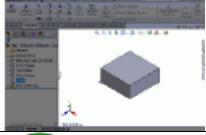
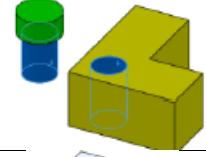
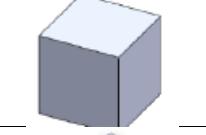
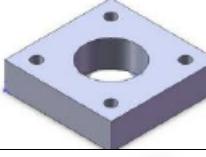
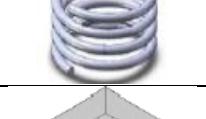
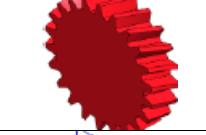
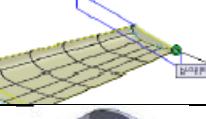
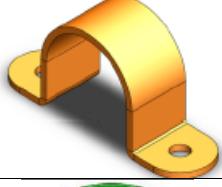
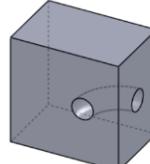
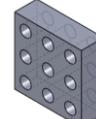
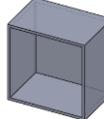
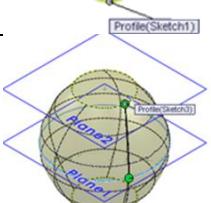


## SolidWorks Tutorial

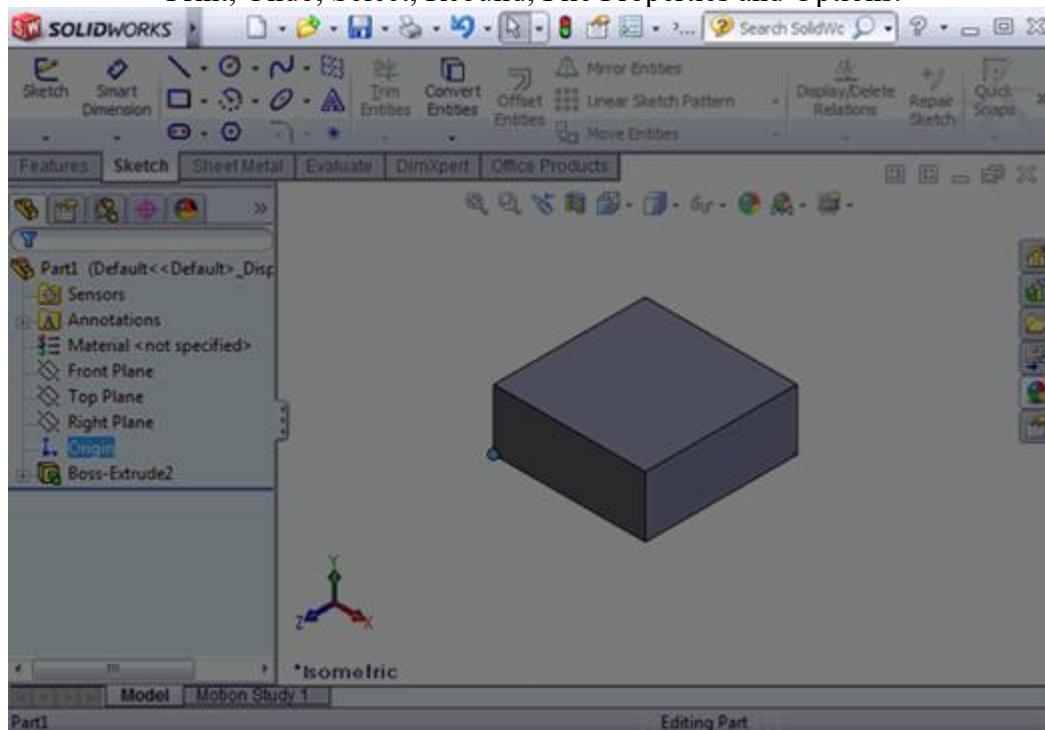
<b>Tutorial</b>	<b>Title (Part 1)</b>	<b>Model</b>	<b>Page</b>
Tutorial 1	SolidWorks User Interface		
Tutorial 2	Introduction to SolidWorks		
Tutorial 3	How to create simple box		
Tutorial 4	How to create simple plate		
Tutorial 5	How to create Allen key		
Tutorial 6	How to create 17" car wheel		
Tutorial 7	How to create simple sheet metal bend		
Tutorial 8	How to create spring		
Tutorial 9	How to engrave text		
Tutorial 10	How to create hexagonal head bolt		
Tutorial 11	How to create helical gear		
Tutorial 12	How to create aero plane wings		
Tutorial 13	How to create turbo fins		

Tutorial 14	How to create U bracket		
Tutorial 15	How to create bottle cap		
Tutorial 16	How to twist phone cord		
Tutorial	Title (Part 2)		Model
Tutorial 17	How to use Revolved Boss Base		
Tutorial 18	How to use Revolved Cut		
Tutorial 19	How to use Linear Pattern		
Tutorial 20	How to use Scale	Scale down to 1/2 .	
Tutorial 21	How to use Hole Wizard		
Tutorial 22	How to use Shell		
Tutorial 23	How to use Swept Boss/Base		
Tutorial 24	How to use Lofted Boss/Base		
Tutorial 25	How to change to metric units	<p>Unit system</p> <ul style="list-style-type: none"> <li><input type="radio"/> MKS (meter, kilogram, second)</li> <li><input type="radio"/> CGS (centimeter, gram, second)</li> <li><input checked="" type="radio"/> MMGS (millimeter, gram, second)</li> <li><input type="radio"/> IPS (inch, pound, second)</li> <li><input type="radio"/> Custom</li> </ul>	

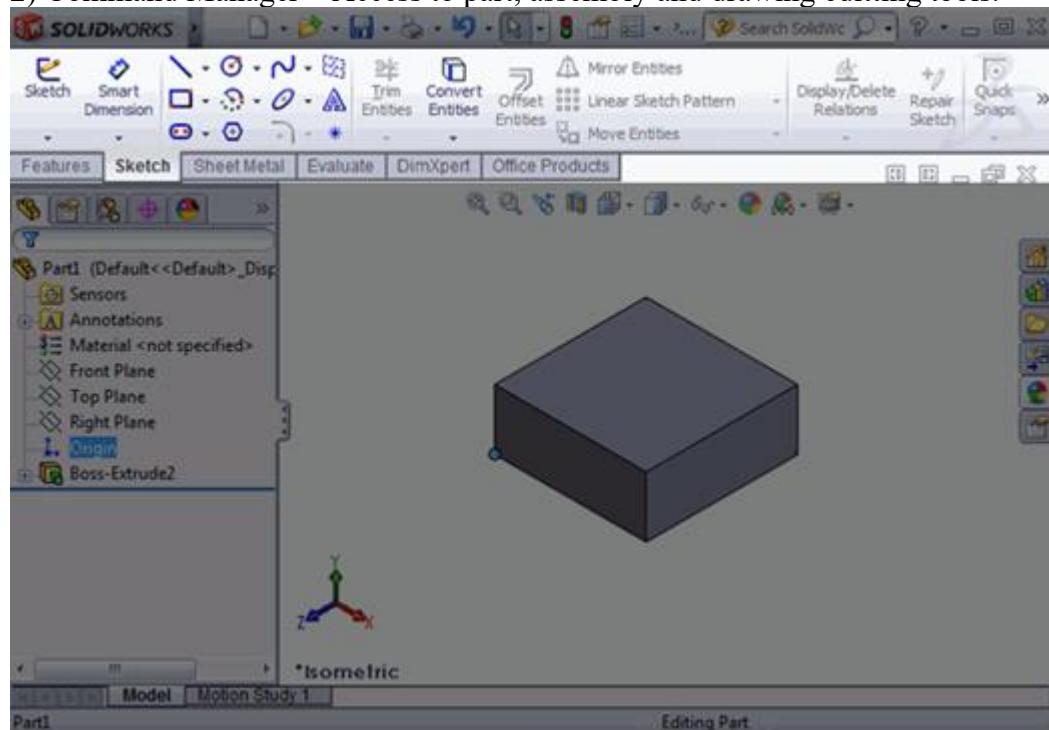
# 1. SolidWorks User Interface

SolidWorks User Interface is pretty simple and straight forward. There is 6 main area of interface you normally work with.

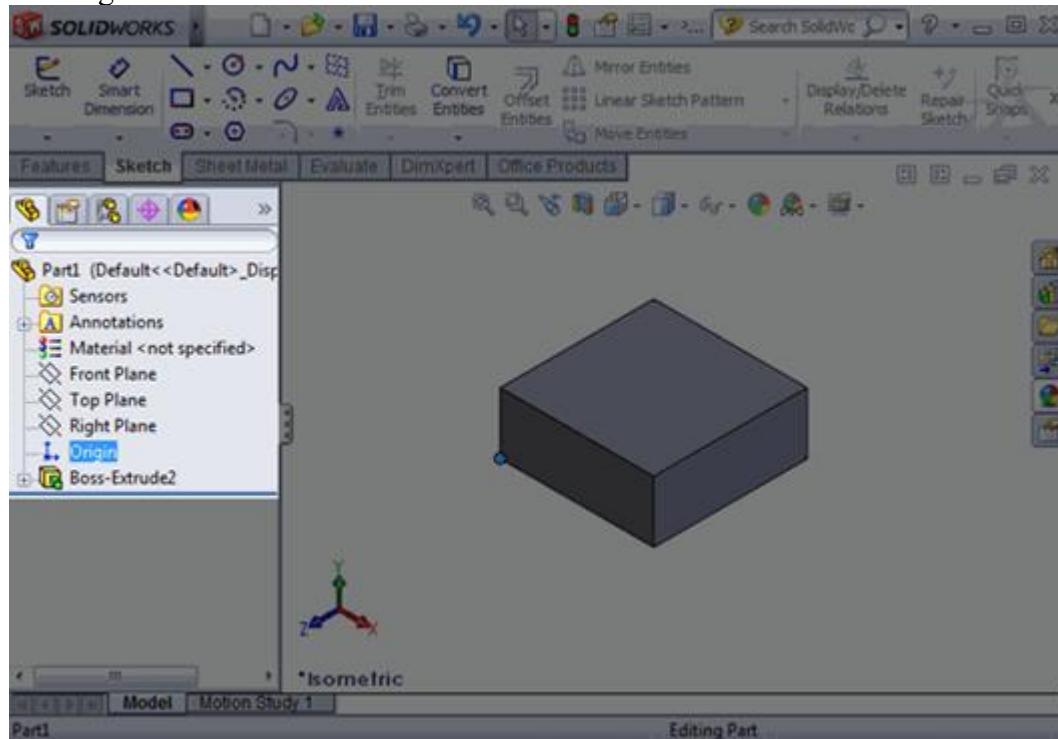
- 1) Menu Bar – Top most of the application, executing New File, Open File, Save, Print, Undo, Select, Rebuild, File Properties and Options.



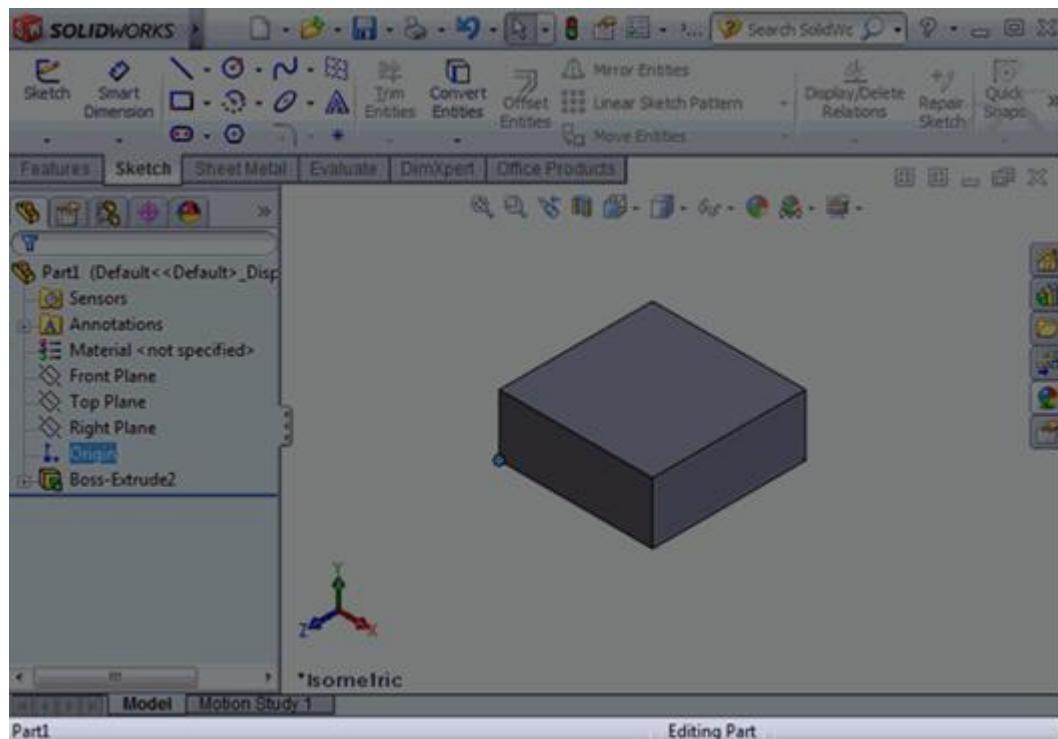
- 2) Command Manager – Access to part, assembly and drawing editting tools.



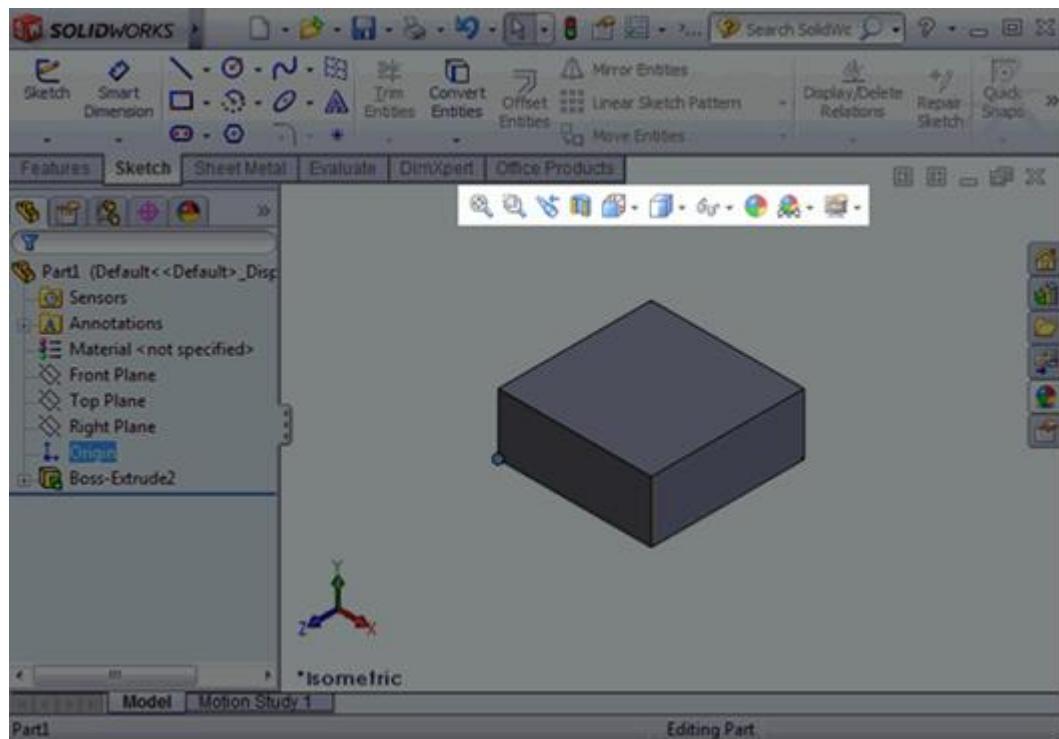
- 3) Feature Manager design tree – Outline overview how your part, assembly and drawing constructed.



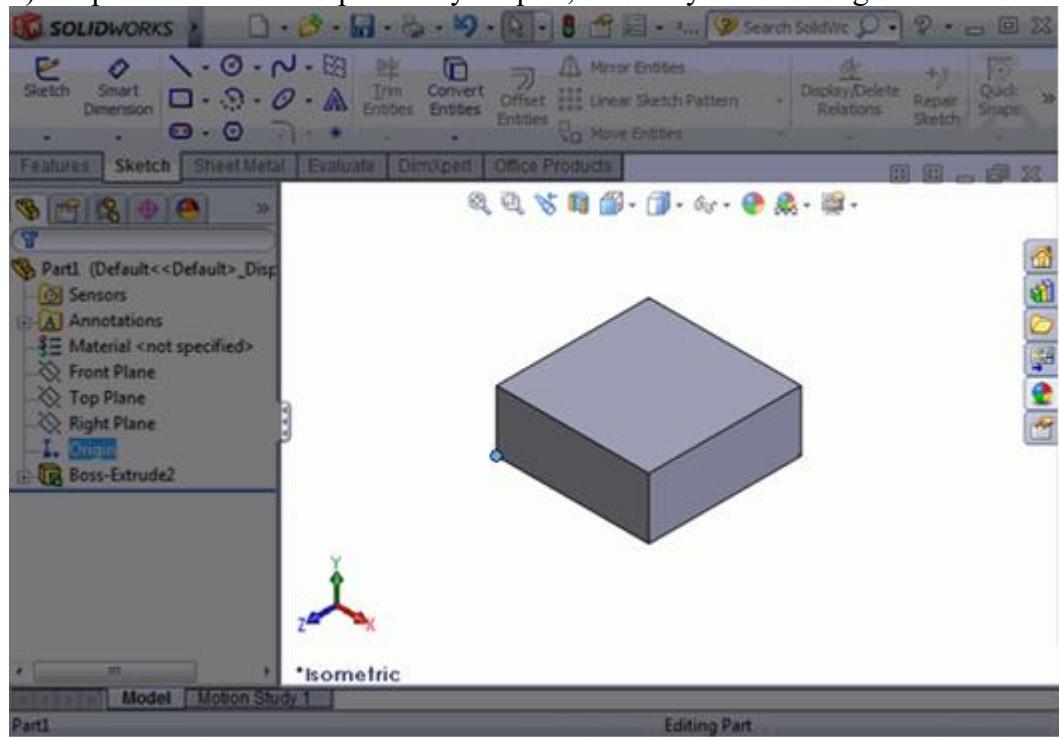
- 4) Status bar – Provide an information about your part, assembly and drawing.



- 5) Head up view toolbar – View tools such as zoom, pan, zoom plane and section view.



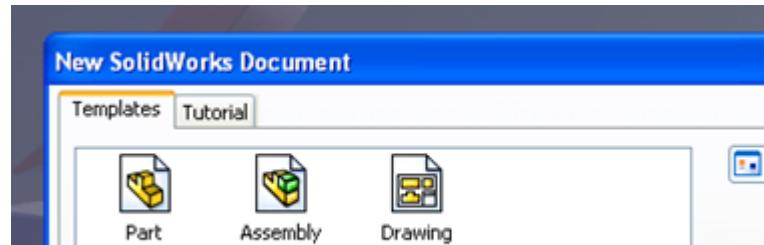
6) Graphics area – Workspace for your part, assembly and drawing.



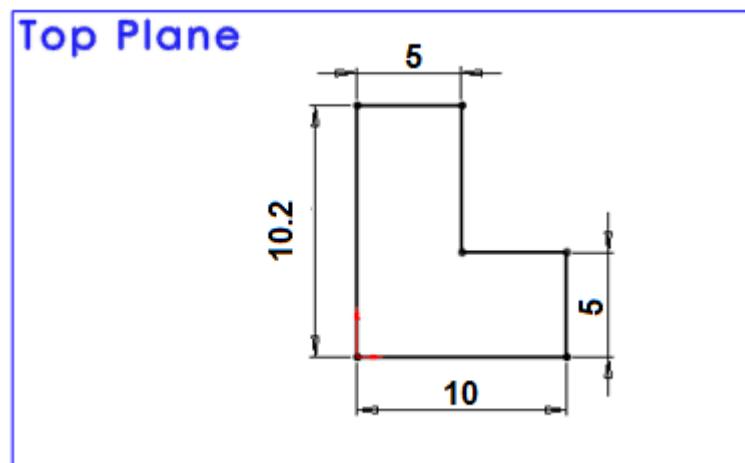
## 2. Introduction to SolidWorks

### Solidworks Overview

Solidworks 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.

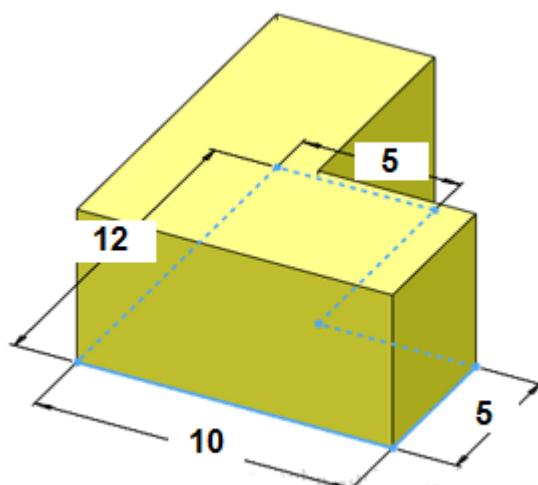


Part

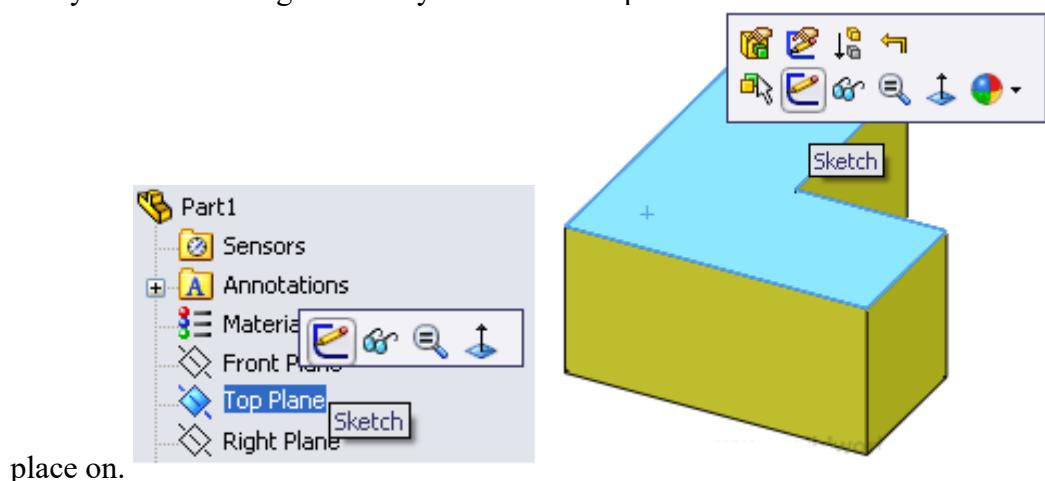


Part is created by sketch.

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

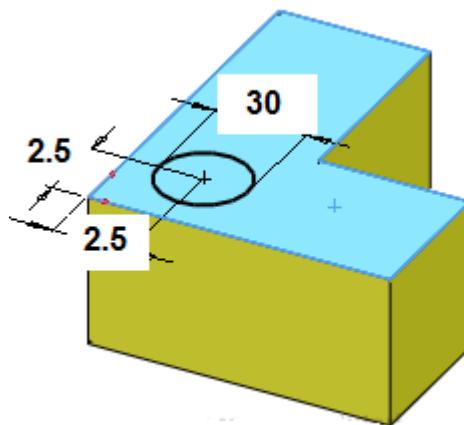


Before you start creating sketches you must select plane or face where the sketch will be

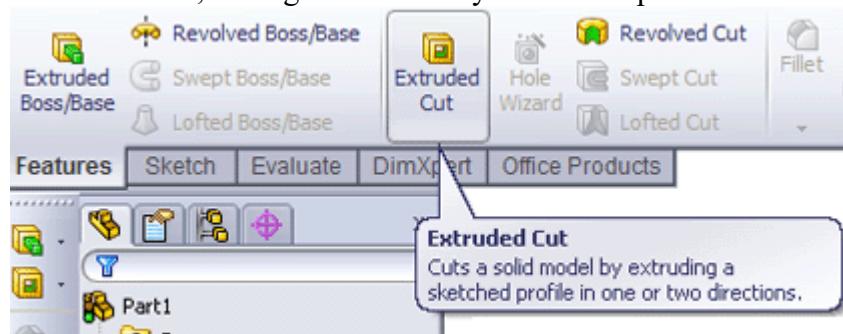


place on.

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

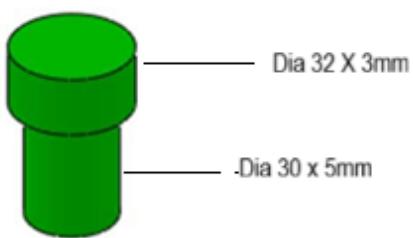
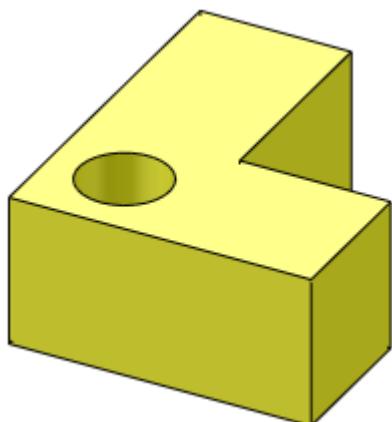
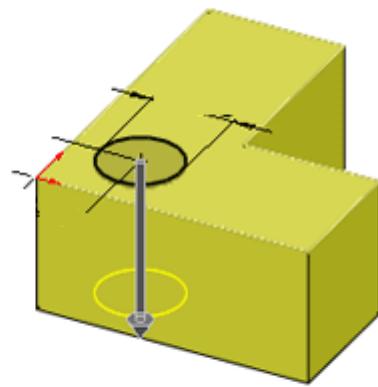
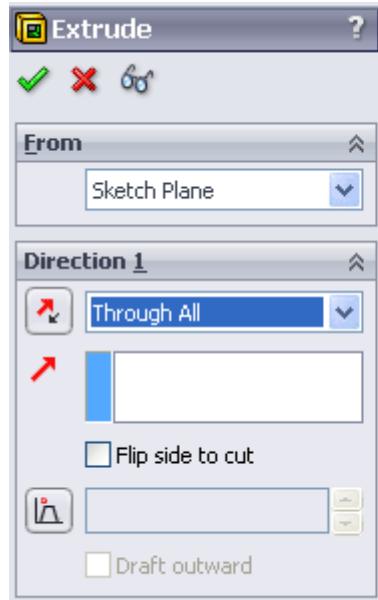


When you done with sketch, adding features it is your next step. Select Feature>Extruded



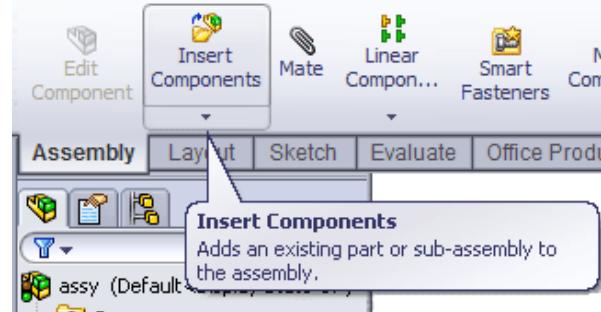
Cut

Select Through All and OK.

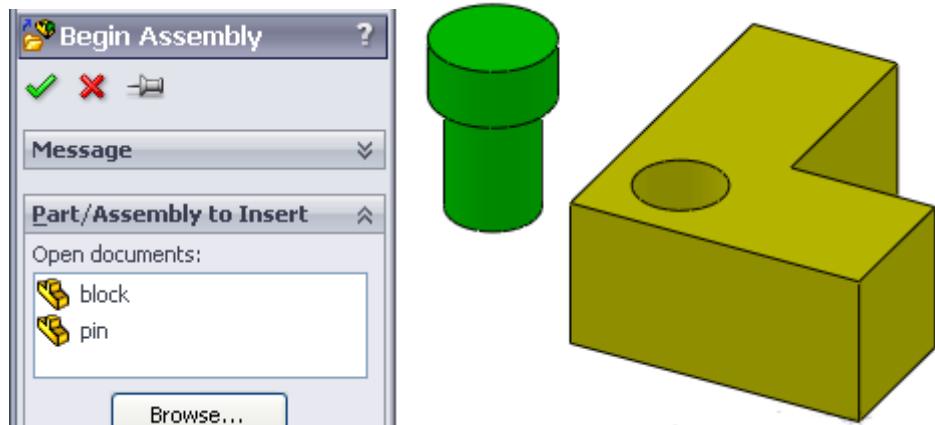


## Assembly

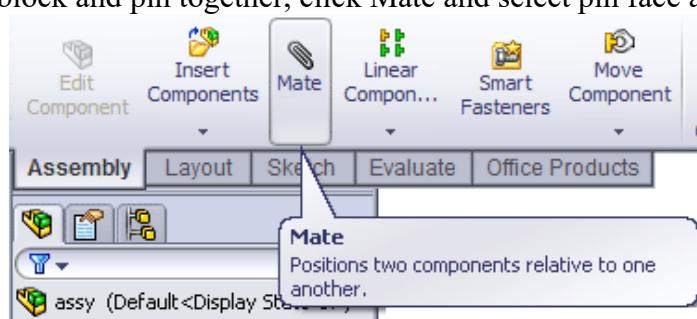
Assembly is how all parts works together in assembly, checking for clashes and it functionality. First all parts inserted in assembly by Insert Component tool.

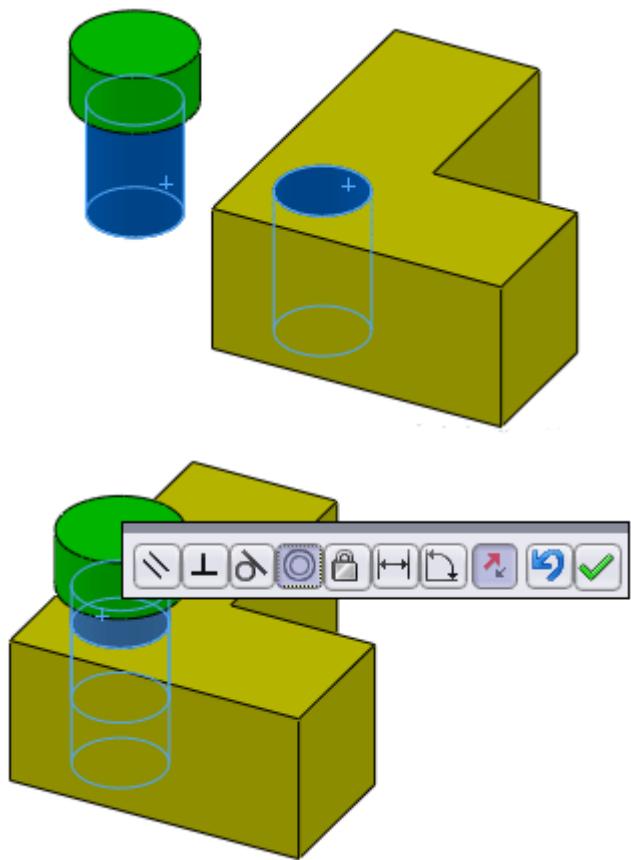


When all parts inserted into workspace, Mate is command to define how parts mate with each other.



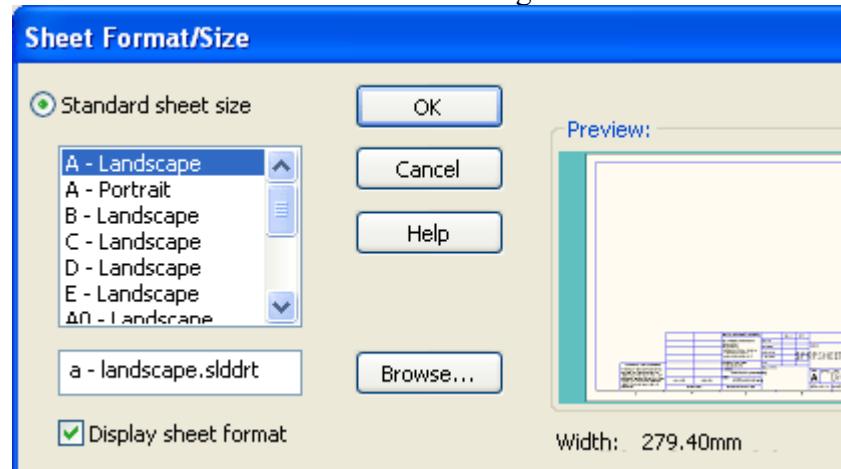
Let's mate this block and pin together, click Mate and select pin face and hole face, OK.



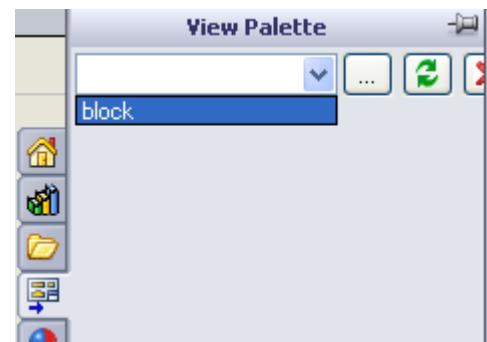


Drawing

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

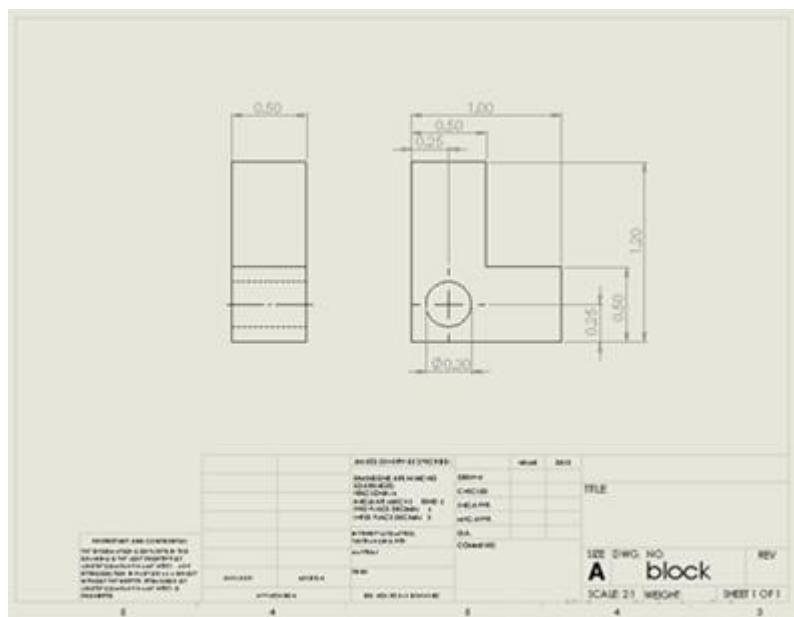
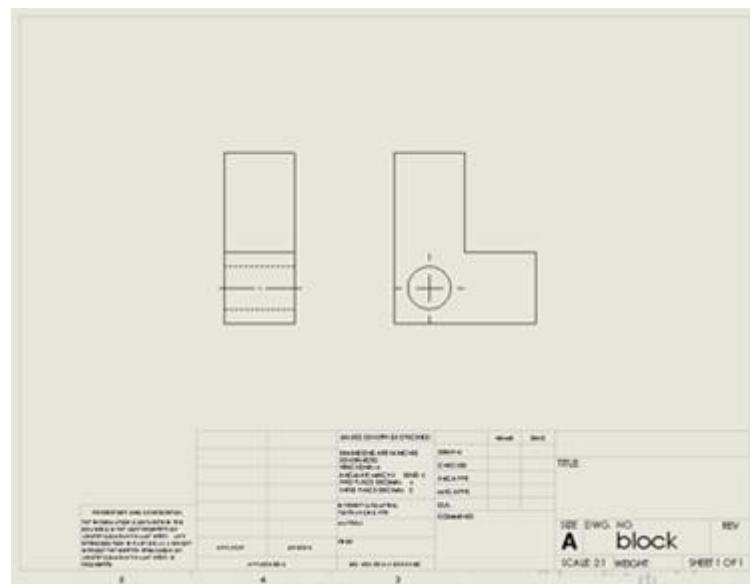
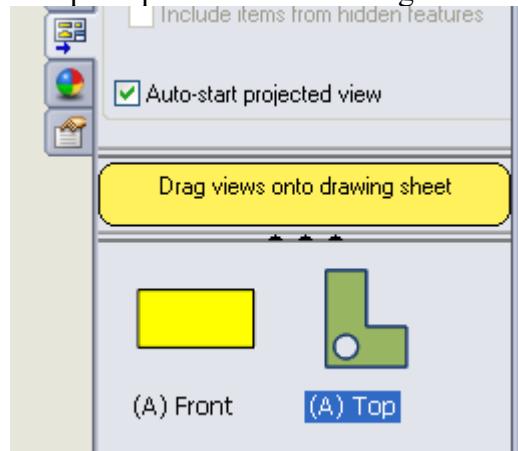


When block inserted, select click view palette to add drawing view.

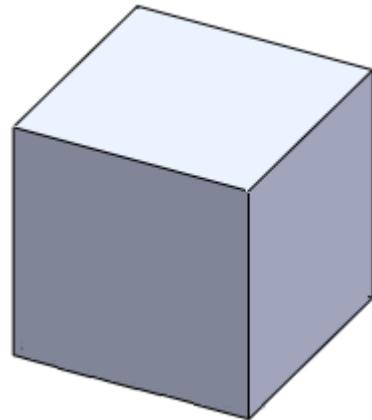


Choose the part you wish to make drawing.

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

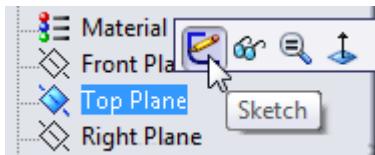


### 3. How to create simple box

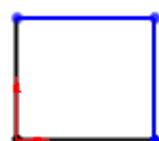


1. Click **New** , Click **Part** and **OK**.

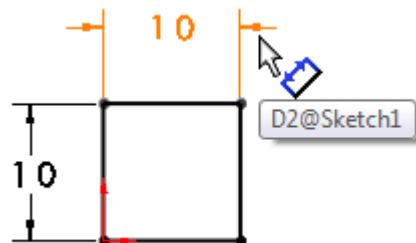
2. Click on **Top Plane** and click **Sketch**.



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



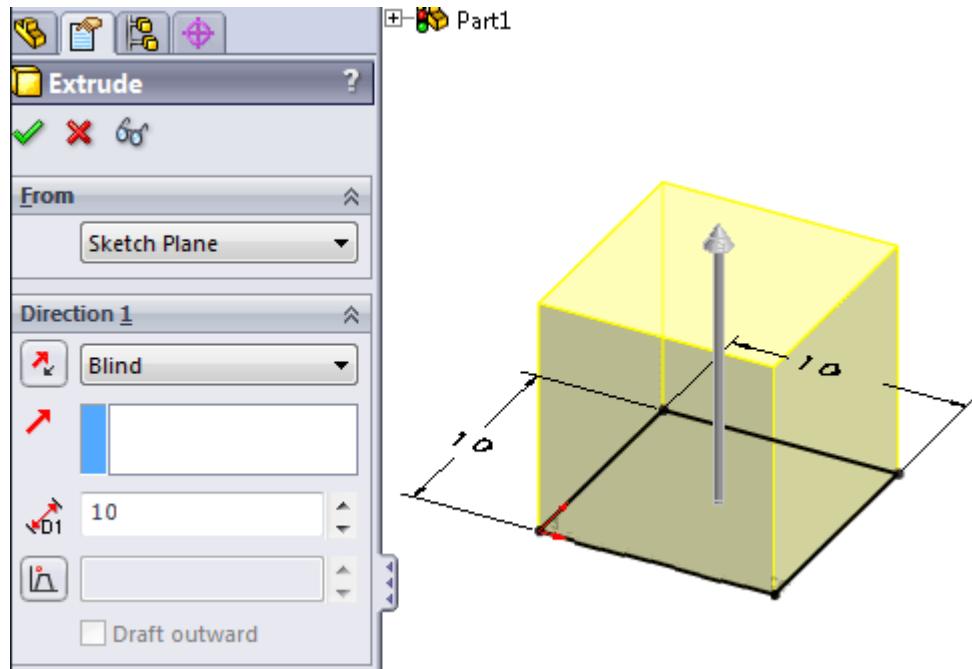
4. Click **Smart Dimension** , click side edge and click top edge to dimension it as **10mm x 10mm**.



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

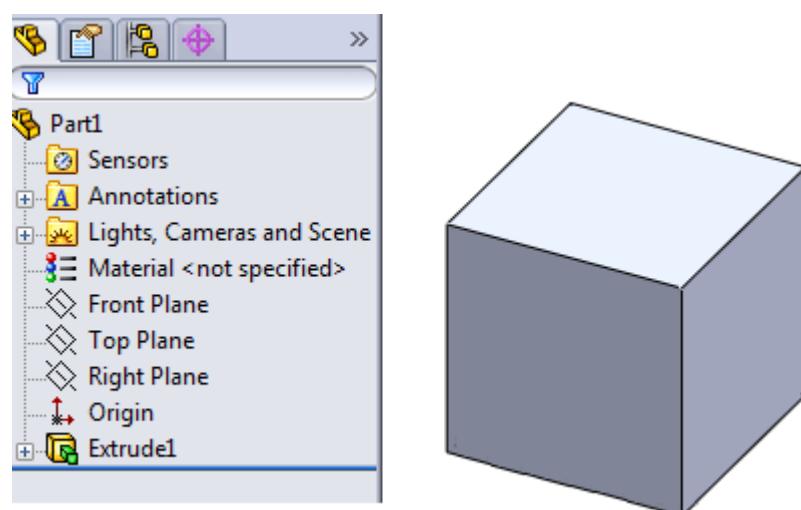


set D1 as 10mm

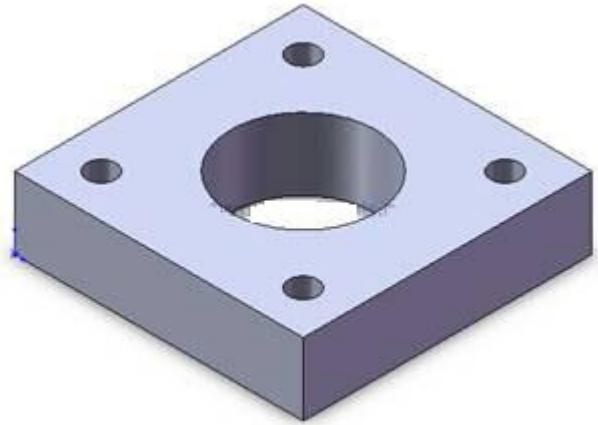


and click .

6. It's done.



#### 4. How to create simple plate



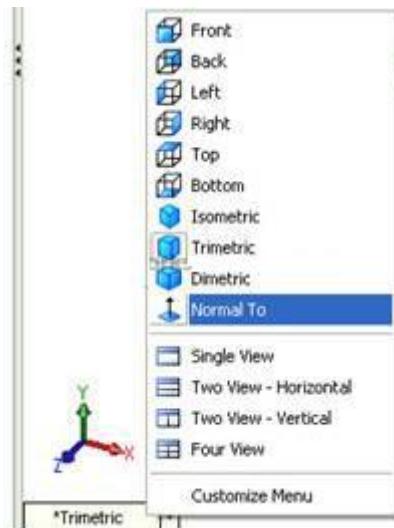
1. Click New  (File>New) , click Part  , OK.



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.

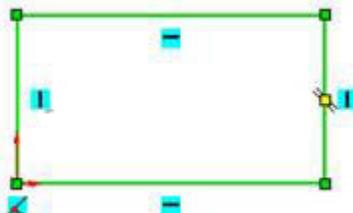


4. Click Sketch in Command Manager, click Rectangle . As you can see on upper right corner sketch icon appear indicate that you're on sketch mode

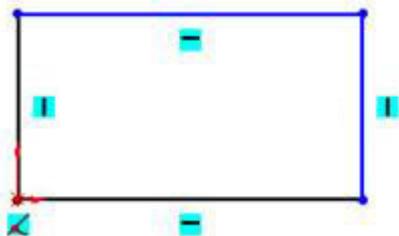




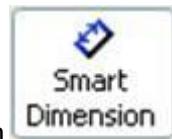
5. Pick Origin point as starting point, drag to right hand side



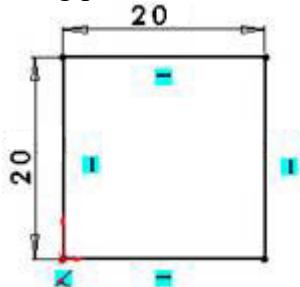
no need to be exact the size will define in later step. Press keyboard ESC to end rectangle sketch.



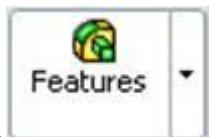
Note: There are two type line generated by your sketching, the one with black line and blue line. Black line is line that fully defined and blue line is under defined.



6. Define sketch with dimension. Click Smart Dimension , and start dimensioning pick vertical line and set to 20mm , pick horizontal line and set to 20mm.



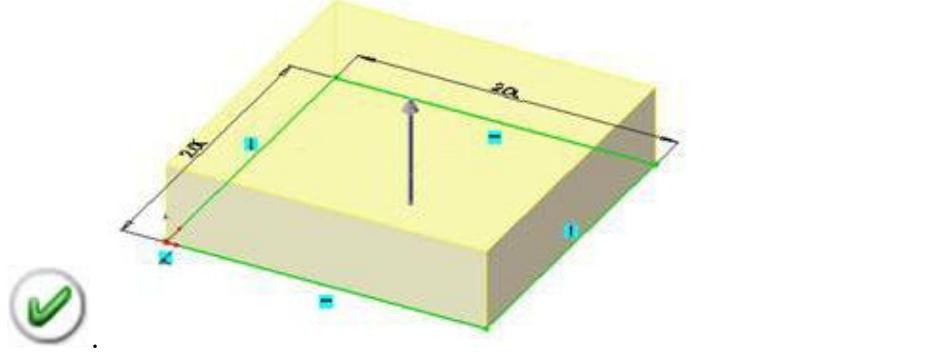
. Press keyboard ESC to end smart dimension

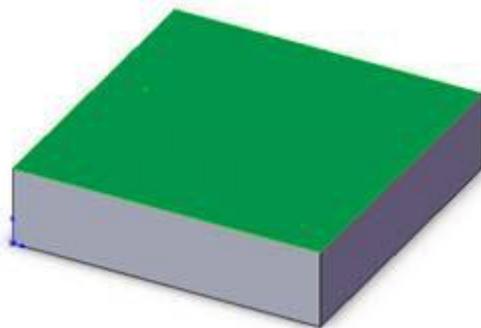


7. Build feature from sketch, click Features and activate features menu.



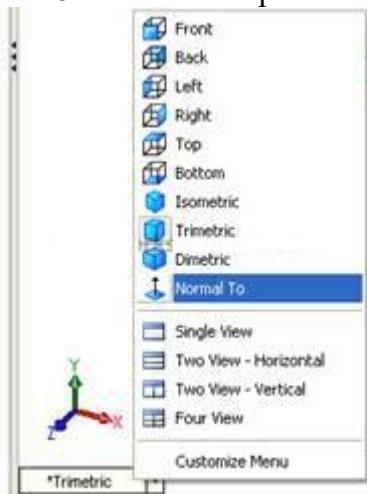
Click Extruded Boss/Base and set D1 to **5mm** and



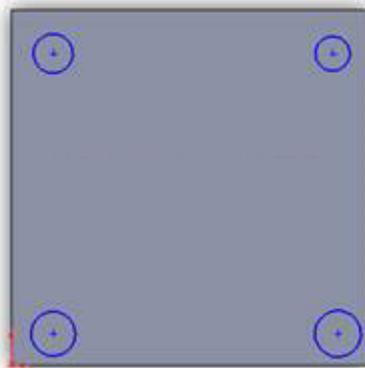


8. Click front top face

, click Normal To



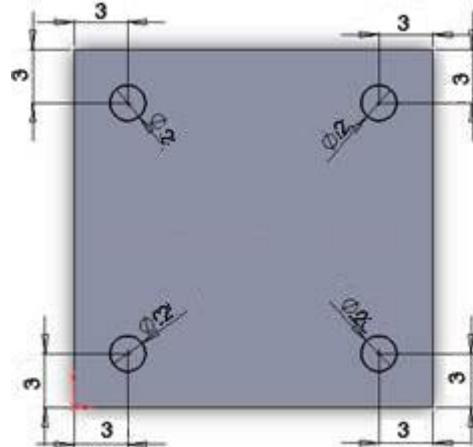
. Activate sketch menu by click Sketch and



select Circle . Sketch 4 circle at four edges.



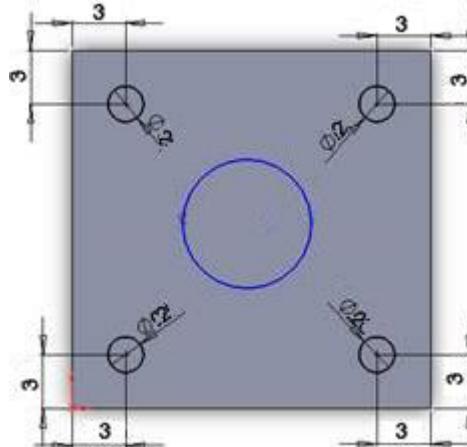
9. Define new circle sketch, click Smart Dimension , set diameter circle to

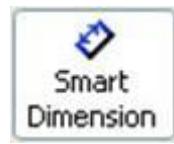


**2mm** . Select distance for edge set to **3mm**

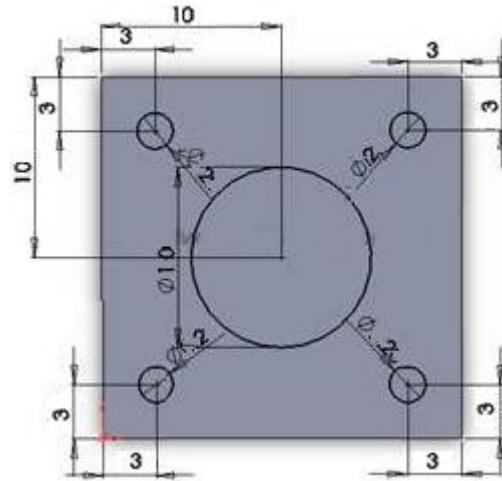


10. Click Circle and sketch one circle at center

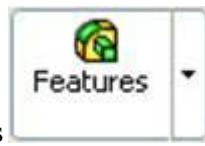




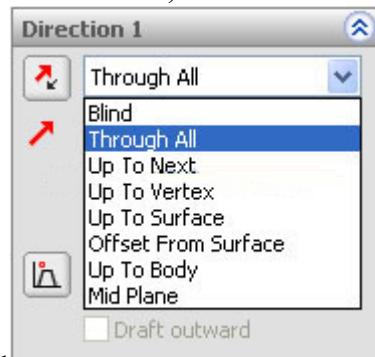
11. Define new circle sketch, click Smart Dimension , set diameter circle to



10mm . Select distance for edge set to **10mm**.



12. For cut click Features , click Extruded Cut , under

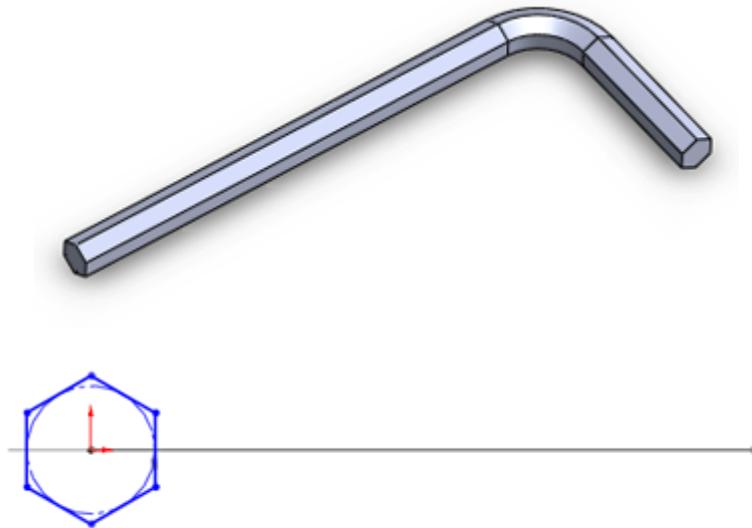


Direction 1, Through All



.Done

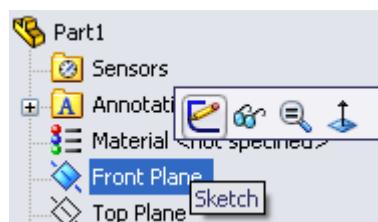
## 5. How to create Allen key



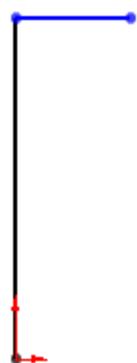
In this solidworks tutorial, you will create simple allen key.

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

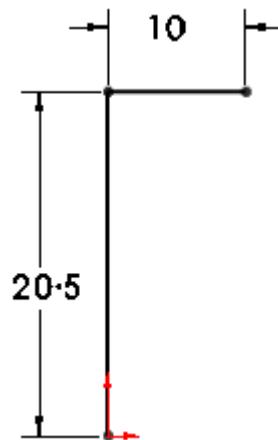
2. Click **Front Plane** and click on **Sketch**.



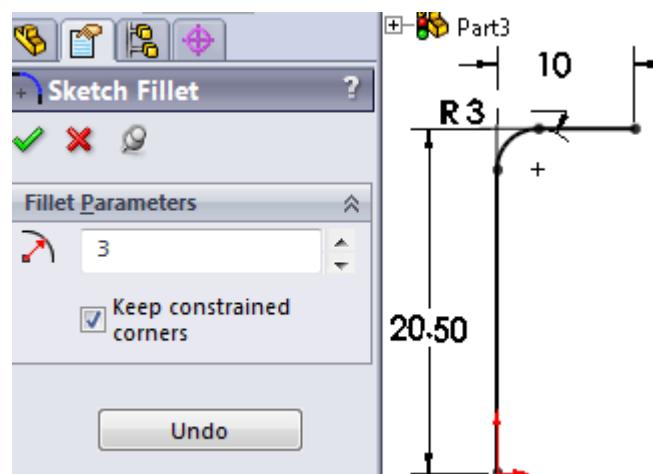
3. Click **Line**, sketch a L shape.



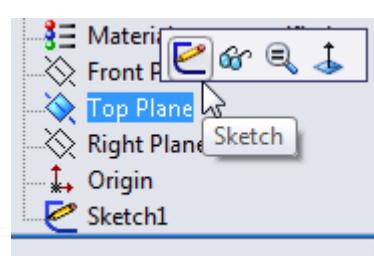
4. Click **Smart Dimension**, and dimension sketch as **25mm** and **10mm**.



5. Click **Sketch Fillet**, add 3mm fillet at L corner.



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



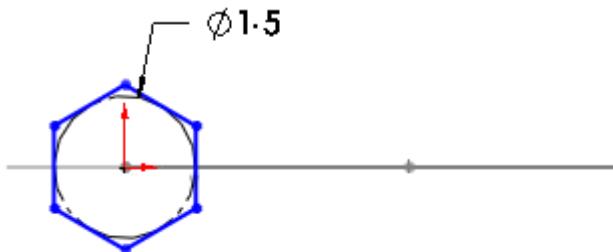
7. Click on **Sketch2** and click **Normal To**.



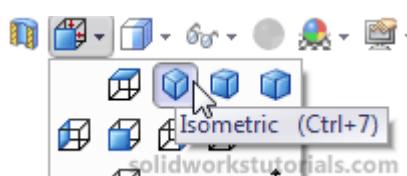
8. Click Polygon,  sketch a polygon at origin.



9. Click **Smart Dimension**,  and dimension sketch diameter to **1.5mm**.

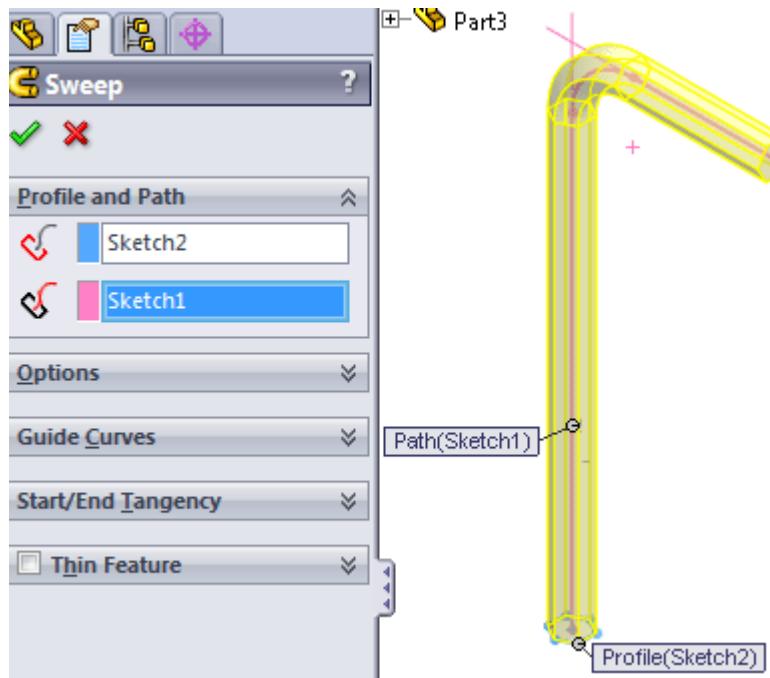


10. Exit sketch,  click on **Isometric** view.



11. Click **Features>Swept Boss/Base**,  for profile click on **Sketch2** and for path click on **Sketch1** and OK.

for profile click on **Sketch2** and for path click on **Sketch1** and OK.

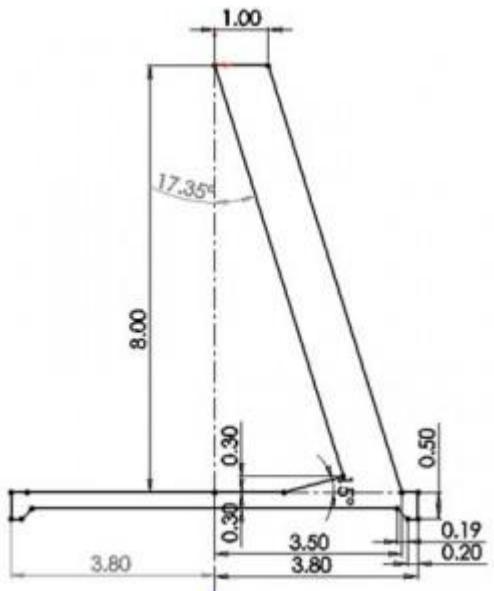


You're done!.

## 6. How to create 17 inch car wheel



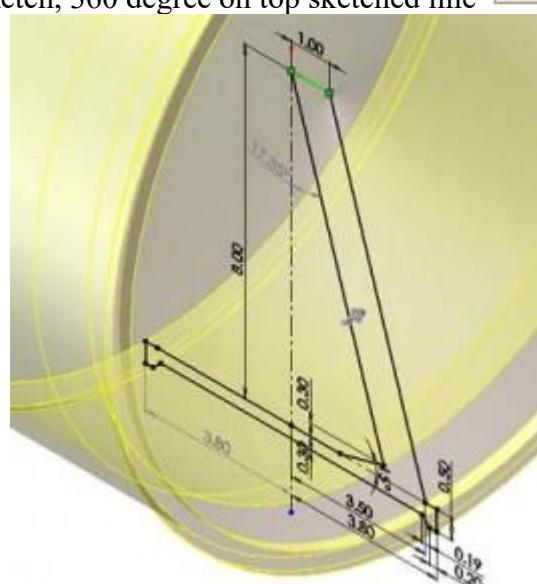
Click Option (Tools>Option...), select Document Properties tab. Select Units , under Unit System select IPS (inch, pound, second) OK.

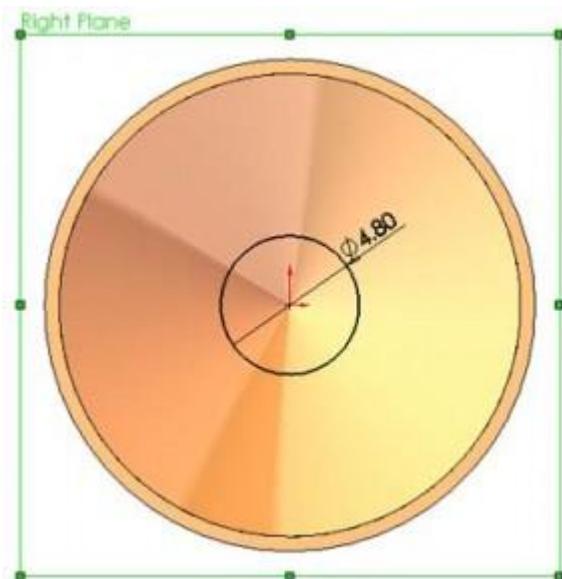


1. Create a sketch as show on Front Plane.

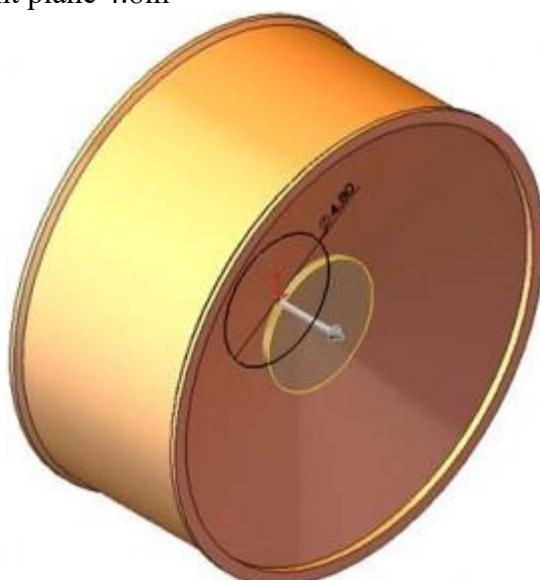


2. Revolve  sketch, 360 degree on top sketched line





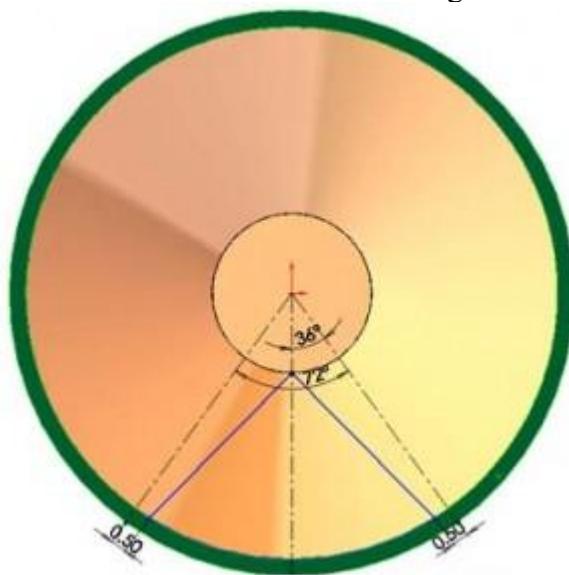
3. Create circle sketch, on right plane 4.8in



extrude 2in

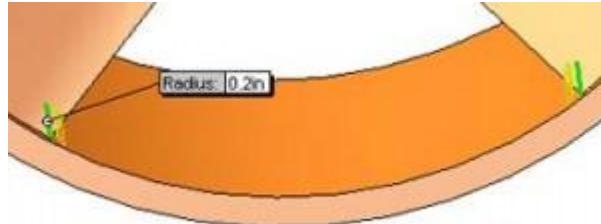
OK.

4. Insert sketch on edge wheel face, sketch for arm hole



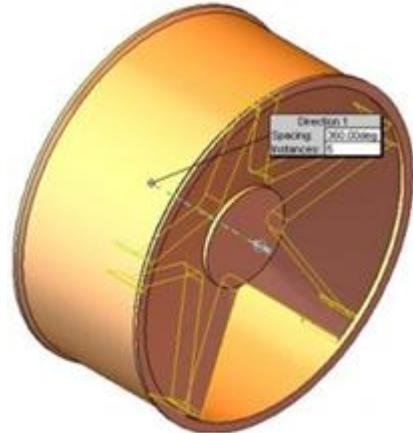
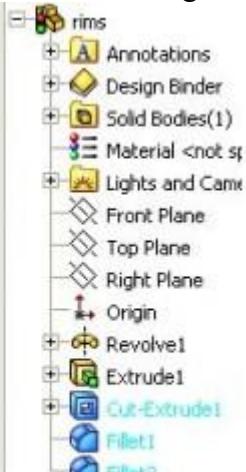
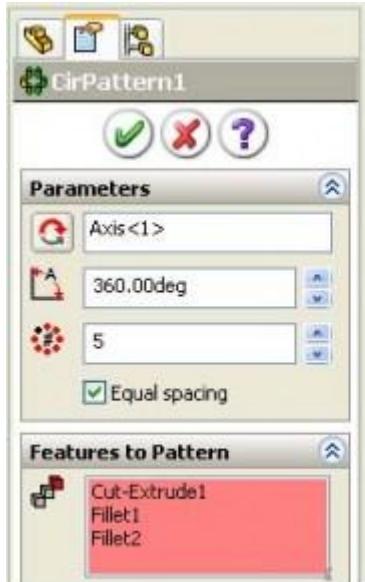
, extruded cut , through all, OK.

5. Add fillet R0.5in inner , add fillet 0.2in



OK.

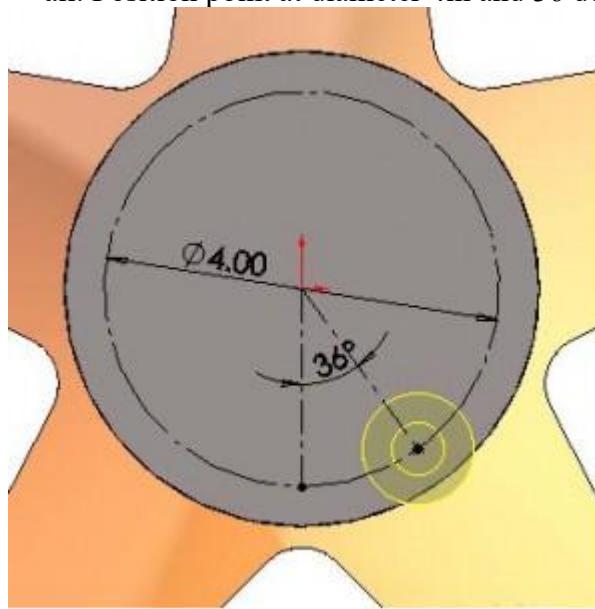
6. Click Circular Pattern , click View>Temporary Axes, select center axis as rotation axis. 360 degree and #5 equal spacing



. Select

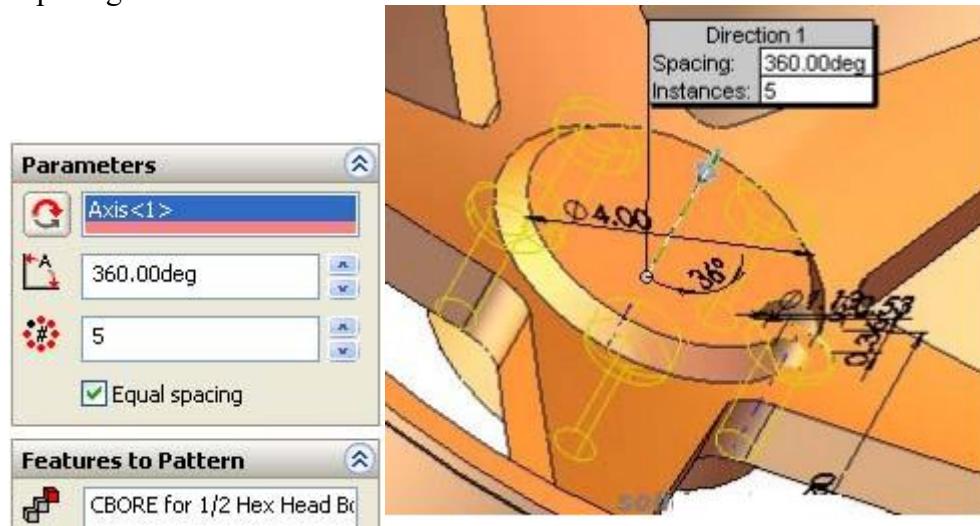
Cut-Extrude1, Fillet1 and Fillet2 as a Features to Pattern. OK.

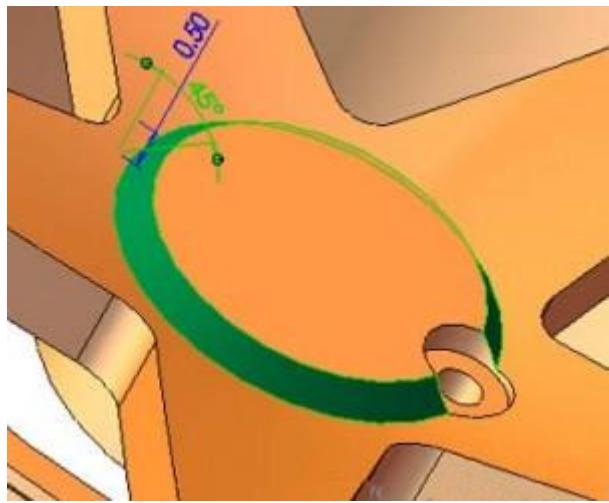
7. Select hub face, click Hole Wizard , select Ansi Inch, Hex Bolt, size 1/2, through all. Position point at diameter 4in and 36 degree



. OK.

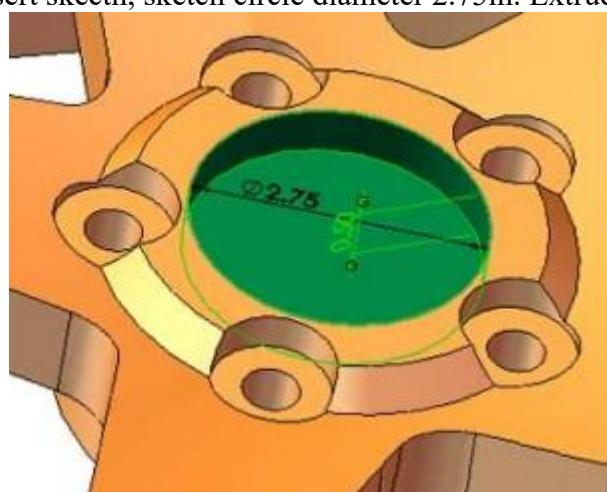
8. Click Circular Pattern , select center temporary axis, 360 degree and #5 equal spacing. Select CBORE for 1/2 Hex Head Bolt as Features to Pattern. OK.



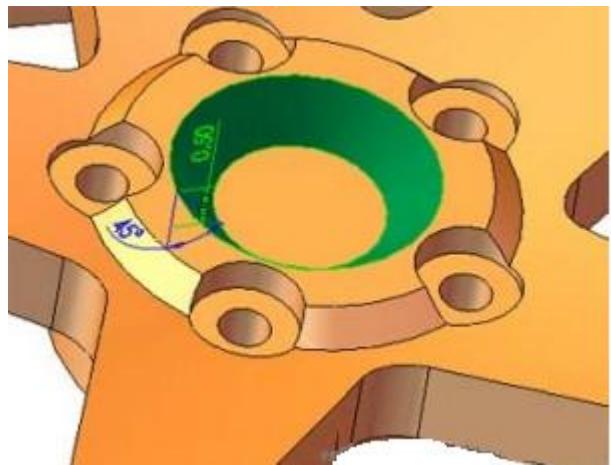


9. Add chamfer 0.5in to hub side.

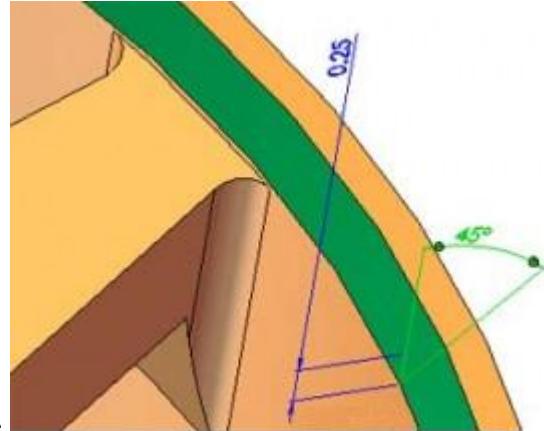
10. Click on hub face, insert sketch, sketch circle diameter 2.75in. Extrude Cut  to



0.5in deep.



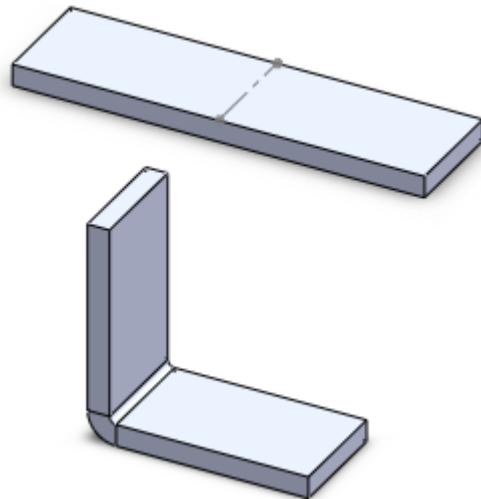
11. Add chamfer 0.5in to inner cut and add



chamfer 0.25in to wheel edge , OK. Done.



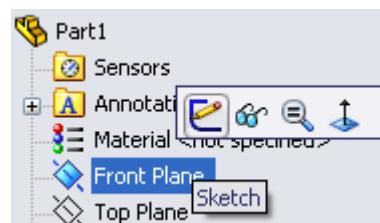
## 7. How to create simple sheet metal bend



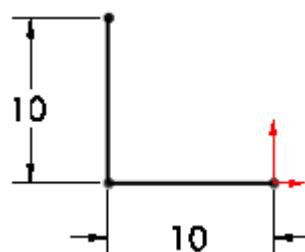
In this tutorials you will learn how to utilize sheetmetal tool such insert bend and flatten.

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

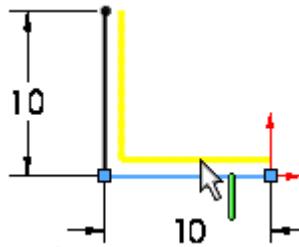
2. Click **Front Plane** and click on **Sketch**.



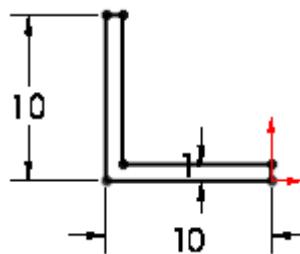
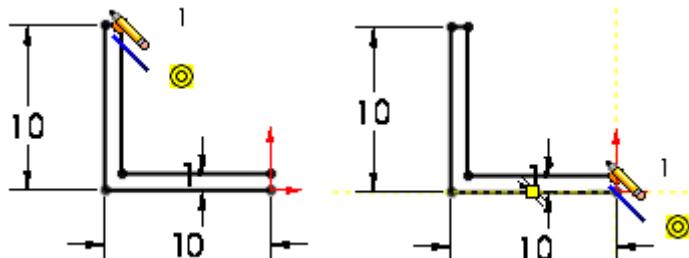
Use **Line** , sketch L shape. Dimension sketch with **Smart Dimension** as **10mm**



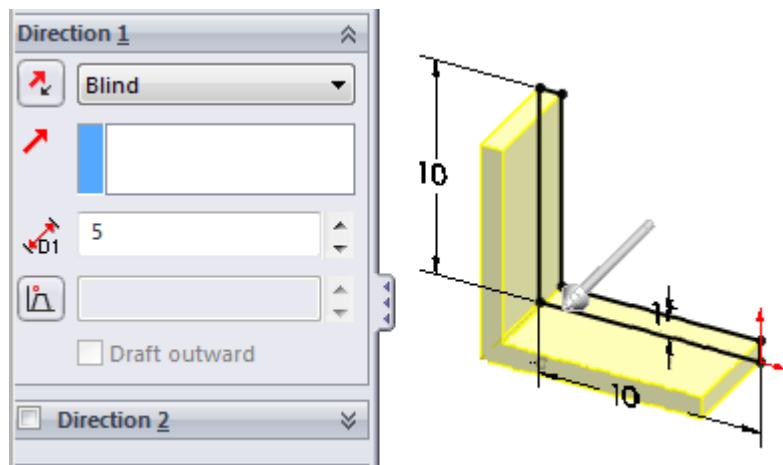
3. Click **Offset Entities** and click L sketch. Set offset distance as **1mm**.



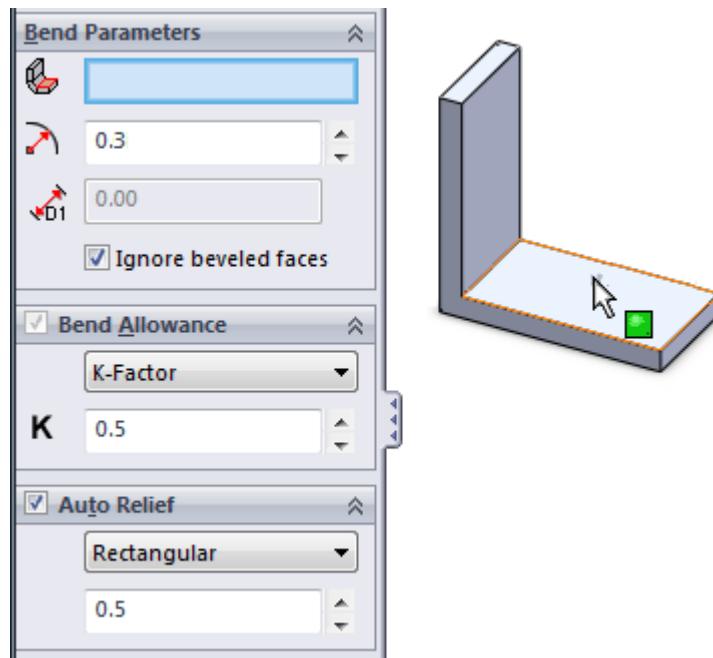
4. Use Line , sketch and connected open end of this sketch and make it close both end.



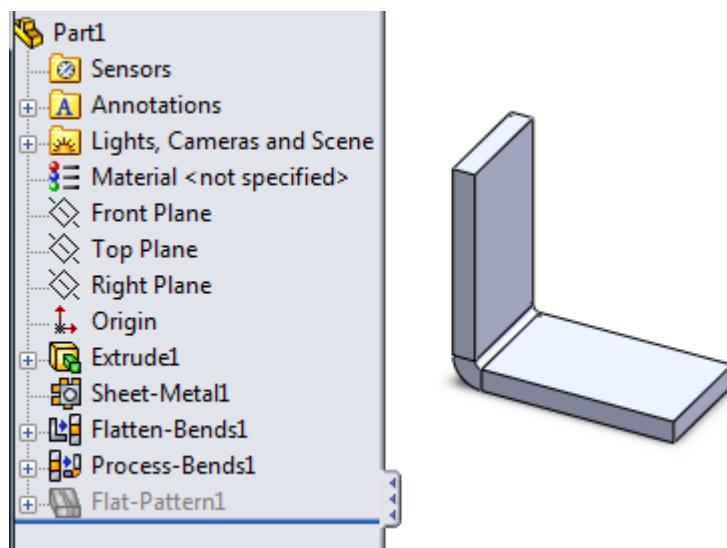
5. Click Features>Extruded Boss/Base set D1 to 5mm and OK.



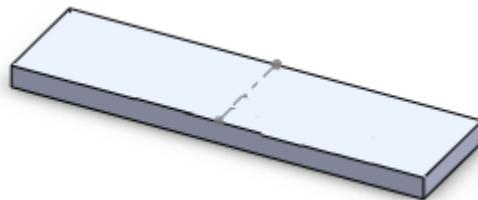
6. Click Sheetmetal>Insert Bends, click flat face as reference when it flatten. Set bend radius to 0.3mm and K factor 0.5 and OK.



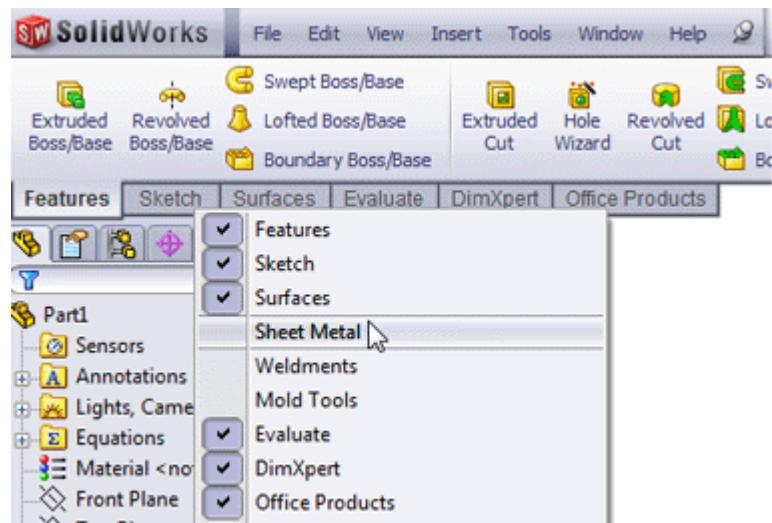
7. Your simple sheetmetal bend is ready. Look at part tree.



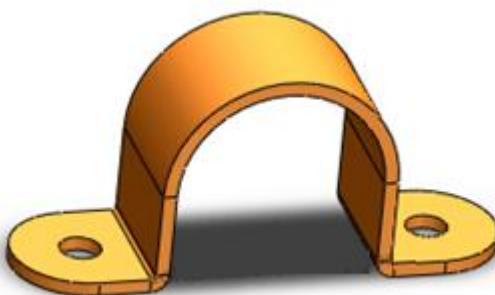
8. To view this part in flatten form click **Sheetmetal>Flatten**.



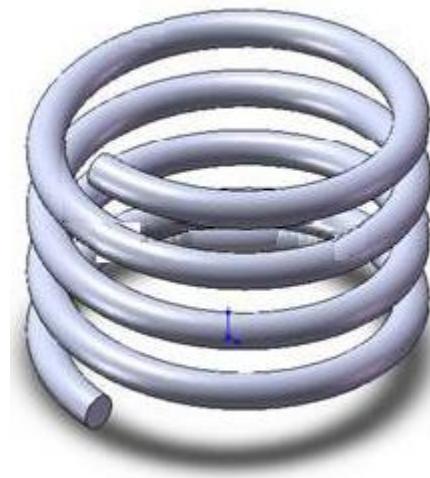
Have fun.. If you cannot find the sheetmetal tool in your main tool menu, you can right click on main menu tab and check Sheetmetal option.



You know the basic, try model this bracket.



## 8. How to create spring



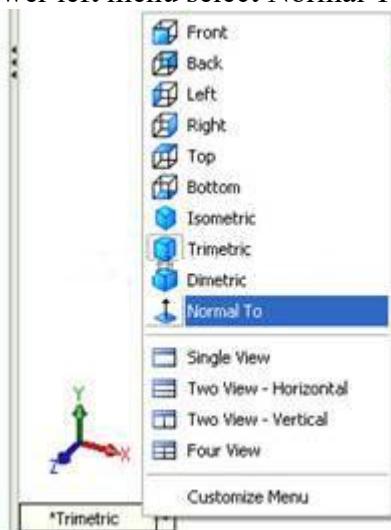
1. Click New (File>New) , click Part , OK .



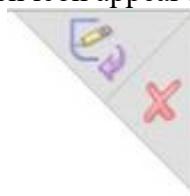
2. Click Option (Tools>Option...) , select Document Properties tab. Select Units , under Unit System select MMGS (millimeter ,gram , second) OK



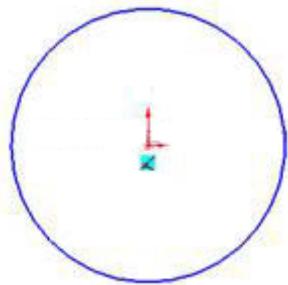
3. Select Top Plane , from lower left menu select Normal To.



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



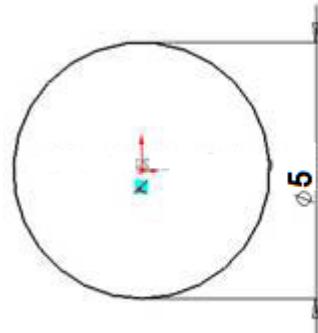
5. Pick Origin point as starting point, drag to right hand side



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

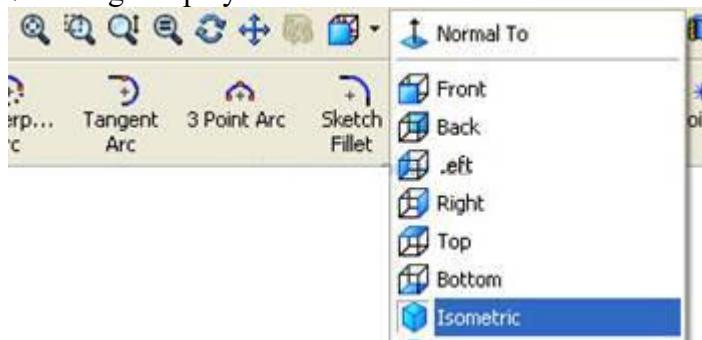
Note: There is two type line generated by in sketching, the one with black line and blue line. Black line is line that fully defined and blue line is under defined..

6. Define sketch with dimension. Click Smart Dimension , and start

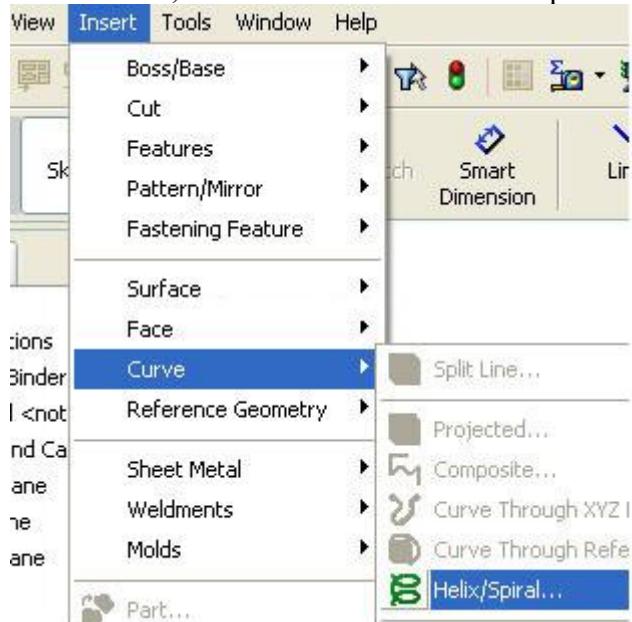


dimensioning pick circle edge and set to 5mm . Press keyboard  
ESC to end smart dimension.

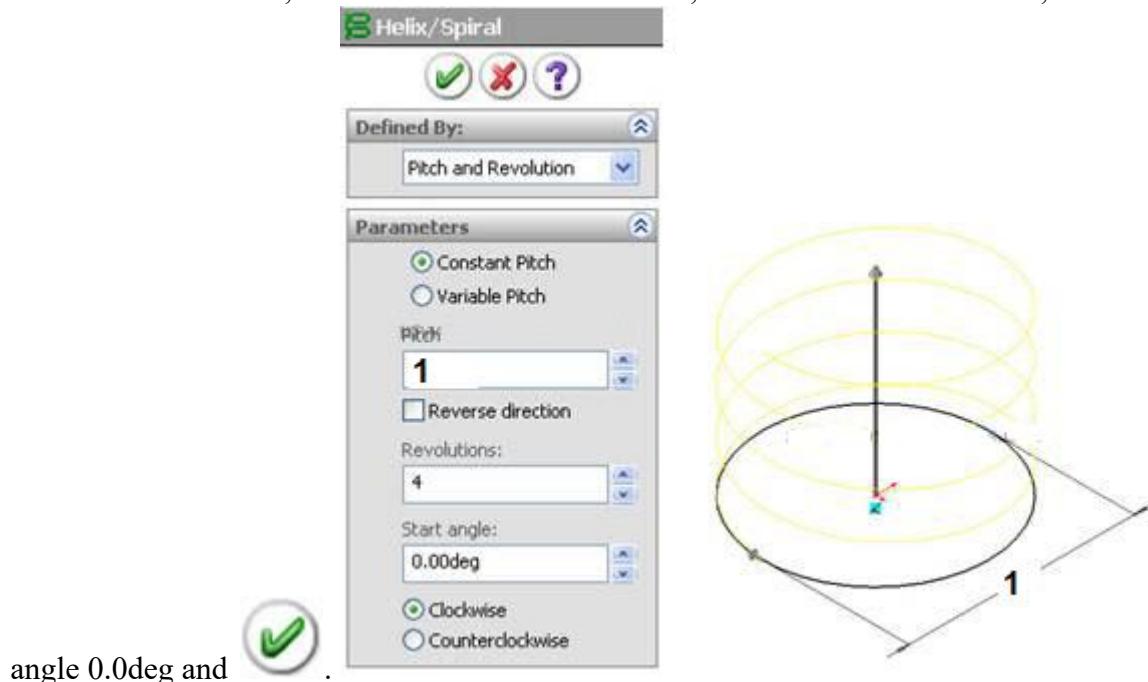
7. Change display to Isometric view.



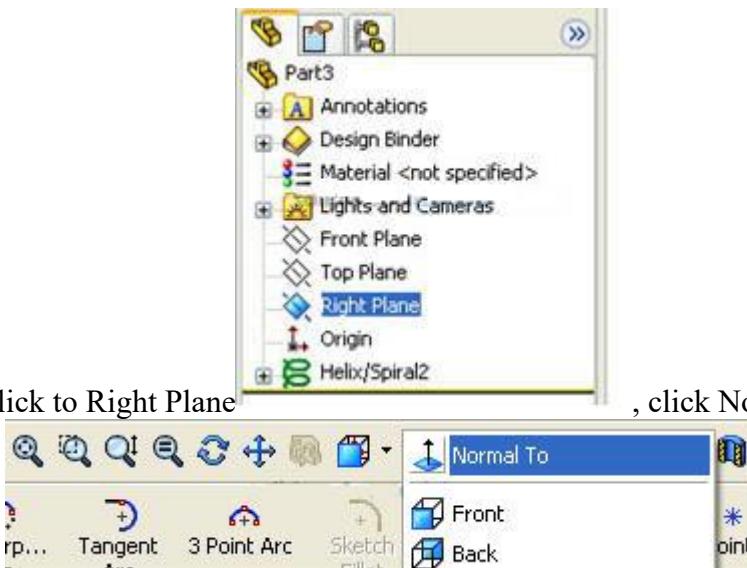
8. Insert coil, Click Insert>Curve>Helix/Spiral



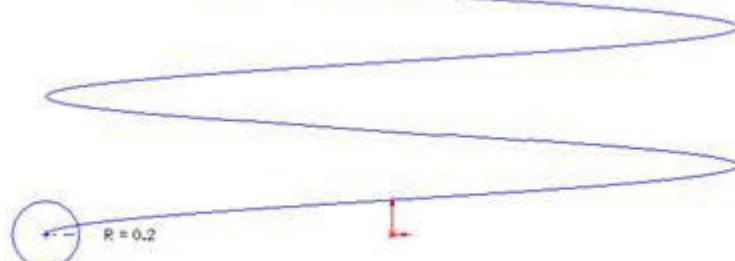
9. Press F to zoom fit, set Parameters Constant Pitch , Pitch 1mm Revolutions 4 , Start angle 0.0deg and



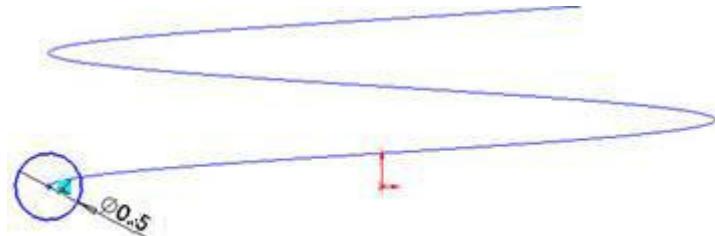
10. Click to Right Plane , click Normal To



11. Click Sketch , click Circle . Sketch circle at start point, then click Smart dimension set



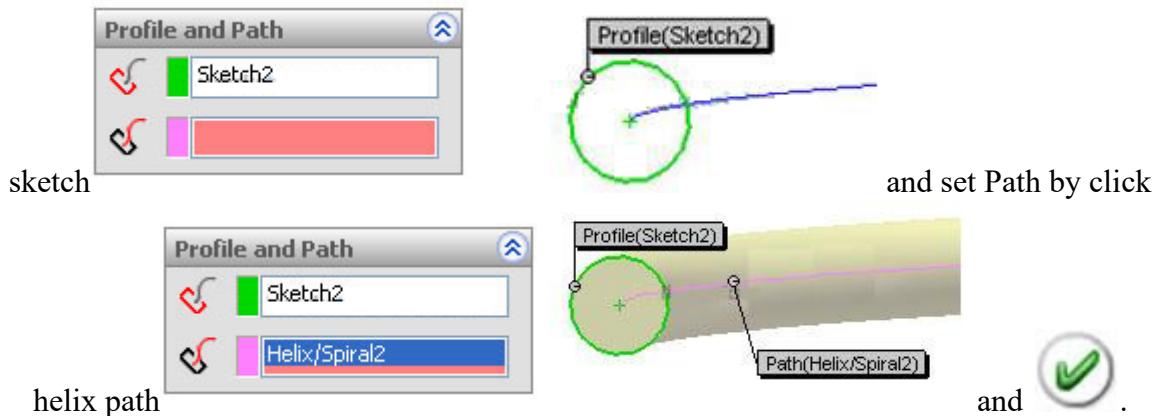
circle diameter to 0.5mm .



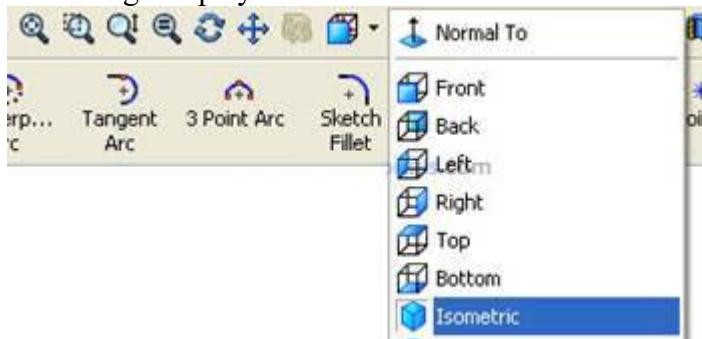
12. Click exit sketch . Click Features and activate features

menu. Click Swept Boss/Base and set Profile to Sketch2 by click on circle



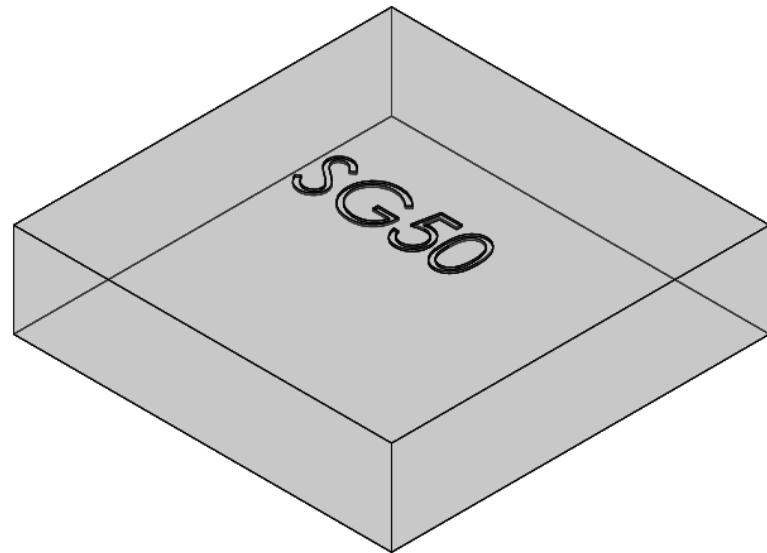


13. Change display to Isometric view.



14. Press F to zoom fit.

## 9. How to engrave text to part

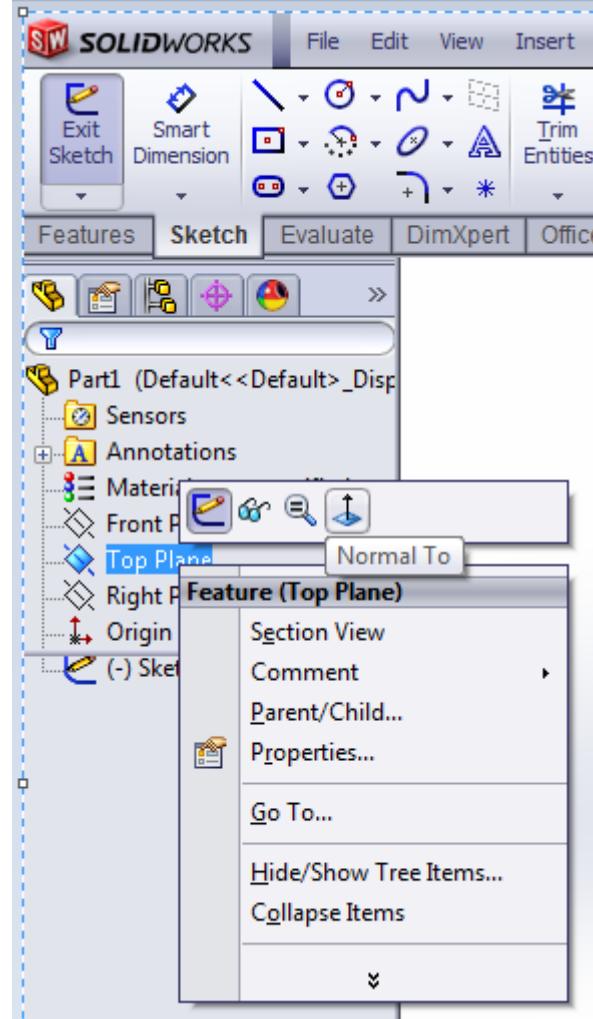


1. Click New (File>New) , click Part , OK .

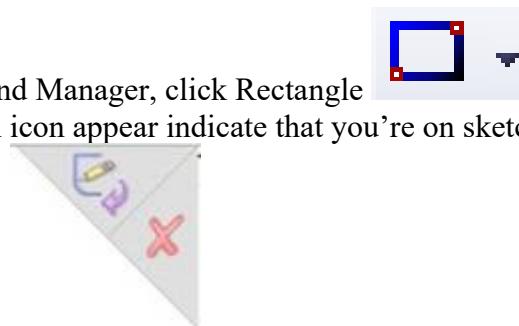


2. Click Option (Tools>Option...) , select Document Properties tab. Select Units , under Unit System select MMGS (millimeter,grm,second) OK .

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

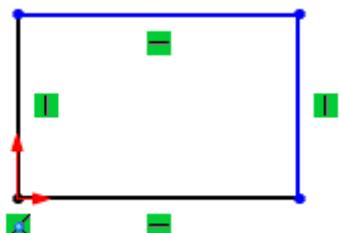


4. Click Sketch in Command Manager, click Rectangle . As you can see on upper right corner sketch icon appear indicate that you're on sketch mode

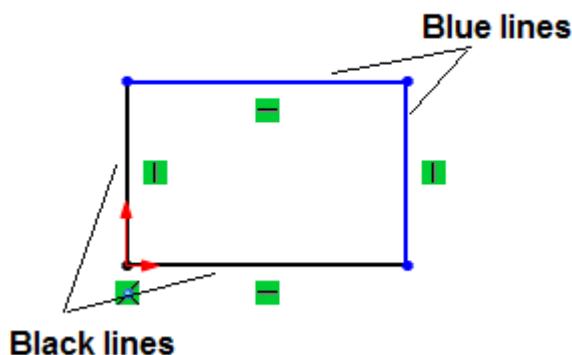




5. Pick Origin point as starting point, drag to right hand side



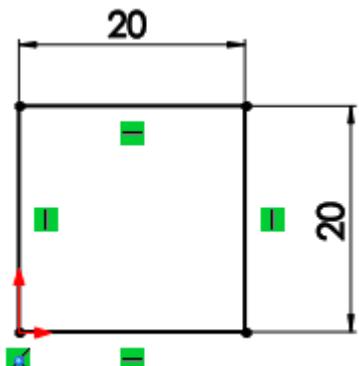
no need to be exact the size will define in later step. Press keyboard ESC to end rectangle sketch.



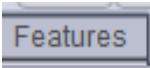
Note: There is two type line generated by your sketching, the one with black line and blue line. Black line is line that fully defined and blue line is under defined.

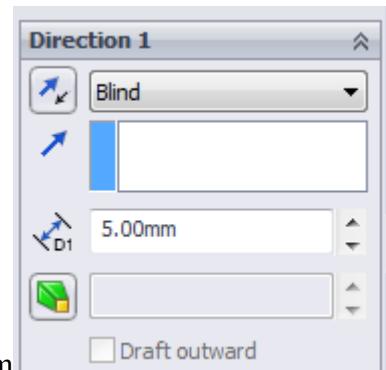


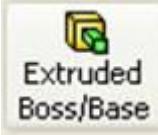
6. Define sketch with dimension. Click Smart Dimension , and start dimensioning pick vertical line and set to **20mm**, pick horizontal line and set to **20mm**

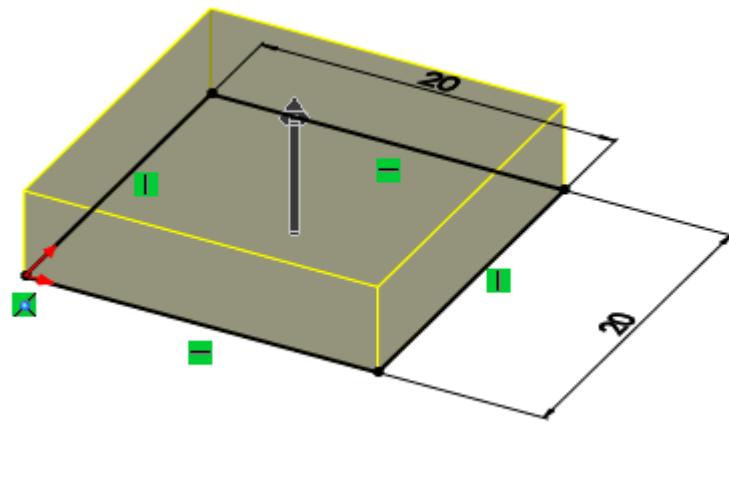


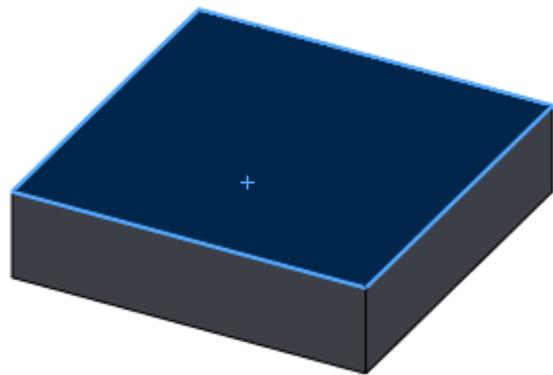
. Press keyboard ESC to end smart dimension.

7. Build feature from sketch, click Features  and activate features menu. Click

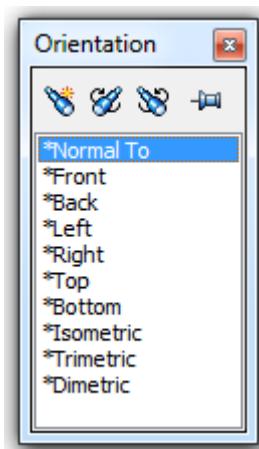


 Extruded Boss/Base and set D1 to 5mm and

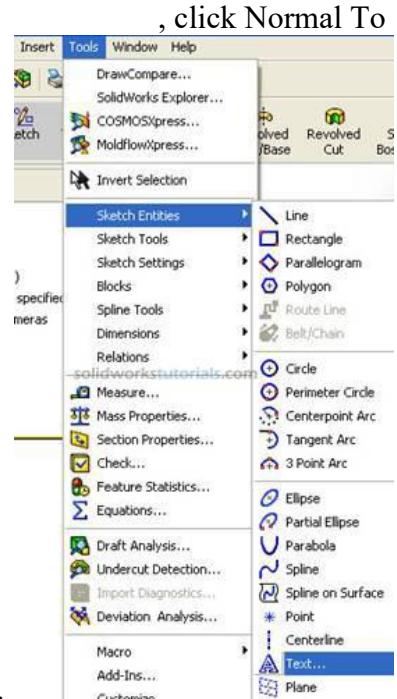


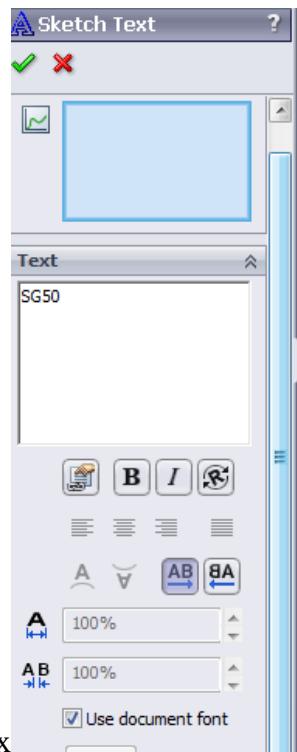


8. Click front top face

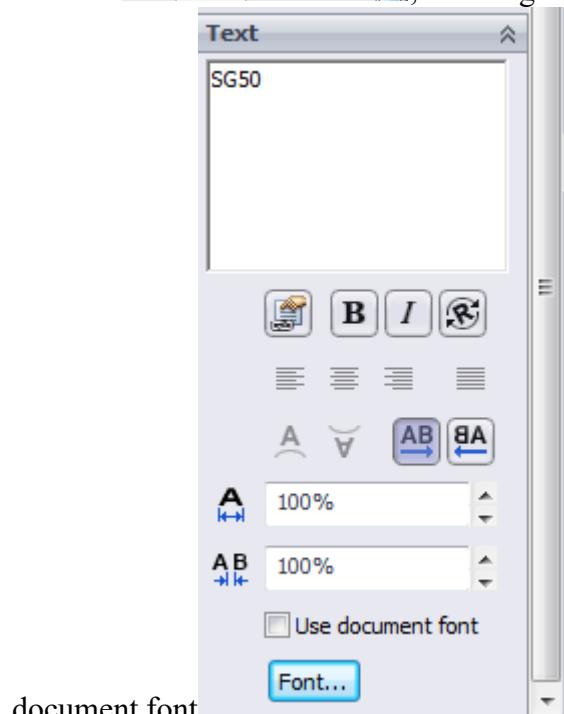


. Click Tools>Sketch Entities>Text...

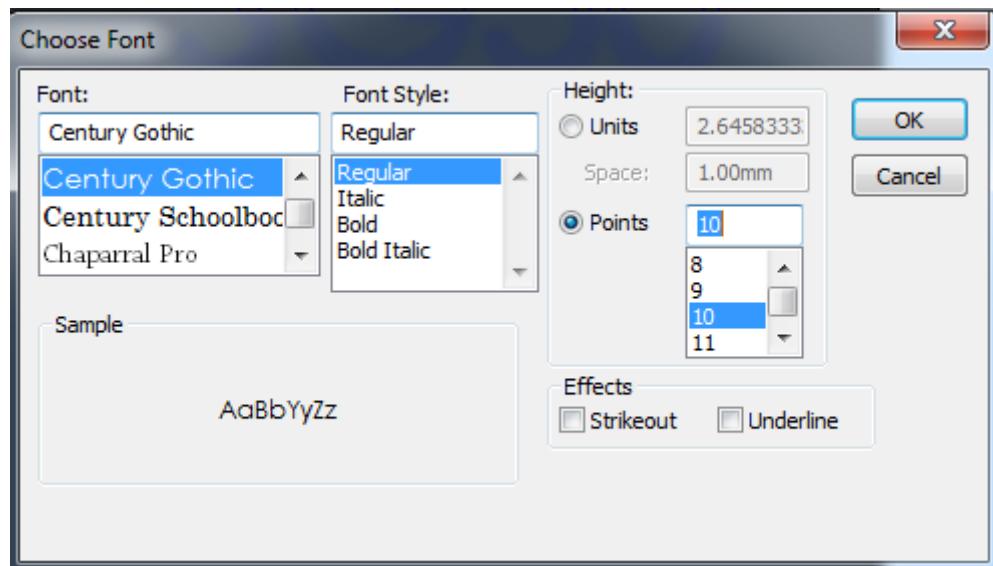




9. Input text in text box, to change font type and size uncheck use



document font . Click Font...

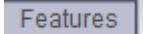
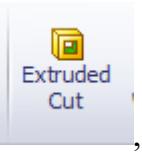


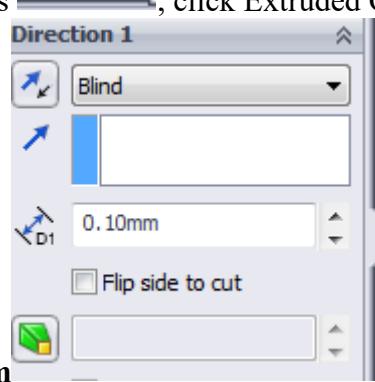
set height to

Points 10 OK.



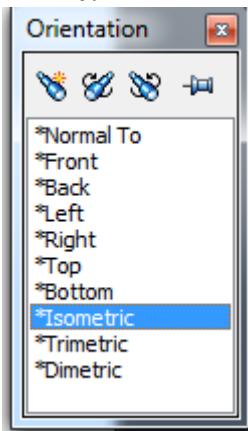
10. Click to part face to relocate text to center.

11. To engrave the text, click Features , click Extruded Cut , under



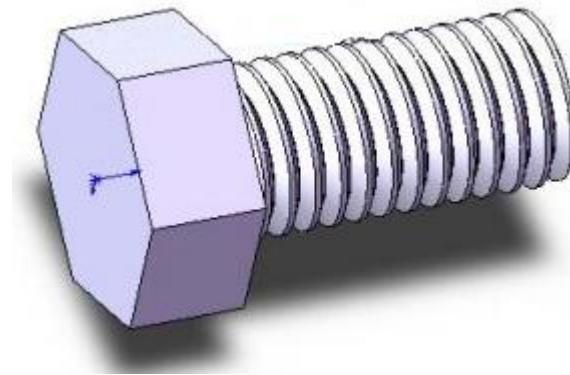
. Click

Direction 1 Blind, set D1 to **0.1mm**

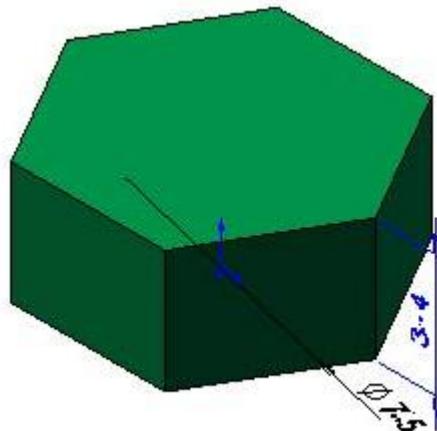
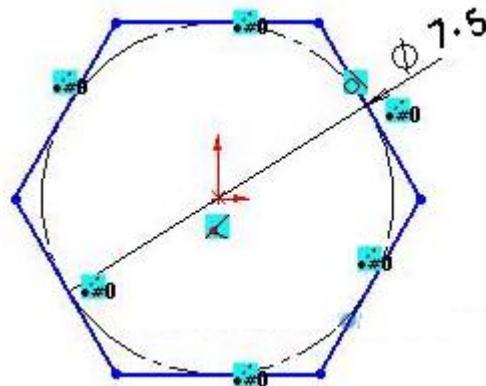


Isometric  from lower left view menu.

## 10.How to create hexagonal head bolt

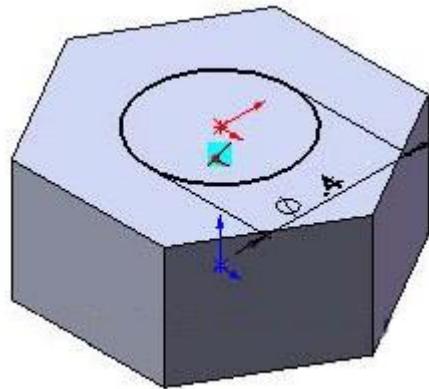


1. Sketch a polygon with 6 side, Tools>Sketch Entities>Polygon

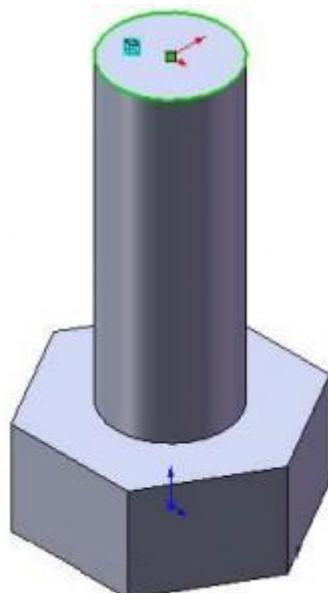


2. Extrude sketch to 3.4mm

3. Create minor diameter for thread, sketch circle on top face, set diameter to 4mm



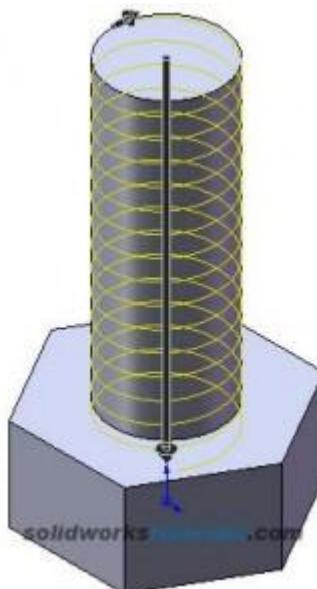
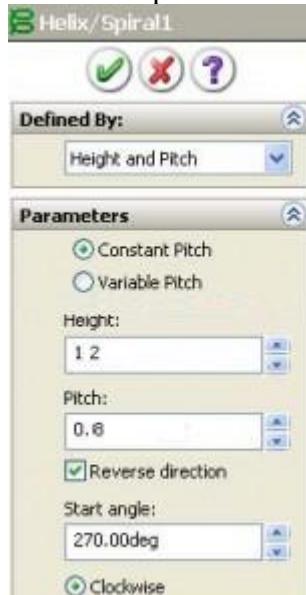
4. Extrude sketch to 11mm



5. Click end edge of thread shaft,

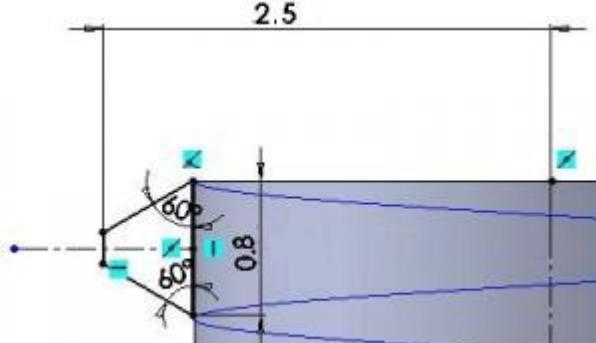
click convert entities .

6. Select Helix/Spiral feature  set height to 12mm, pitch 0.8mm

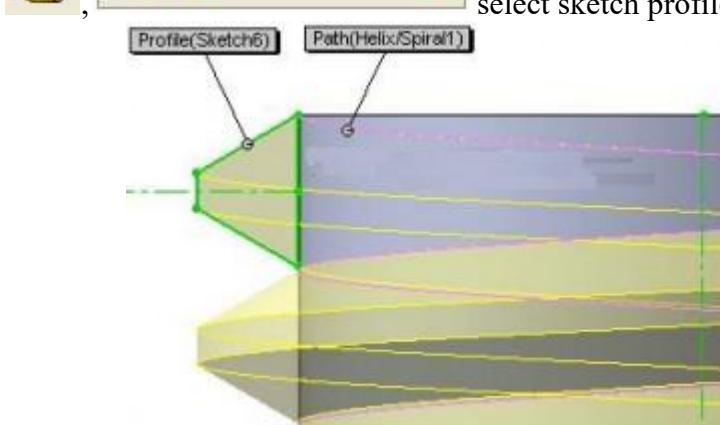


Ok.

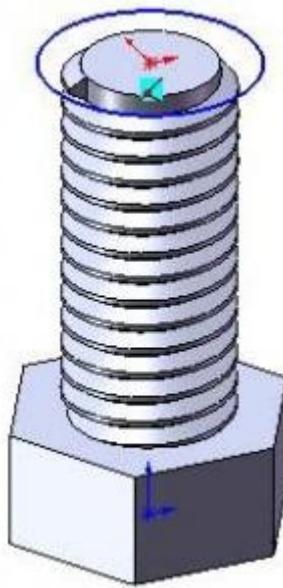
7. Right click on Front plane, Insert sketch  sketch thead profile.



8. Click sweep feature  , select sketch profile as sketch



and helix as a path, OK.



9. Create sketch a circle on the end shaft,

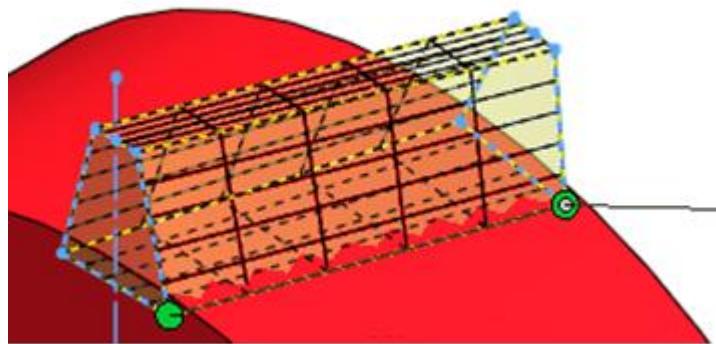
extrude cut 1mm.



10. Finish.

## 11. How to create helical gear

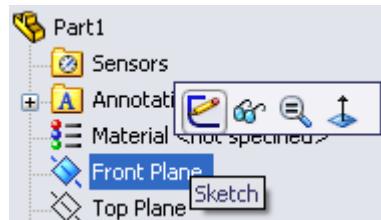




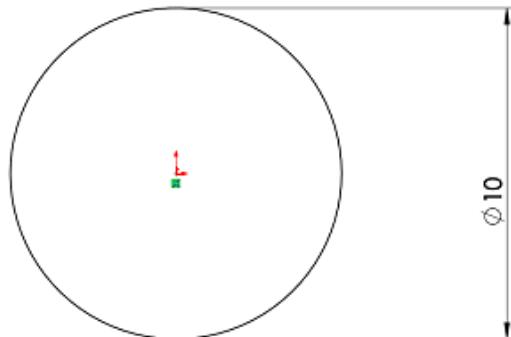
In this solidworks tutorial, you will create helical gear.

1. Click New. Click Part, OK.

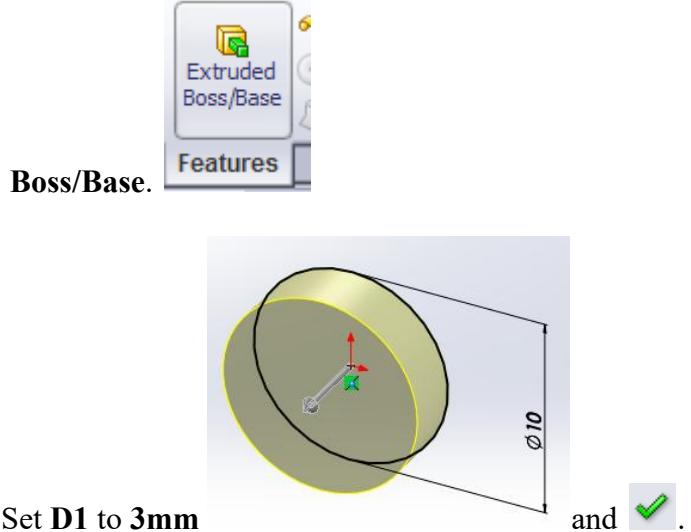
2. Click **Front Plane** and click on **Sketch**.



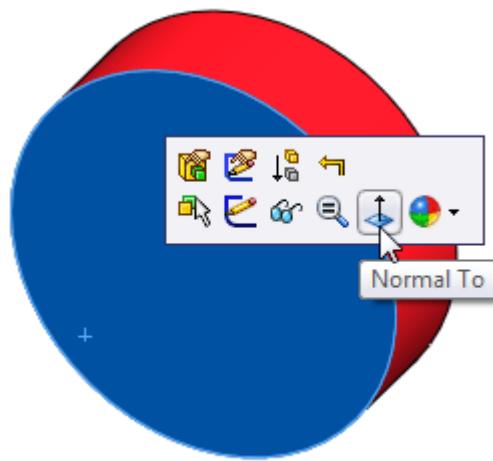
3. Click **Circle** and sketch a circle center at origin. Click **Smart Dimension**, click sketched circle and set it diameter to **10mm**



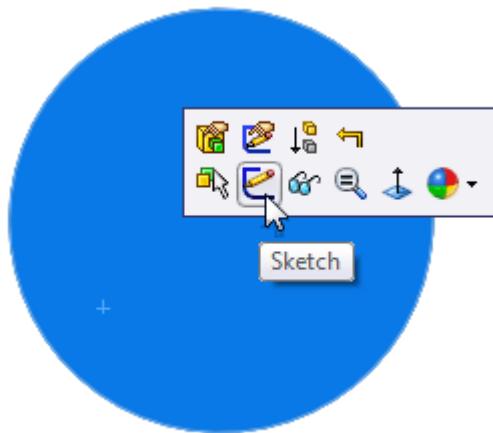
4. You just completed your sketch, let's build feature from it. Click **Features>Extruded Boss/Base**.



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

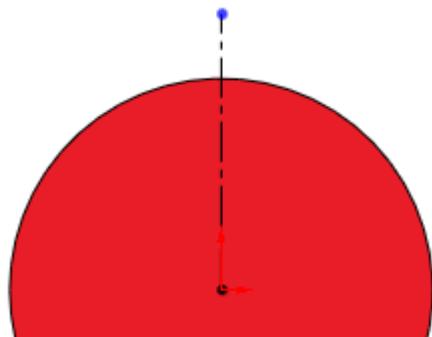


6. Click on front face and click **Sketch**.

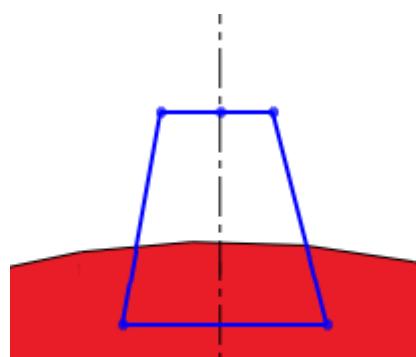




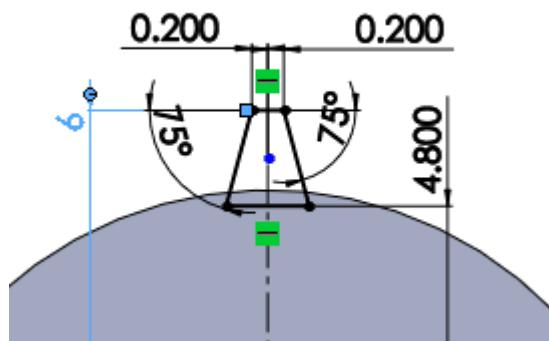
7. Click on **Centerline** and sketch vertical Centerline.



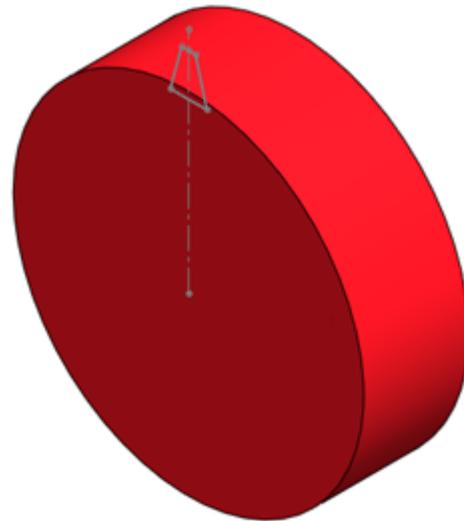
8. Click **Line** and sketch gear teeth profile.



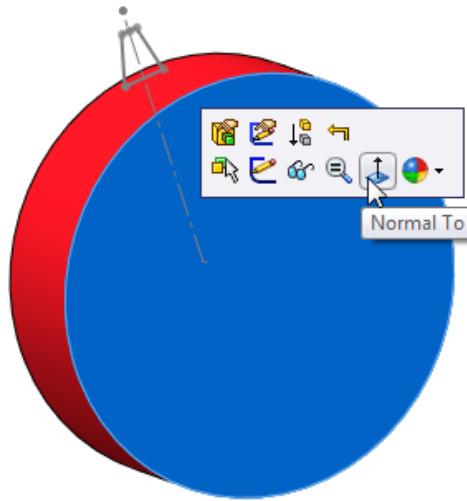
9. Click **Smart Dimension**, dimension sketch as sketched below.



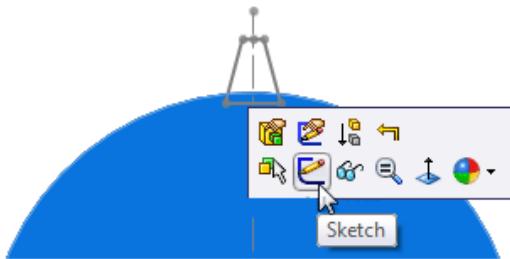
10. Click **Exit Sketch**, change view to **Isometric**.



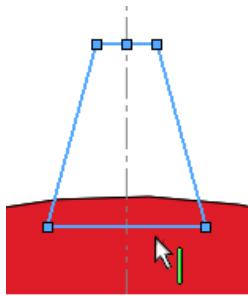
11. Click scroll mouse button and rotate the part to back side.



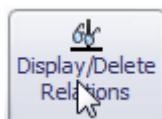
Click the back face and select **Normal To**. Click on this face again and click **Sketch**.



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

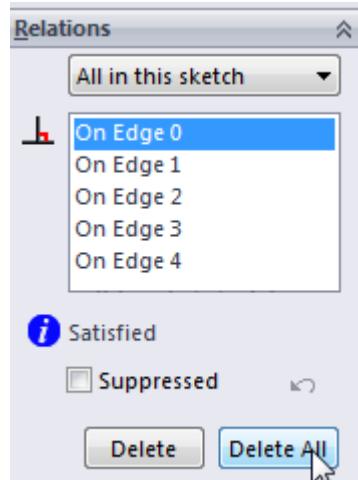


and click **Convert Entities**. Now we need removed all relation between this sketch and the other sketch, click **Display/Delete Relations**



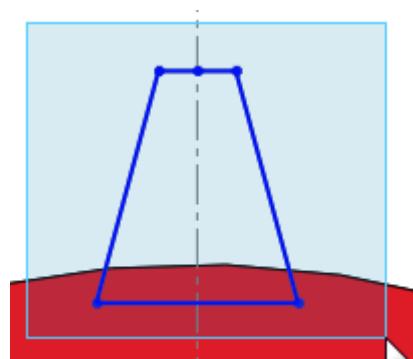
click **Delete All**

(Alternative method to use Project Curve (**Sketch on faces**) > Select line drawn from Top Plane > and Select face diameter)

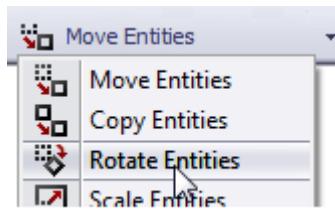


and

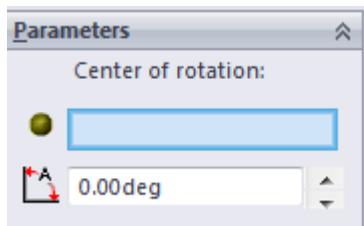
13. Click and drag select all the sketch line.



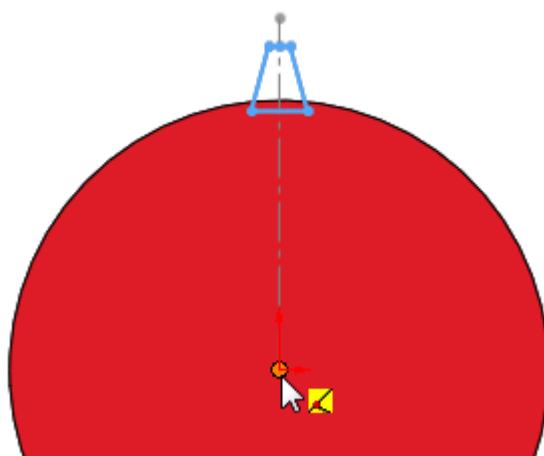
Click on **Rotate Entities**,



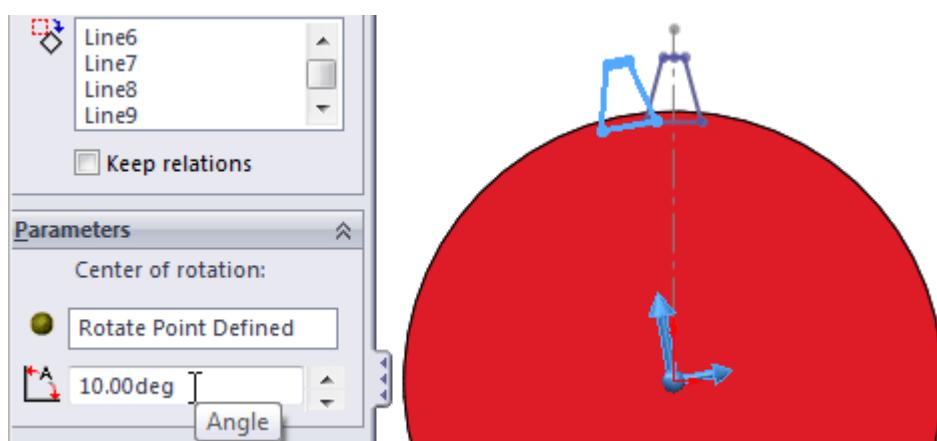
Click Center of Rotation box



and click origin (center part).

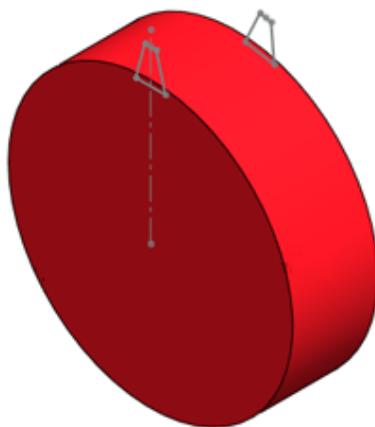
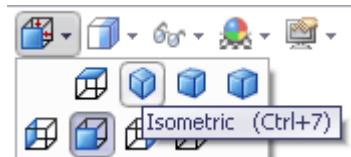


On Parameter option enter **10 deg** rotation.

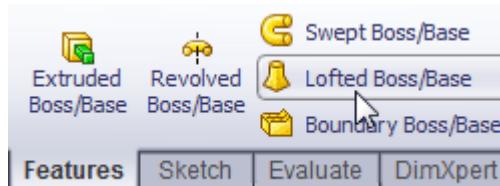


and .

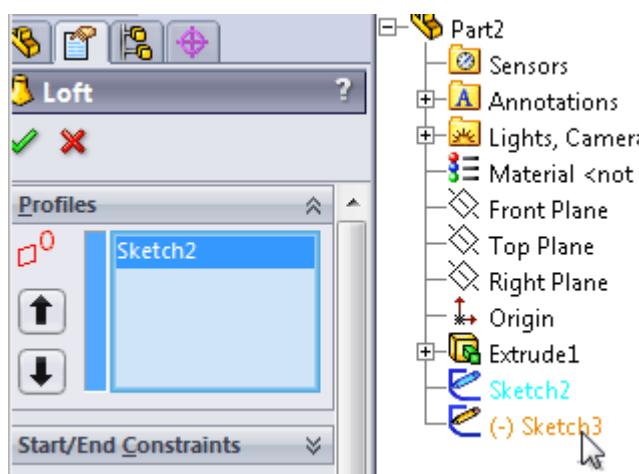
14. Click **Exit Sketch**,  change view to **Isometric**.

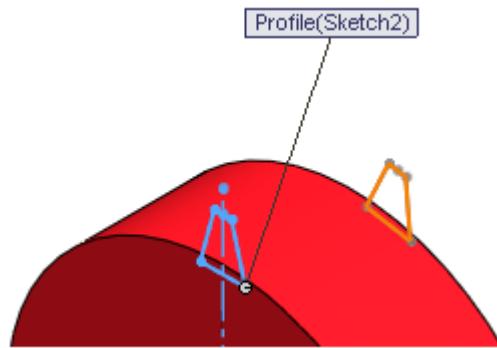


15. Click **Features>Lofted Boss/Base**,

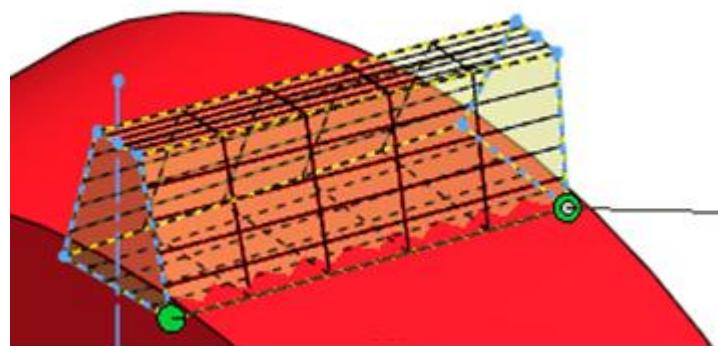


open up part tree and double click **Sketch2** and **Sketch3** to add for lofted features.

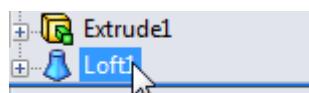
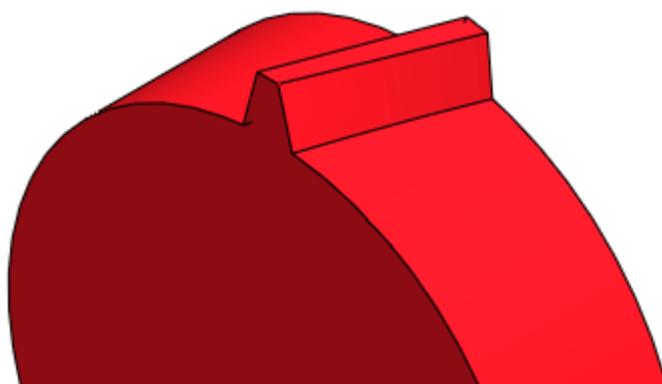




Make sure two green point is at the same edge as other sketch, if not drag and relocate it.



and

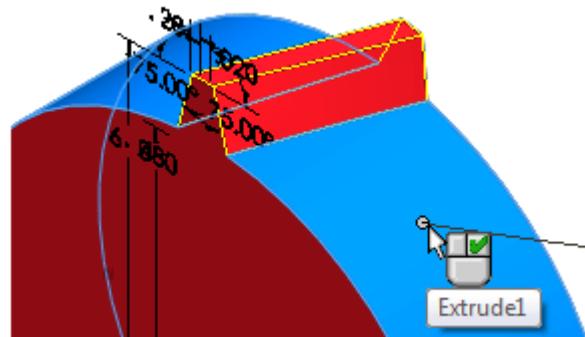


12. Click on **Loft1** (gear teeth) and

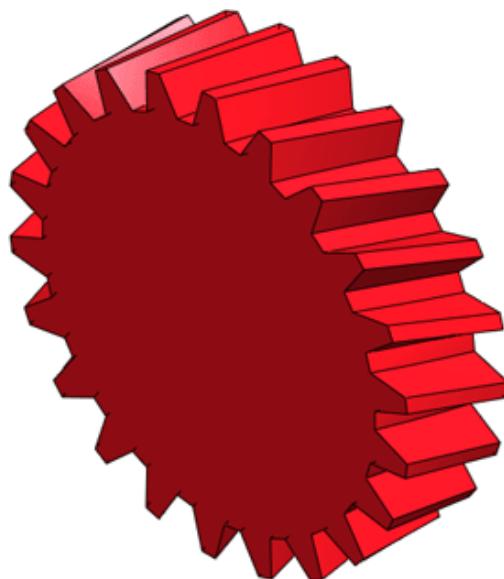
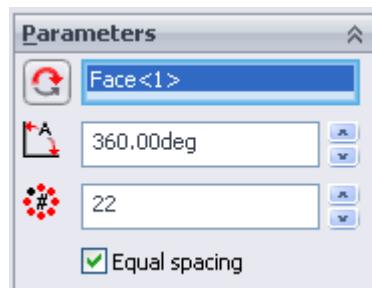


click **Circular Pattern**.

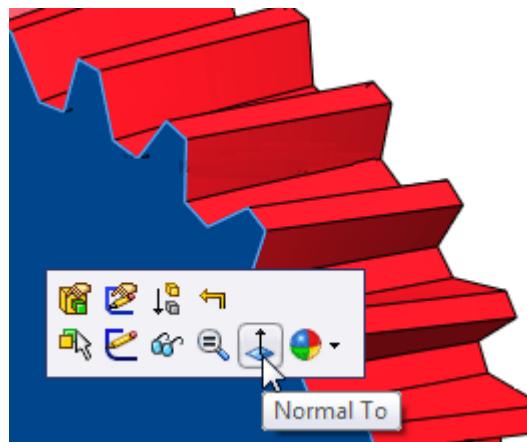
Click on the cylinder face as axis of rotation (or click on View>Temporary Axes select the temporary axis as axis of rotation).



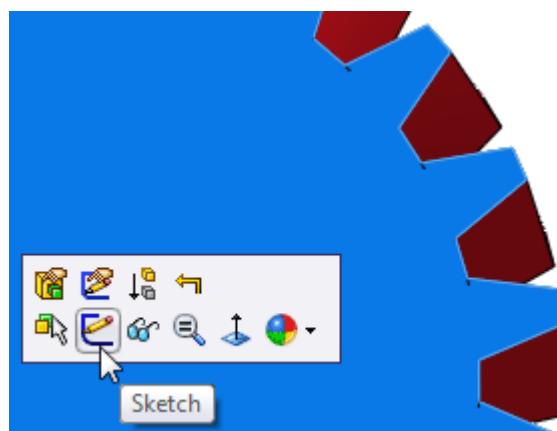
Set Instances to 22 and  .



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

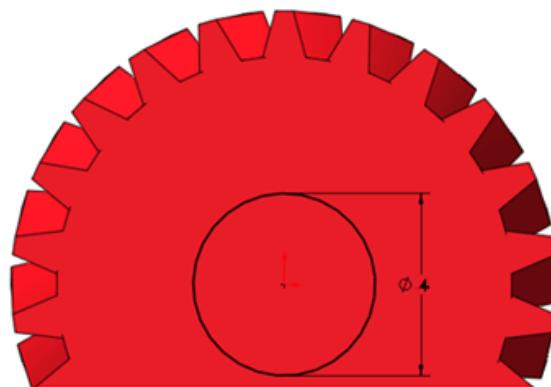


14. Click on front face and select **Sketch**.



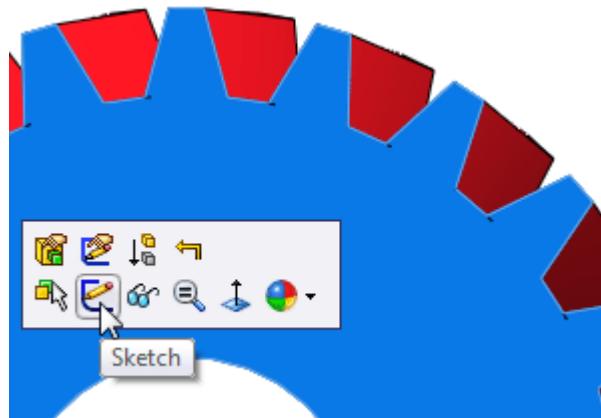
15. Sketch a **Circle** and sketch a circle center at origin. Click **Smart Dimension**,

dimension sketch as **4mm** circle.



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

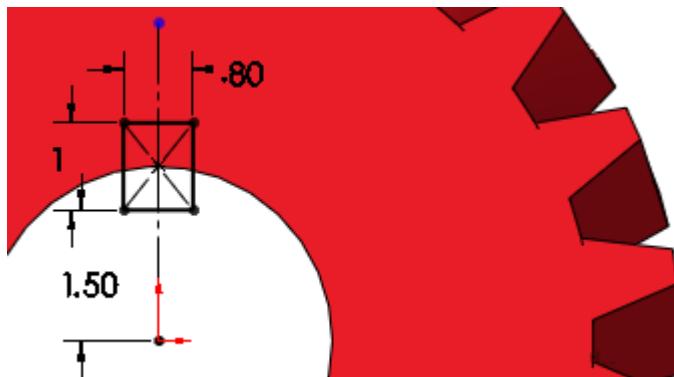
17. Click on front face and select **Sketch**.



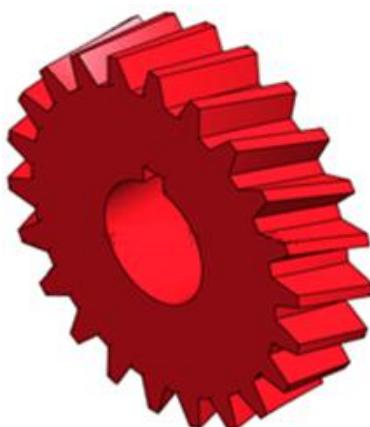
18. Click **Rectangle** and sketch a rectangle as sketched. Click **Smart Dimension**,



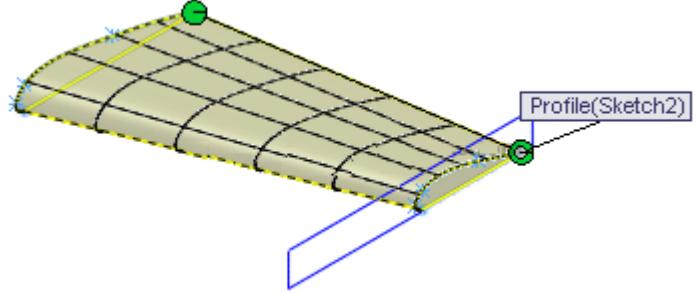
dimension rectangle as sketched below.



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



## 12.How to create aero plane wings

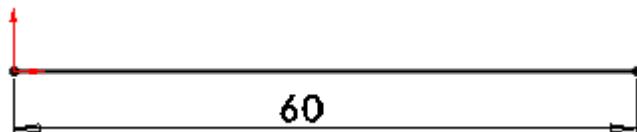


1. Click New, Part and OK.

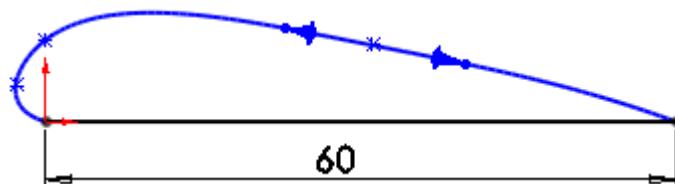
2. Click on **Right Plane** and click **Sketch**.



3. Sketch a center aerofoil profile at this plane. Click **Line**, sketch a horizontal line, click **Smart Dimension** and dimension the line as **60mm**.

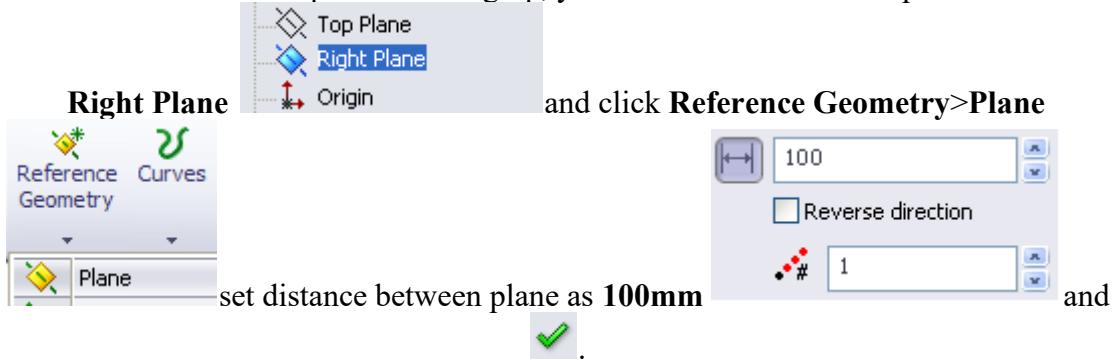


4. To create top curve of aerofoil, click **Spline**, and sketch top curve as sketched below, to end Spline press **Esc** key.



Exit the sketch.

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



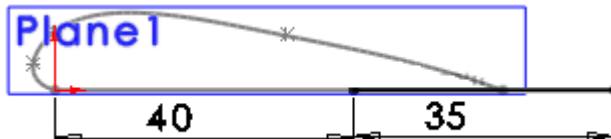
set distance between plane as **100mm** and

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

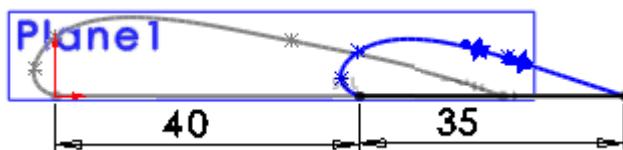


7. Click **Line**, sketch a horizontal line on same level as first sketch a bit off set from

origin, click **Smart Dimension** and dimension sketch as sketched below.

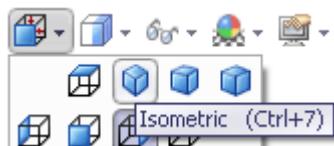


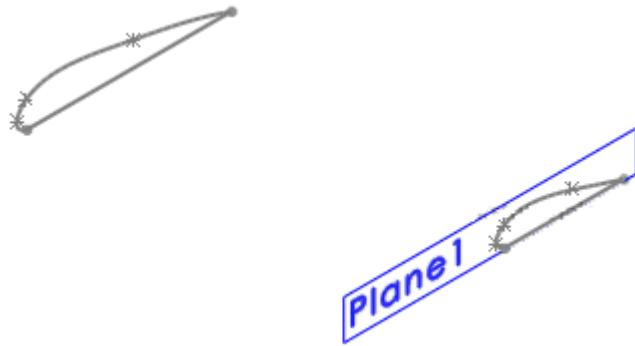
8. To create top curve of aerofoil, click **Spline**, and sketch top curve as sketched below, to end Spline press **Esc** key.



**Exit** the sketch.

9. Click **View Orientation>Isometric**.

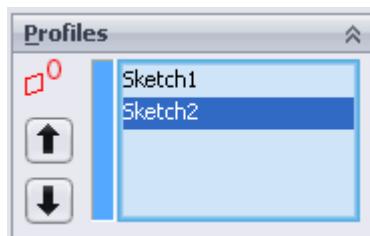
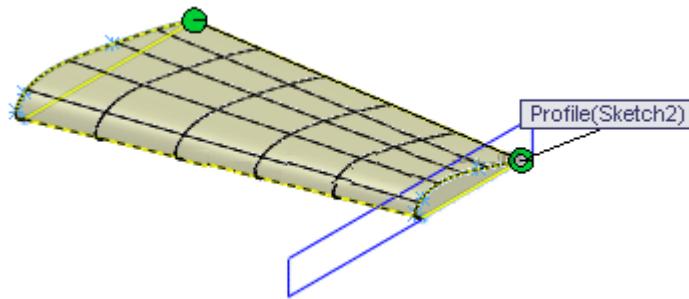




10. Click **Features>Lofted Boss/Base**,



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

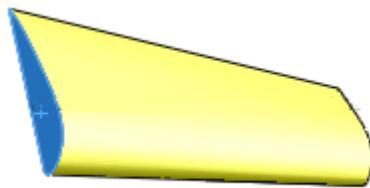


and .

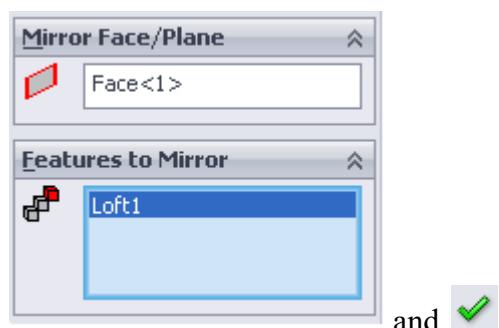
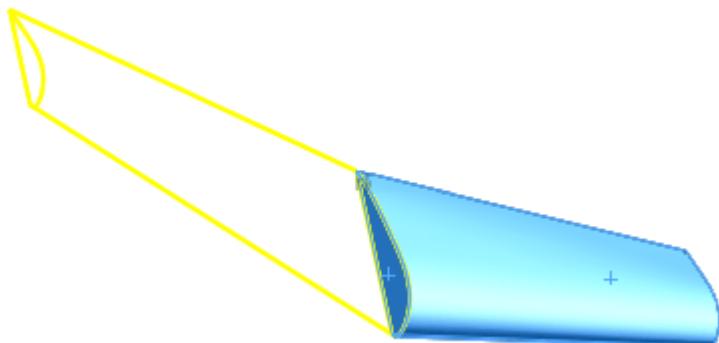
11. To hide Plane 1, click **Plane 1** and click **Hide**.



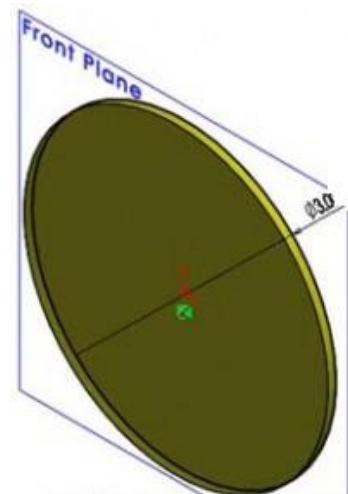
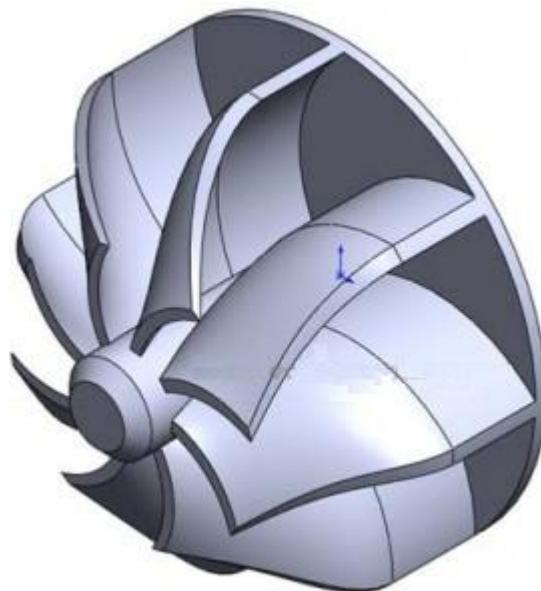
12. Now let make the full wings, click on **Mirror**.  Turn the wings to right side and select center face as a **Mirror Face/Plane**.



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

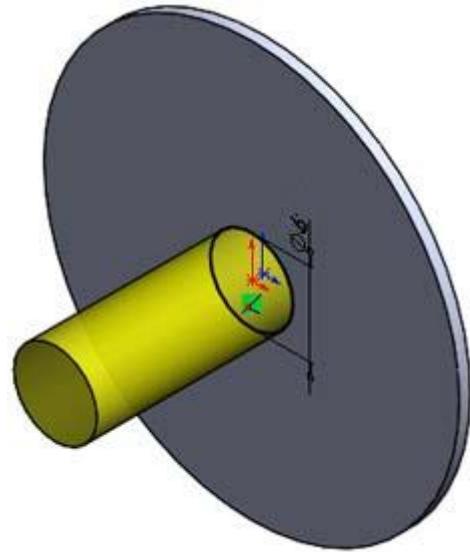


### 13.How to create turbo fins

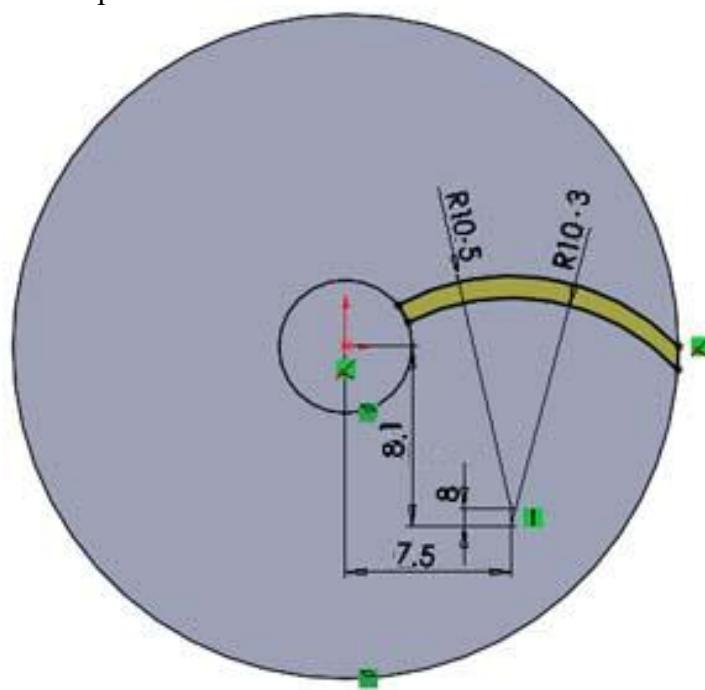


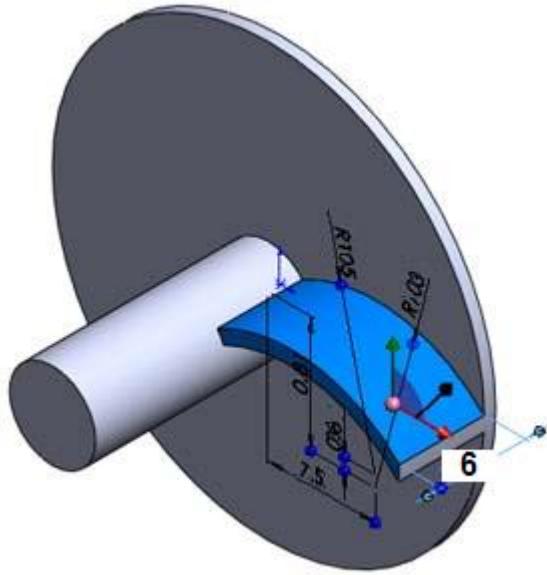
1. Sketch **30mm** circle and extrude to **0.8mm** on front plane.

2. Sketch **6mm** circle on top extruded face and extrude to **15mm**.



3. Sketch fin profile at extruded face as shown and extrude to 6mm.

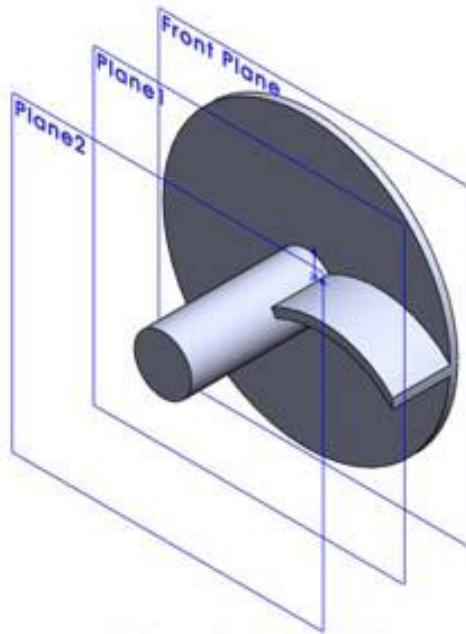




4. Add Plane 1 with 6.7

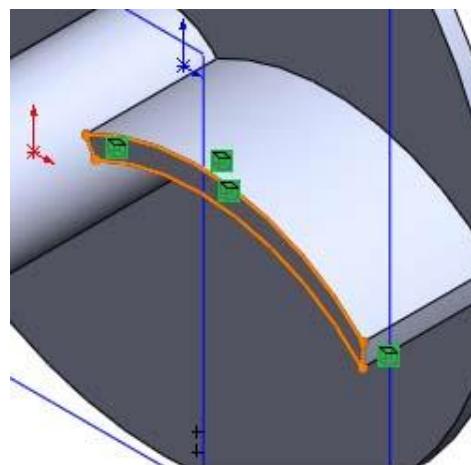
offset from Front plane and

Plane 2 with **8.5mm** from Plane 1.

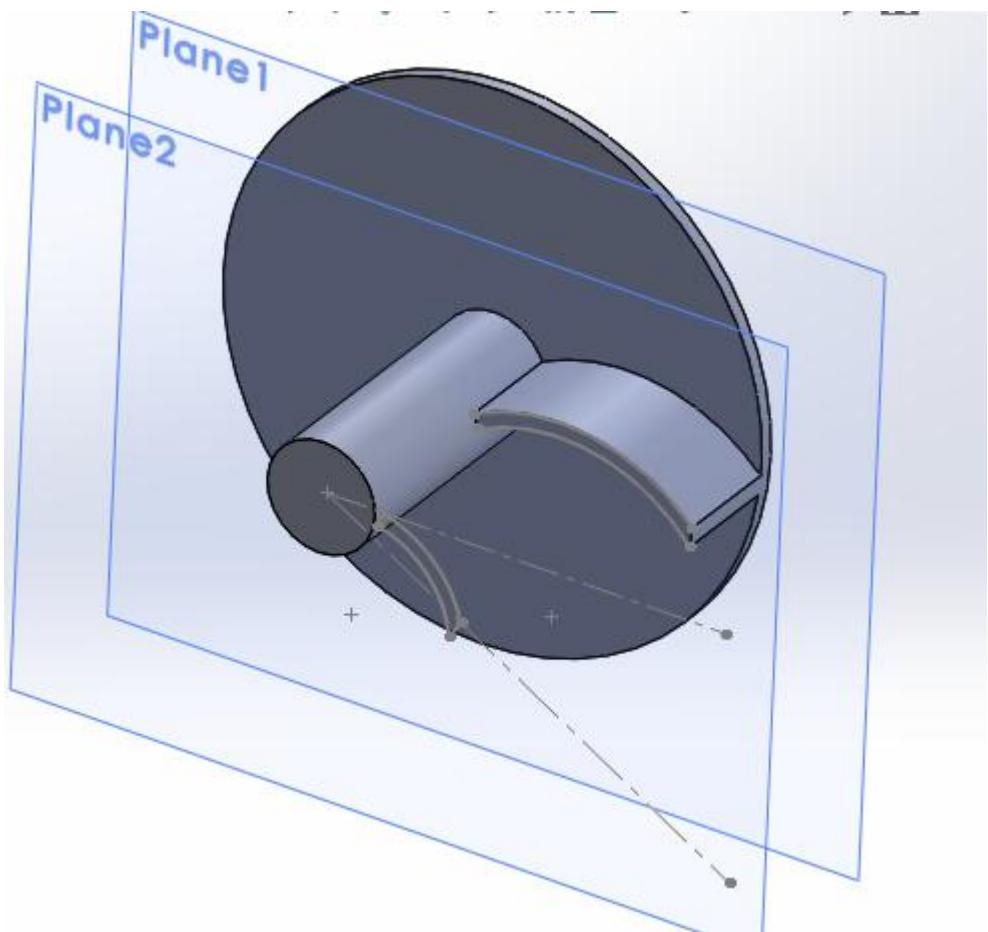
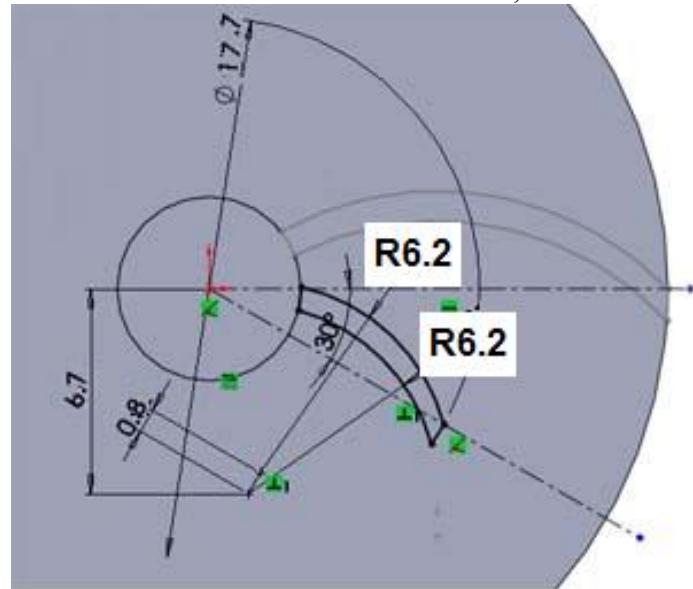


5. Insert sketch on Plane 1, select all edges to extruded fin and convert it to entities.

(Convert Entities>Entities to convert: Click the geometry>Ok>Select Plane1>OK)



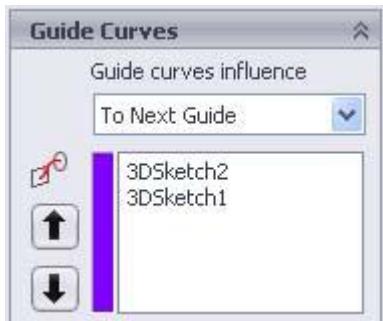
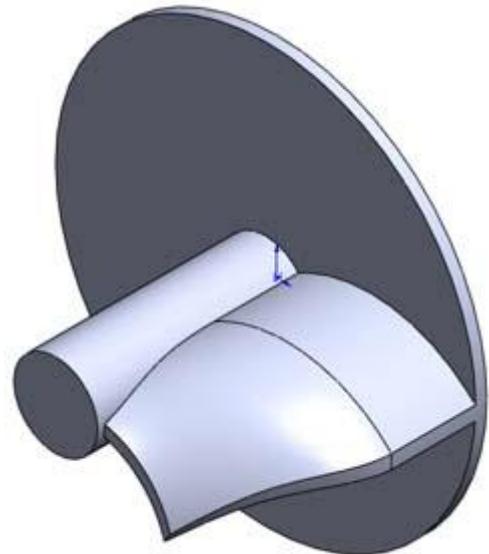
6. Insert another sketch on Plane 2, as shown.



7. . Click Lofted Boss/Base  , select profile Sketch5 and sketch6



and for guide curves select 3DSketch1 and 3DSketch2



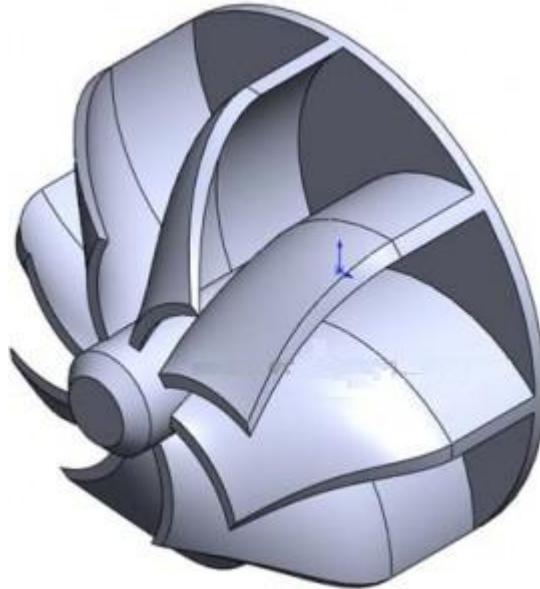
, OK.



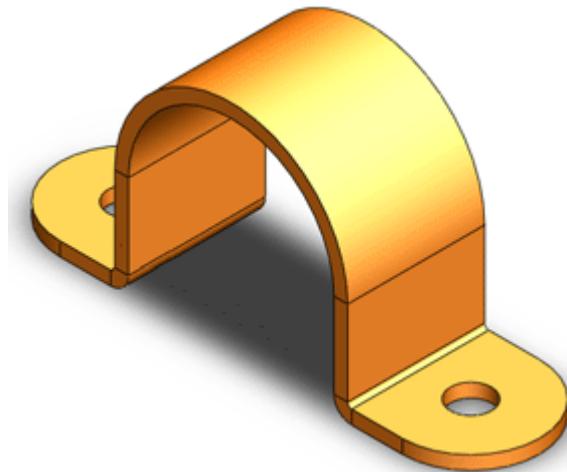
9. Click Circular Pattern Tools>Temporary Axes. Select center axis, 360 degree, #8, Equal Spacing, OK



. Done!

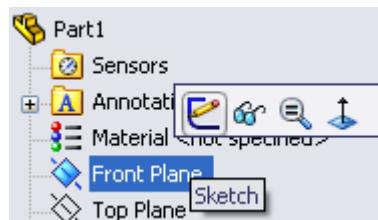


## 14.How to create U bracket sheet metal

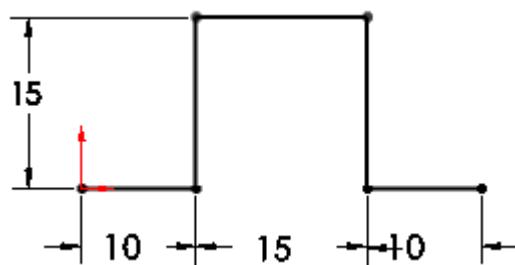


1. Click New. Click Part, OK.

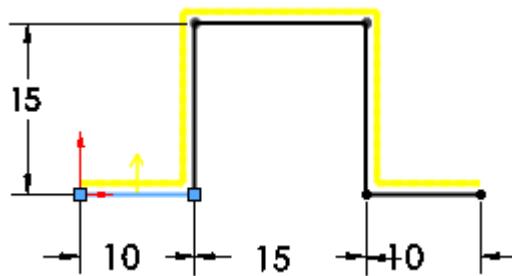
2. Click Front Plane and click on Sketch.



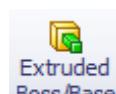
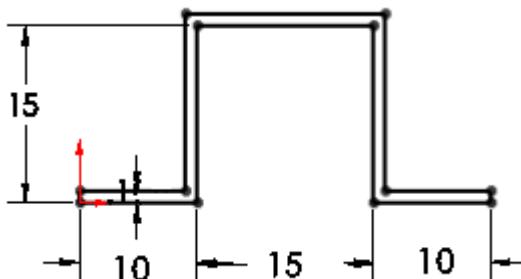
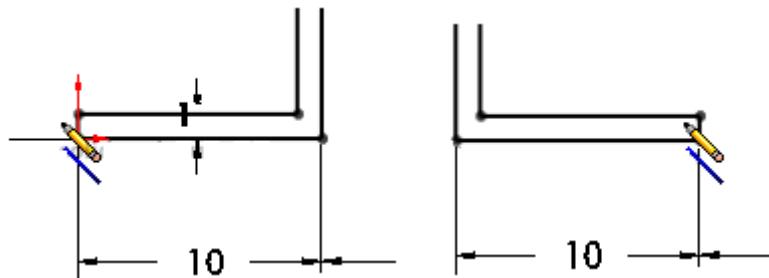
Use **Line** , sketch U shape. Dimension sketch with **Smart Dimension** **10mmx 15mm x10mm and 15mm height.**

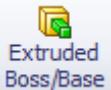


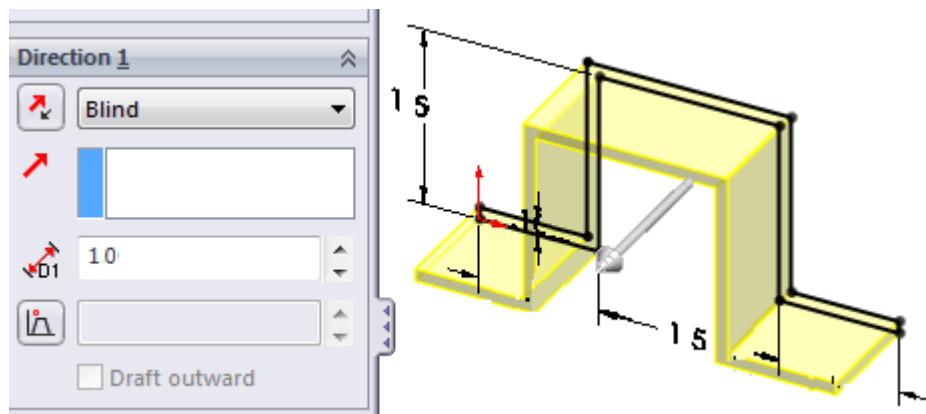
3. Click **Offset Entities** and click U sketch. Set offset distance as **10mm**, check **Reverse** box and **OK**.

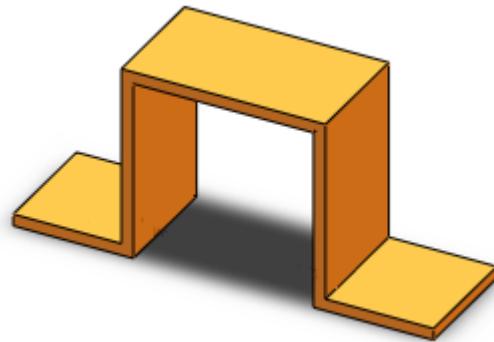


4. Use Line , sketch and connected open end of this sketch and make it close both end.

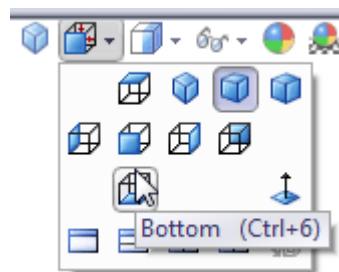


5. Click Features>Extruded Boss/Base  set D1 to 1in and OK.

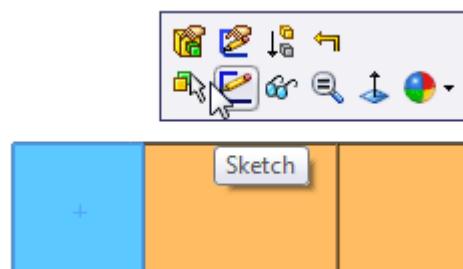




6. Click **View>Bottom**



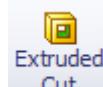
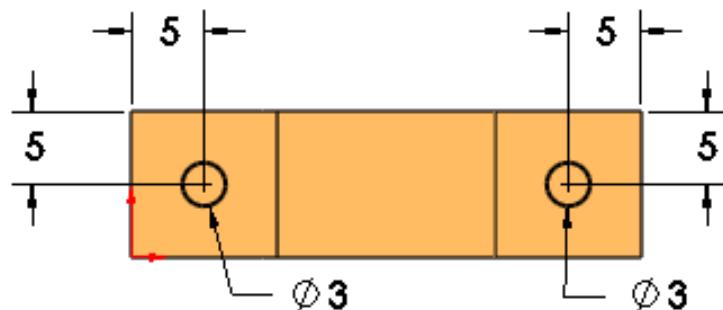
click on bottom face and click **Sketch**.



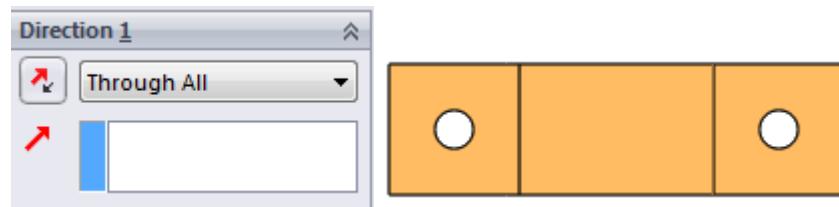
7. Click **Circle** and sketch 2 circle on bottom face each side. Use **Smart Dimension**



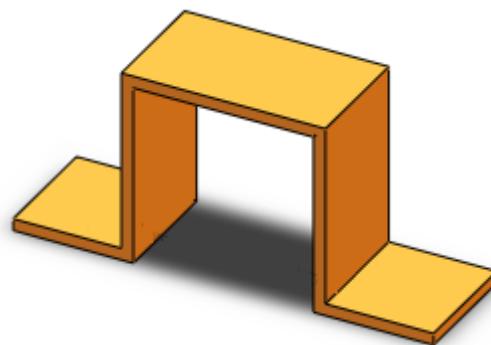
to dimension this sketch as sketched below.



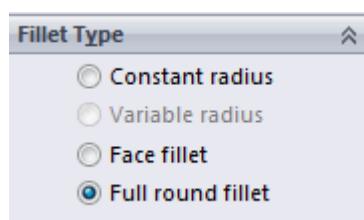
8. Click **Features>Extruded Cut** and cut **Through All** this circle.



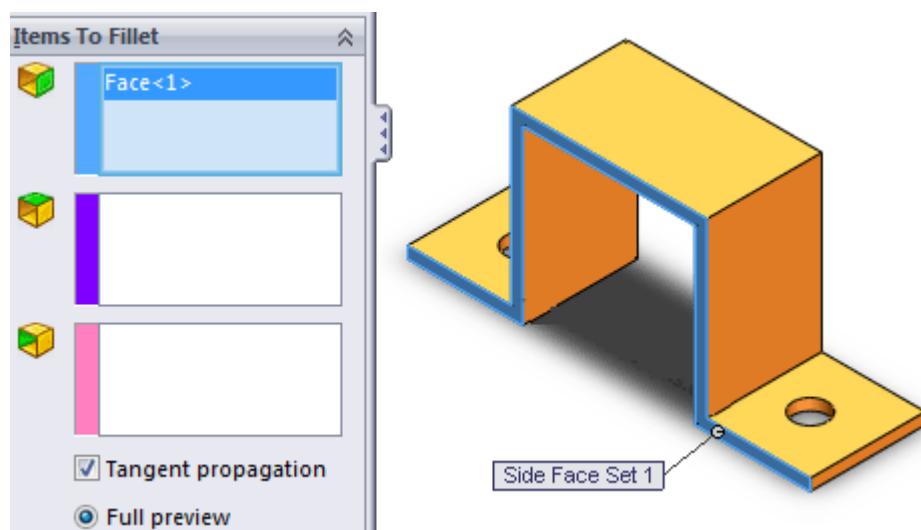
9. Click **View>Isometric**.



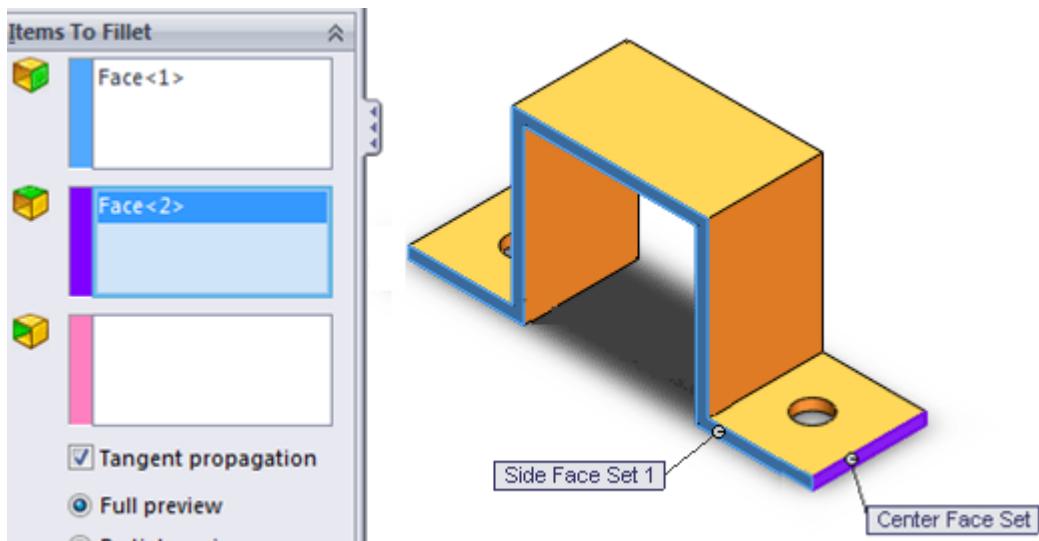
10. Click **Fillet** , check box **Full round fillet**.



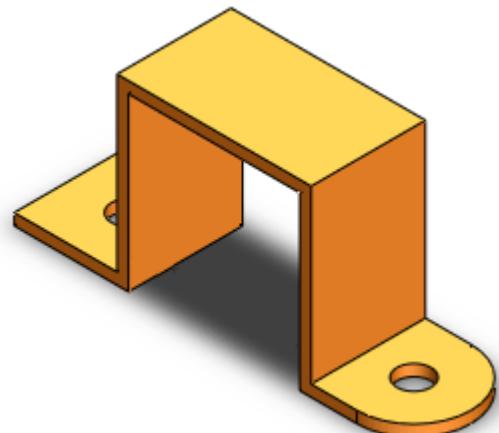
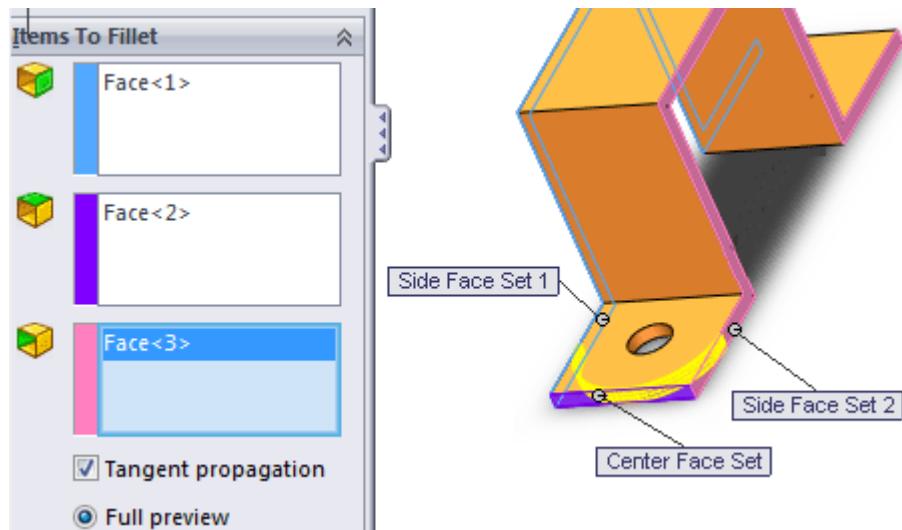
11. Click side left side face as **Side Face 1**.



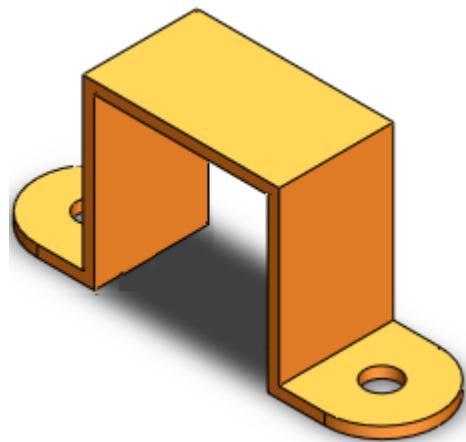
12. Click on purple box and click center face as **Center Face Set**.



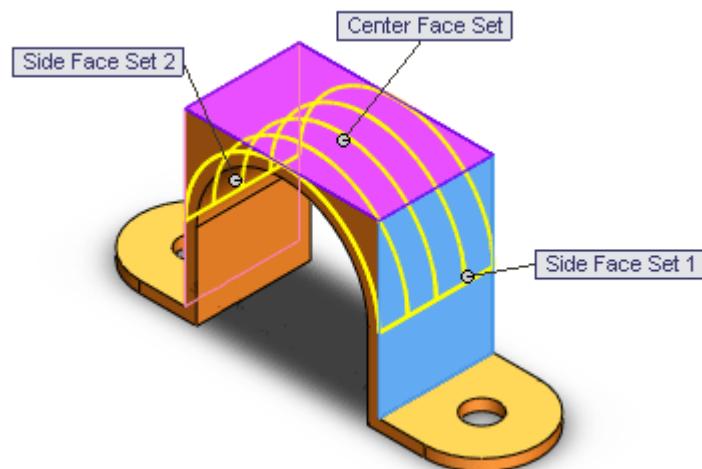
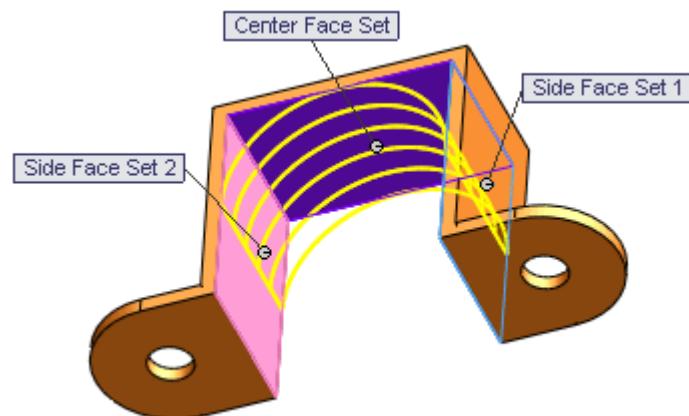
13. Click on pink box and click right side face as **Side Face Set2** and **OK**.

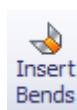
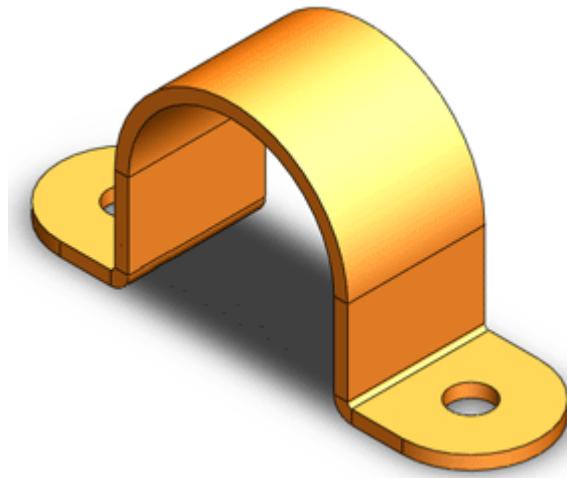


14. Repeat step 11 – 13 for the other side.

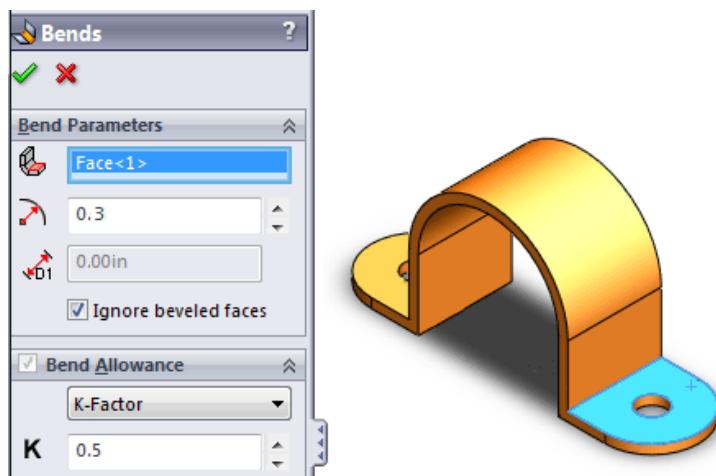


15. Repeat step 11 – 13 for inner face and outer face of U bracket.

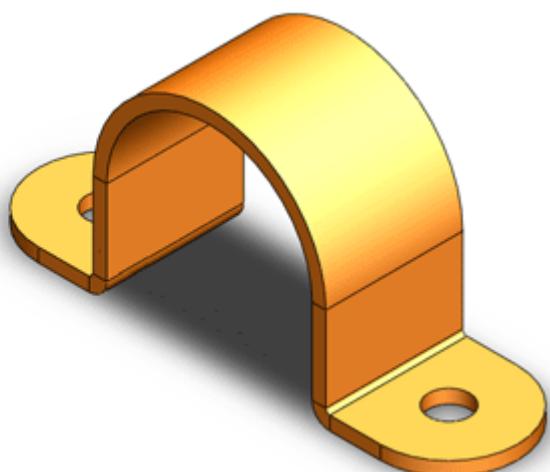
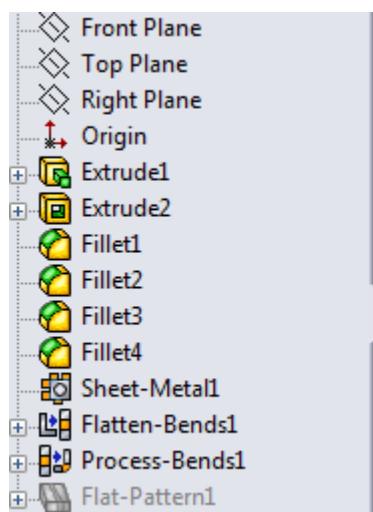




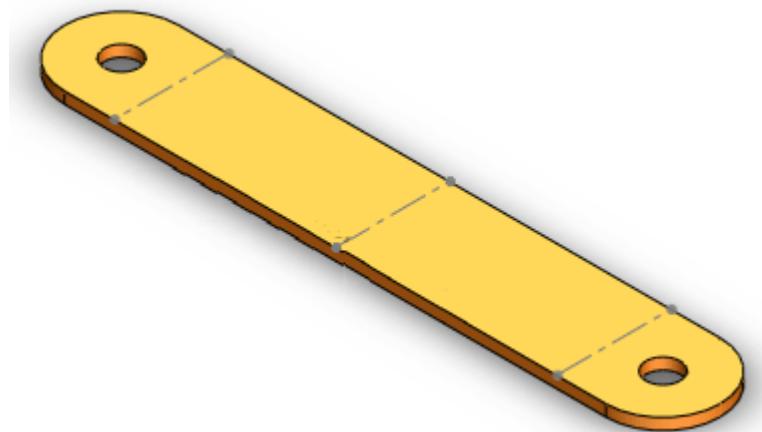
16. Click **Sheetmetal>Insert Bends**, click flat face as reference when it flatten. Set bend radius to **0.3mm** and **K factor 0.5** and **OK**.



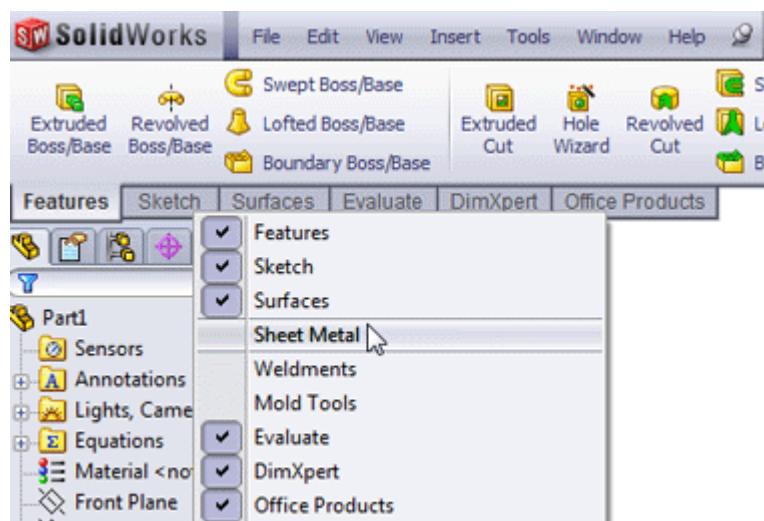
17. Your simple sheetmetal bend is ready. Look at part tree.



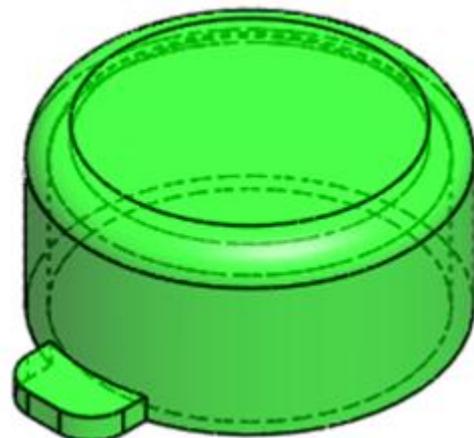
18. To view this part in flatten form click **Sheetmetal>Flatten**



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

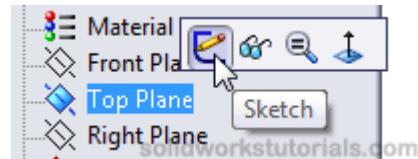


## 15. How to create bottle cap

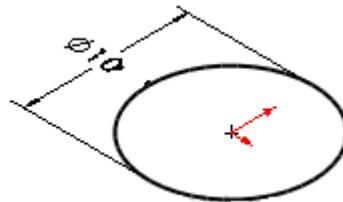


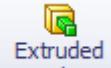
1. Click **New**  , Click **Part**  and **OK**.

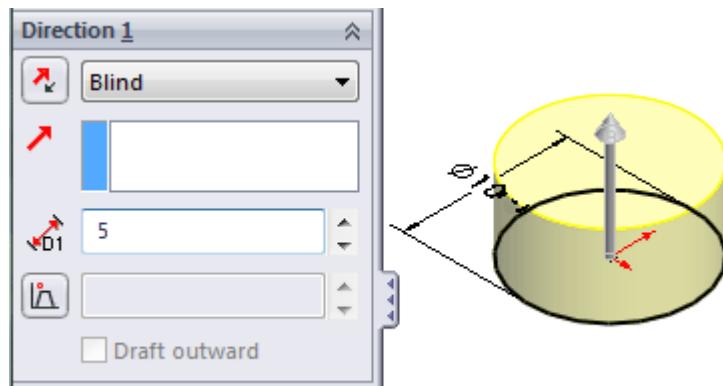
2. Click on **Top Plane** and click **Sketch**.



3. Click **Circle**  and sketch start at Origin, click **Smart Dimension**  and dimension the circle as **10mm** diameter.

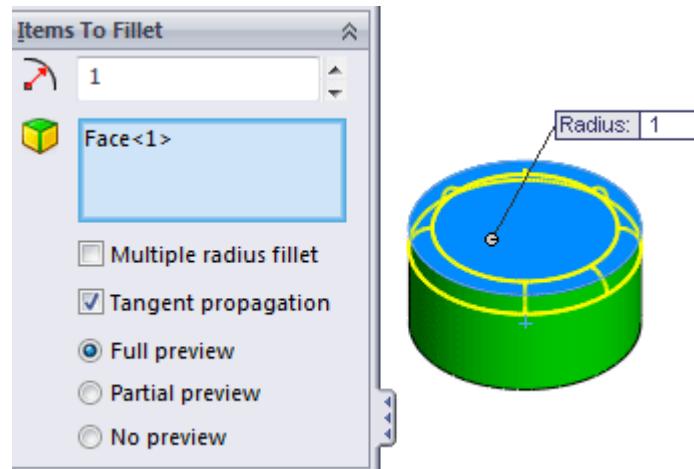


4. Click **Features>Extrude Boss/Base**  set the D1 to **5mm**.



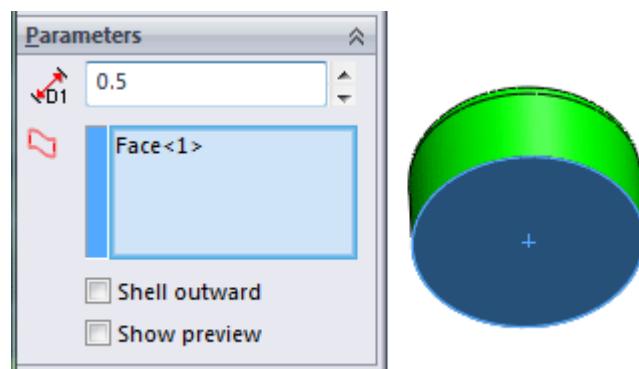
and .

5. Click **Fillet**  , set fillet size as **1mm**, select **top face** of the part

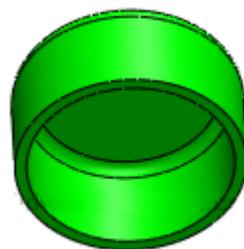


and

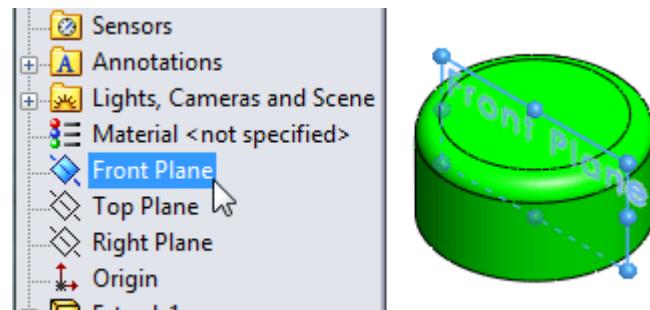
6. Turn the part to view bottom side, set D1 as **0.5mm**, click **Shell** , select **bottom face**



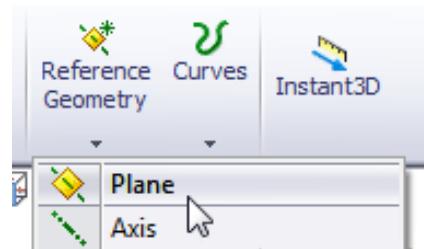
and



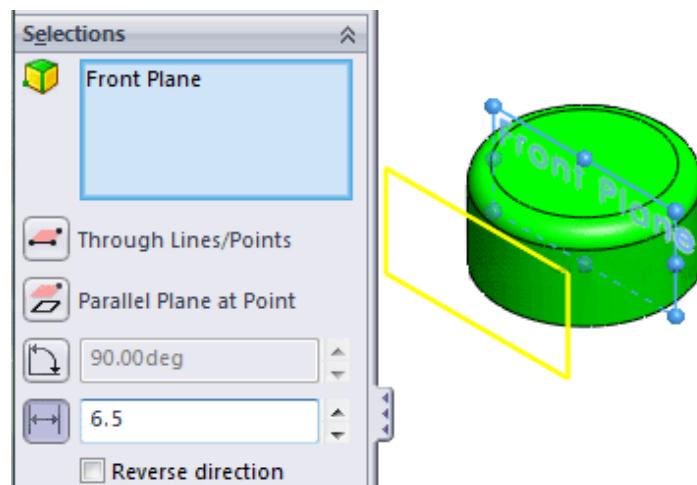
7. Click **Isometric View** , click on **Front Plane**



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



Set distance to **6.5mm**

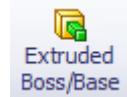
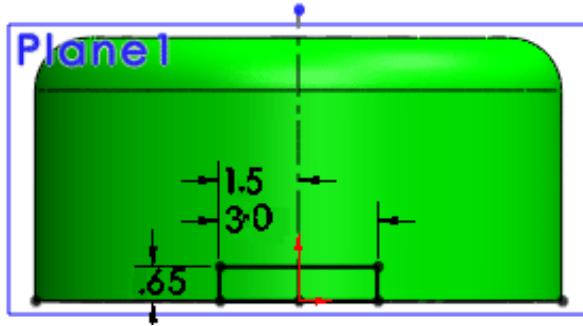


and

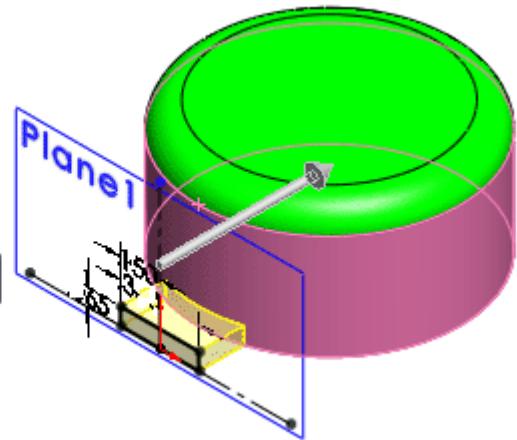
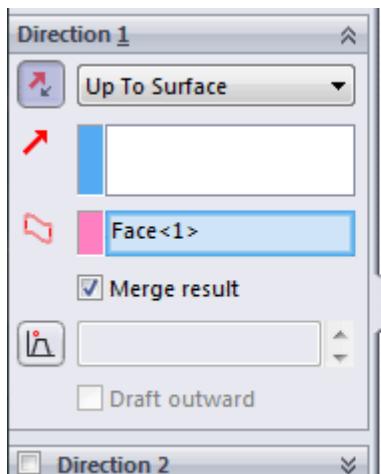
8. Click **Plane1** and click **Sketch**.



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



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



and .

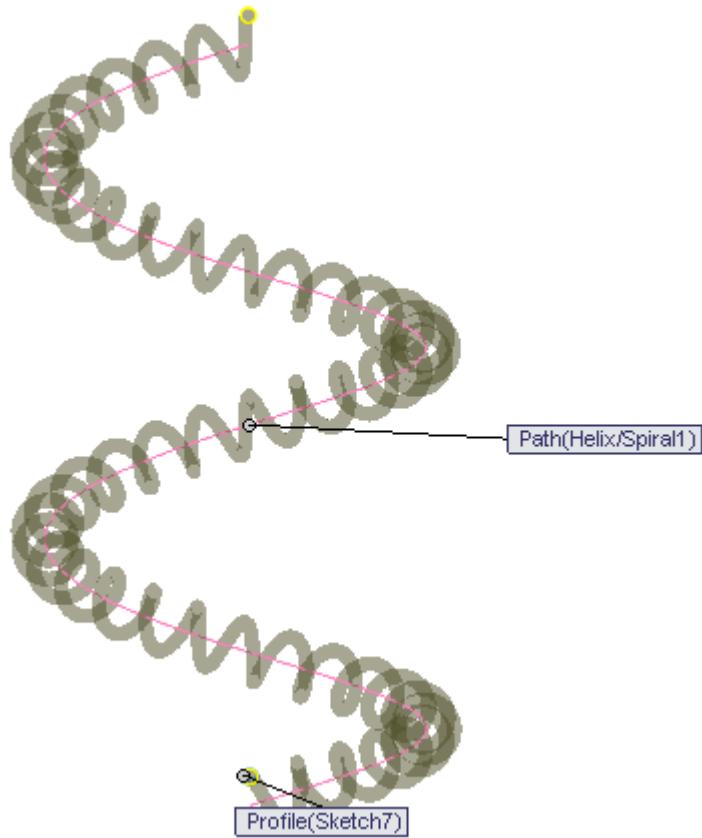


11. Click **Fillet**, set fillet size as **1mm**, select **side edge of the lid**.

and .

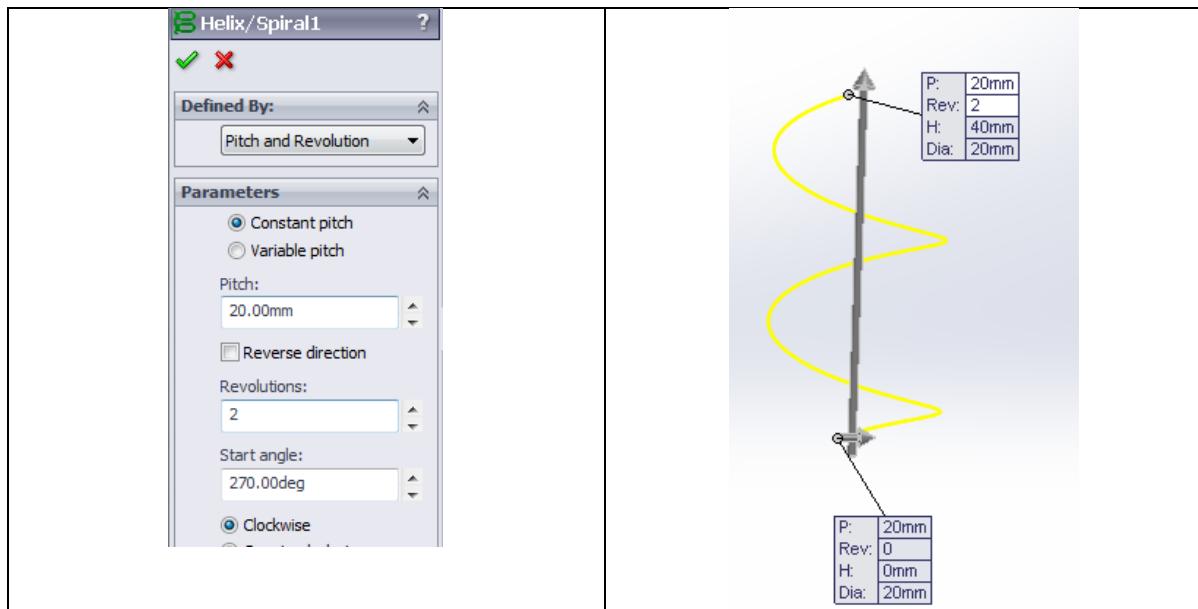
12. And you're done!

## 16. How to twist phone cord

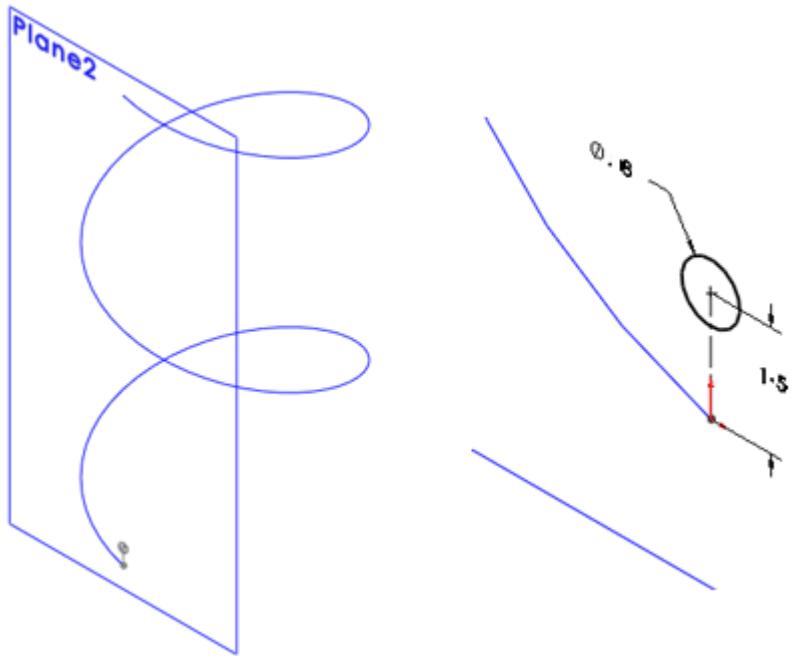


First you need to have spiral, with circle base 20mm, 2 revolutions and 20 pitch.

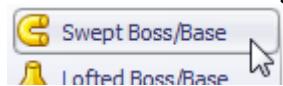
Refer to tutorial: How to create a simple spring helix/spiral from the right plane.



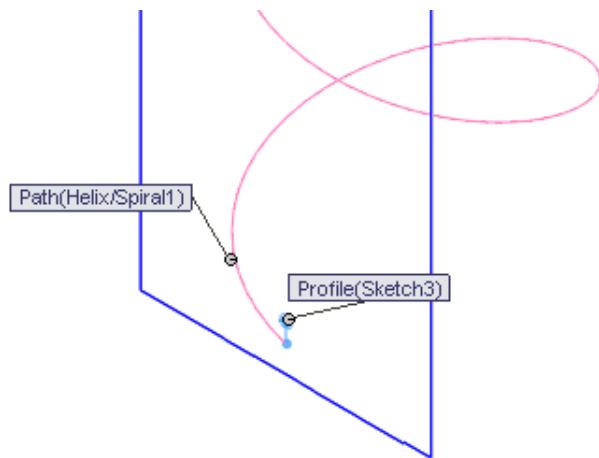
Now add a plane at end of spiral, select parallel to front plane.



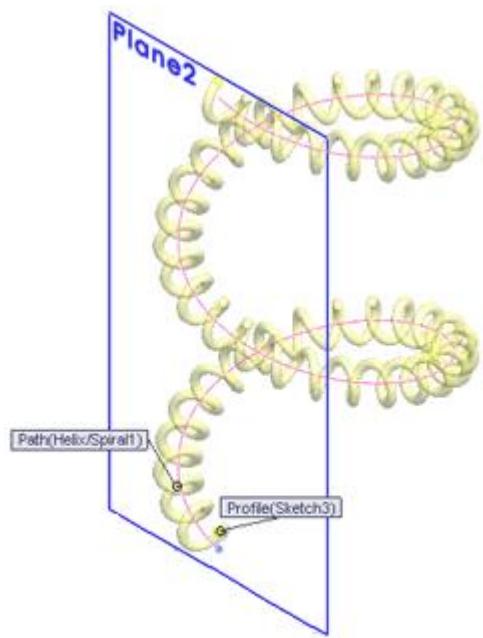
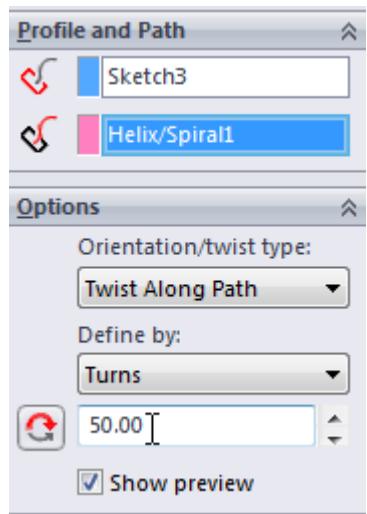
Sketch a circle on Plane2, 0.8mm and 1.5mm height. Click Swept Boss/Base.



Select Sketch3 as profile and Helix/Spiral1 as path.

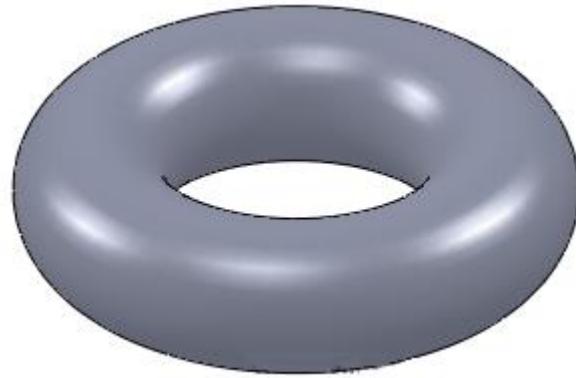


Open up Options and set Twist Along Path, define by Turns and 50 turns.



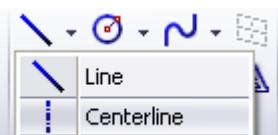
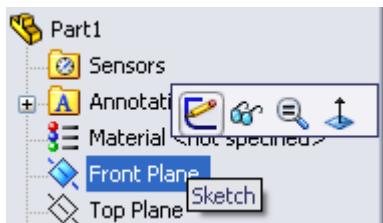
Done!

## 17. How to use Revolved Boss/Base



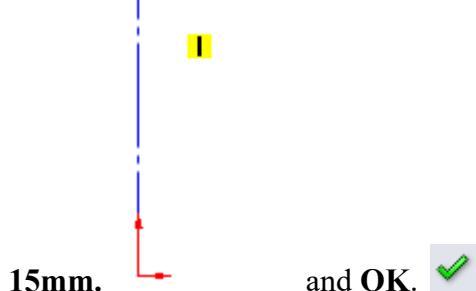
1. Click New. Click Part, OK.

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



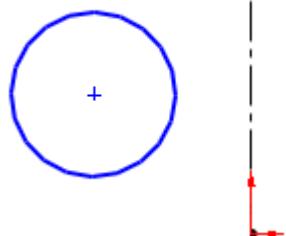
3. Select **centerline**, sketch vertical line start from origin, roughly

1.51, 90°



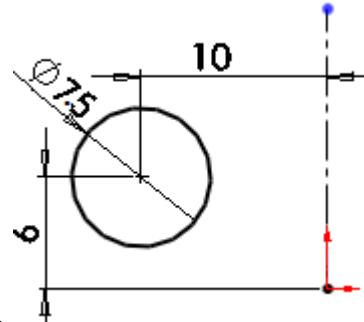
and .

4. Click circle and sketch a circle on left side of the centerline.





5. Click **Smart Dimension**, click sketched circle and set it diameter to **7.5mm**

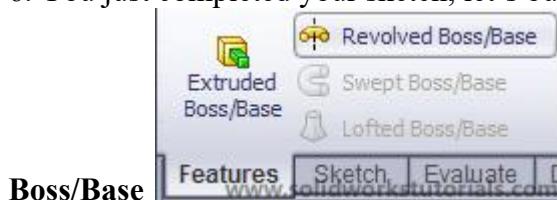


and add dimension for its location as below sketch

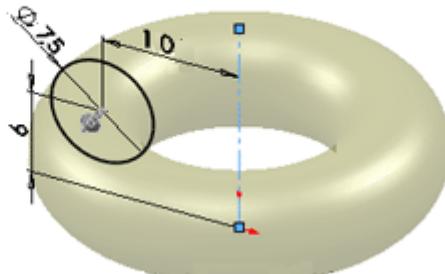


and OK.

6. You just completed your sketch, let's build feature from it. Click **Feature>Revolved**



7. Click centerline as axis

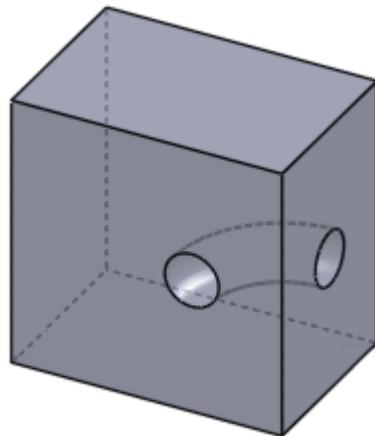


OK.



8. You're done!

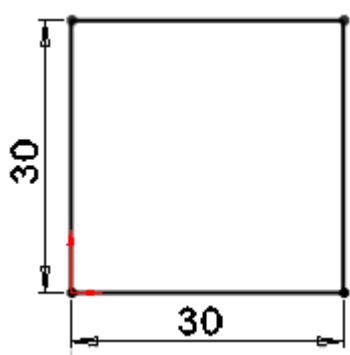
## 18.How to use Revolved Cut



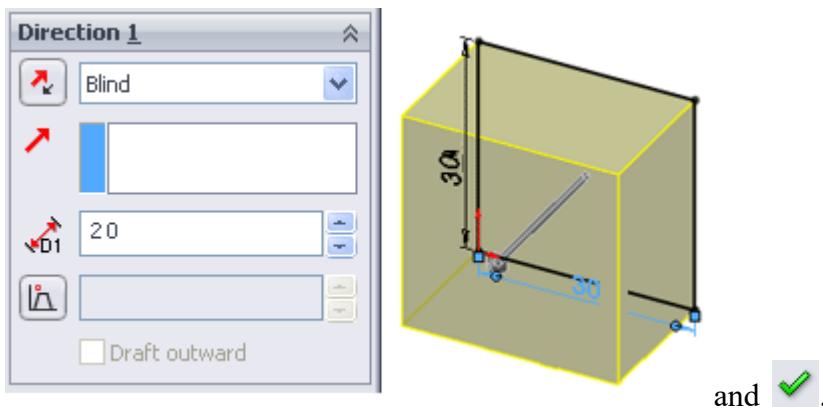
1.Click New. Click Part, OK.

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

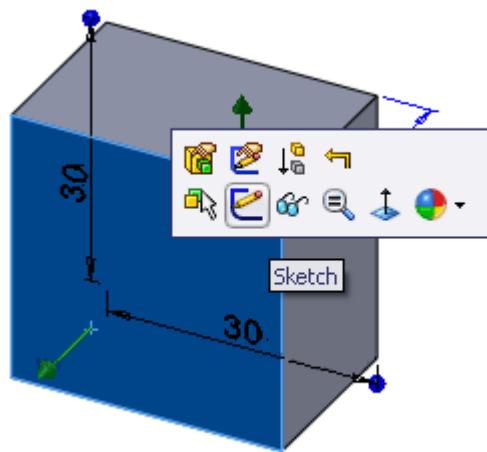
3.Click **Rectangle**, sketch rectangular. Click **Smart Dimension**, dimension rectangular **30mm x 30mm**.



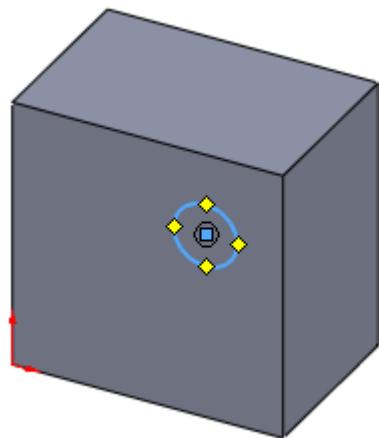
4.Click Feature>**Extruded Boss/Base**, set D1 to **20mm**.



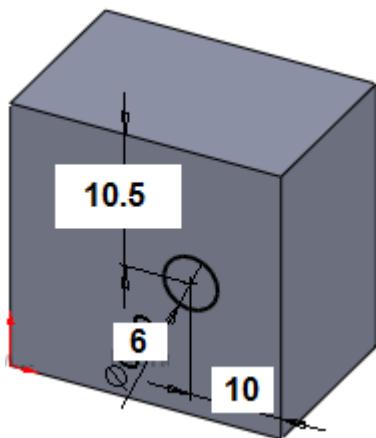
5. Click on front face and click **Sketch**.



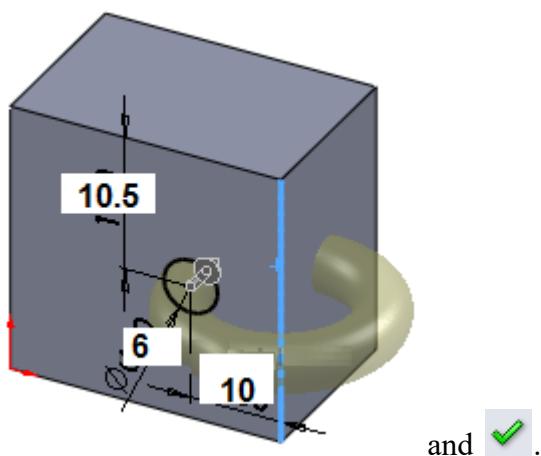
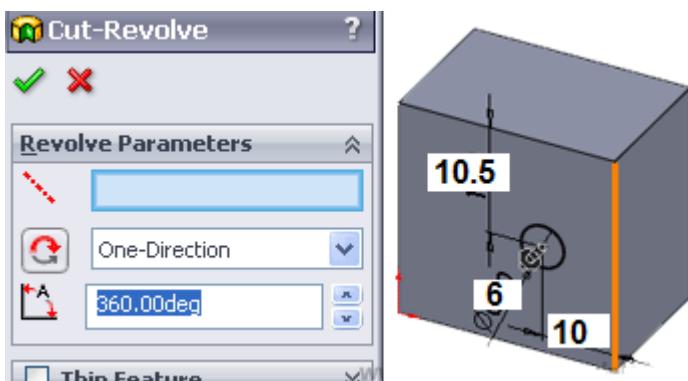
6. Click **Circle**, and sketch a circle on front face.



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

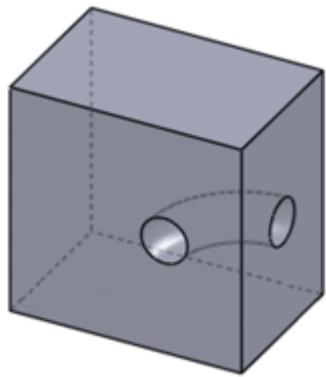


8. Click **Features>Revolved Cut** click on **right side edge** as axis of revolution,

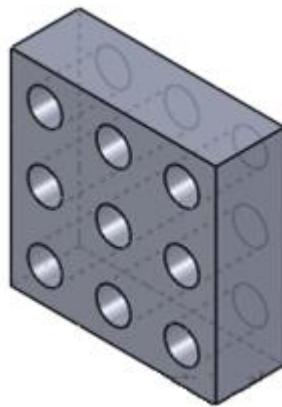


and .

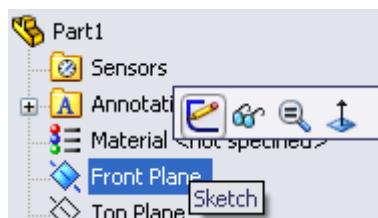
9. You're done!



## 19. How to use Linear Pattern

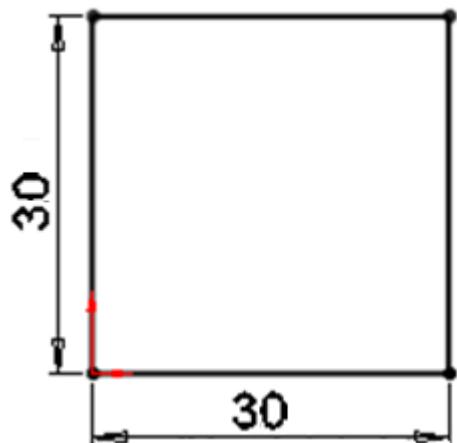


1.Click New. Click Part, OK.

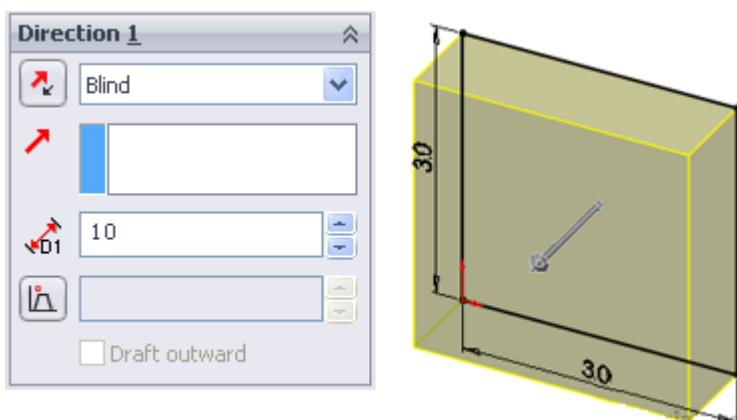


2.Click Front Plane and click on Sketch.

3.Click Rectangle, sketch rectangular. Click Smart Dimension, dimension rectangular 30mm x 30mm

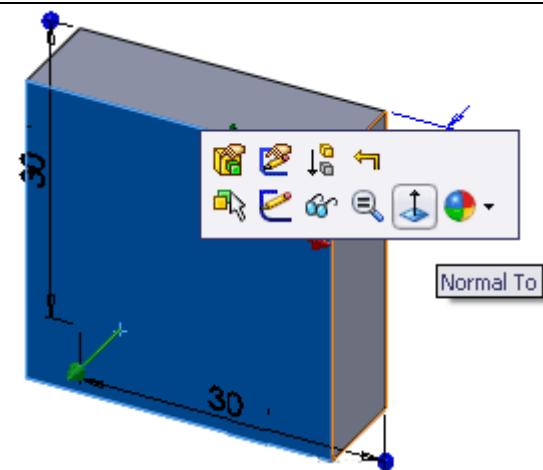


4. Click Feature>Extruded Boss/Base,

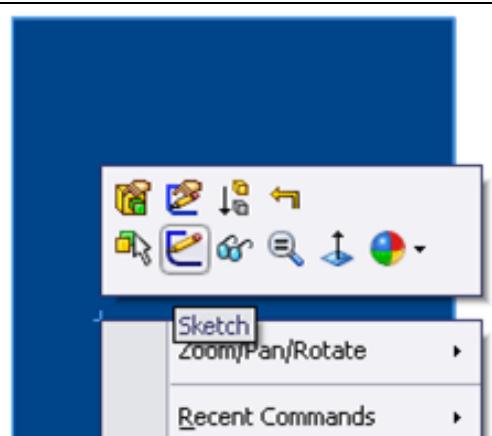


set D1 to 10mm and OK. ✓

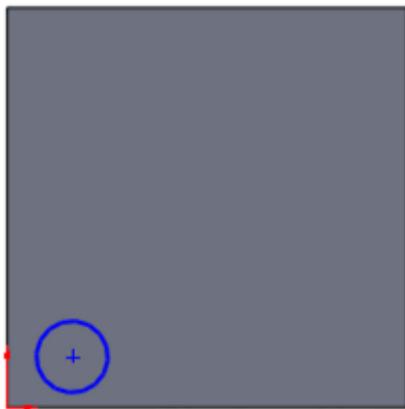
5. Click on front face and select Normal to.



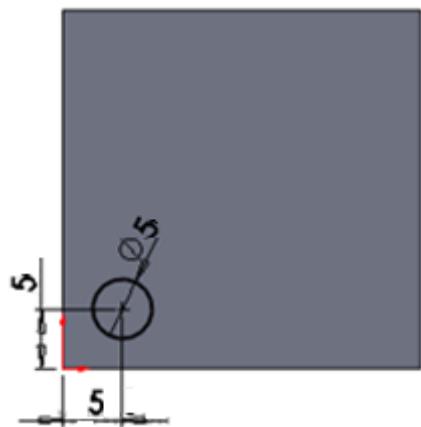
6. Click front face and Insert Sketch.



7.Click Circle,  sketch circle at one edge.



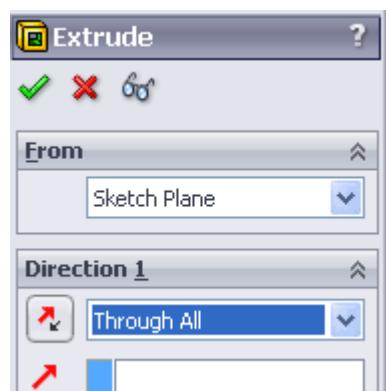
8.Click Smart Dimension,  dimension circle as below sketch.



9.Click Features>Extruded Cut,



set Direction 1, Through All and OK. 



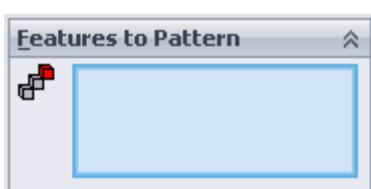
10.Click Linear Pattern,	click left edge, on Direction 1 set spacing D1 to 10mm and Instances # to 3.

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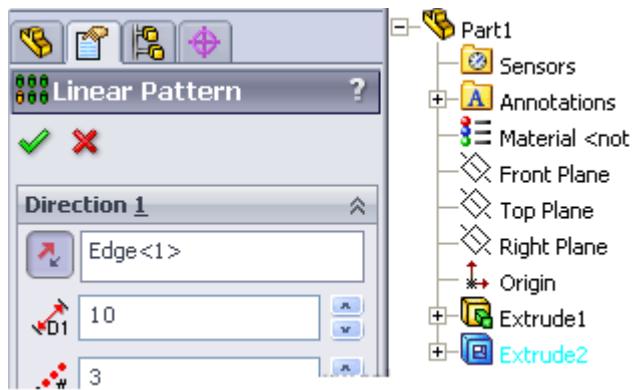
11.Click bottom edge,	on Direction 2 set spacing D2 to 10mm and Instances # to 3.
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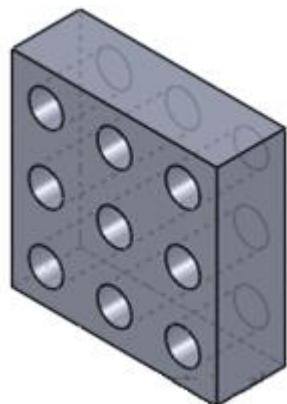
12.Click inside white box Features to Pattern.



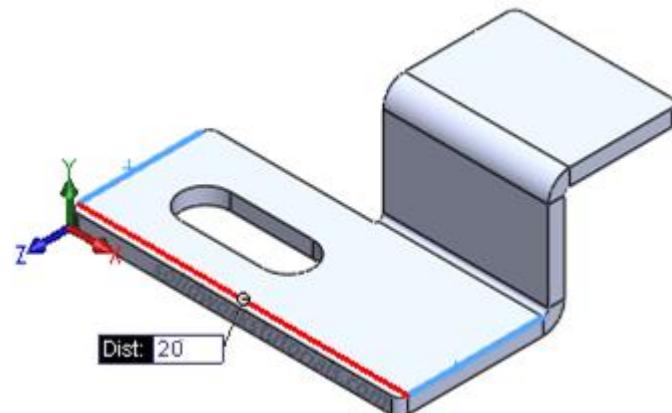
Open up part tree, select Extrude 2



and OK. . You're done!

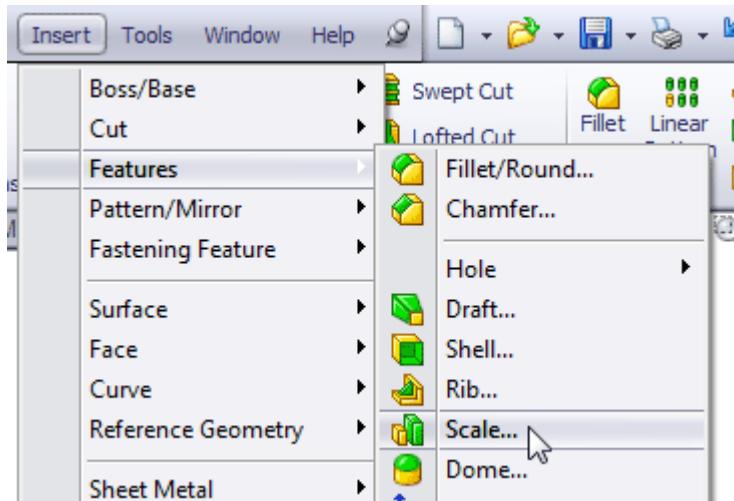


## 20. How to use Scale

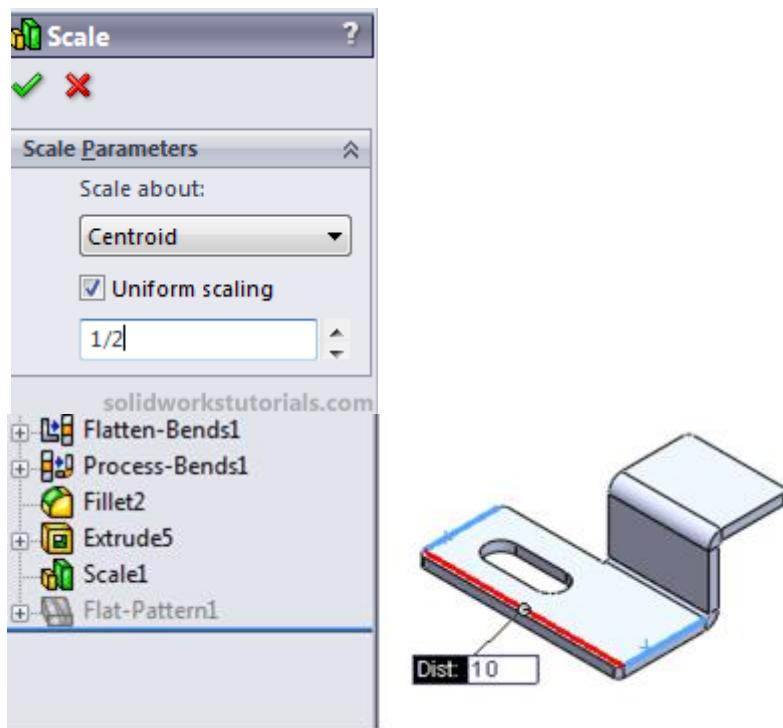


Scale down to 1/2 ...

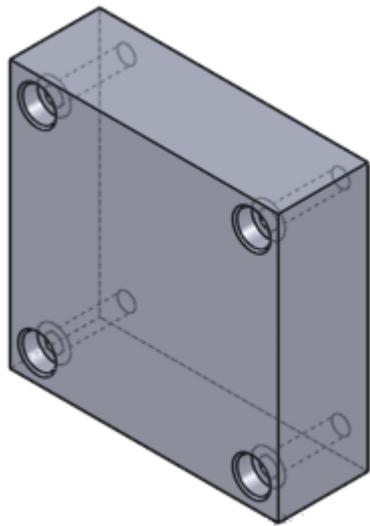
First click Insert>Features>Scale...



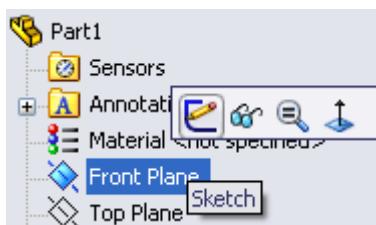
Scale down your part 1/2 of it original size and OK.



## 21. How to use Hole Wizard



1.Click New. Click Part, OK.



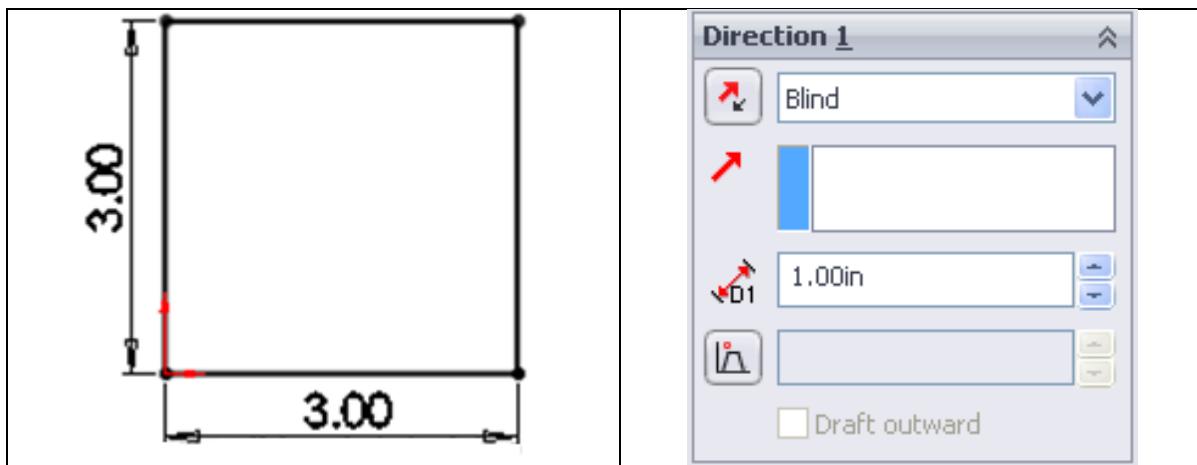
2.Click Front Plane and click on Sketch.

3.Click Rectangle, sketch rectangular.  
Click Smart Dimension, dimension rectangular 3.0in x 3.0in.

4.Click Feature>Extruded Boss/Base,



set D1 to 1.0in and OK.

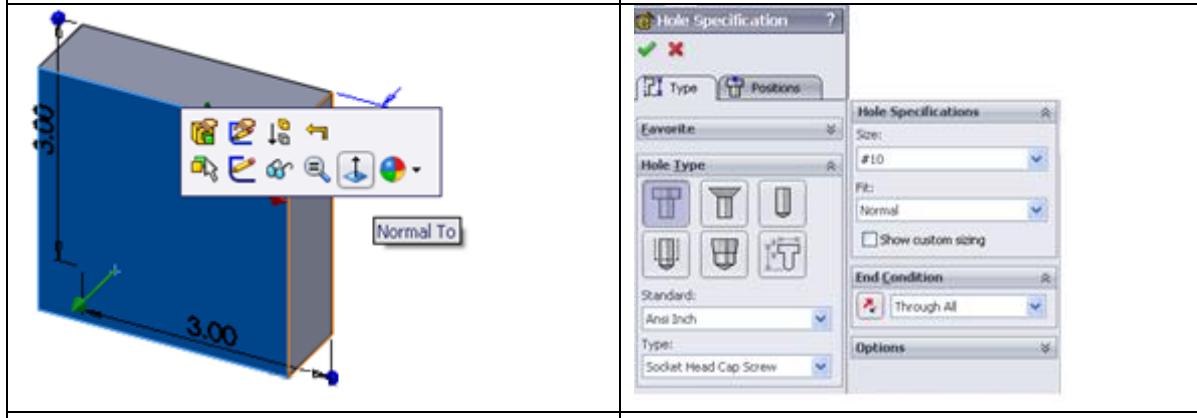


5. Click on front face and select **Normal to**.

6. Click **Hole Wizard**,

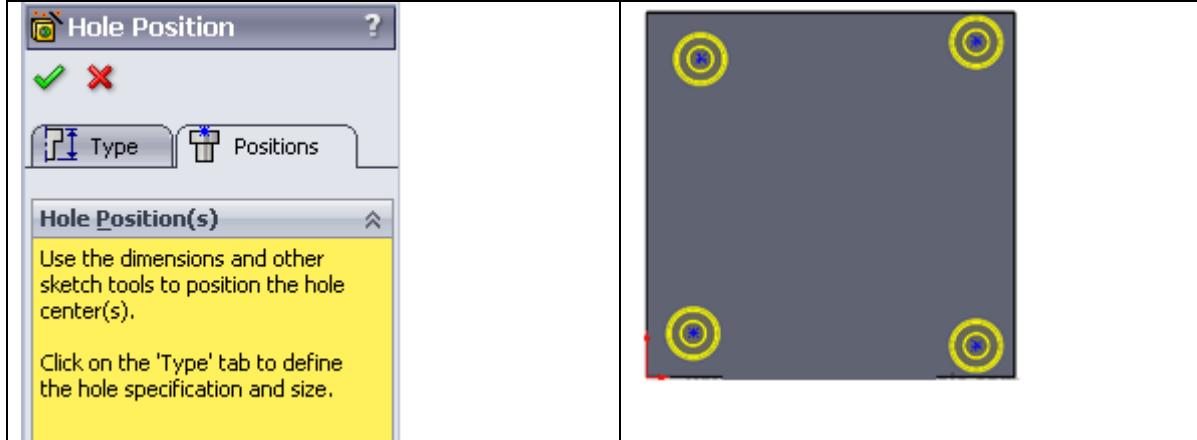


for Hole Type  
select **Counterbore, Standard ANSI**  
**Inch**, Type **Socket Head Cap Screw** and  
**Size #10**.



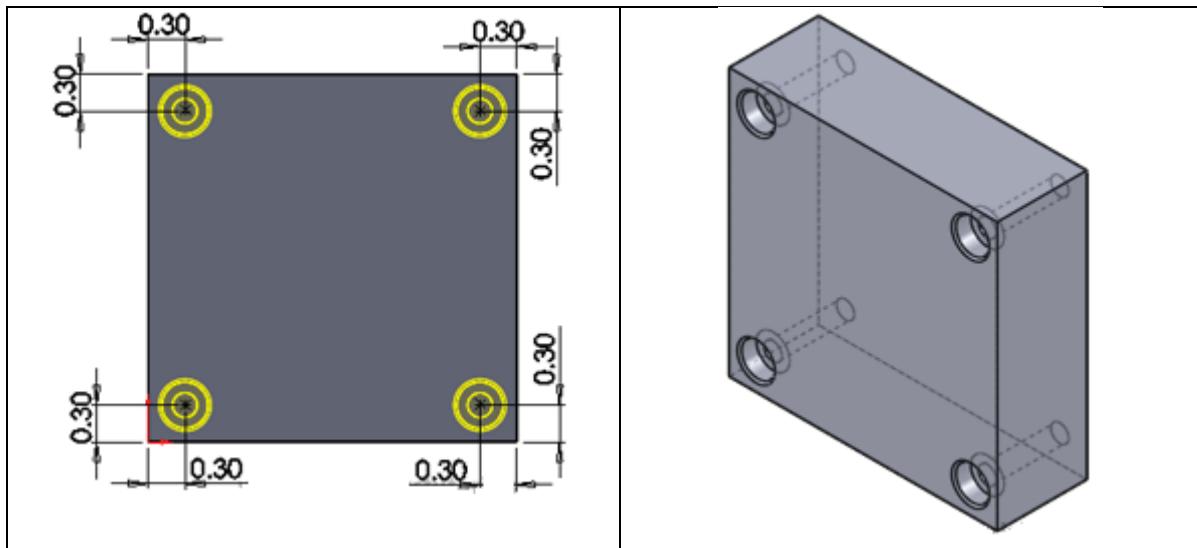
Click **Positions** tab,

click 4 points at edges,



click **Smart Dimension** and  
dimension all 4 points **0.3in** from edge.

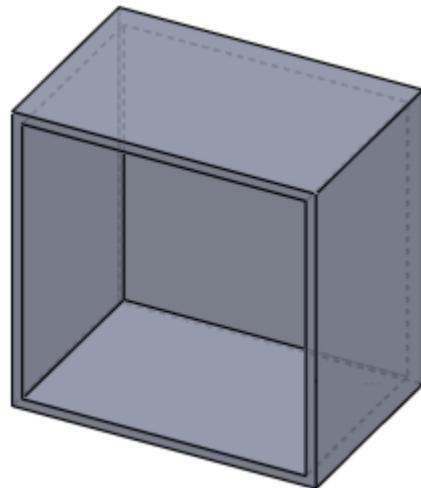
Click **OK**.



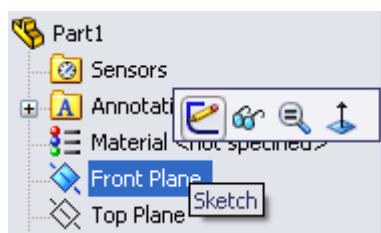
7.Done!

## 22. How to use Shell

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



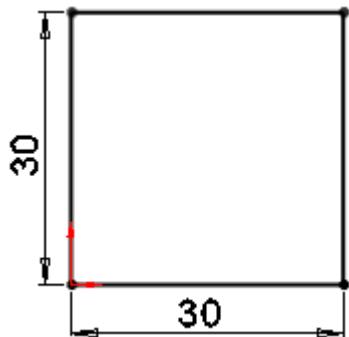
1.Click New.  Click Part,  OK.



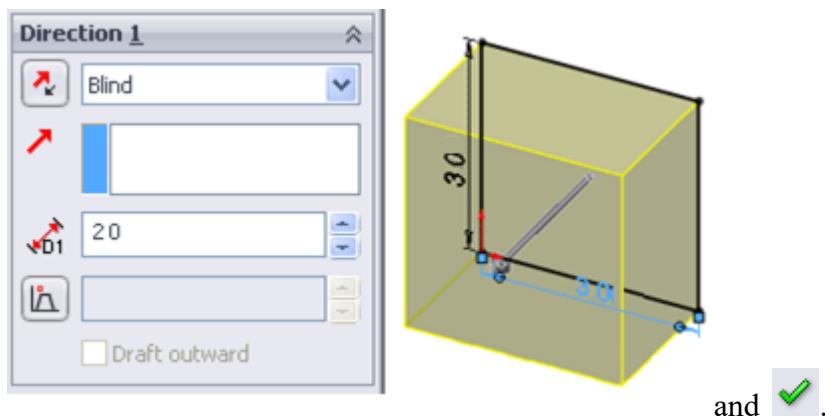
2.Click **Front Plane** and click on **Sketch**.



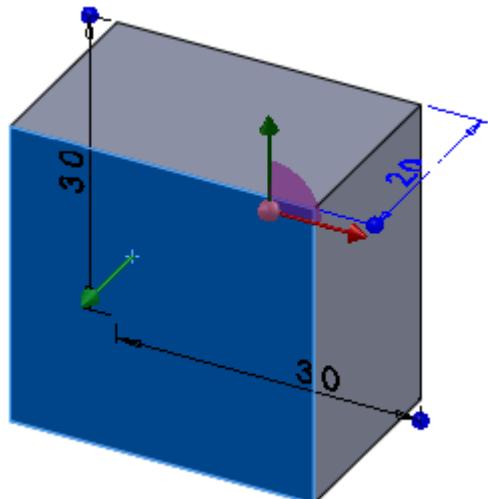
3.Click Rectangle, sketch rectangular. Click Smart Dimension, dimension rectangular 30mm x 30mm.



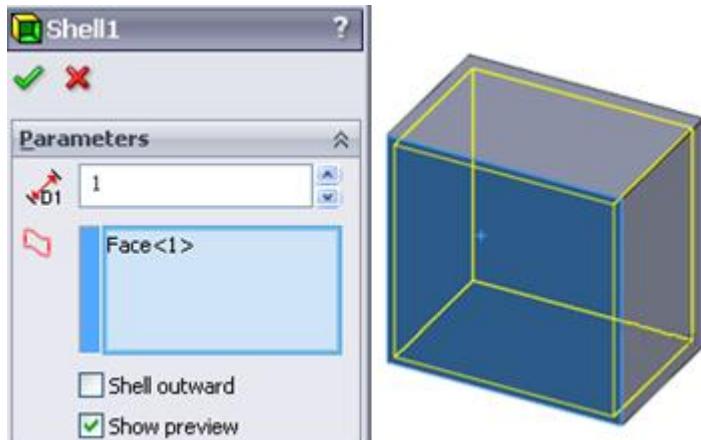
4.Click Feature>Extruded Boss/Base, set D1 to 20mm



5.Click on front face,

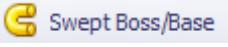


and click **Shell**  on shell Parameter, set D1 to 1mm, check show preview and 

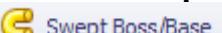
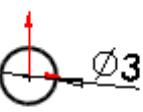
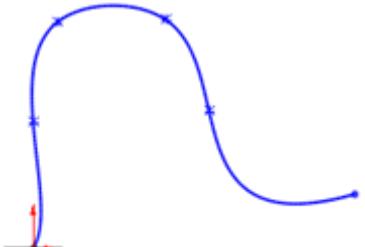
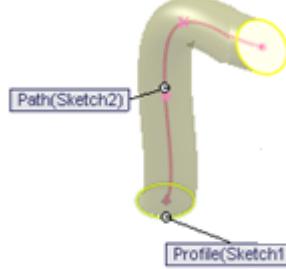


6. Done!

## 23. How to use Swept Boss/Base

In this tutorials you will create swept part using Swept Boss/Base tool. 

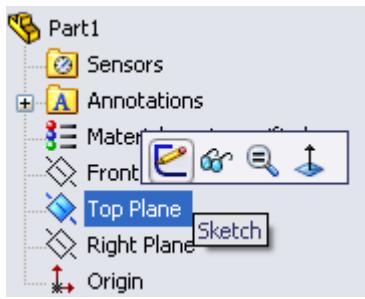
**Overview:**

Top Plane	Front Plan , Spline 	Features>Swept Boss/Base 
		

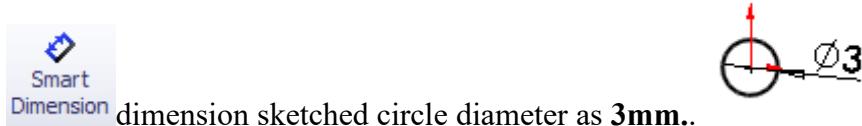


1.Click New.  Click Part,  OK.

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



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



dimension sketched circle diameter as **3mm..**

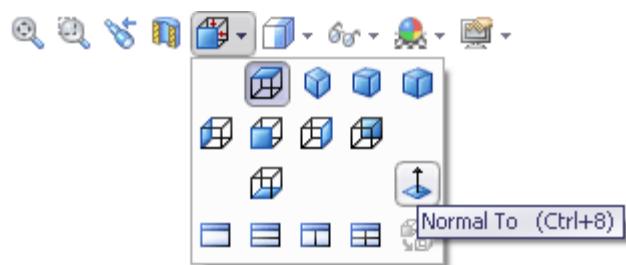


4. Exit Sketch.

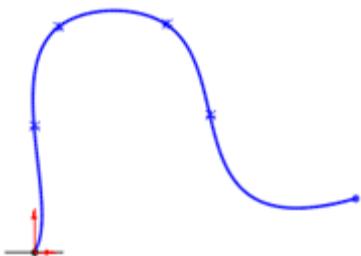
5. Click on **Front Plane** and click **Sketch**.



6. Click on **View Orientation>Normal To**.

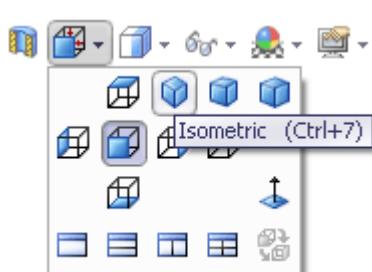


7. Zoom out the sketch, click on **Spline** and sketch a curve as sketched below.



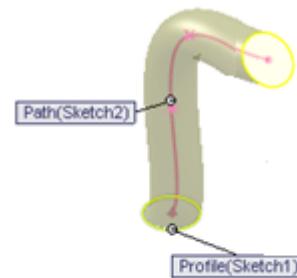
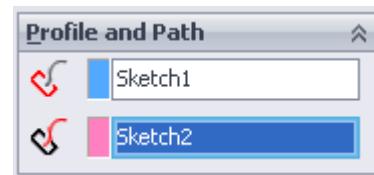
8. Exit Sketch.

9.Click View Orientation>Isometric.



10. Click Features>Swept Boss/Base.

For swept profile select Sketch1 (circle) and for path click on Sketch2 (curve). and .



## 24. How to use Lofted Boss/Base

In this tutorial you will create this part using loft feature

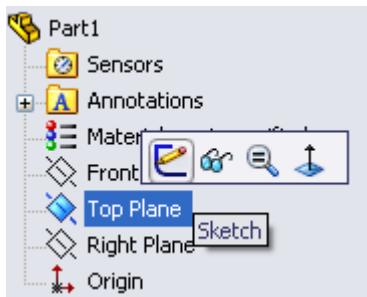
**Overview:**

Top Plane	Create two plane 1 & plane 2	Lofted Boss/Base
A 2D sketch of a circle with a radius of 10 units, centered on the top plane. The sketch is defined by a red dimension line and a red arrow indicating the radius.	A 3D diagram showing two parallel planes, 'Plane1' and 'Plane2'. A circle is drawn on each plane. The distance between the planes is indicated by a vertical dimension line.	A 3D view of the Lofted Boss/Base feature. A green shaded surface represents the profile, and a blue shaded surface represents the path. The feature connects the two parallel planes.

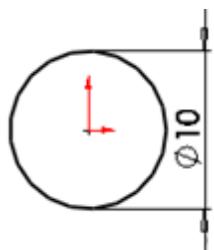


1. Click New. Click Part, OK.

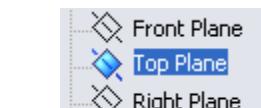
2. Click **Top Plane** and click on **Sketch**.



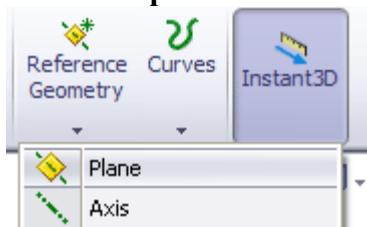
3. Click **Circle**, sketch a circle start at origin. Click **Smart Dimension**, click on circle and set dimension to **10mm**.

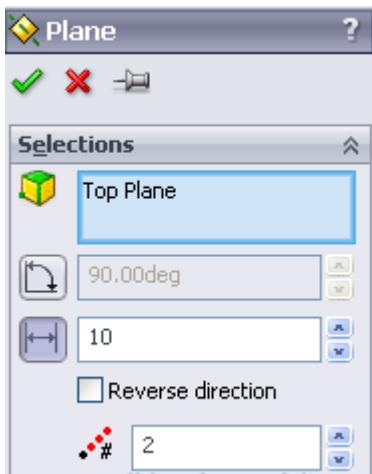


4. Exit sketch.

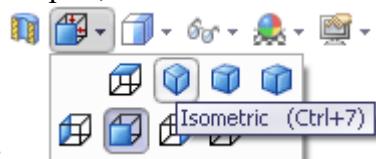


5. Click **Top Plane** and click **Features>Reference Geometry>Plane**.

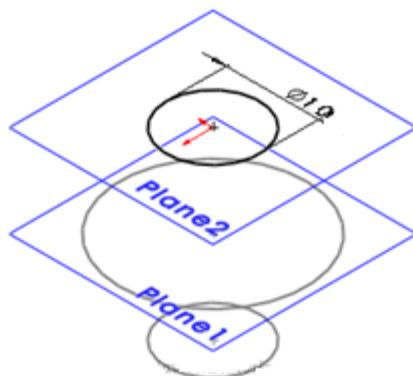




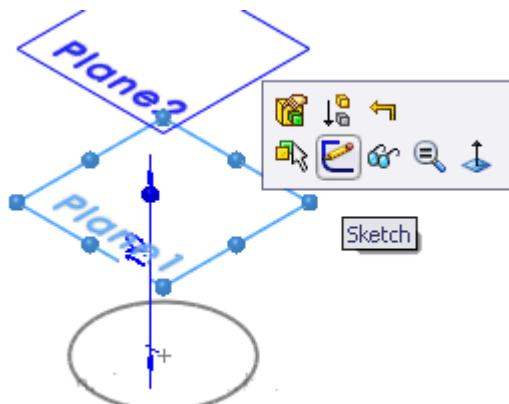
Set distance to 10mm apart, set # to 2 and OK. Two more



plane added, click View **Orientation>Isometric**



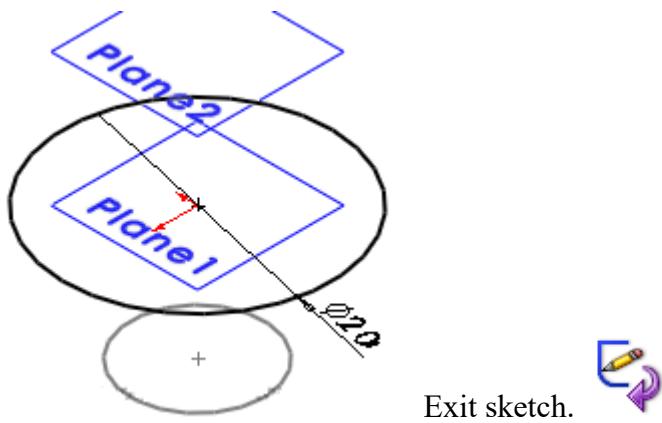
Plane 1 and Plane2.



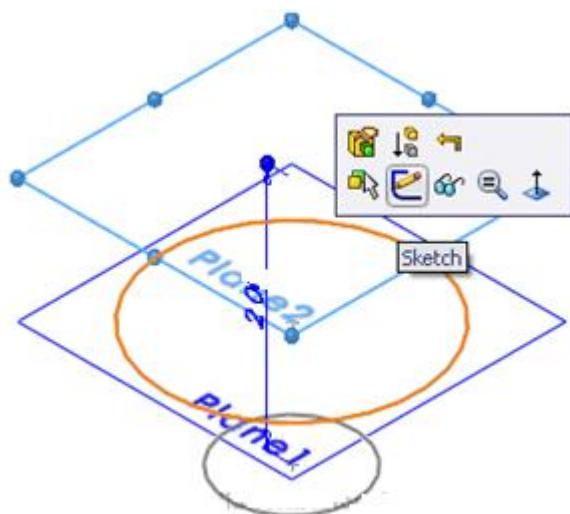
6. Click on **Plane 1**, click **Sketch**.

7. Click **Circle**, sketch a circle start at origin. Click **Smart Dimension**, click on circle and set dimension to **20mm**.

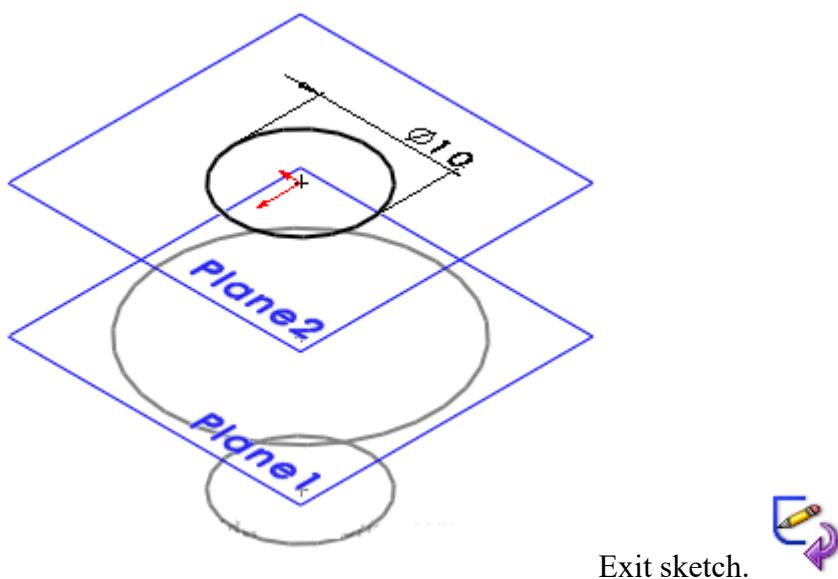


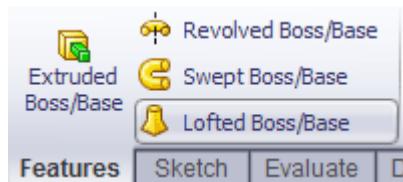


8. Click on **Plane 2**, click **Sketch**.

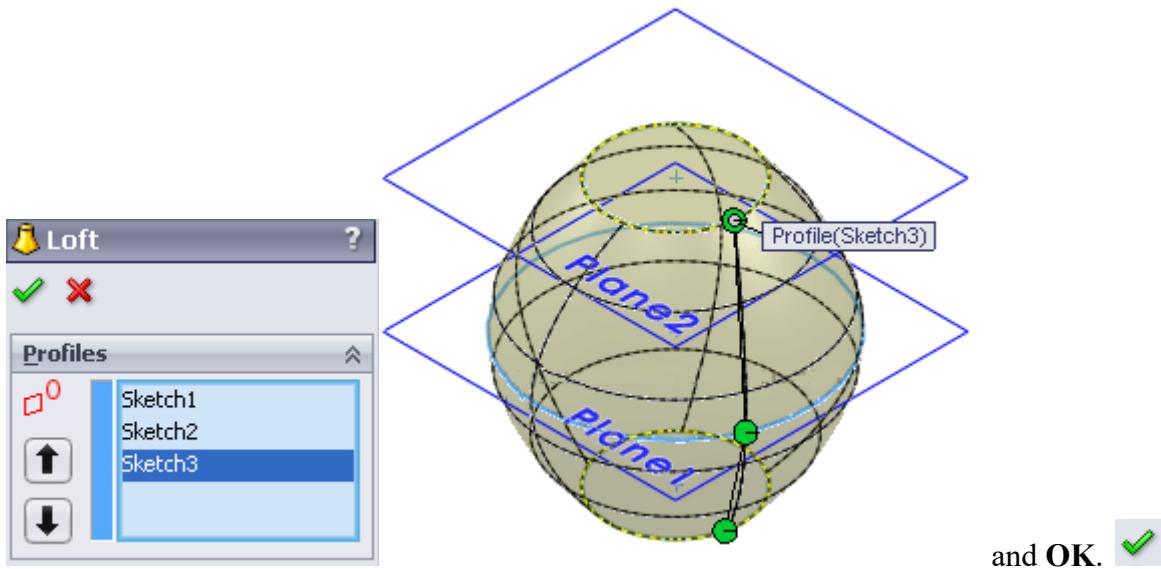


9. Click **Circle**, sketch a circle start at origin. Click **Smart Dimension**, click on circle and set dimension to **10mm**.





10. Click Features>Lofted Boss/Base click bottom circle, middle circle, top circle



11. Hide Plane 1, hide Plane 2.

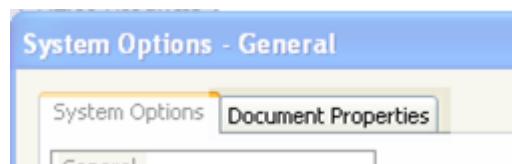


12. Done.

## 25. How to change to metric units

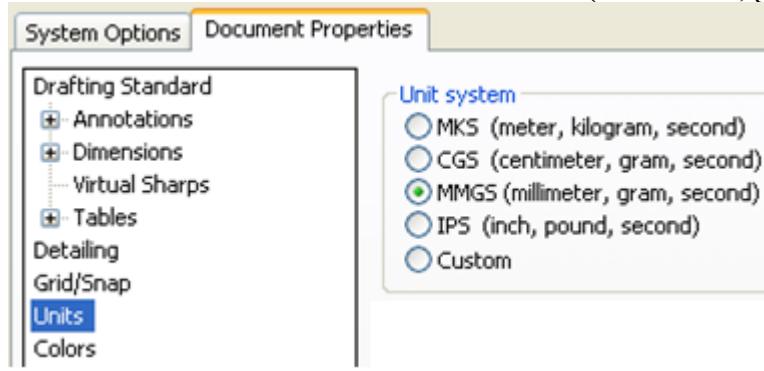


First click **Option** on top of main menu, open



**Document Properties** tab,

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



Ok, done!