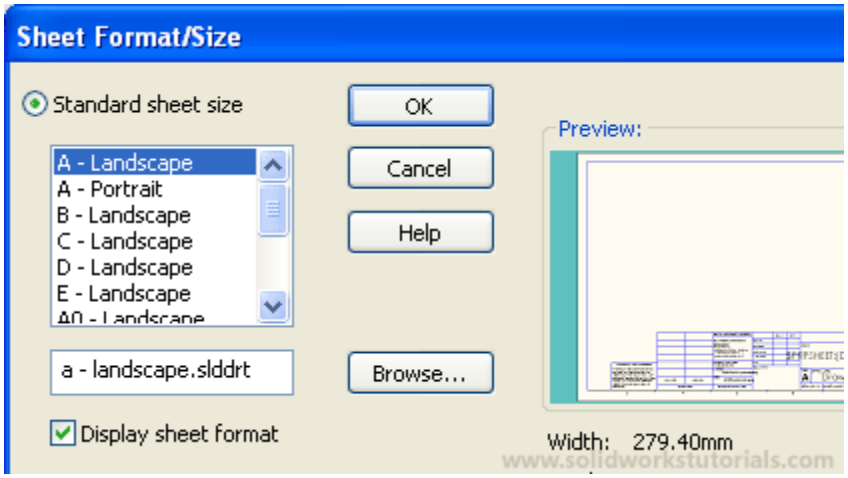




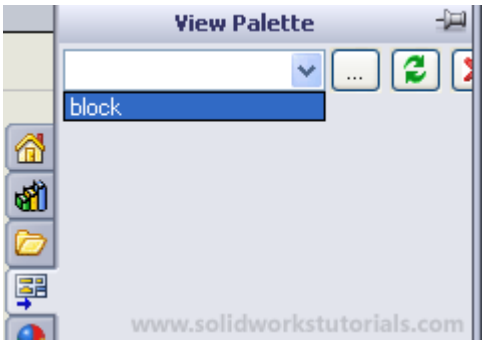


Drawing

Drawing is use for detailing part by adding dimension to it. To create a drawing first you need to select drawing block.

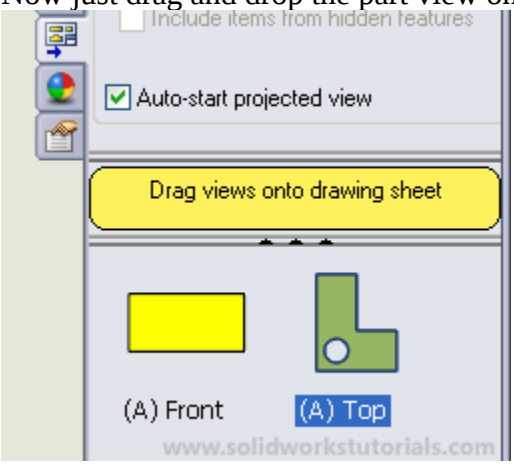


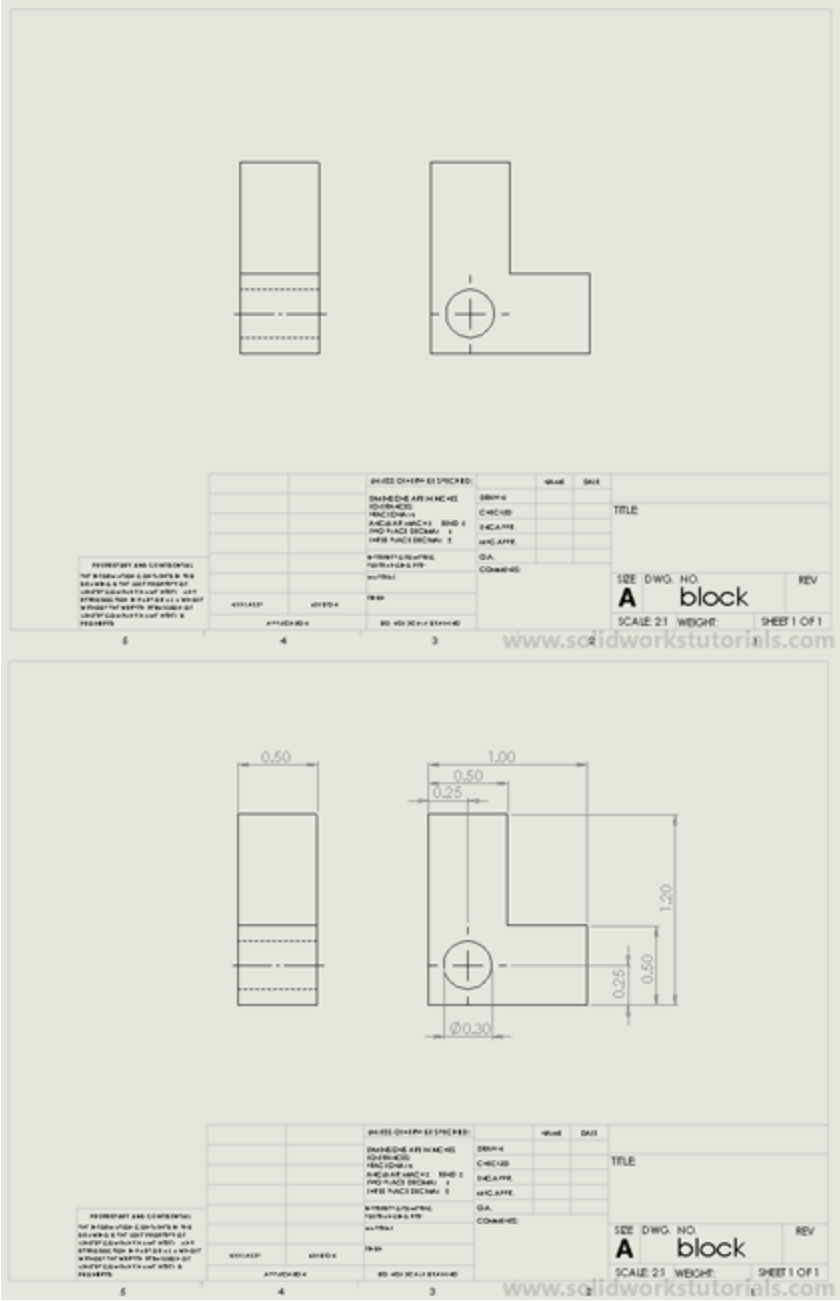
When block inserted, select click view palette to add drawing view.



Choose the part you wish to make drawing.

Now just drag and drop the part view on drawing block and add dimensions.





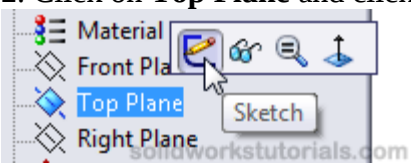
Summary

Solid works works by it user creating part in 3D or solid form. Three solid works component is Part, Assembly and Drawing. Part define by it sketch and selected feature. Assembly is how all parts assemble in one unit; parts assemble by user adding mate between parts. Drawing is for detailing and adding dimensions to part.

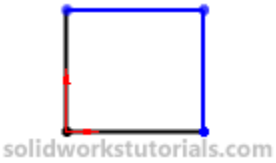
1. How to create simple cube

1. Click **New**, Click **Part** and **OK**.

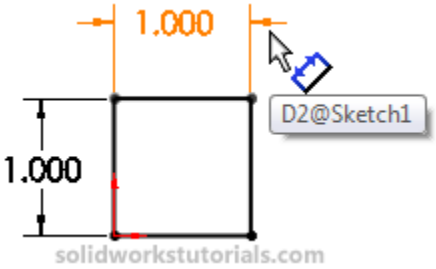
2. Click on **Top Plane** and click **Sketch**.



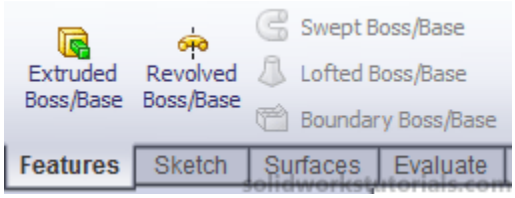
3. Click **Rectangle** , sketch a rectangle start from origin.



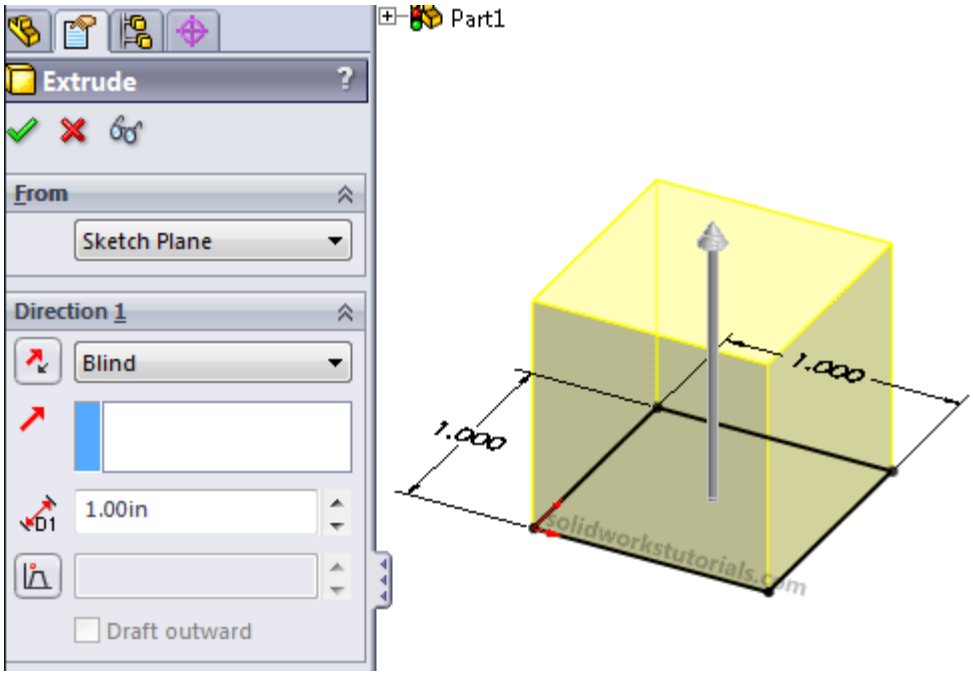
4. Click **Smart Dimension** , click side edge and click top edge to dimension it as **1.0in x 1.0in**.



5. Click **Features>Extruded Boss/Base**

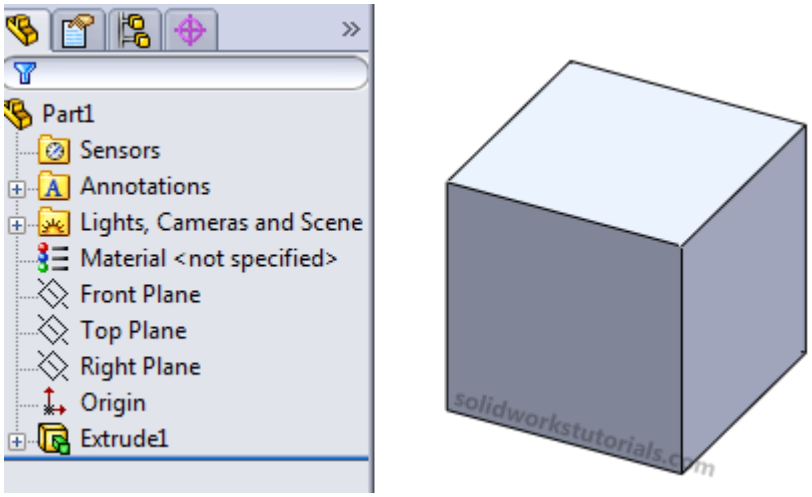


Set **D1** as **1.0in**



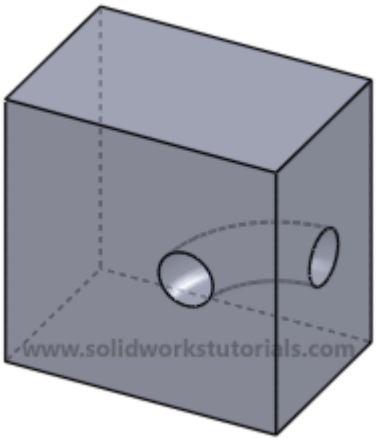
And click .

6. it’s done. Simple Right?



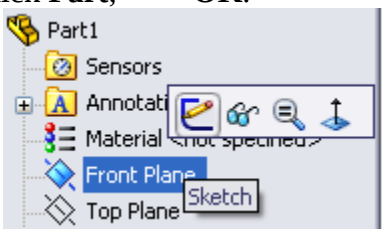
2. How to use revolved cut

In this tutorial, you will create this part using revolved feature tools.



1.Click **New**.  Click **Part**,  

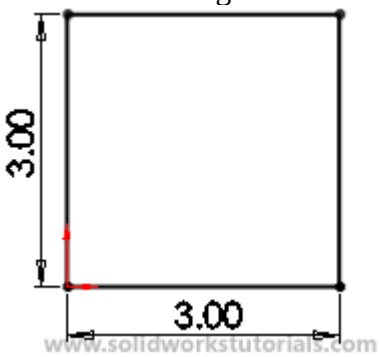
OK.

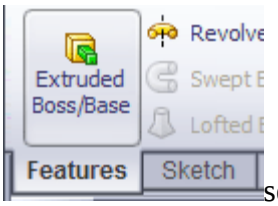
2.Click **Front Plane** 

and click on **Sketch**.

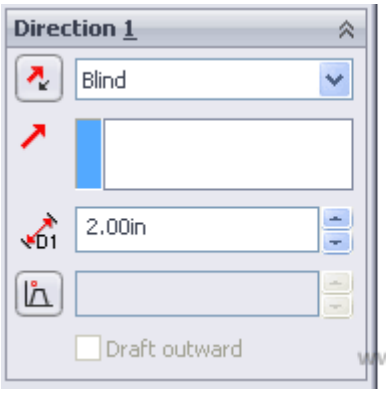
3.Click **Rectangle**,  sketch rectangular. Click **Smart Dimension**, 

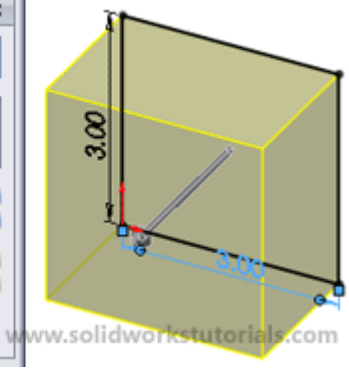
dimension rectangular **3in x 3in**.



4.Click **Feature>Extruded Boss/Base**, 

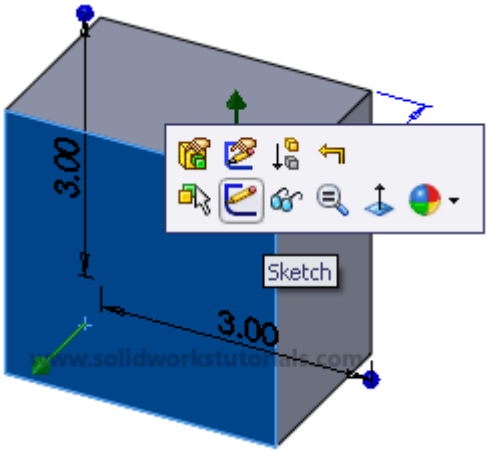
set **D1** to **2in**



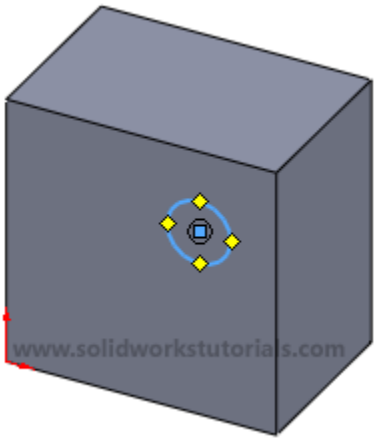


and  .

5.Click on front face and click **Sketch**.

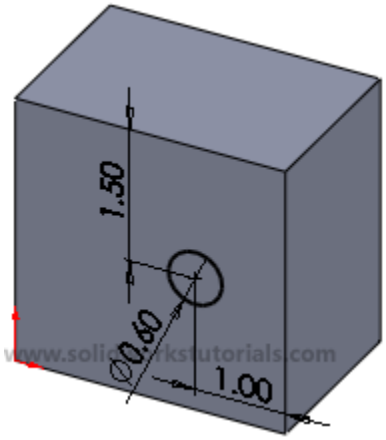


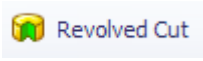
6. Click **Circle**,  and sketch a circle on front face.



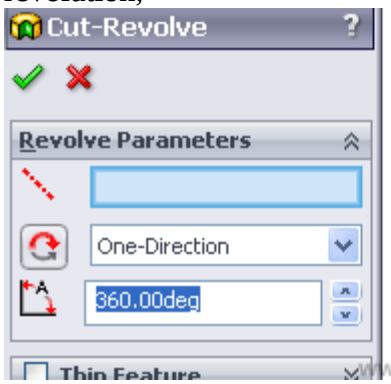
7. Click **Smart Dimension**, 

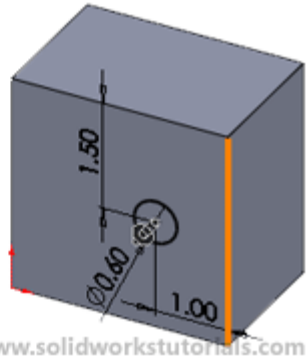
dimension sketch as below sketched.

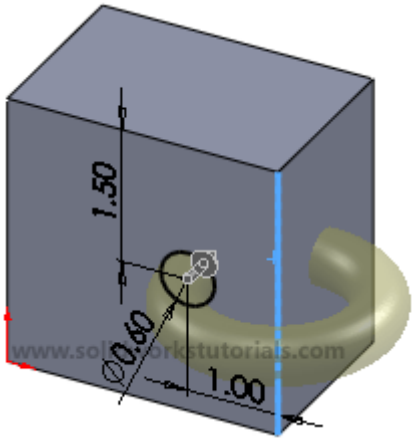


8.Click **Features**>**Revolved Cut** 

click on **right side edge** as axis of revolution,

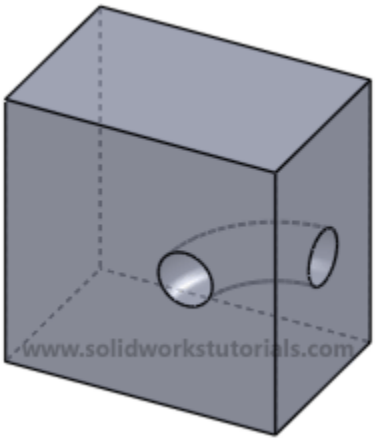






and  .

9. You’re done!



3. Create spring

1. Click New  (File>New) , click Part 

, OK .

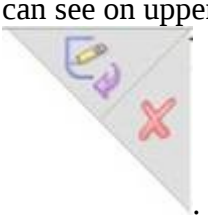
2. Click Option  (Tools>Option…) , select Document Properties tab. Select Units , under Unit System select IPS (inch, pound, second) OK.

3. Select Top Plane , from lower left menu select Normal To.

4. Click Sketch 

in Command Manager, click Circle 

. As you can see on upper right corner sketch icon appear indicate that you're on sketch mode


.

5. Pick Origin  point as starting point, drag to right hand side

no need to be exact the size will define in later step. Press keyboard ESC to end circle sketch.

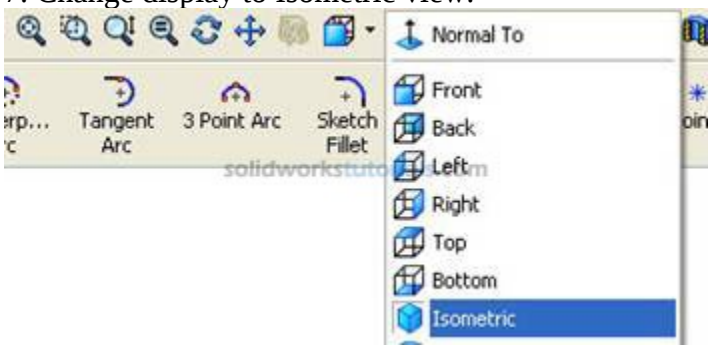
Note: There is two type line generated by in sketching, the one with black line and blue line. Black line is line that fully defined and blue line is under defined..

6. Define sketch with dimension. Click Smart Dimension 

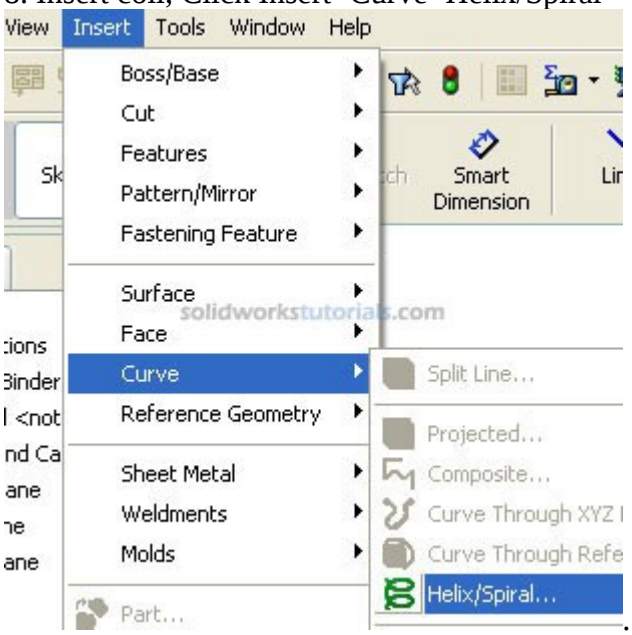
, and start

dimensioning pick circle edge and set to 0.50in . Press keyboard ESC to end smart dimension.

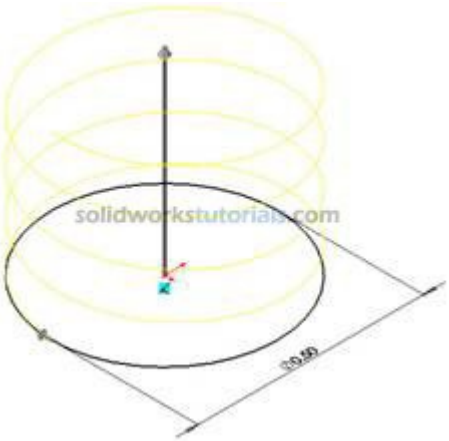
7. Change display to Isometric view.

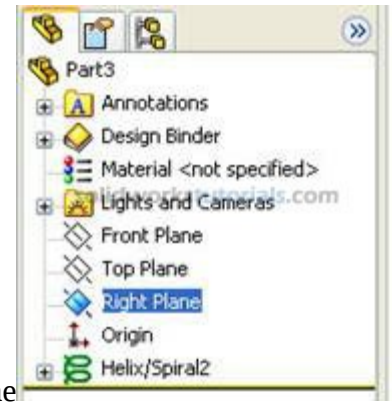


8. Insert coil, Click Insert>Curve>Helix/Spiral



9. Press F to zoom fit, set Parameters Constant Pitch , Pitch 0.10in Revolutions 4 , Start angle 0.0deg and .

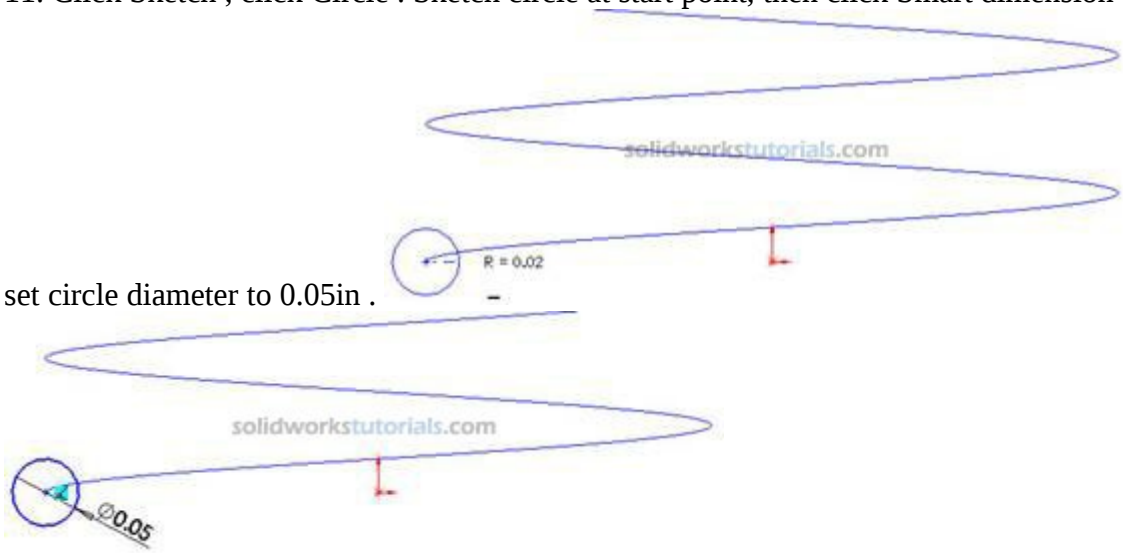


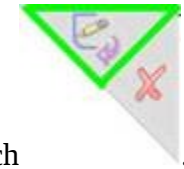
10. Click to Right Plane 

, click Normal To

.

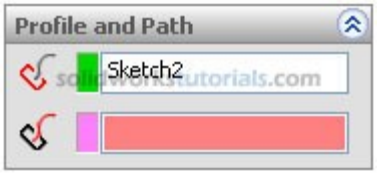
11. Click Sketch , click Circle . Sketch circle at start point, then click Smart dimension set circle diameter to 0.05in .



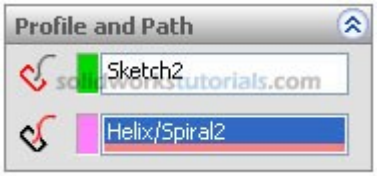
12. Click exit sketch  . Click Features 

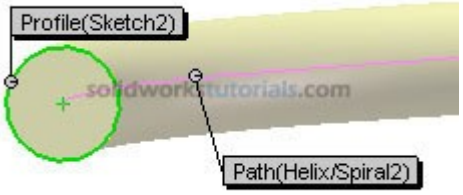
and activate features menu. Click Swept Boss/Base 

and set Profile to Sketch2 by click on circle

sketch 

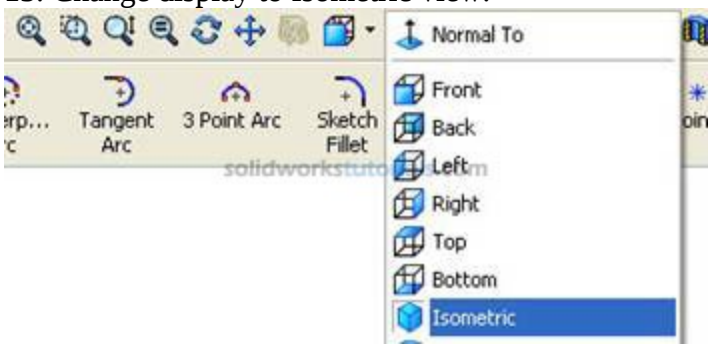
and set Path by click

helix path 



and  .

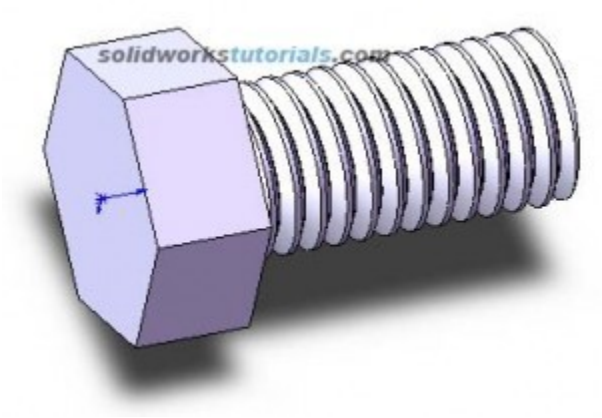
13. Change display to Isometric view.



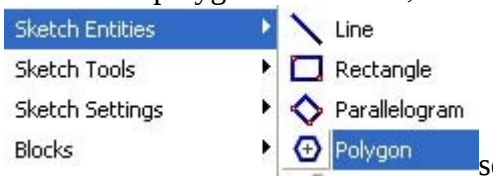
14. Press F to zoom fit.

Done. Pat yourself on back.

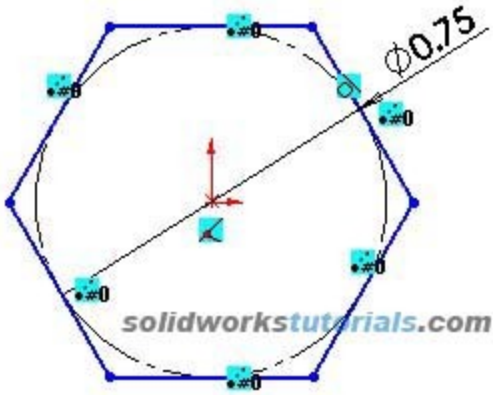
4. Create 1/2" Hex Bolt

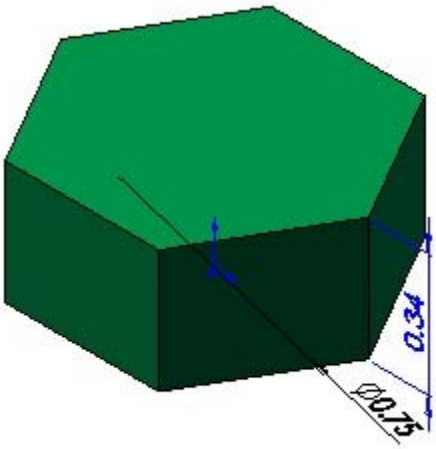


1. Sketch a polygon with 6 side, Tools>Sketch Entities>Polygon

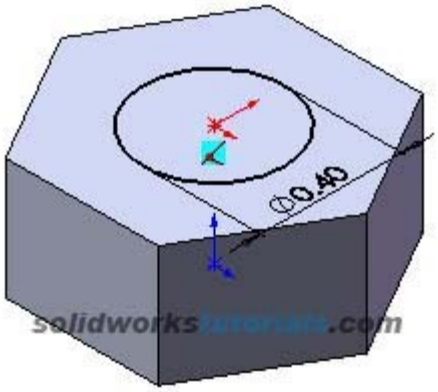


set diameter to 0.75in.

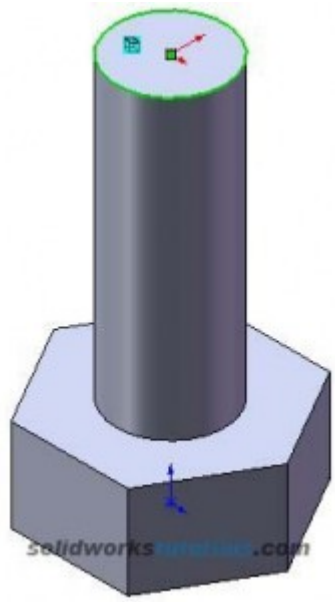




2. Extrude  sketch to 0.34in.
3. Create minor diameter for thread, sketch circle on top face, set diameter to 0.4in.

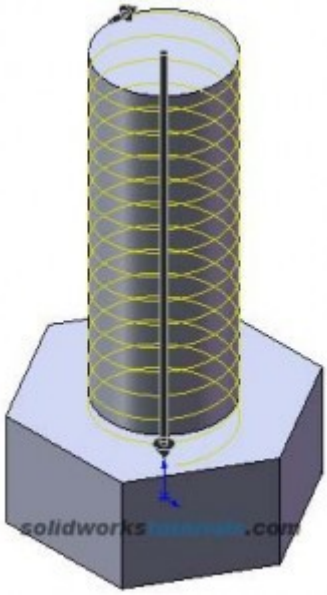


4. Extrude sketch to 1.1in.



5. Click end edge of thread shaft, click convert entities  .

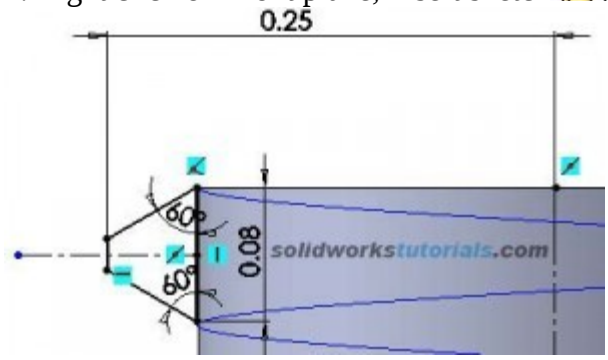
6. Select Helix/Spiral feature  set height to 1.2in, theap per inch=pitch 13/1in



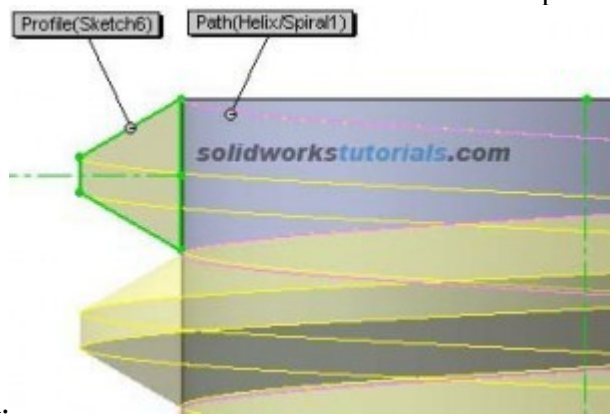
Ok.

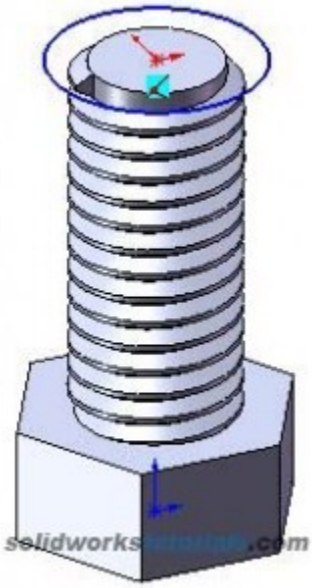
7. Right click on Front plane, Insert sketch 

sketch thead profile.

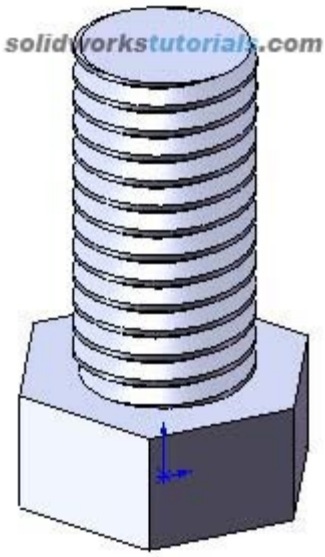


8. Click sweep feature  , 

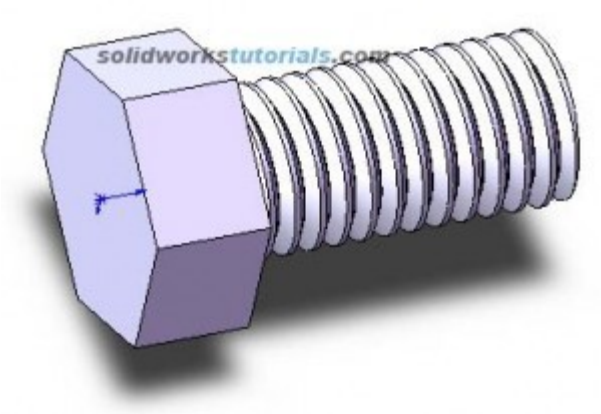
select sketch profuile as sketch and helix as a path, OK. 



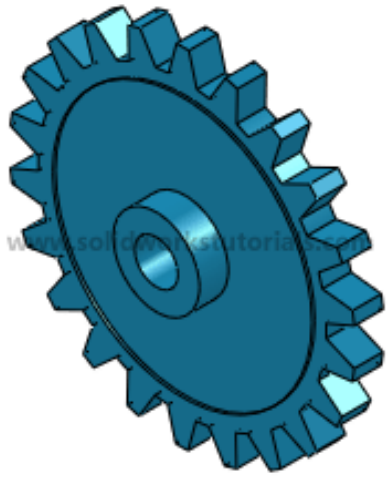
9. Create skecth a circle on the end shaft, extrude cut 0.1in  .



10. Finish.

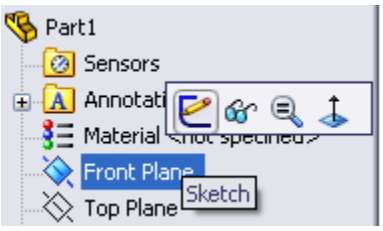


5. How to create gear

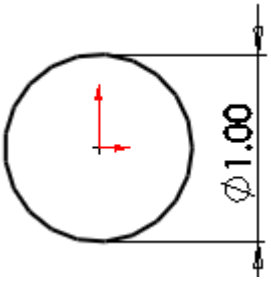


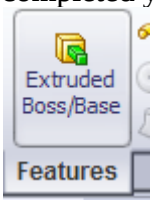
In this solidworks tutorial, you will create gear.

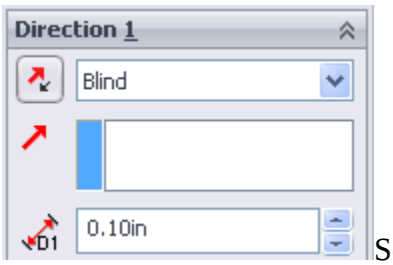
1. Click **New**.  Click **Part**,  **OK**.
2. Click **Front Plane** and click on **Sketch**.

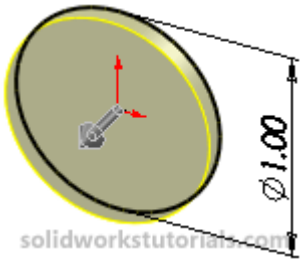


3. Click **Circle**  and sketch a circle center at origin. Click **Smart Dimension**,  click sketched circle and set it diameter to **1in**.

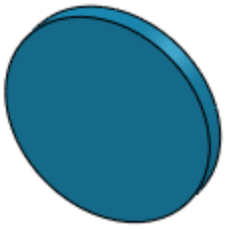


4. You just completed your sketch, let's build feature from it. Click **Features>Extruded Boss/Base**. 

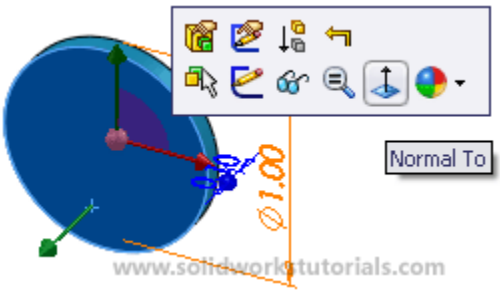


Set **D1** to **0.1in** 

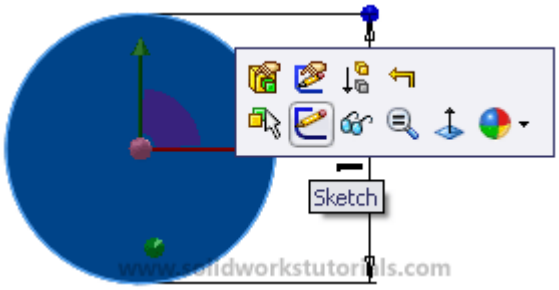
and  .

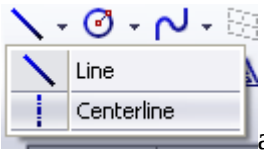


5. Click on front face and click **Normal To**.

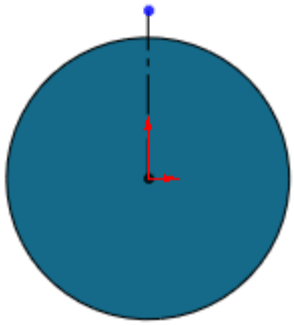


6. Click on front face and click **Sketch**.

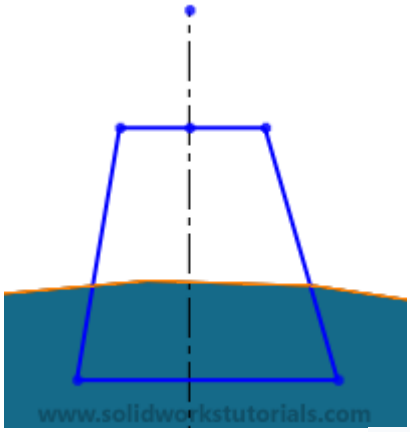


7. Click on **Centerline** 

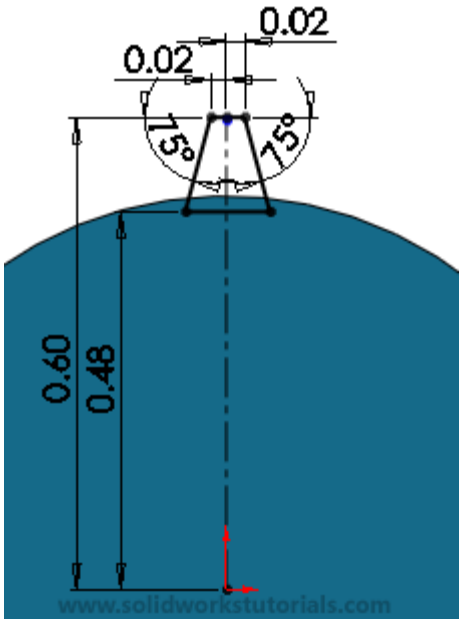
and sketch vertical Centerline.



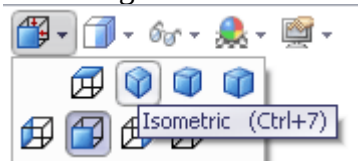
8. Click **Line** and sketch gear teeth profile.

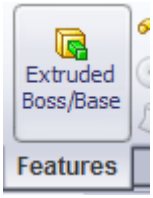


9. Click **Smart Dimension**, dimension sketch as sketched below.

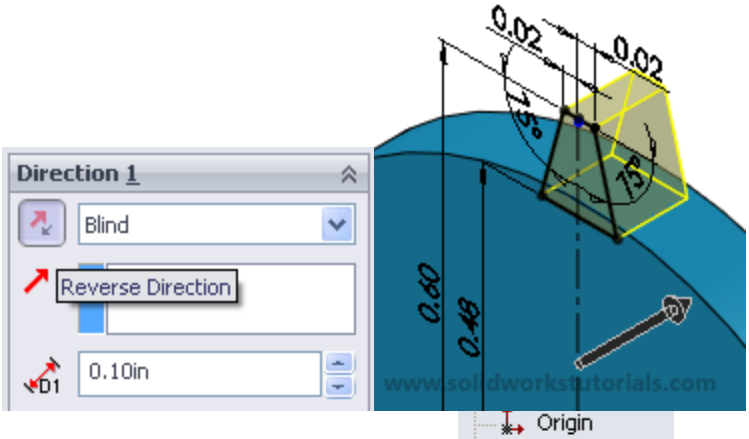


10. Change view to **Isometric**.



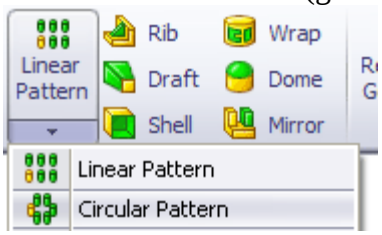
11. Click **Feature>Extruded Boss/Base**. 

Set **D1** to **0.1in**, click **Reverse Direction** and  .

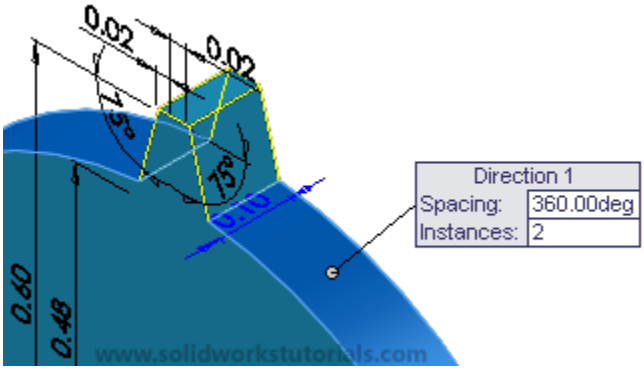


12. Click on **Extrude2** (gear teeth) 

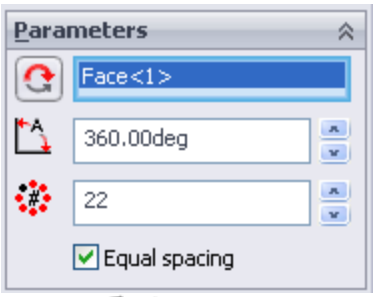
and click **Circular Pattern**.

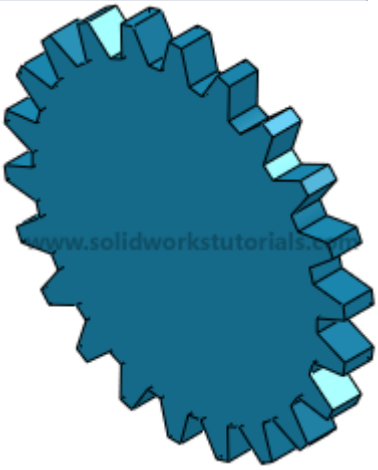


Click on the cylinder face as axis of rotation (or click on View>Temporary Axes select the temporary axis as axis of rotation).

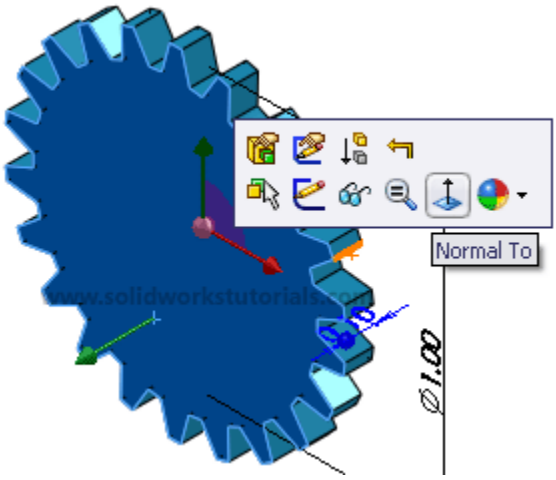


Set **Instances** to **22** and .

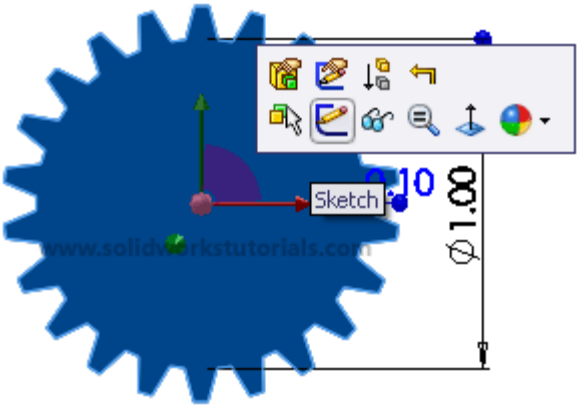




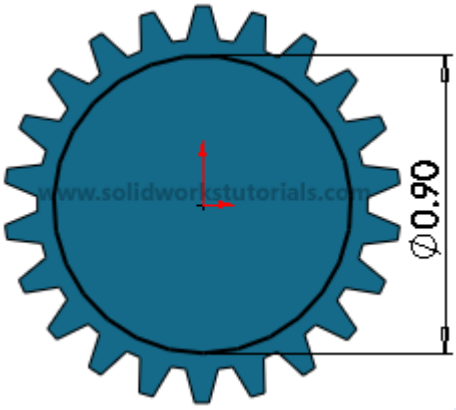
13. Click on Front face and select **Normal To**.



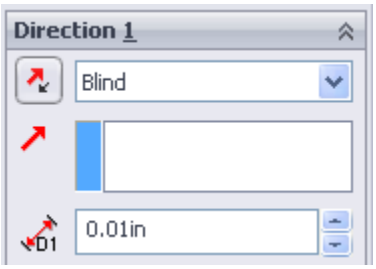
14. Click on front face and select **Sketch**.



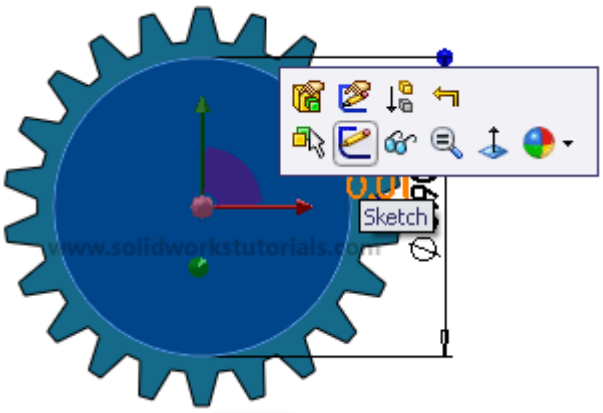
15. Sketch a **Circle** and sketch a circle center at origin. Click **Smart Dimension**, Smart Dimension dimension sketch as **0.9in** circle.



16. Click **Features>Extruded Cut** and set **D1** to **0.01in** and .

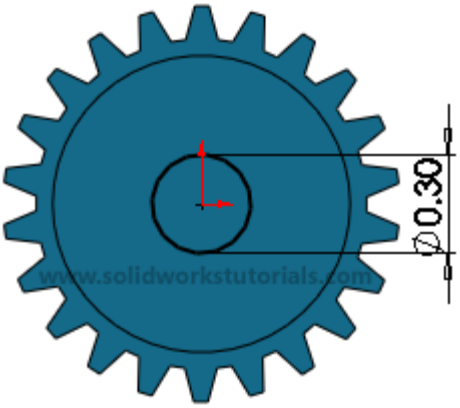


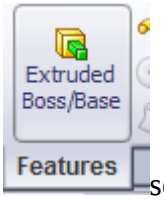
17. Click on inner front face and select **Sketch**.



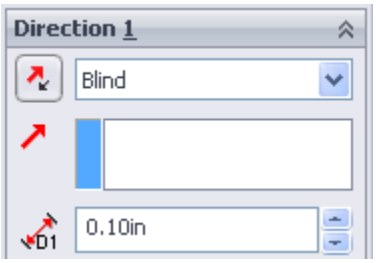
18. Click **Circle**  and sketch a circle center at origin. Click **Smart Dimension**, 

dimension circle as **0.3in** circle.

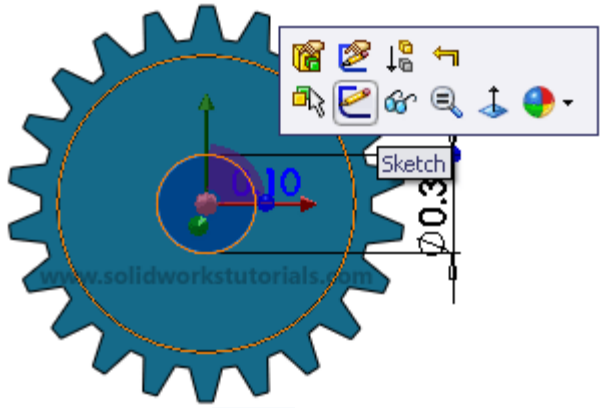


19. Click **Features>Extruded Boss/Base** 

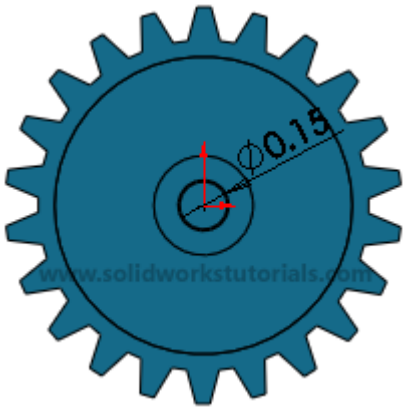
set **D1** to **0.1in** and  .



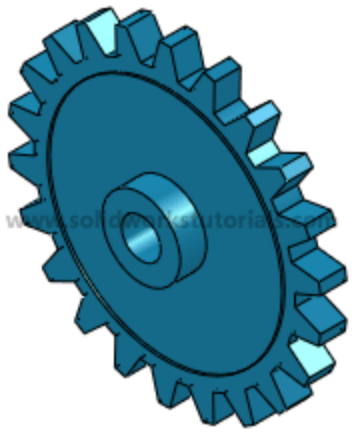
20. Click on center face and select **Sketch**.



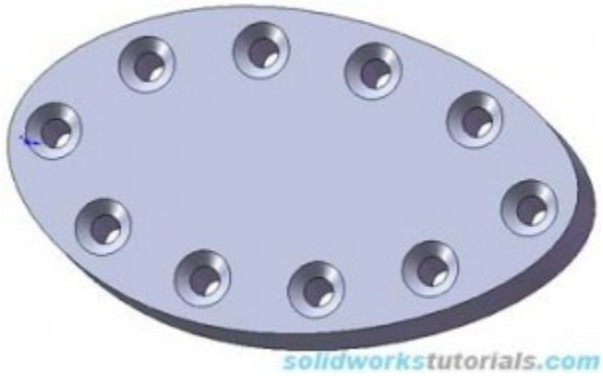
21. Click **Circle** and sketch a circle center at origin. Click **Smart Dimension**, Smart Dimension dimension circle as **0.15in** circle.



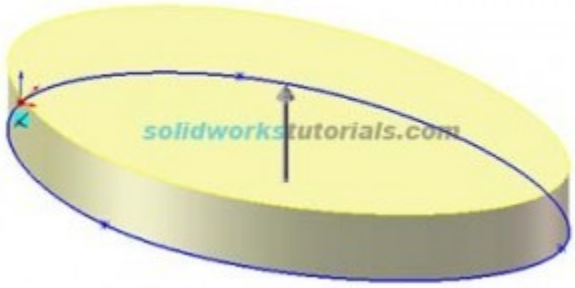
22. Click **Features>Extruded Cut** and set **Direction** to **Through All** and .
23. Repeat Step 13 – 22 to back side face and you're done!



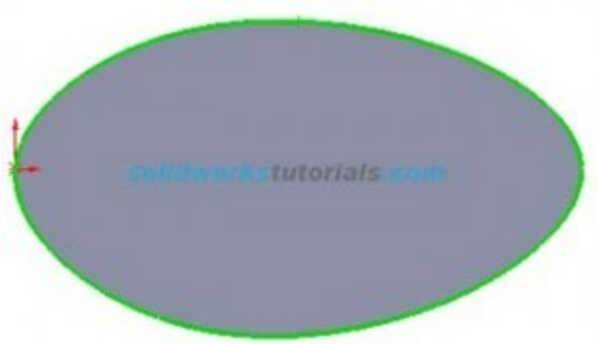
6. Curve Driven Pattern



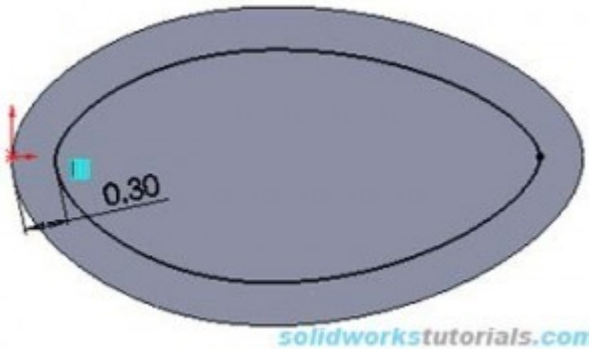
1. Create new part, sketch egg shape on top plane using spline  .

2. Extrude  shape to 0.3in 

, click top face and right click Insert Sketch. 



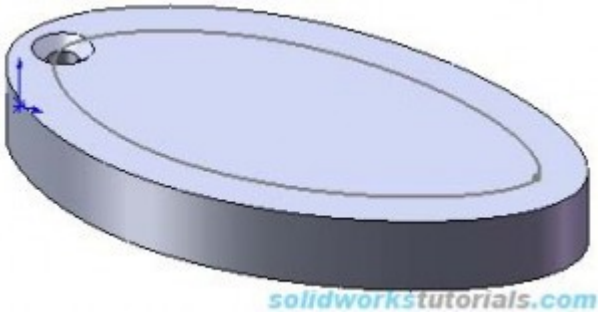
3. Select part edge and click offset 



to 0.3in . Exit Sketch  .

4. Click hole wizard  , select Countersink, Ansi Inch, Flat Head Screw, #10, Normal,

Through All . Click Positions , click screw point at curve edge.

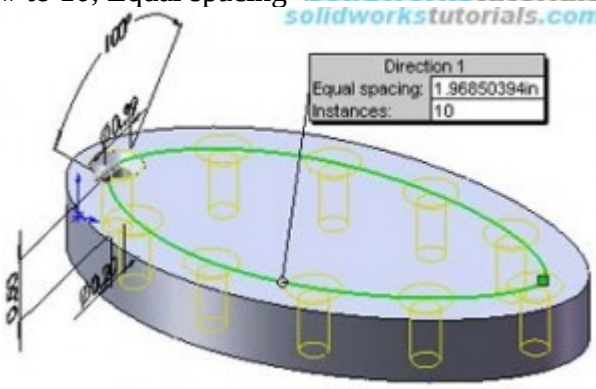


OK.

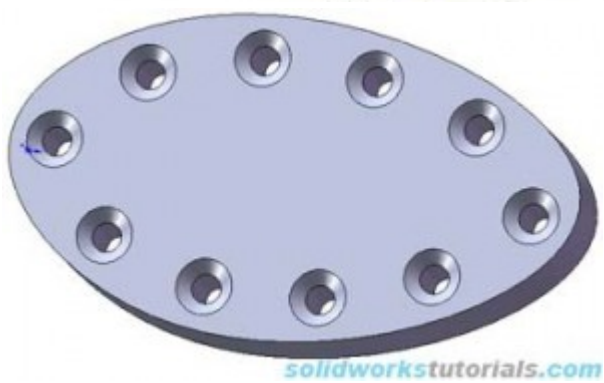
5. Select CSK for #10 Flat Head Machine Screw , Click Insert>Pattern/Mirror>Curve Driven Pattern…

Define Pattern, select spline sketch and set

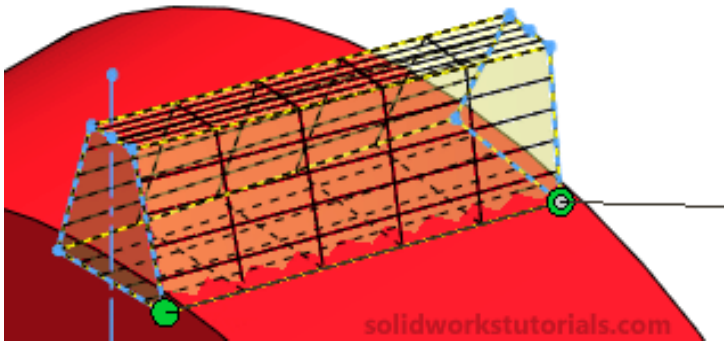
to 10, Equal spacing 



. OK, done.

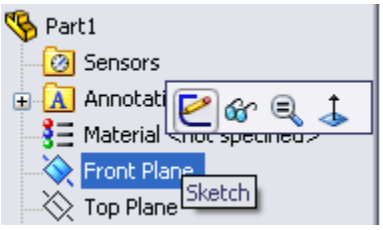


7. How to create helical gear



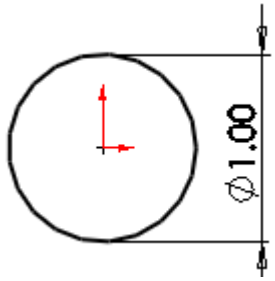
In this solidworks tutorial, you will create helical gear.

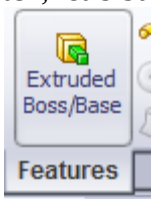
1. Click **New**.  Click **Part**,  **OK**.
2. Click **Front Plane** and click on **Sketch**.

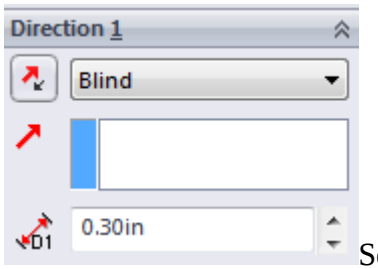


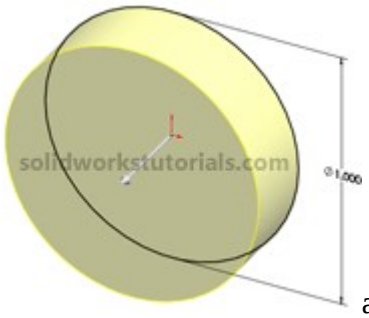
3. Click **Circle**  and sketch a circle center at origin. Click **Smart Dimension**, 

click sketched circle and set it diameter to **1.0in**.



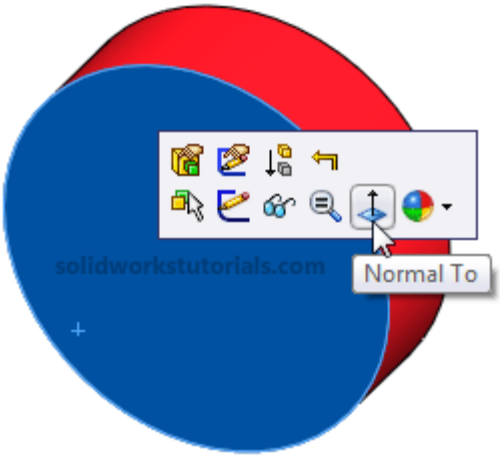
4. You just completed your sketch, let's build feature from it. Click **Features>Extruded Boss/Base**. 



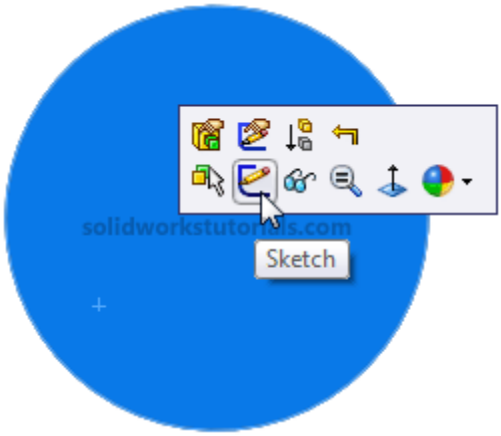
Set **D1** to **0.3in** 

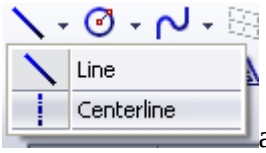
and  .

5. Click on front face and click **Normal To**.

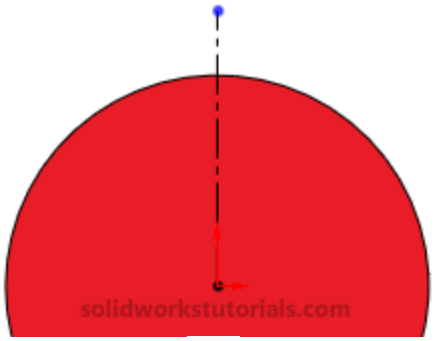


6. Click on front face and click **Sketch**.

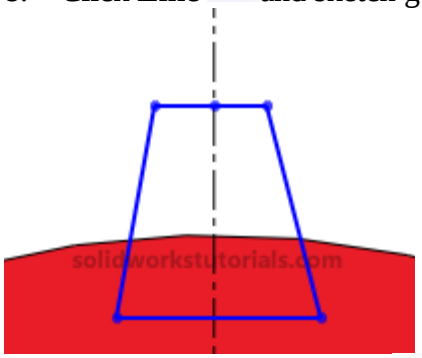


7. Click on **Centerline** 

and sketch vertical Centerline.

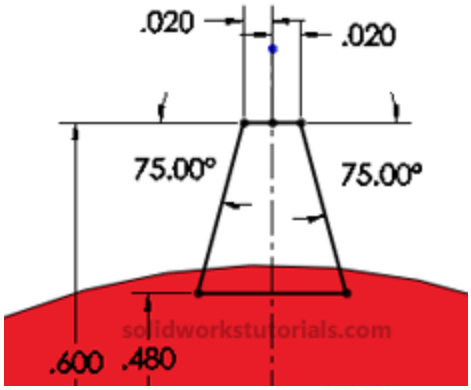


8. Click **Line** and sketch gear teeth profile.

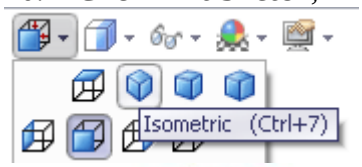


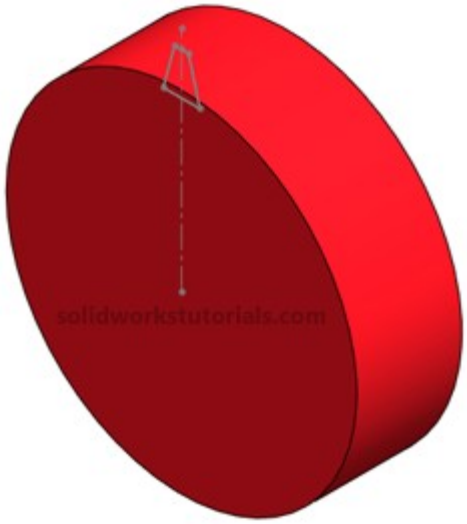
9. Click **Smart Dimension**, 

dimension sketch as sketched below.

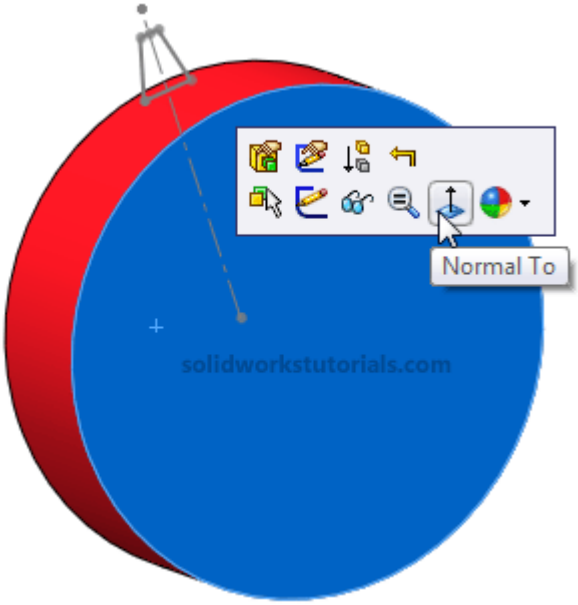


10. Click **Exit Sketch**,  change view to **Isometric**.

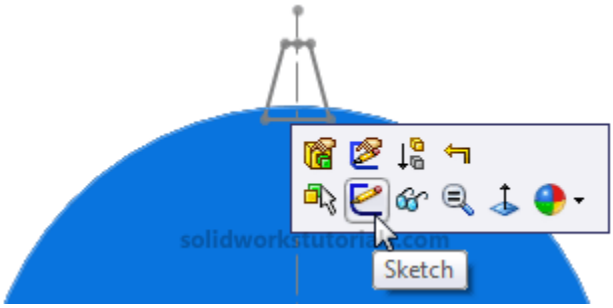




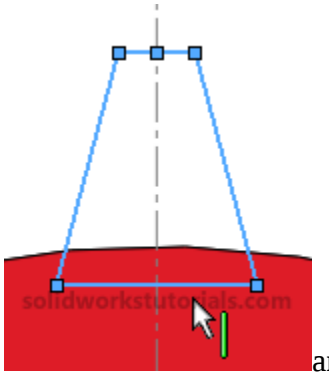
11. Click scroll mouse button and rotate the part to back side.



Click the back face and select **Normal To.** Click on this face again and click **Sketch.**



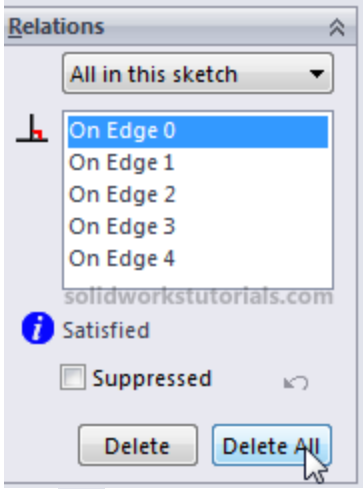
12. We will trace last sketch to this face, while holding **CTRL** click **all sketched line**



and click **Convert Entities** 

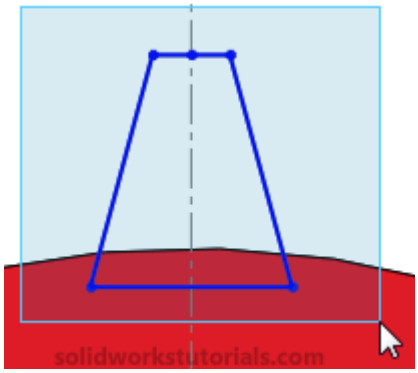
. Now we need removed all relation between this sketch and the other sketch, click **Display/Delete Relations**

click **Delete All**

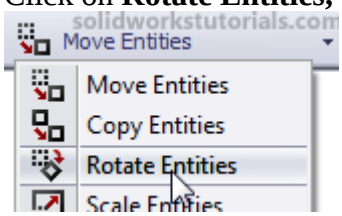


and  .

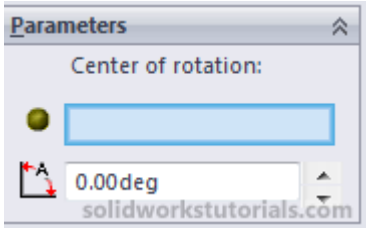
13. Click and drag select all the sketch line.



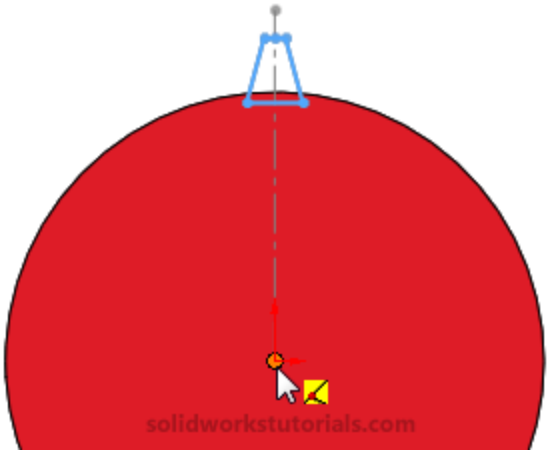
Click on **Rotate Entities,**



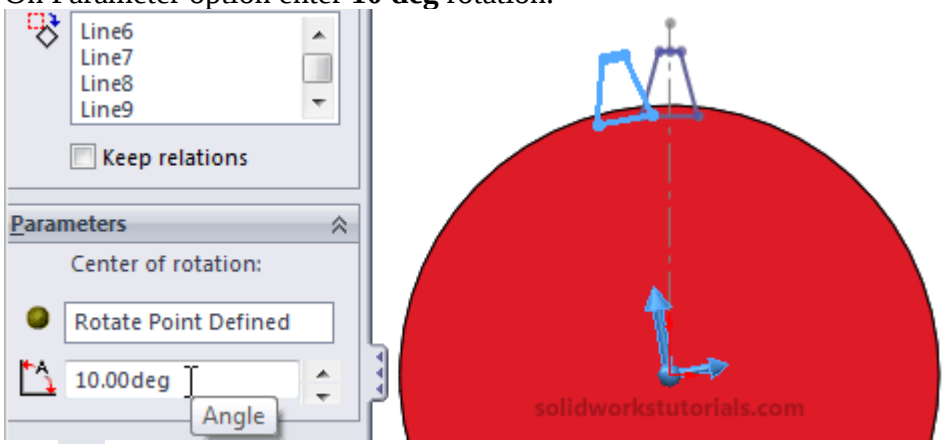
Click **Center of Rotation box**



and click **origin (center part).**

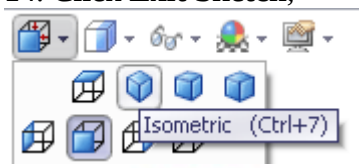


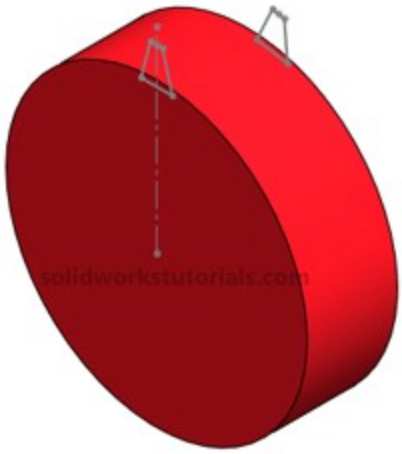
On Parameter option enter **10 deg** rotation.



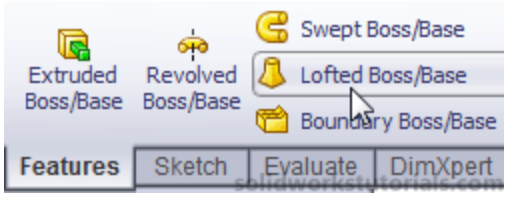
and  .

14. Click **Exit Sketch**,  change view to **Isometric**.

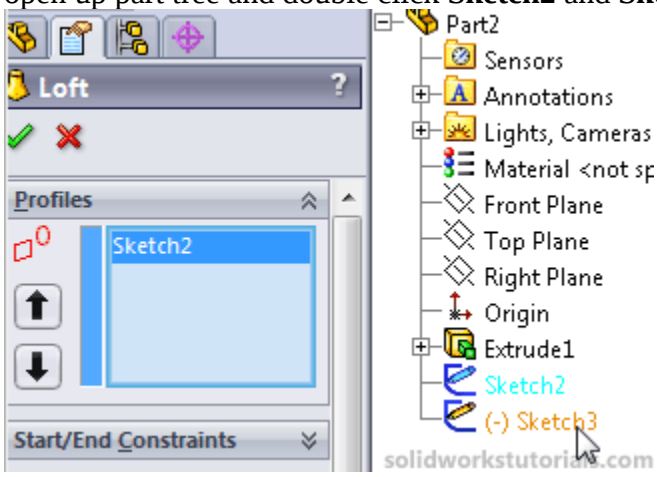


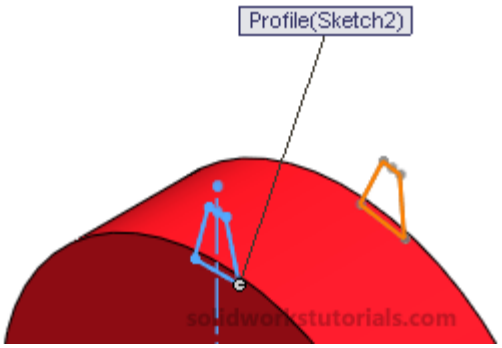


15. Click **Features>Lofted Boos/Base,**

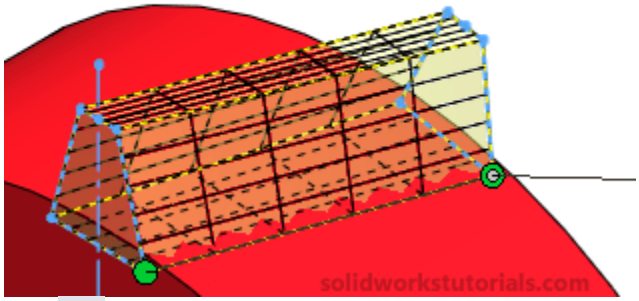


open up part tree and double click **Sketch2** and **Sketch3** to add for lofted features.

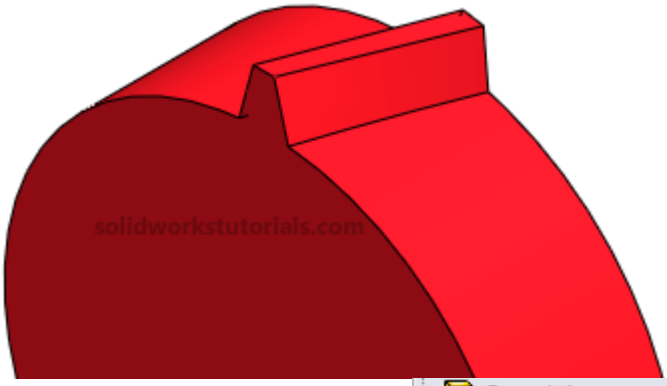


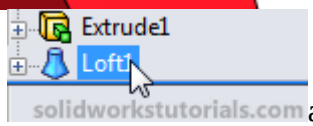


Make sure two green point is at the same edge as other sketch, if not drag and relocate it.

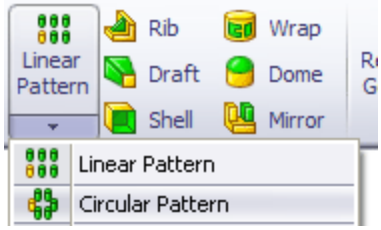


and  .



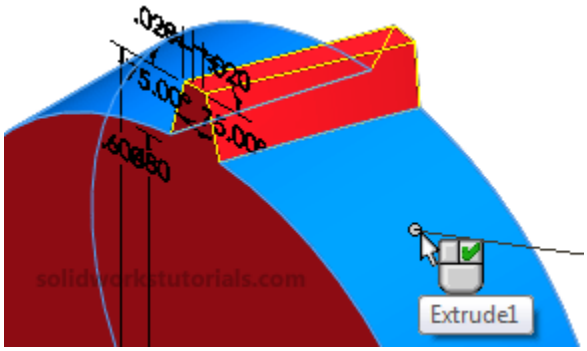
12. Click on **Loft1** (gear teeth) 

and click **Circular Pattern**.

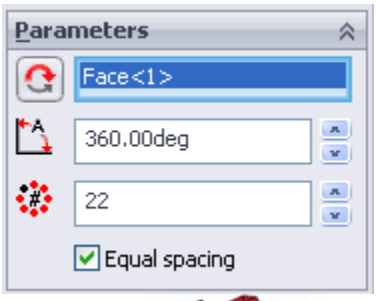


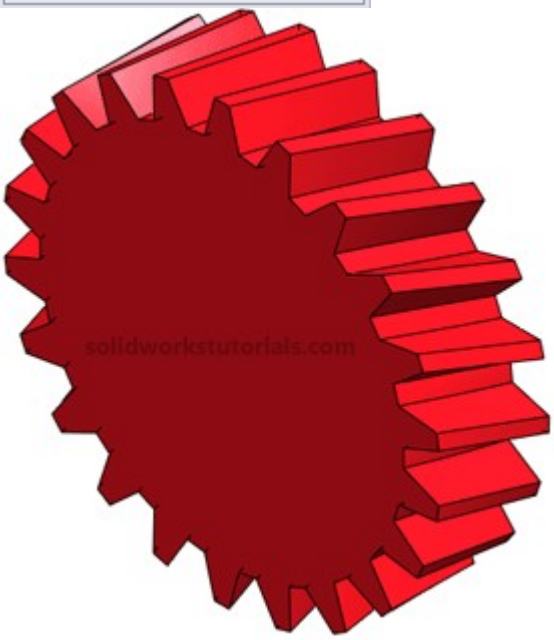
Click on the cylinder face as axis of rotation (or click on View>Temporary Axes select

the temporary axis as axis of rotation).

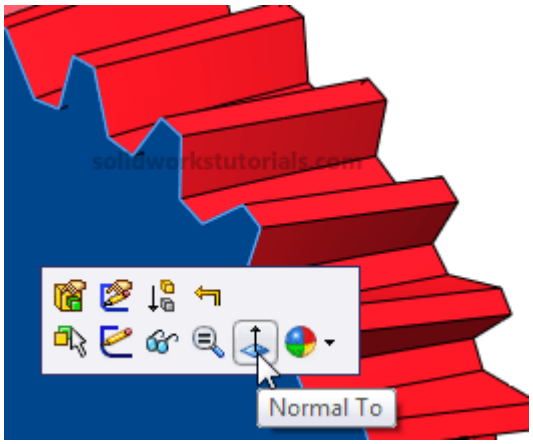


Set **Instances** to **22** and  .

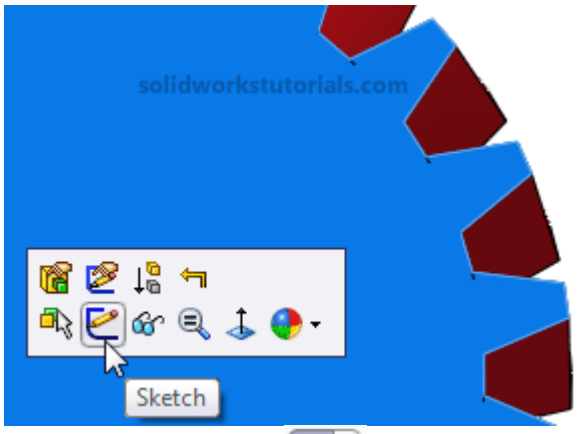




13. Click on Front face and select **Normal To**.

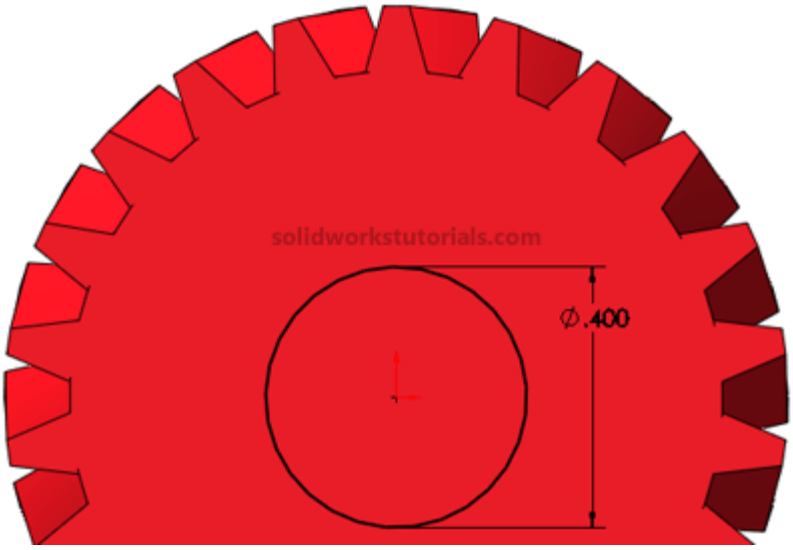


14. Click on front face and select **Sketch**.



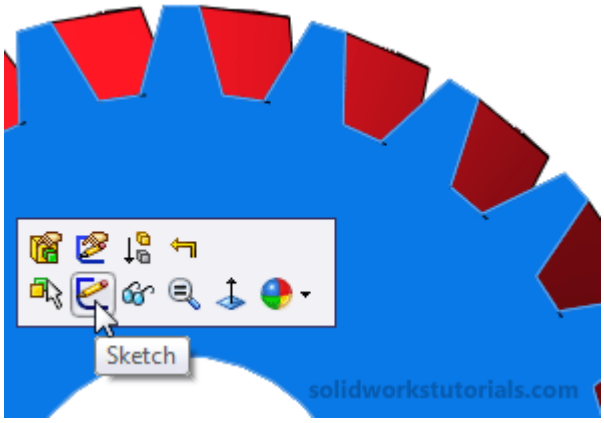
15. Sketch a **Circle**  and sketch a circle center at origin. Click **Smart Dimension**, 

dimension sketch as **0.40in** circle.

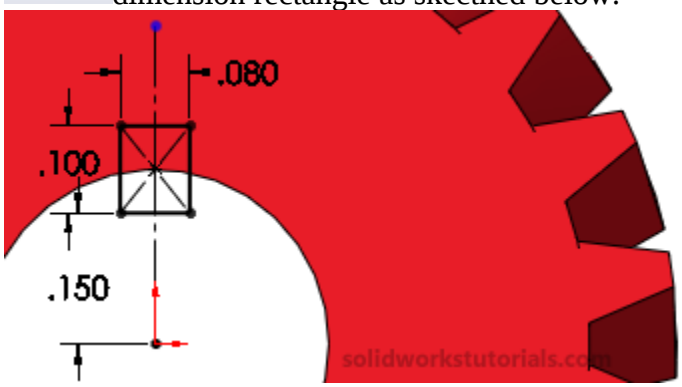


16. Click **Features>Extruded Cut** and set **Direction** to **Through All** and .

17. Click on front face and select **Sketch**.



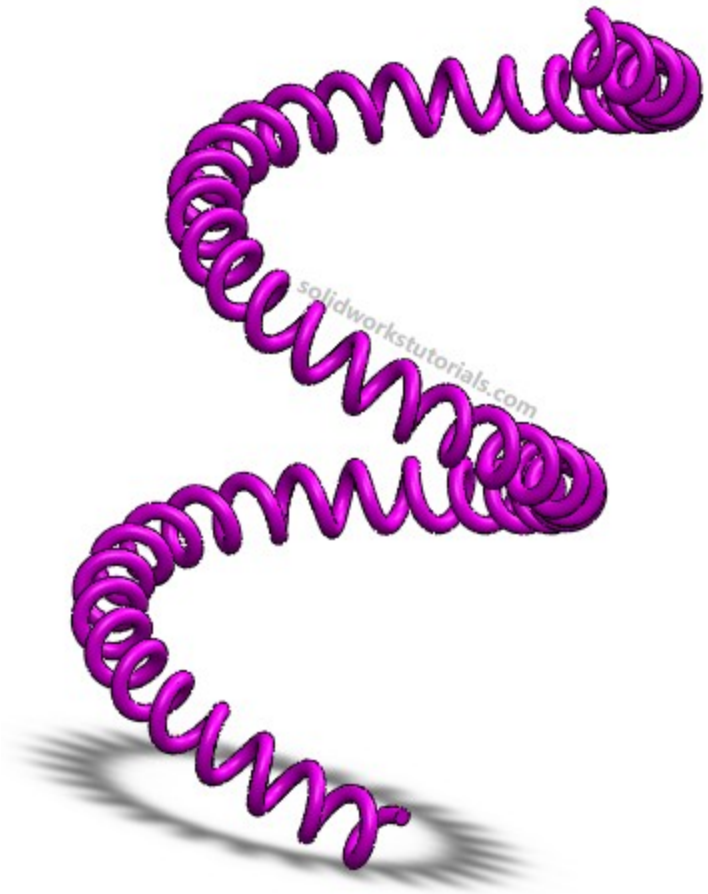
18. Click **Rectangle** and sketch a rectangle as sketched. Click **Smart Dimension**, Smart Dimension dimension rectangle as skecthed below.

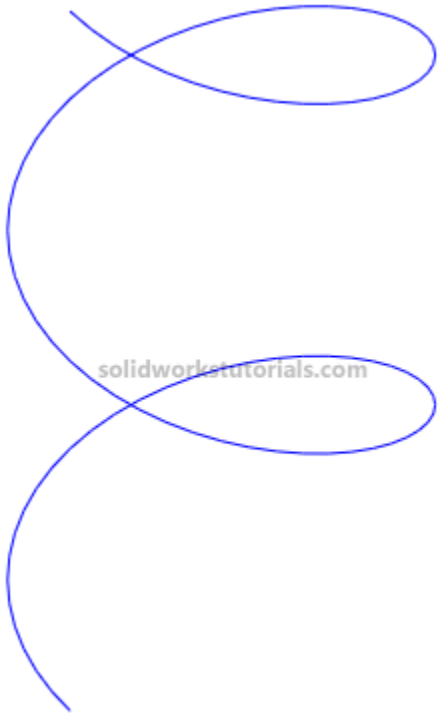


16. Click **Features>Extruded Cut** and set **Direction** to **Through All** and . You're done!

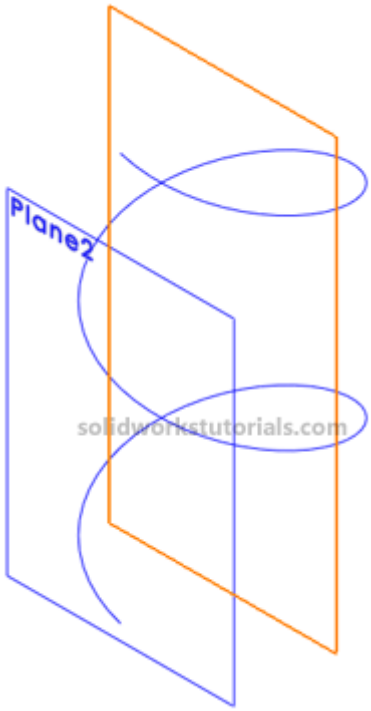
8. How to twist phone cord

Learn how to create twist phone cord… with you mouse

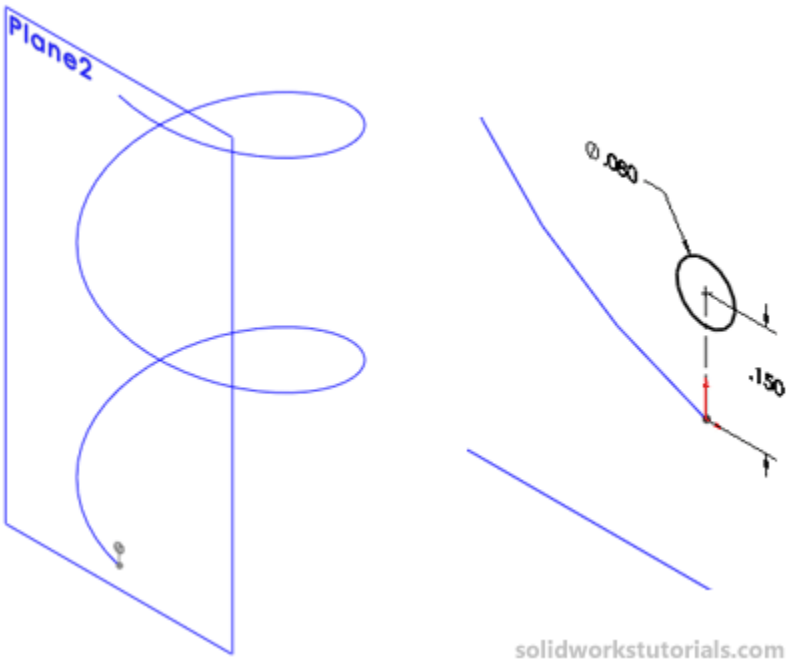

solidworkstutorials.com


solidworkstutorials.com

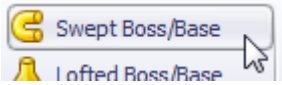
First you need to have spiral, with circle base 2″ , 2 revolution and 2 pitch. Don't know how? Refer this tutorial; [Tutorial #2: How to create simple spring]



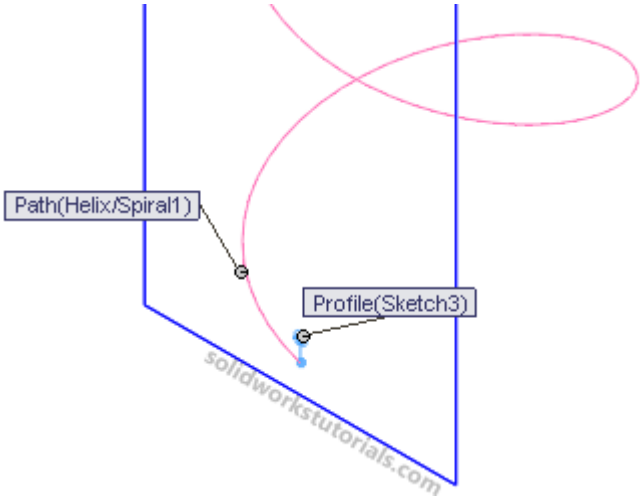
Now add a plane at end of spiral, select parallel to front plane.



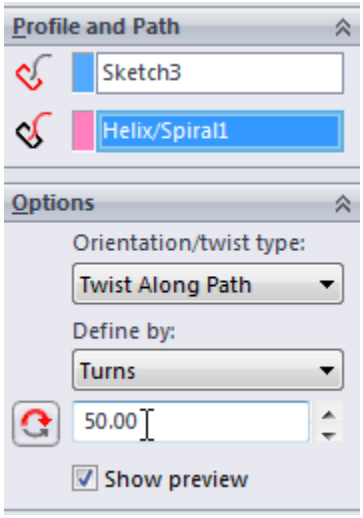
Sketch a circle on Plane2, 0.08″ and 0.15 height. Click Swept Boss/Base.

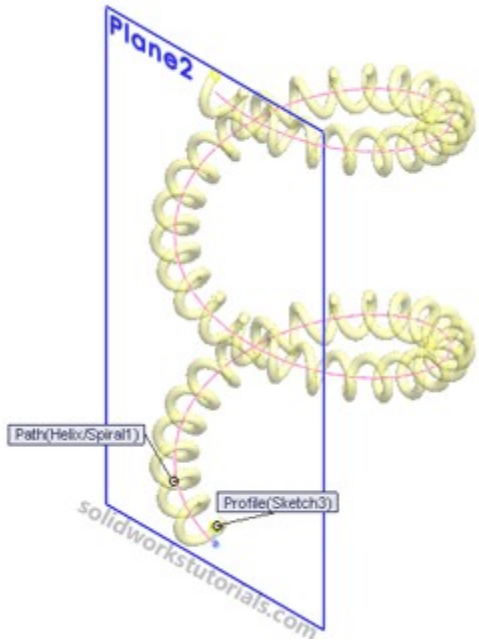


Select Sketch3 as profile and Helix/Spiral1 as path.



Open up Options and set Twist Along Path, define by Turns and 50 turns.

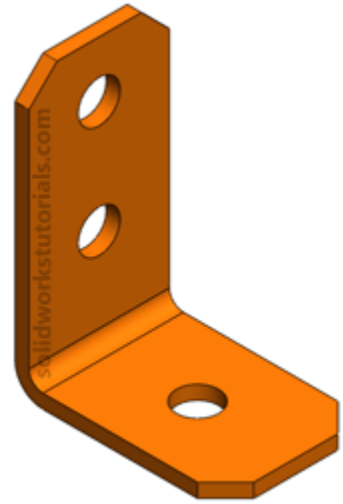




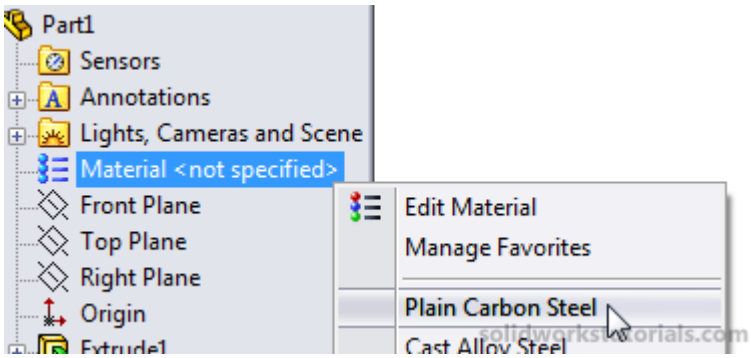
And OK you're done!

9. How to check part mass

You can check part mass by assigning part material first, example for this bracket

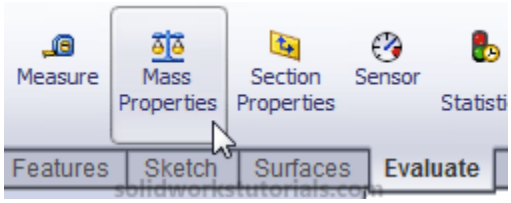


I set it material as plain carbon steel

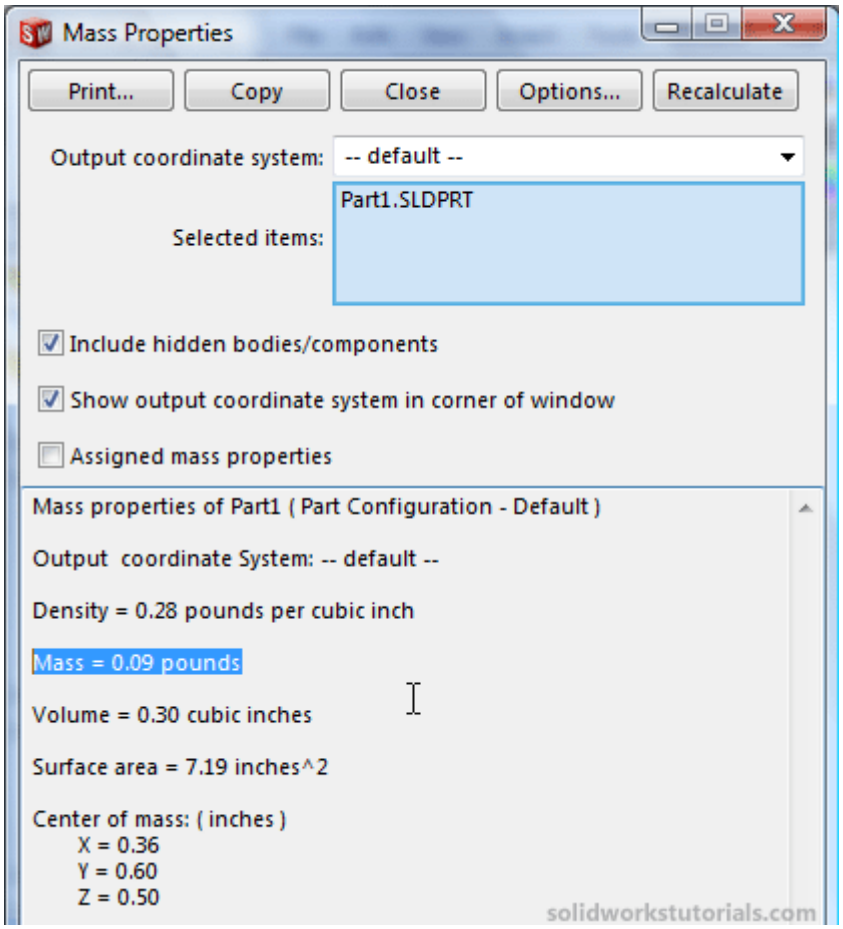


Now material set as Plain Carbon Steel

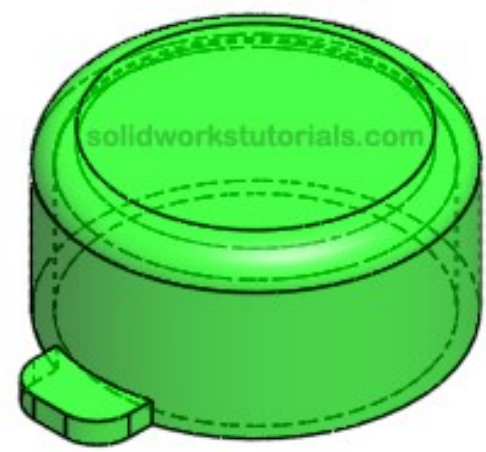
Click Evaluate>Mass properties



and you got the mass is 0.09 pounds. You also can check it volume, surface area and center of mass.



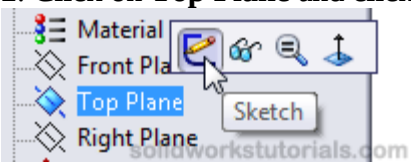
10. How to create bottle cap



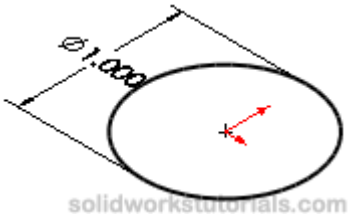
I get this idea from my medicine bottle cap, the tips here show you how you can use extrude up to the face function.

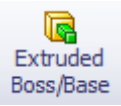
1. Click **New**  , Click **Part**  and **OK**.

2. Click on **Top Plane** and click **Sketch**.

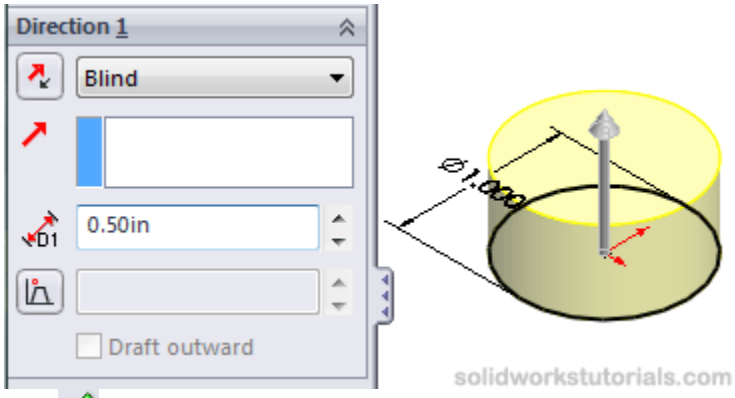


3. Click **Circle**  and sketch start at Origin, click **Smart Dimension**  and dimension the circle as **1.0in** diameter.



4. Click **Features>Extrude Boss/Base** 

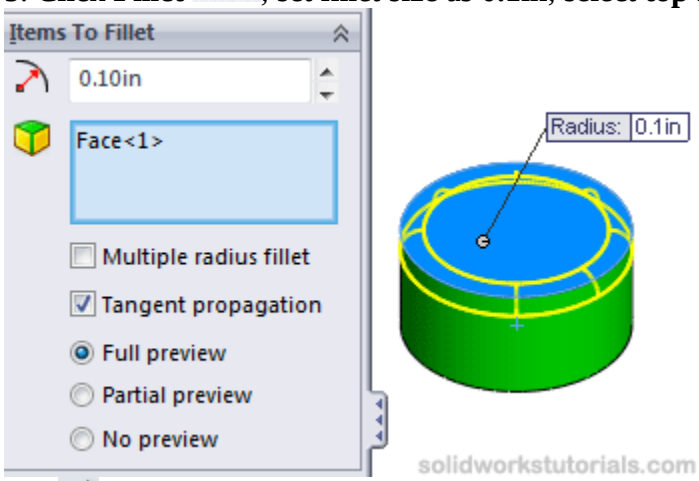
set the D1 to **0.5in**



and ✔.

5. Click **Fillet** 

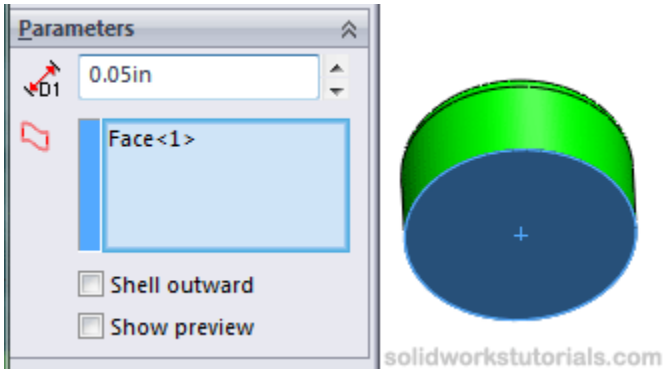
, set fillet size as **0.1in**, select **top face** of the part



and ✔.

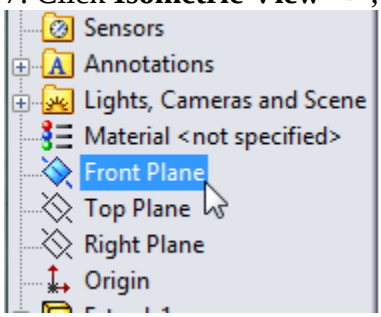
6. Turn the part to view bottom side, set D1 as **0.05in**, click **Shell** 

, select **bottom face**

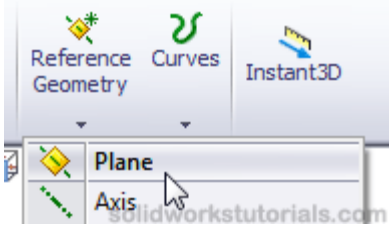


and .

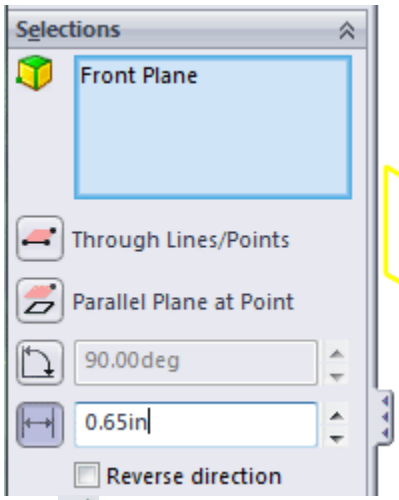
7. Click **Isometric View** , click on **Front Plane**

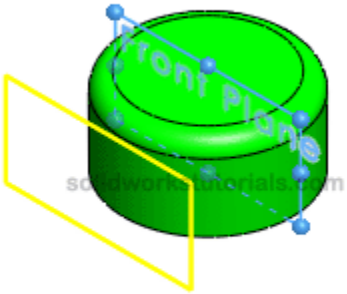


and click on **Reference Geometry>Plane**.



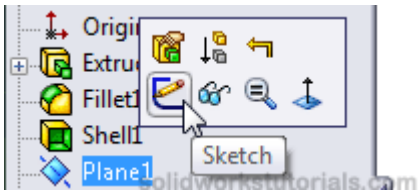
Set distance to **0.65in**



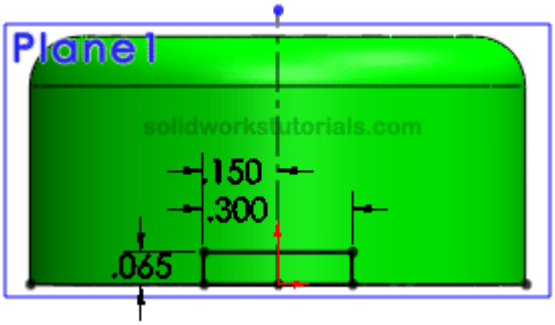


and .

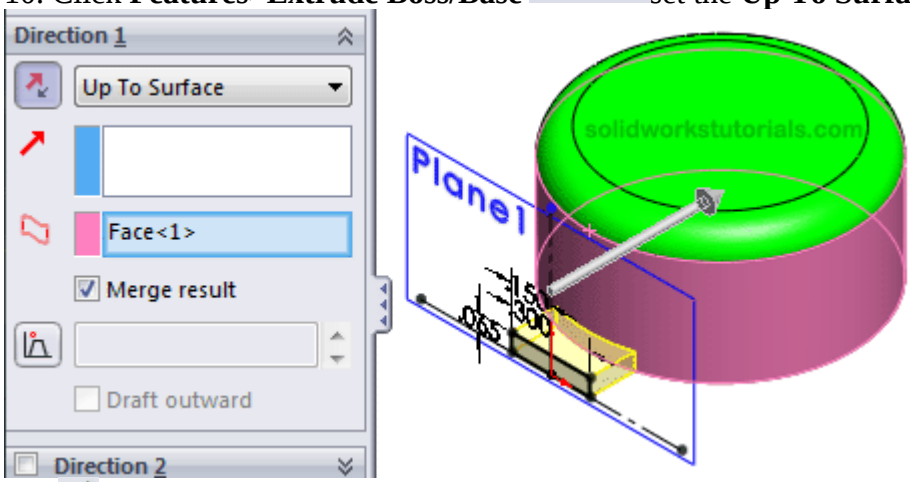
8. Click **Plane1** and click **Sketch**.



9. Click **Rectangle** , sketch on **Plane1** as sketched below and use **Smart Dimension** for your dimensioning.



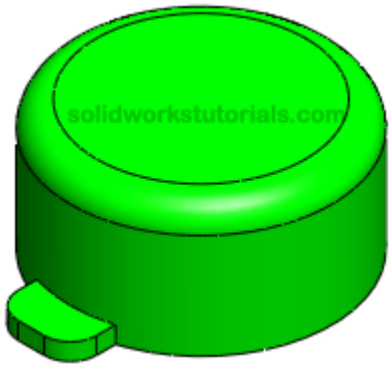
10. Click **Features>Extrude Boss/Base** Extruded Boss/Base set the **Up To Surface**



and .

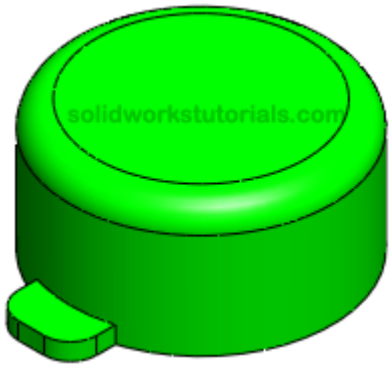
11. Click **Fillet** 

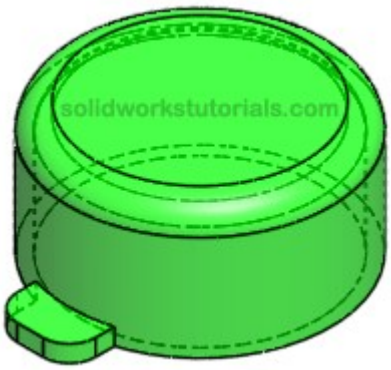
, set fillet size as **0.1in**, select **side edge of the lid**.



and  .

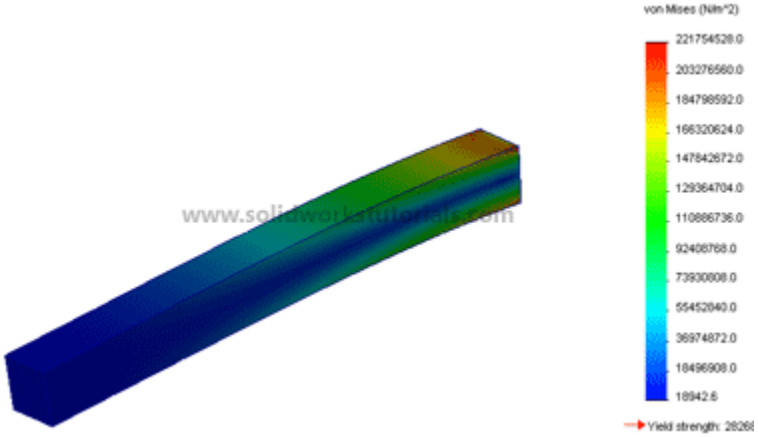
12. And you're done!



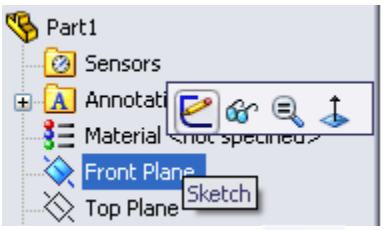


11. Stress strength test in Solid Works

In this tutorial, you will analyze this part using **SimulationXpress** in solidworks

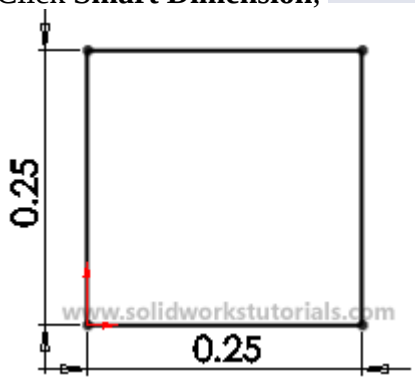


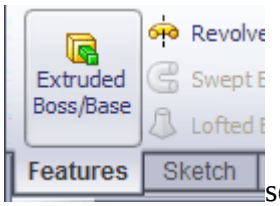
1. Click **New**.  Click **Part**,  **OK**.
2. Click **Front Plane** and click on **Sketch**.



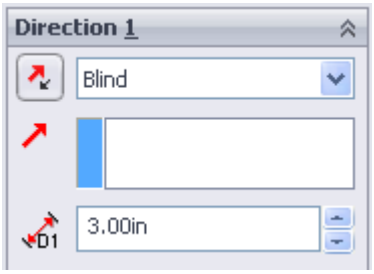
3. Click **Rectangle**,  sketch a rectangular.

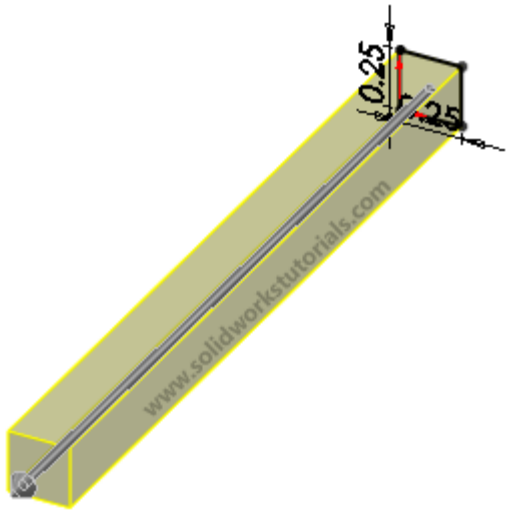
Click **Smart Dimension**,  dimension rectangular **0.25in x 0.25in**.



4. Click **Feature**>**Extruded Boss/Base**, 

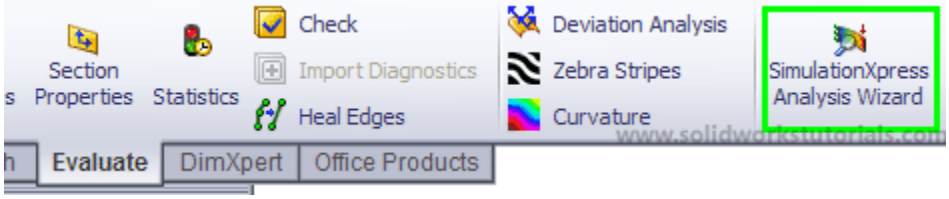
set **D1** to **3.0in**



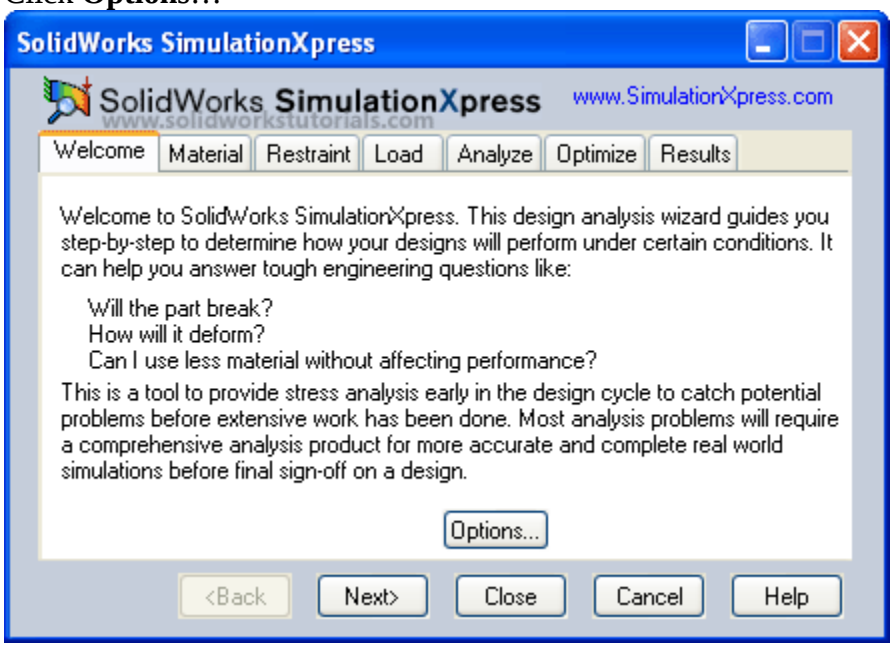


and ✓.

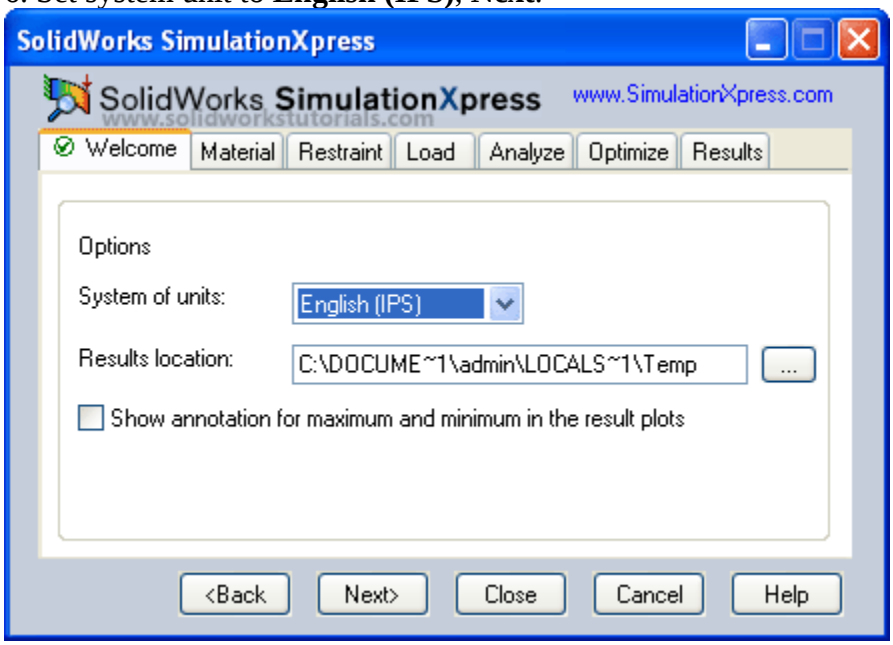
5. For analysis, click **Evaluate**>**SimulationXpress Analysis Wizard**



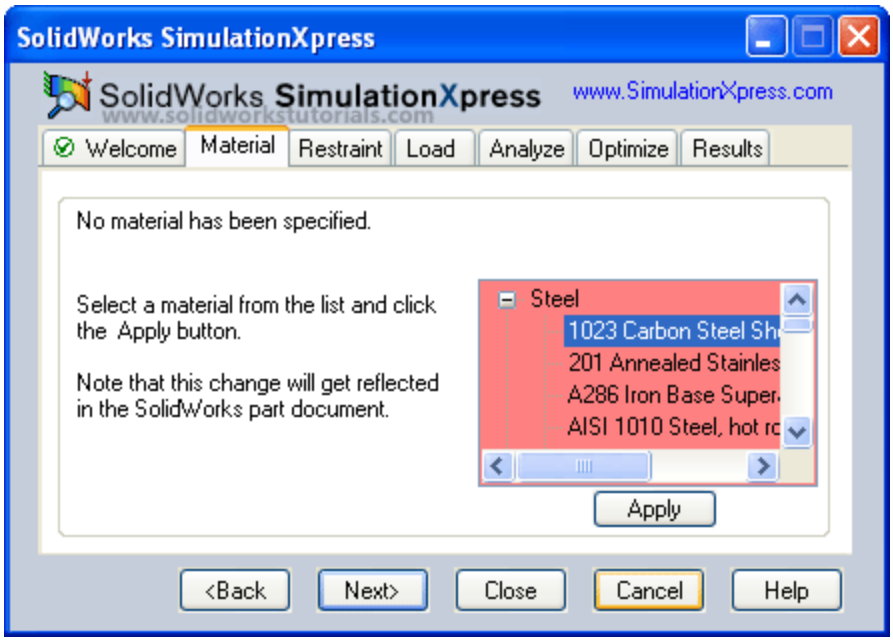
Click **Options**…



6. Set system unit to **English (IPS)**, **Next**.



7. Set materials type, select **Steel**, **1023 Carbon Steel**, **Apply**. **Next**.


SolidWorks SimulationXpress
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Results
No material has been specified.
Select a material from the list and click the Apply button.
Note that this change will get reflected in the SolidWorks part document.
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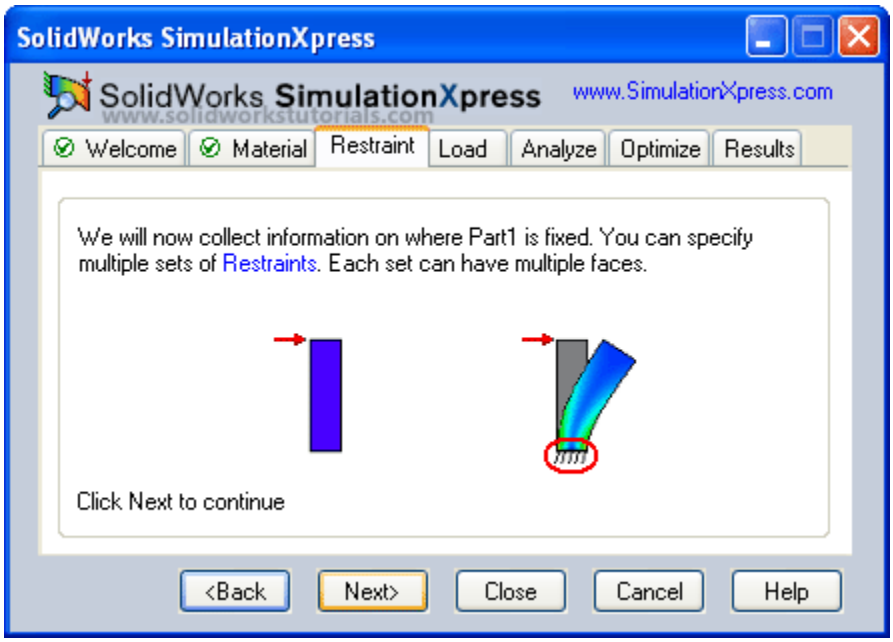
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Welcome
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Optimize
Results
We will now collect information on where Part1 is fixed. You can specify multiple sets of Restraints. Each set can have multiple faces.
Click Next to continue
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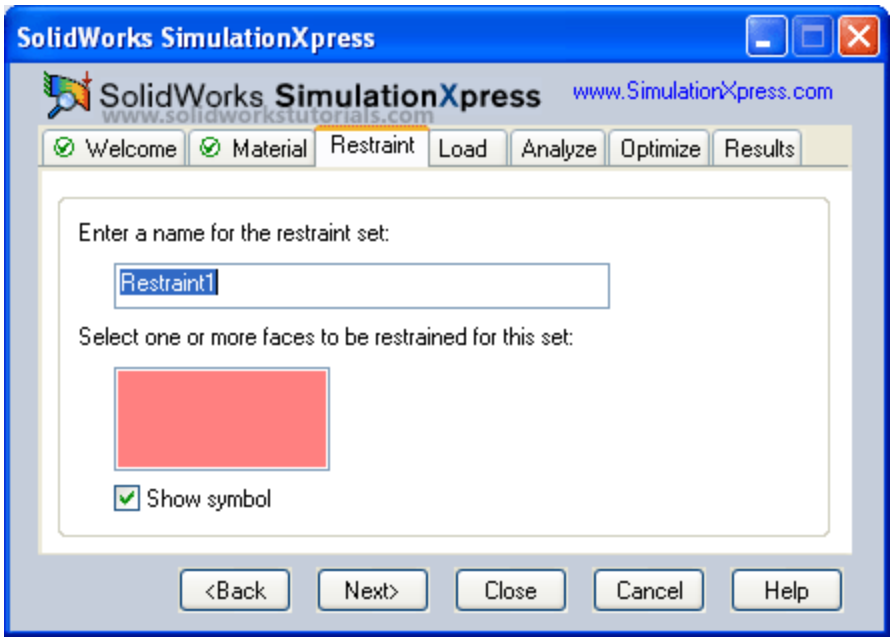
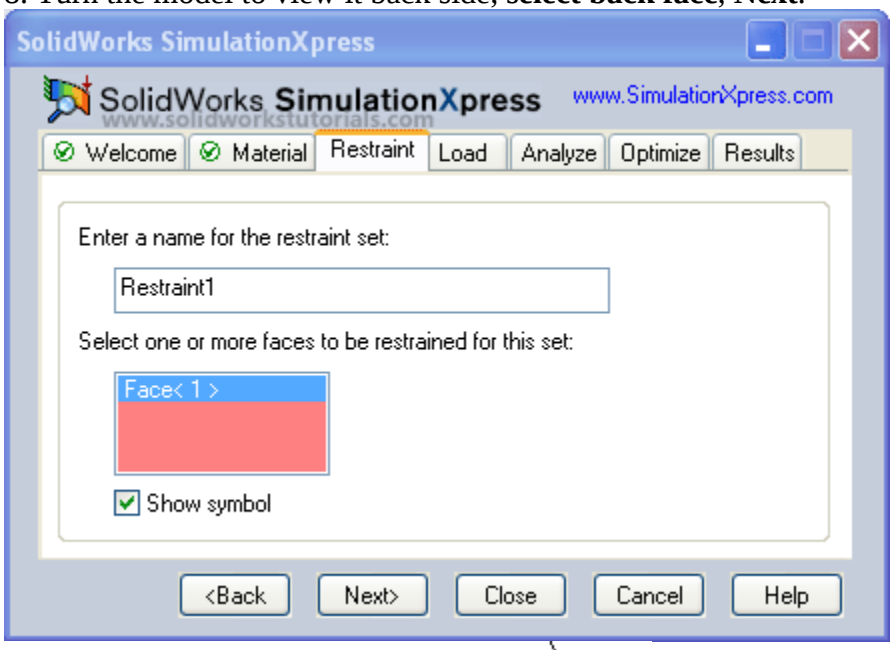
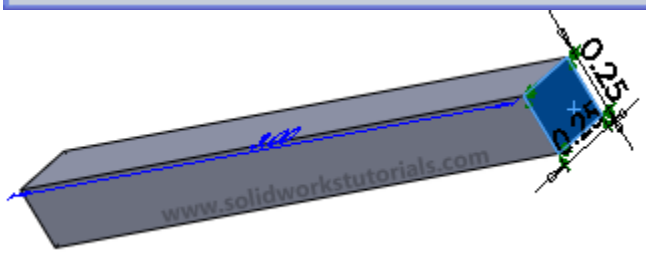
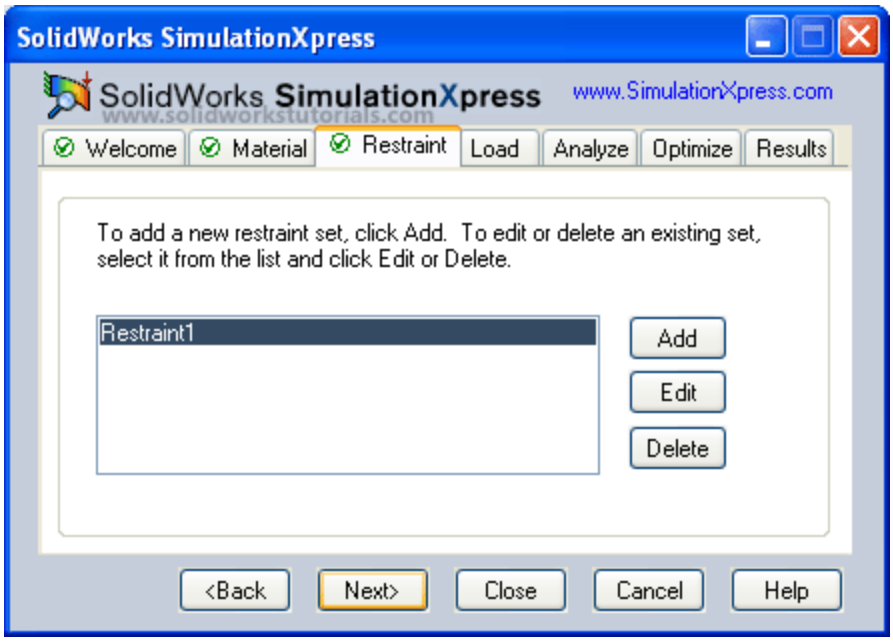


8. Turn the model to view it back side, **select back face**, **Next**.

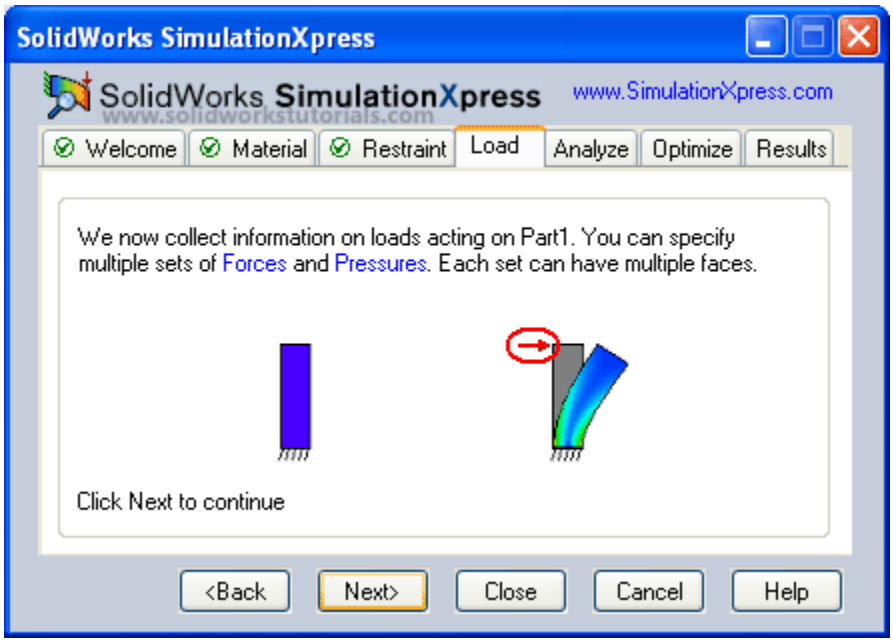




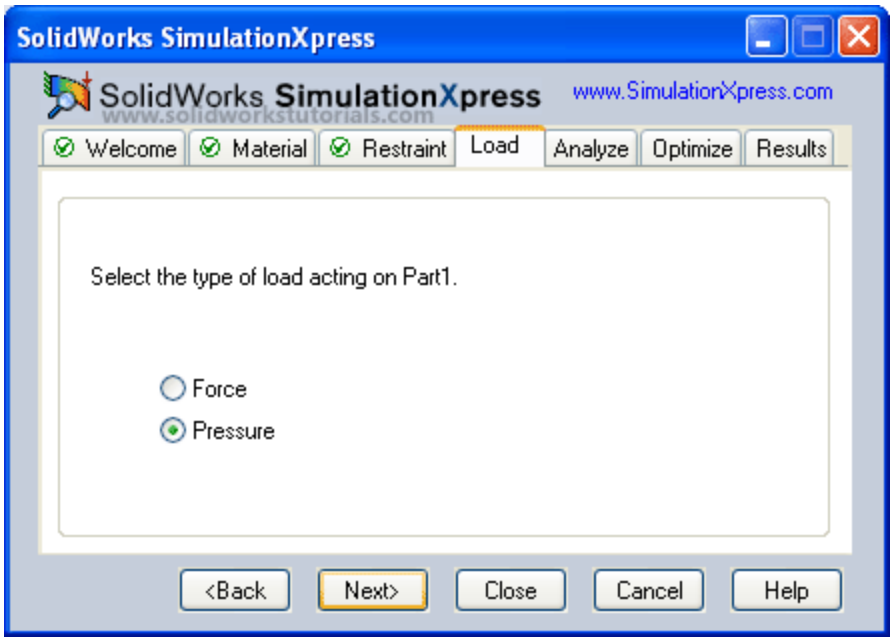
Next.



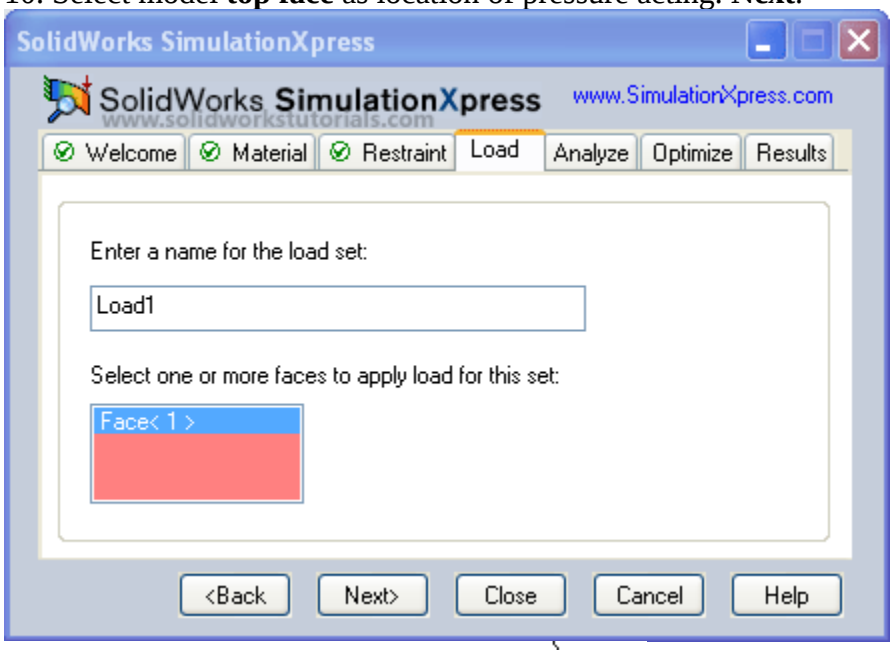
Next.

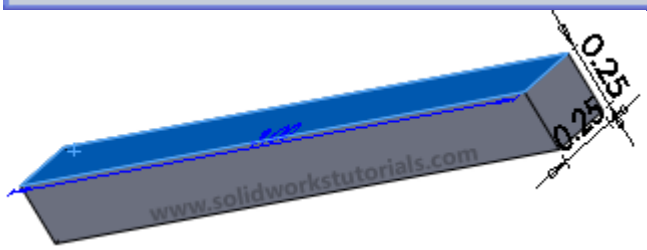


9. Select load type, Click **Pressure**. **Next**.

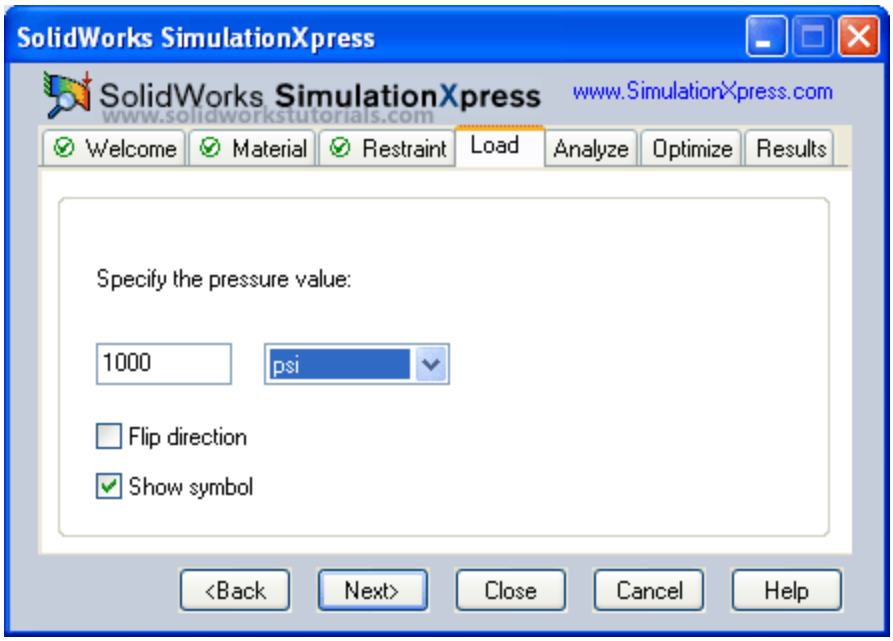


10. Select model **top face** as location of pressure acting. **Next**.





11. Set pressure value to **1000psi**, **Next**.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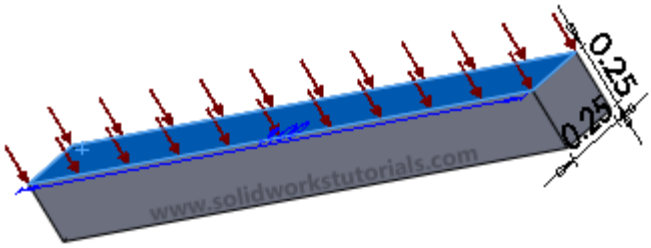
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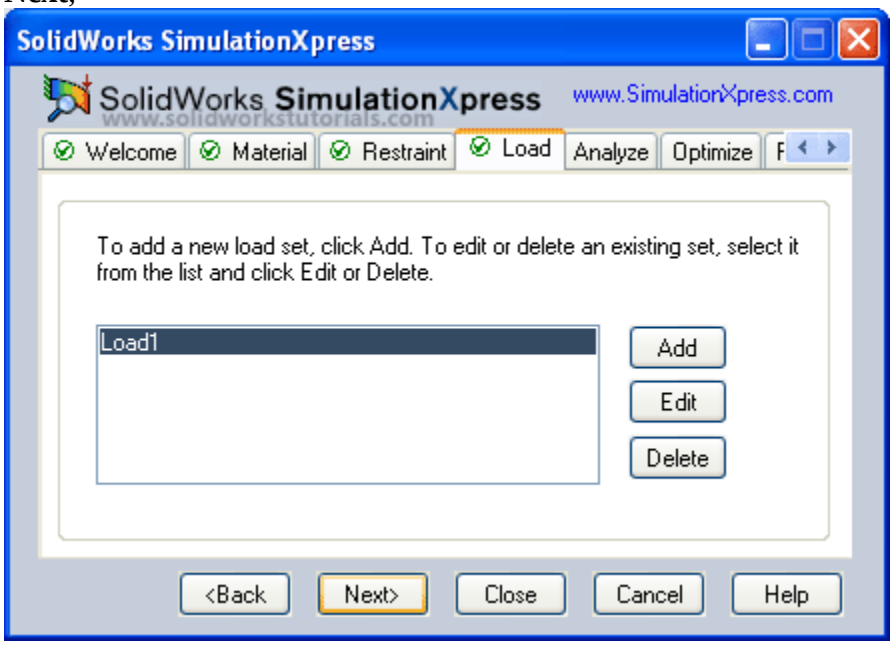
SolidWorks SimulationXpress
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Welcome
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To add a new load set, click Add. To edit or delete an existing set, select it from the list and click Edit or Delete.
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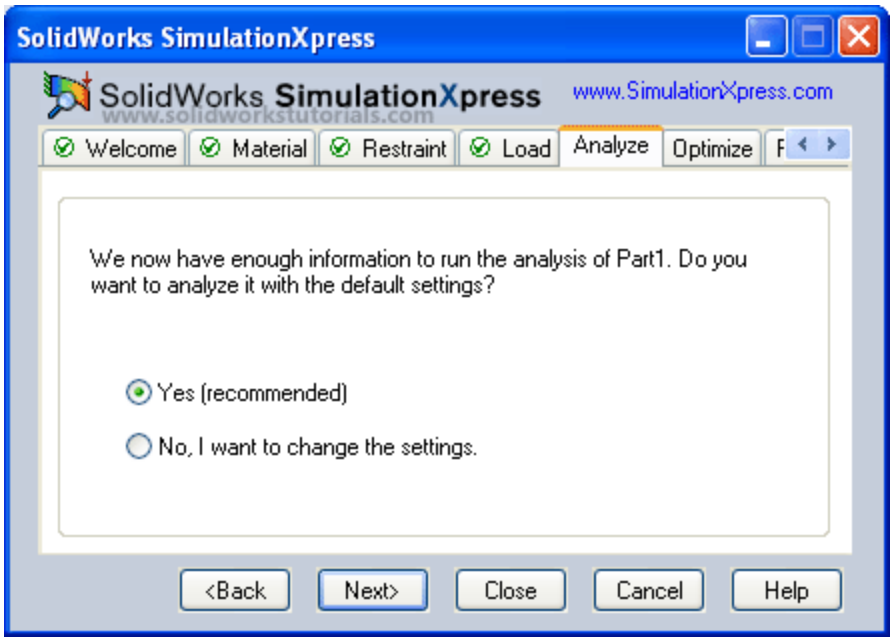
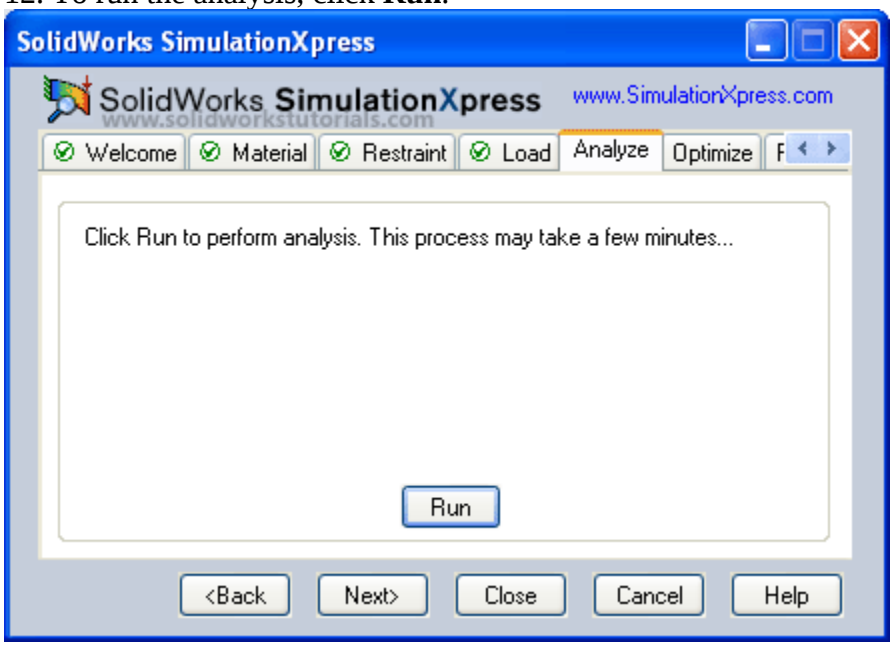
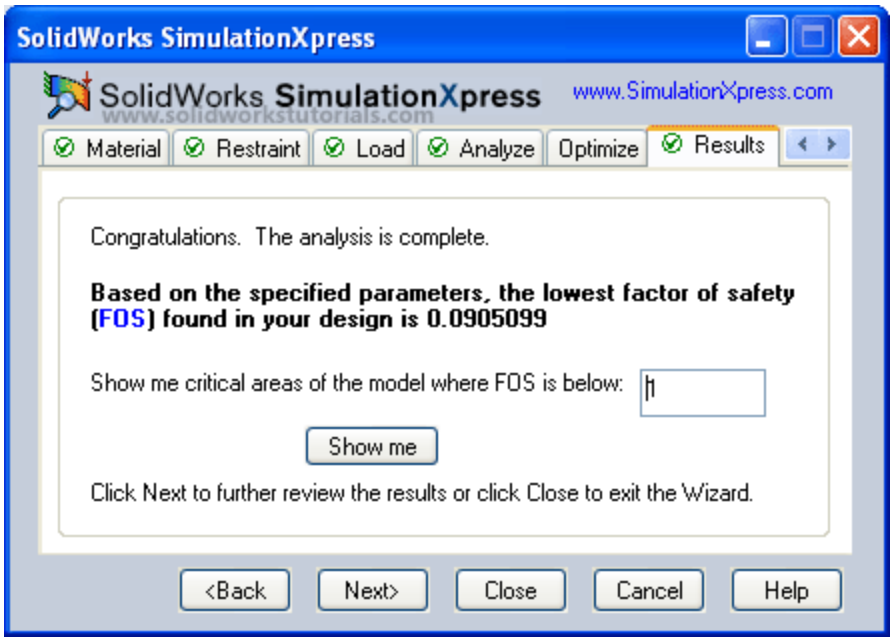


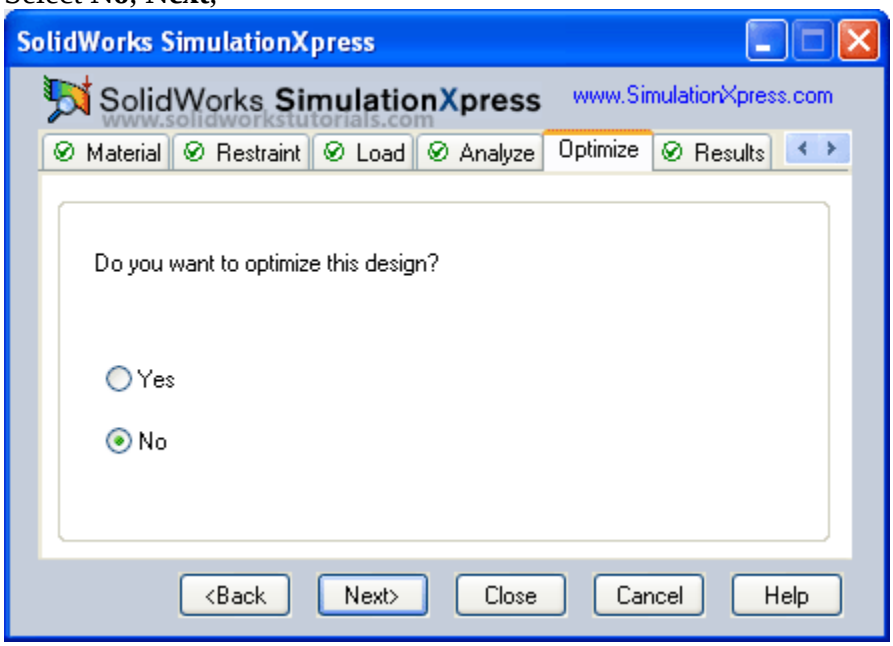
12. To run the analysis, click **Run**.



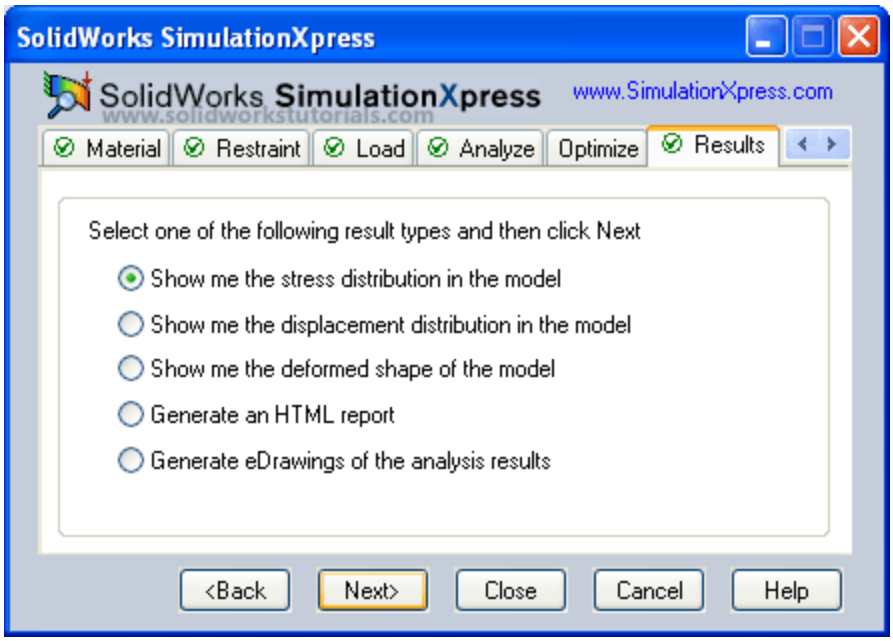
Next,



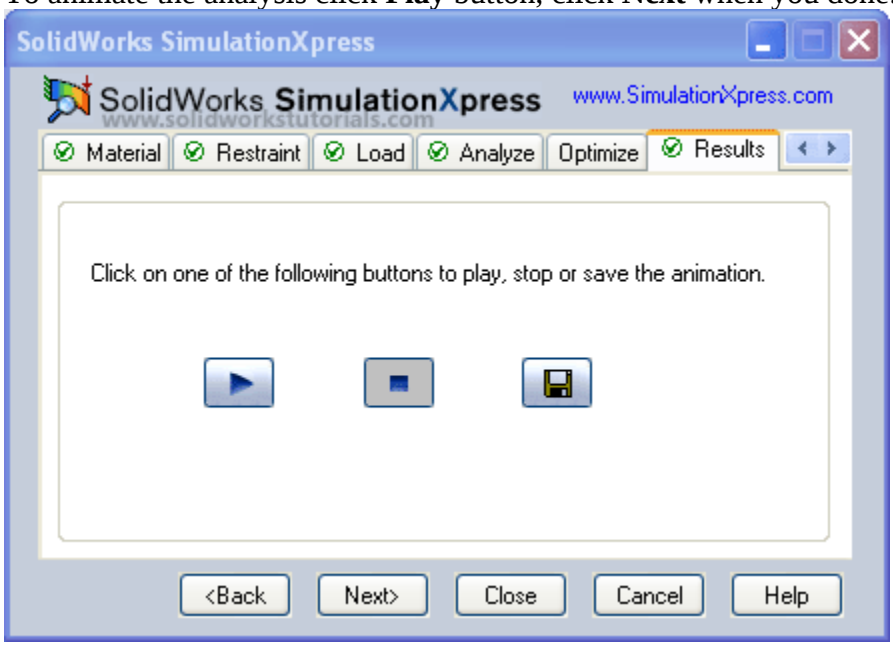
Select **No**, **Next**,

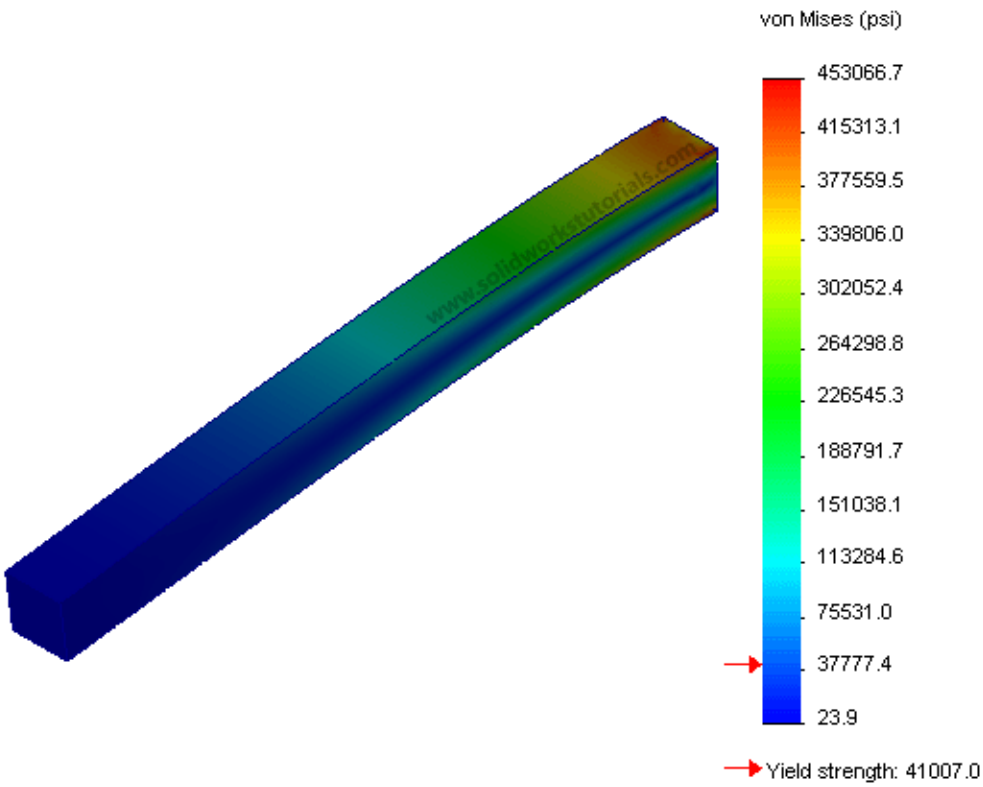


Click **Next** to view stress distribution in the model,

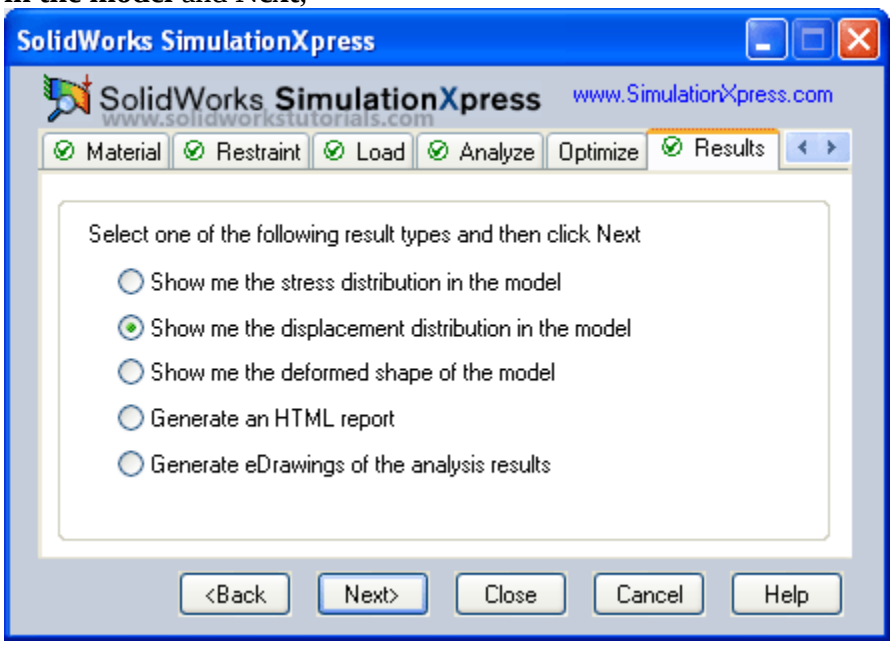


To animate the analysis click **Play** button, click **Next** when you done.

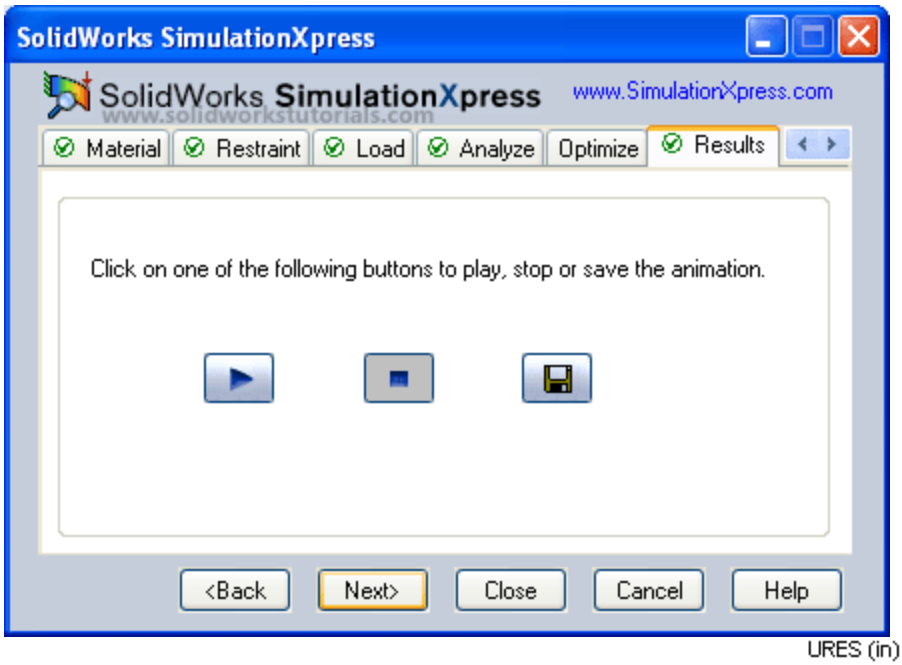


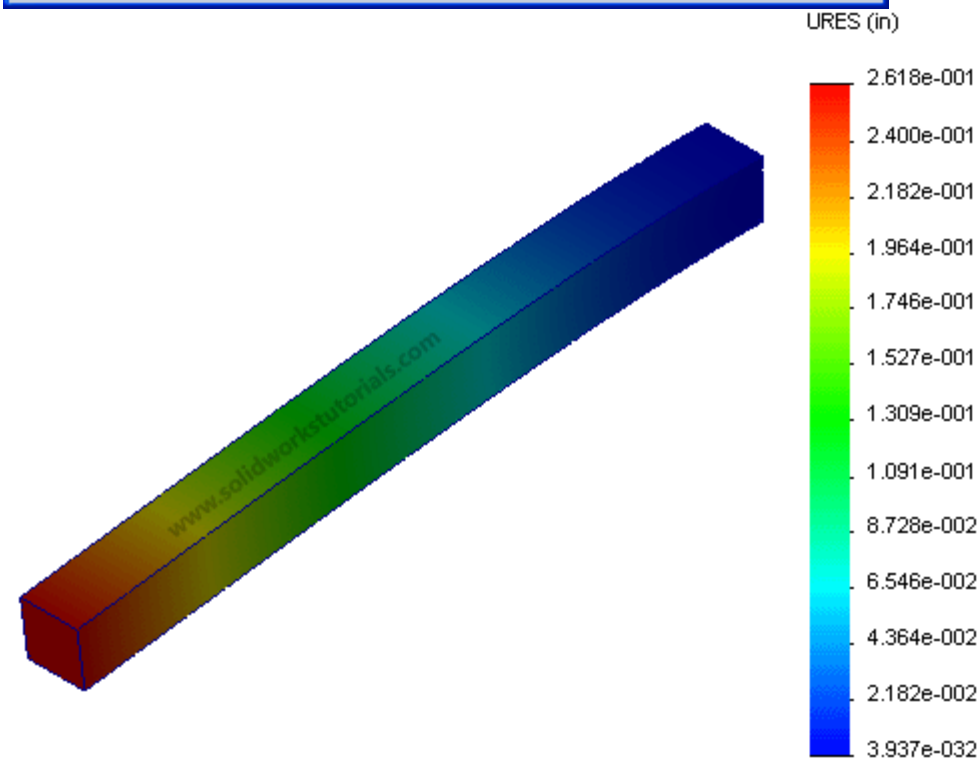


13. To view the displacement distribution, click **Show me the displacement distribution in the model** and **Next**,

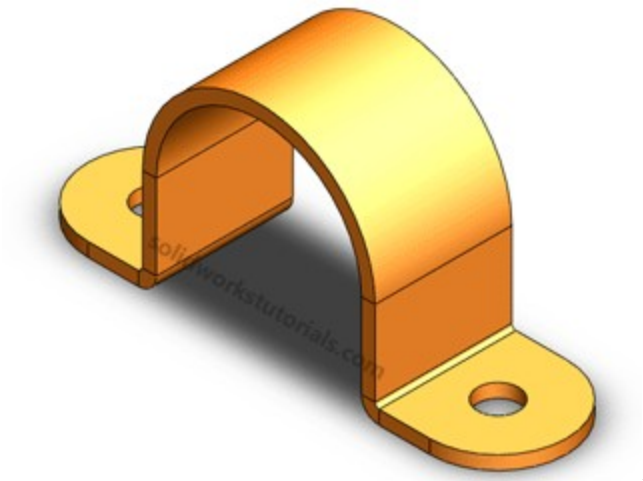


To animate analysis click **Play** button. Done!



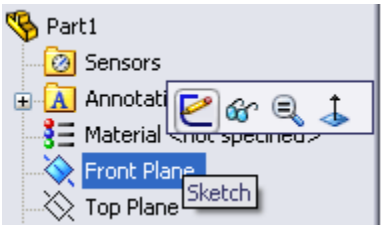


12. How to create U bracket sheet metal

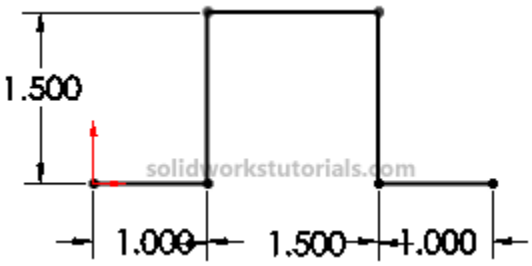


In this tutorials you will learn how to create U bracket sheetmetal.

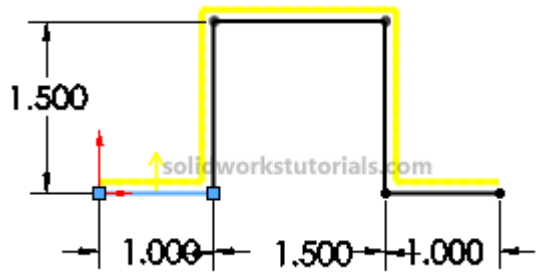
1. Click **New**.  Click **Part**,  **OK**.
2. Click **Front Plane** and click on **Sketch**.



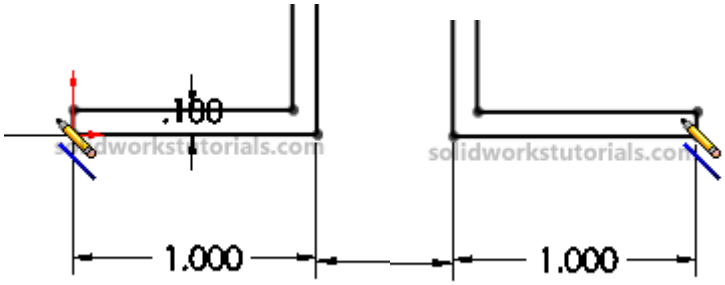
Use **Line**  , sketch U shape. Dimension sketch with **Smart Dimension**  as **1in x 1.5in x1in and 1.5in height.**

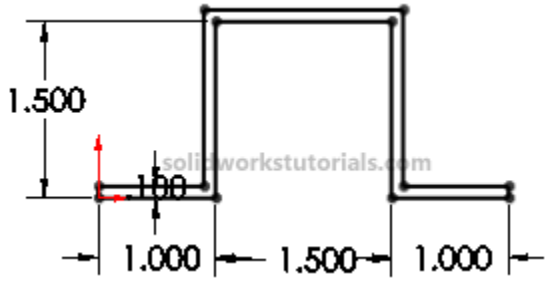


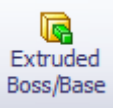
3. Click **Offset Entities**  and click U sketch. Set offset distance as **0.1in,** check **Reverse** box and **OK**.



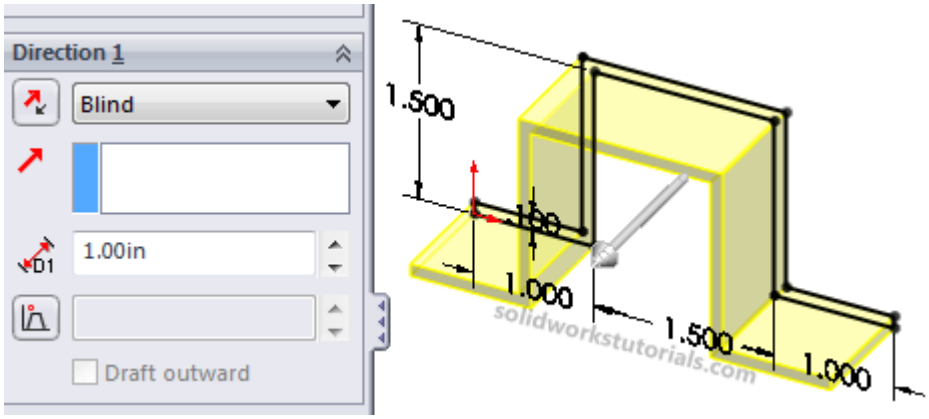
4. Use **Line**  , sketch and connected open end of this sketch and make it close both end.

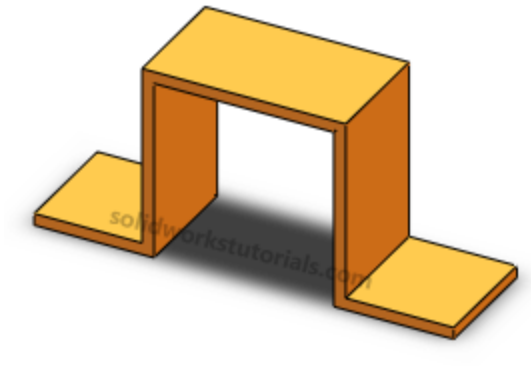




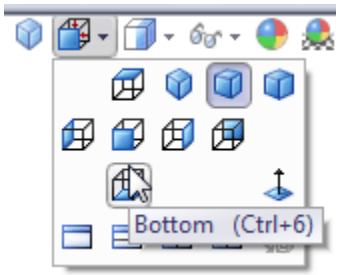
5. Click **Features>Extruded Boss/Base** 

set **D1** to **1in** and **OK**.

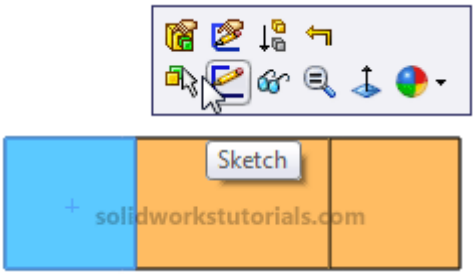




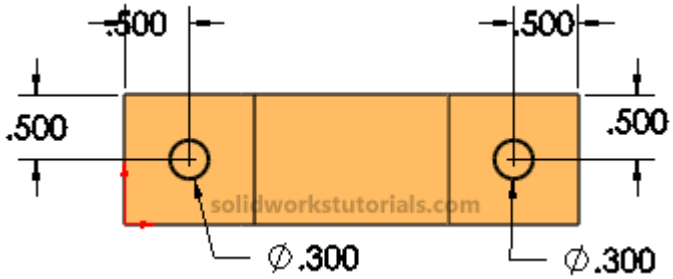
6. Click **View>Bottom**

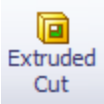


click on bottom face and click **Sketch**.

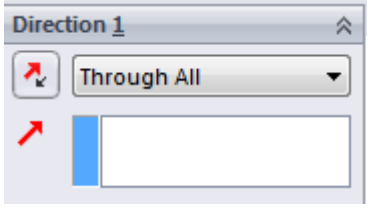


7. Click **Circle** and sketch 2 circle on bottom face each side. Use **Smart Dimension** Smart Dimension to dimension this sketch as sketched below.

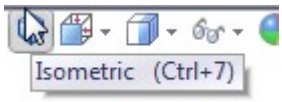


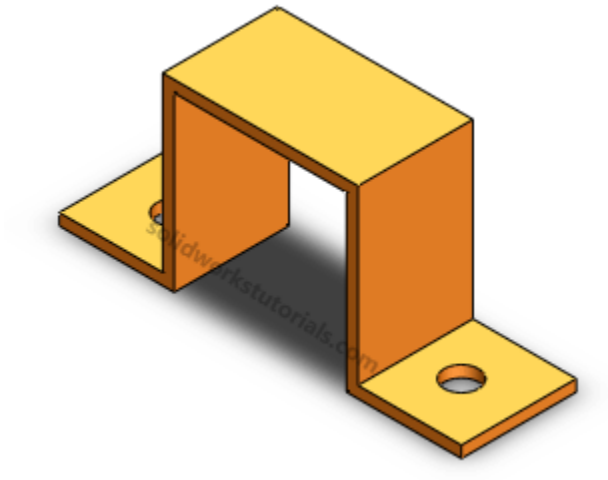
8. Click **Features>Extruded Cut** 

and cut **Through All** this circle.



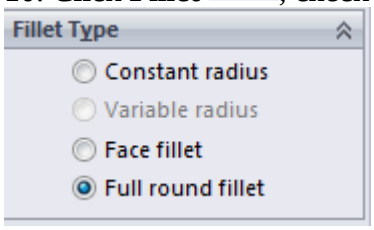
9. Click **View>Isometric.**



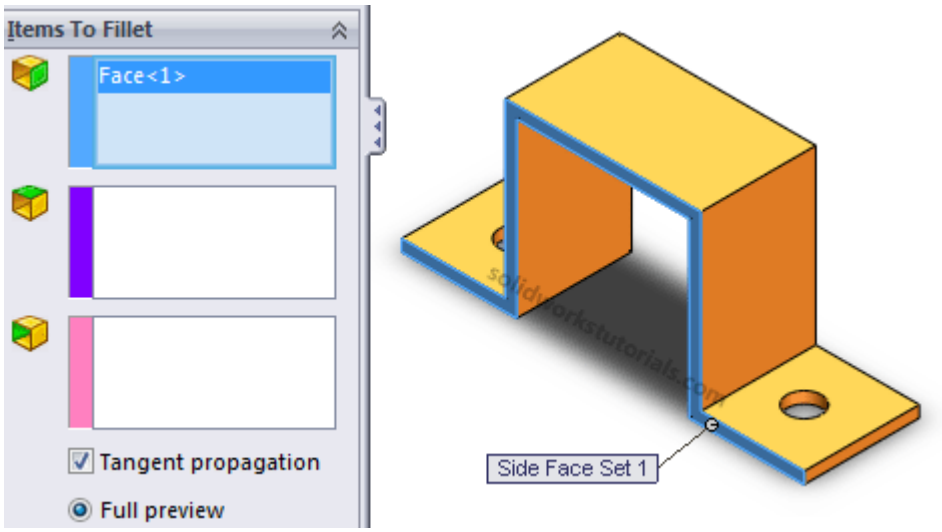


10. Click **Fillet** 

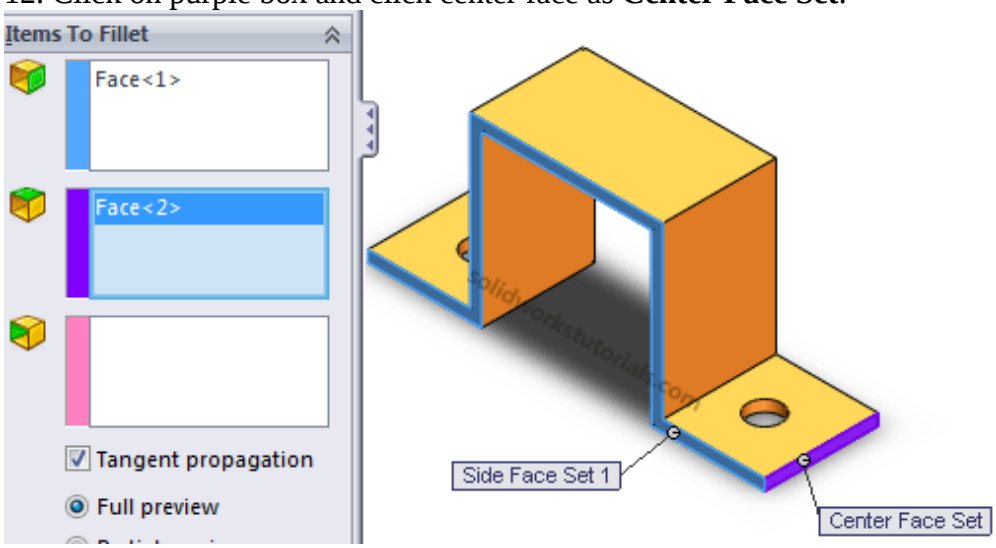
, check box **Full round fillet**.



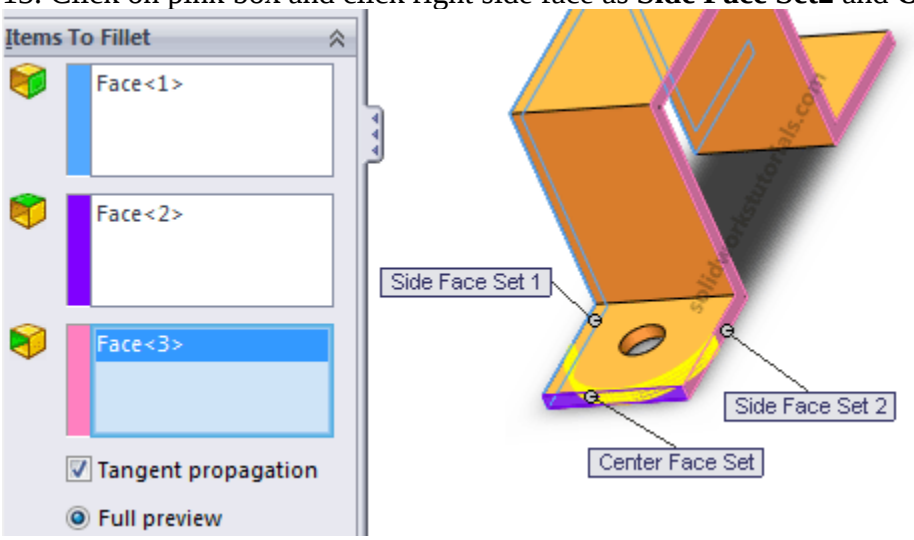
11. Click side left side face as **Side Face 1**.

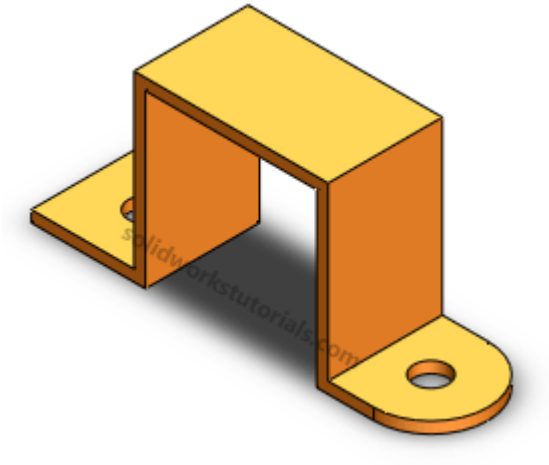


12. Click on purple box and click center face as **Center Face Set**.

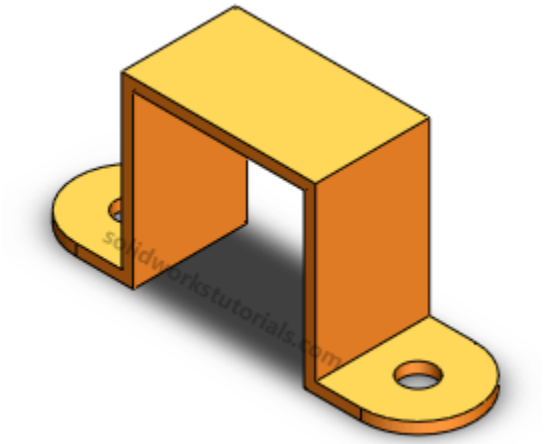


13. Click on pink box and click right side face as **Side Face Set2** and **OK**.

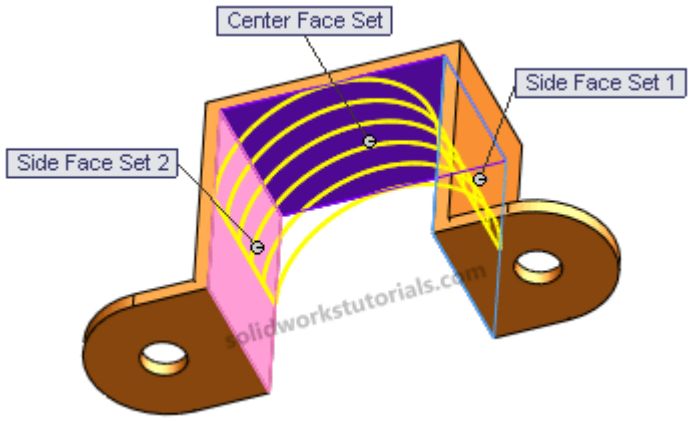


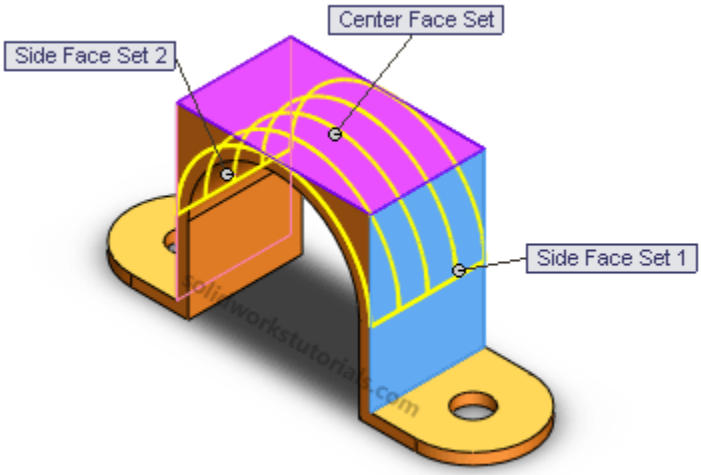


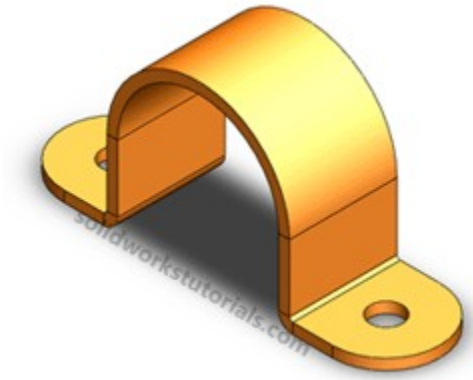
14. **Repeat step 11 - 13** for the other side.



15. **Repeat step 11 - 13** for inner face and outer face of U bracket.

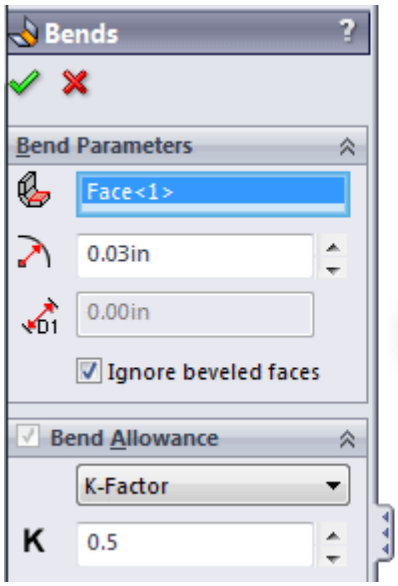


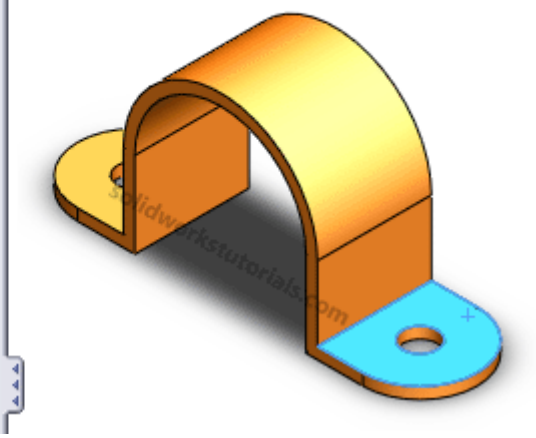




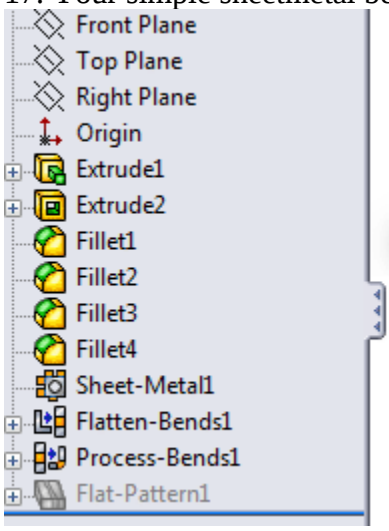
16. Click **Sheetmetal>Insert Bends**, 

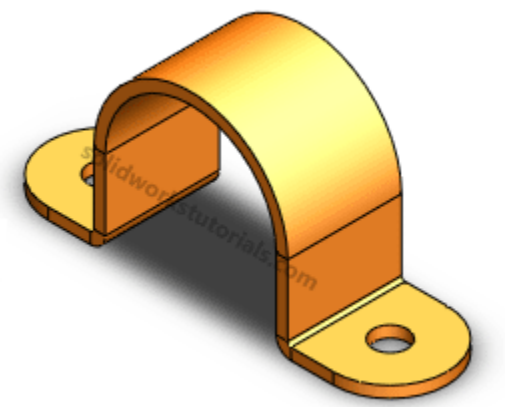
click flat face as reference when it flatten. Set bend radius to **0.03in** and **K factor 0.5** and **OK**.



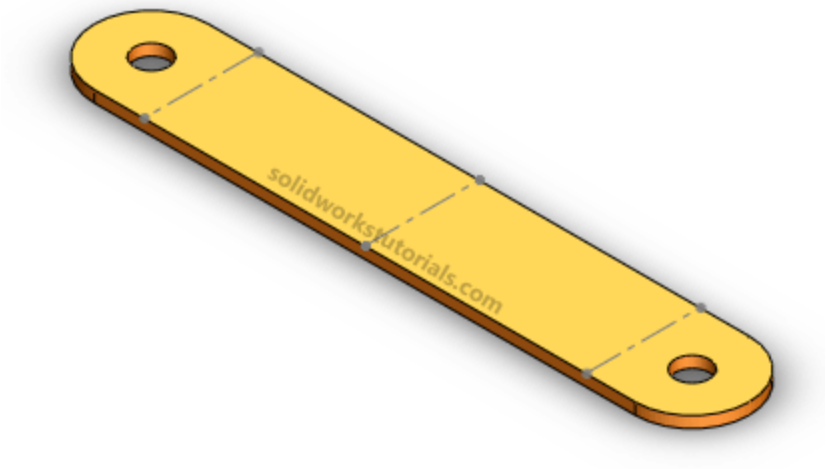


17. Your simple sheetmetal bend is ready. Look at part tree.

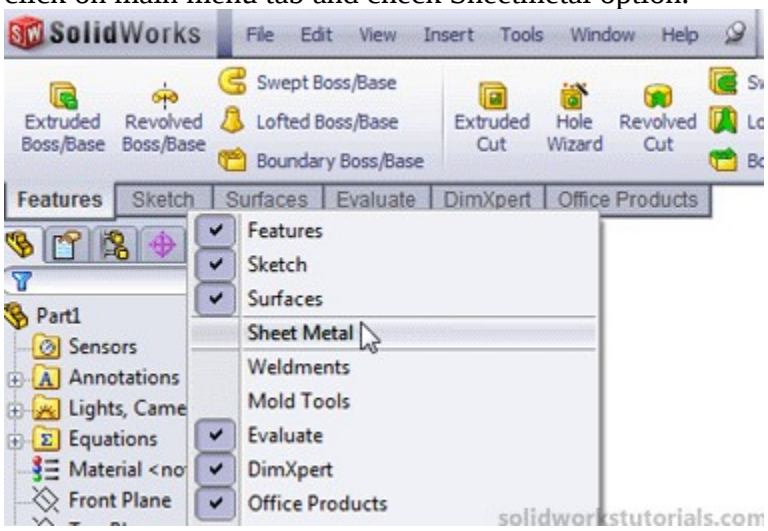




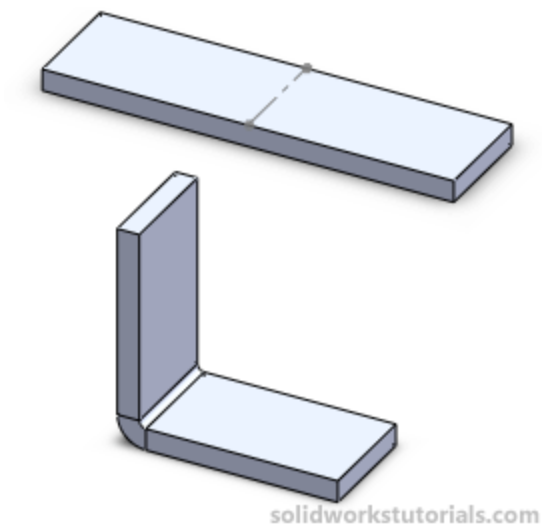
18. To view this part in flatten form click **Sheetmetal>Flatten** 
.



Have fun.. If you cannot find the sheetmetal tool in you main tool menu, you can right click on main menu tab and check Sheetmetal option.

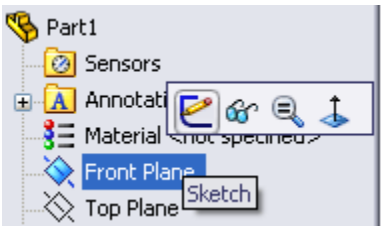


13. Create simple sheet metal bend

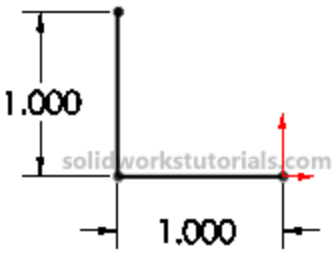


In this tutorials you will learn how to utilize sheetmetal tool such insert bend and flaten.

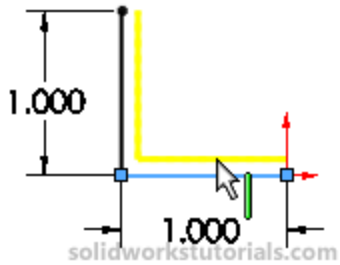
1. Click **New**. Click **Part**, **OK**.
2. Click **Front Plane** and click on **Sketch**.



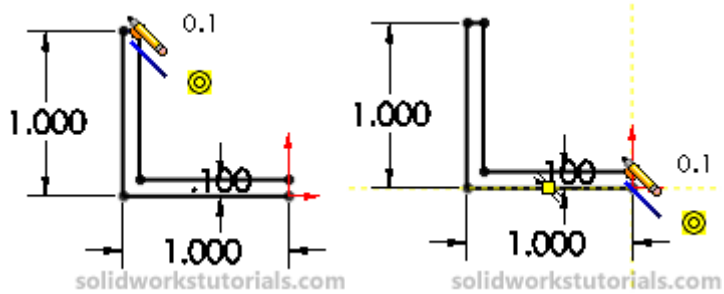
Use **Line** , sketch L shape. Dimension sketch with **Smart Dimension** as **1in x 1in**.

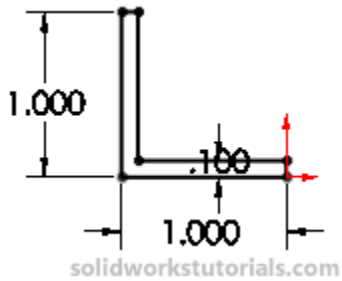


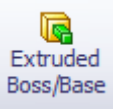
3. Click **Offset Entities** and click L sketch. Set offset distance as **0.1in**.



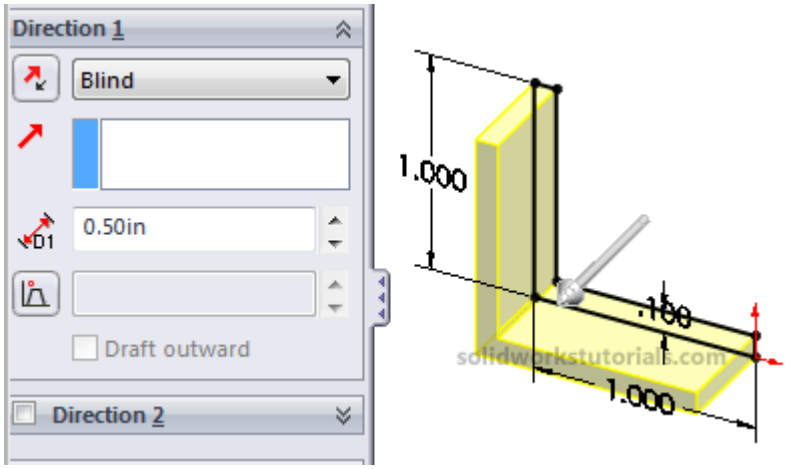
4. Use **Line**  , sketch and connected open end of this sketch and make it close both end.





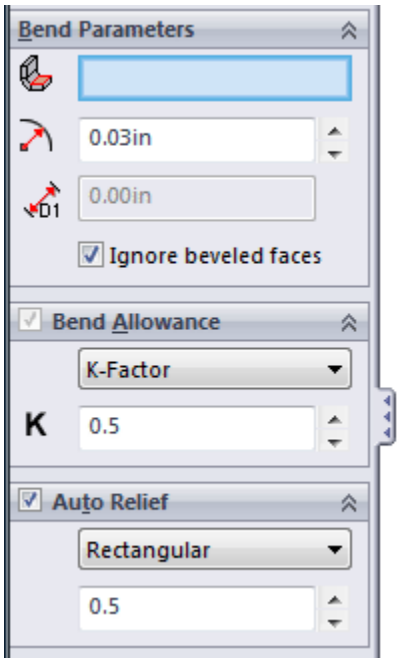
5. Click **Features>Extruded Boss/Base** 

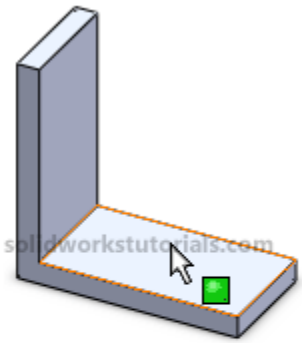
set **D1** to **0.5in** and **OK**.



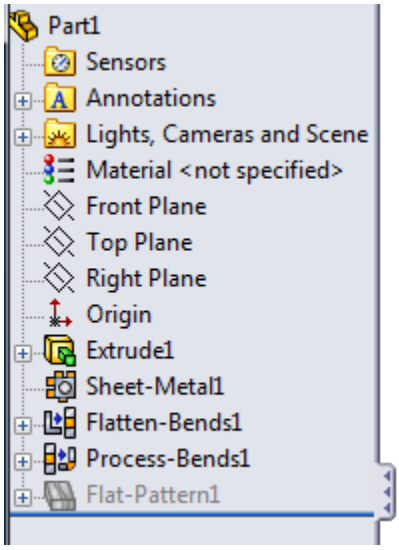
6. Click **Sheetmetal>Insert Bends**, 

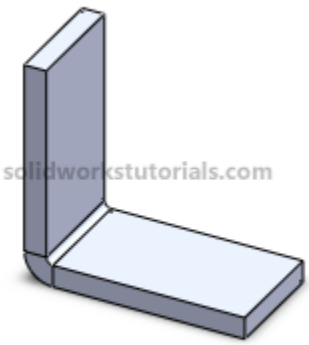
click flat face as reference when it flatten. Set bend radius to **0.03in** and **K factor 0.5** and **OK**.



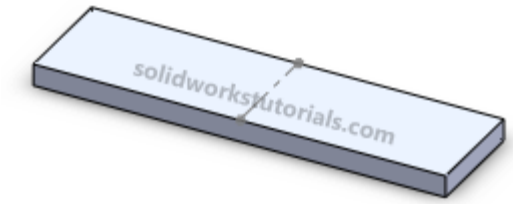


7. Your simple sheetmetal bend is ready. Look at part tree.

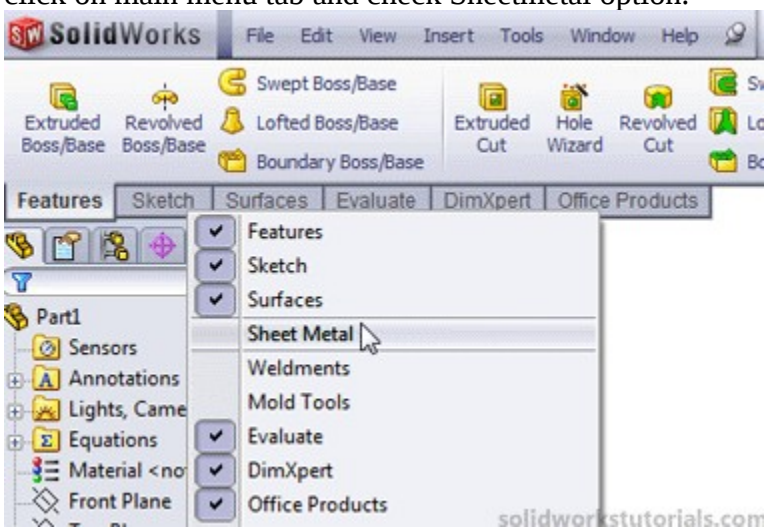




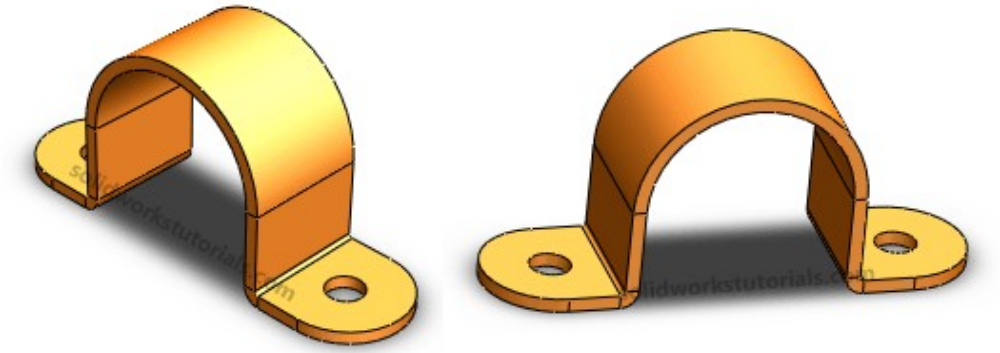
8. To view this part in flatten form click **Sheetmetal>Flatten**.



Have fun.. If you cannot find the sheetmetal tool in you main tool menu, you can right click on main menu tab and check Sheetmetal option.



You know the basic, try model this bracket.



No idea? Wait for this SolidWorks tutorial on my next post..

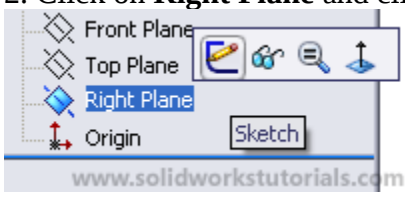
14. How to model aero plane wings

Last week my friends ask me how to model RC (remote control) wings in solidworks? He tried to model by extruding the sketch but it didn't reflect what the real wings. So he email me this picture of RC wings for me to look at. After reviewing the wings shape, I told him he can model these wings by loft features. Let's model these wings together.

1. Click **New**,  **Part** 

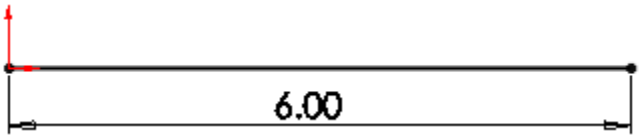
and **OK.**

2. Click on **Right Plane** and click **Sketch**.



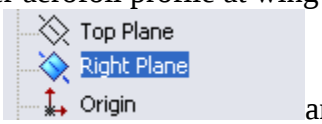
3. Sketch a center aerofoil profile at this plane. Click **Line**,  sketch a horizontal line, click **Smart Dimension** 

and dimension the line as **6in**.

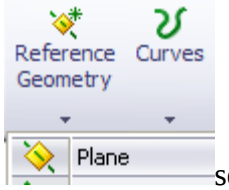


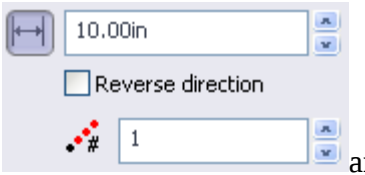
4. To create top curve of aerofoil, click **Spline**,  and sketch top curve as sketched below, to end Spline press **Esc** key.

Exit the sketch. 

5. For another aerofoil profile at wing tip, you need to create another plane. Click on **Right Plane** 

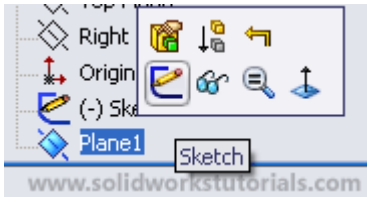
and click **Reference Geometry**>**Plane**



set distance between plane as **10in** 

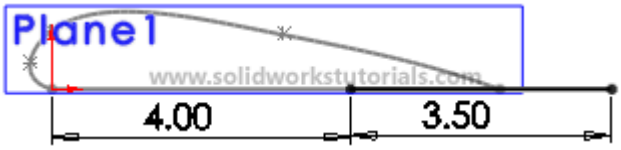
and  .

6. Click on **Plane 1** and click **Sketch**.

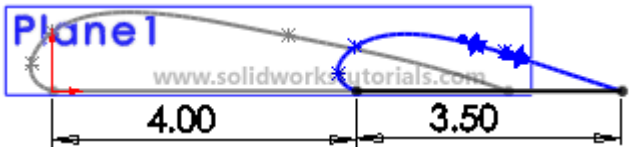


7. Click **Line**,  sketch a horizontal line on same level as first sketch a bit off set from origin, click **Smart Dimension** 

and dimension sketch as sketched below.

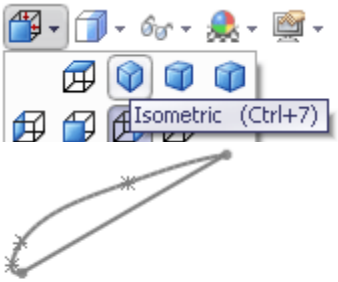


8. To create top curve of aerofoil, click **Spline**,  and sketch top curve as sketched below, to end **Spline** press **Esc** key.

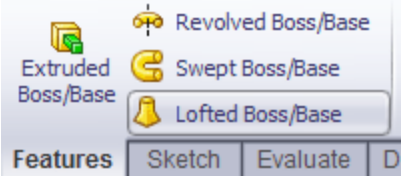


Exit the sketch. 

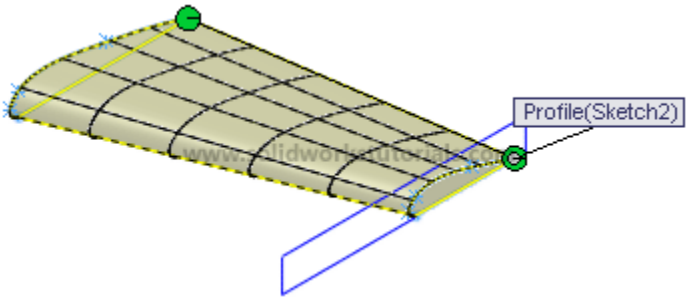
9. Click **View Oreintation**>**Isometric**.

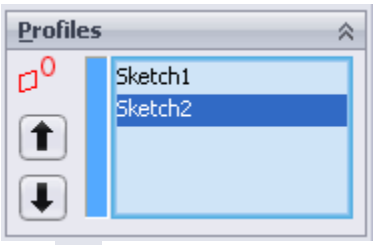


10. Click **Features**>**Lofted Boss/Base**,



click **Sketch1** and then **Sketch2**.





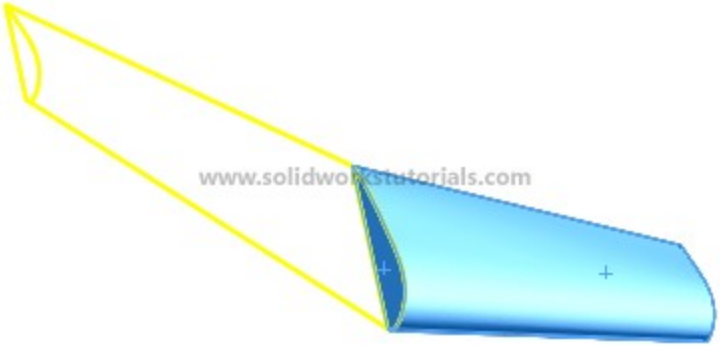
and  .

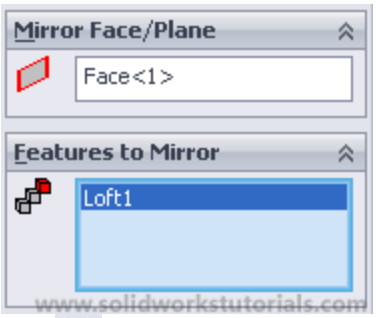
11. To hide Plane 1, click **Plane 1** and click **Hide**.

12. Now let make the full wings, click on **Mirror**. 

Turn the wings to right side and **select center face** as a **Mirror Face/Plane**.

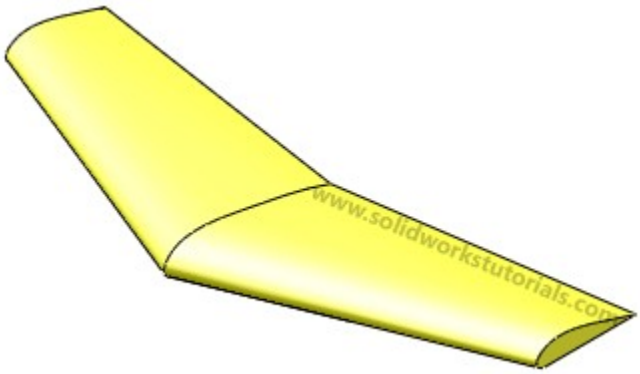
Click on wing body as **Features to Mirror**



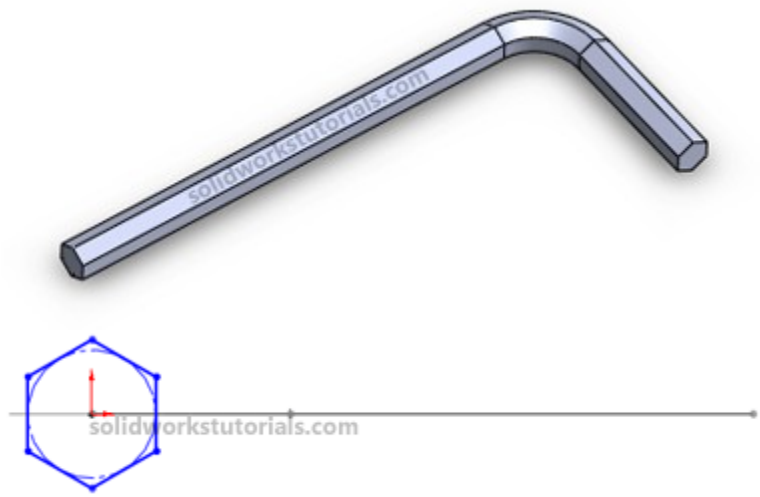


and  .

13. You're done.



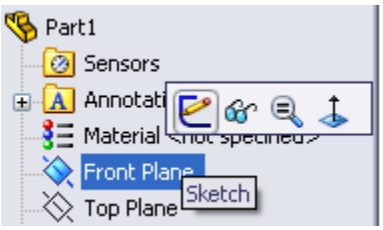
15. How to create Allen key



In this solidworks tutorial, you will create simple allen key.

1. Click **New**.  Click **Part**,  **OK**.

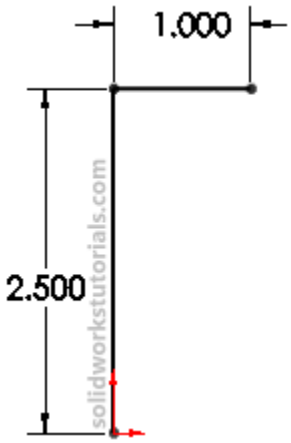
2. Click **Front Plane** and click on **Sketch**.



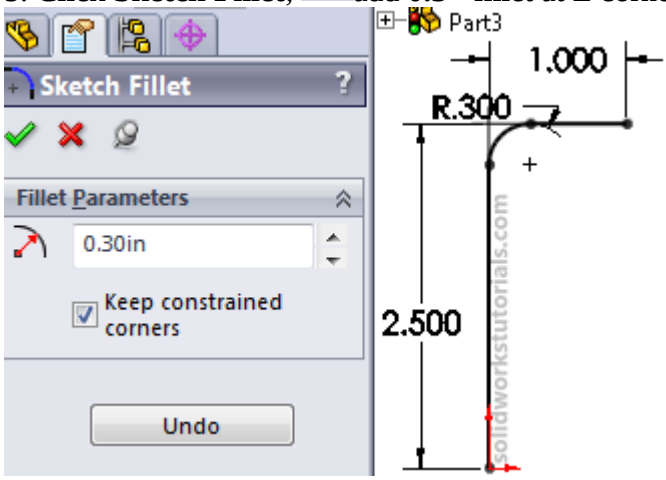
3. Click **Line**, skecth a L shape.

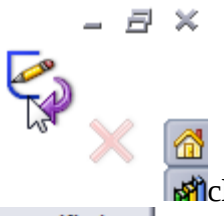
4. Click **Smart Dimension**, 

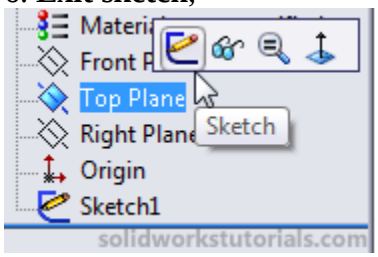
and dimension sketch as **2.5″** and **1″**.



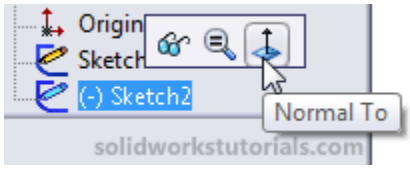
5. Click **Sketch Fillet**, 
add **0.3″** fillet at L corner.



6. **Exit sketch**,  click on **Top Plane** and click **Sketch**.



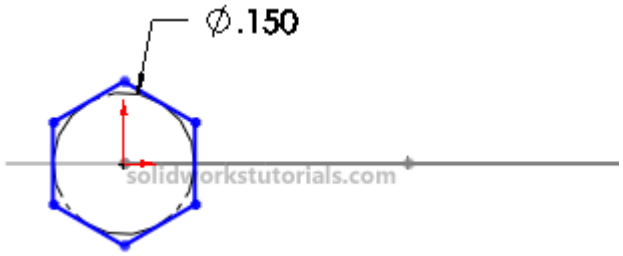
7. Click on **Sketch2** and click **Normal To**.



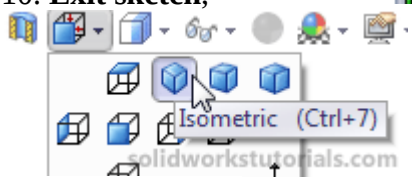
8. Click Polygon,  sketch a polygon at origin.

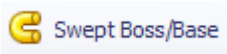
9. Click **Smart Dimension**, 

and dimension sketch diameter to **0.15″**.

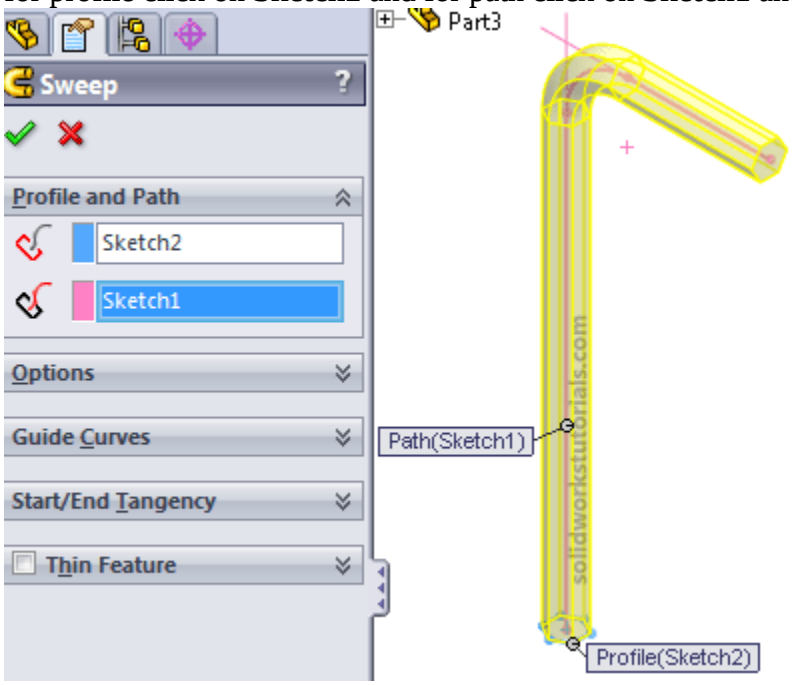


10. **Exit sketch**,  click on **Isometric** view.

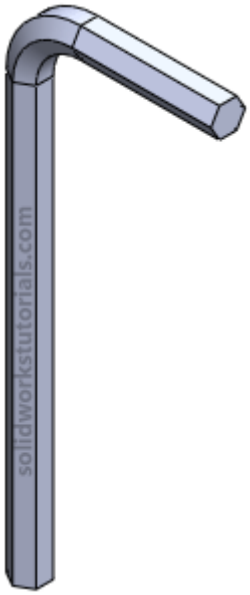


11. Click **Features>Swept Boss/Base**, 

for profile click on **Sketch**2 and for path click on **Sketch1** and OK.



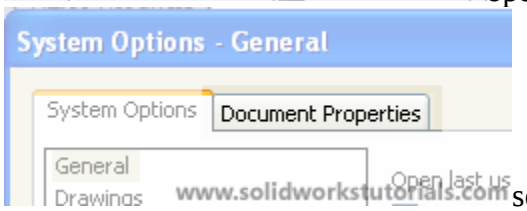
You’re done!.



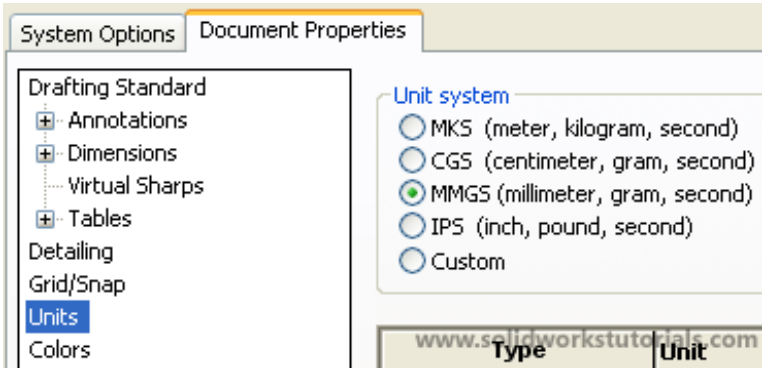
16. How to change to metric units

There is time we need to change our parts to metric units, but how? It’s very simple just few clicks its done. First click **Option** on top of main menu,

open **Document Properties** tab,



select **Units** in menu tree and check **MMGS (millimeter, gram, second)**.



Ok, done!