Notes

1- اساليب التعامل مع المسائل Approaches (advantages & disadvantages)
- Experimental خاص مضبوط ولكن يحتاج اجهزة وكلفة ودقة وامان ووقت طويل.
- Theoretical عام دقيق ولكن للمشاكل او المسائل البسيطة.
- Numerical (Program or Package) واسع التطبيق وكلفة ووقت وخطورة اقل ولكن معقد وتقريبي.

2- لماذا نستخدم هذه البرامج (Packages)؟

ج: تعلّم برامج رسم وتصميم ومحاكاة متطورة وقابلة للتعديل او التغيير في اي وقت ـ المعرفة قبل التنفيذ او استحالة التنفيذ – توقع ما حصل او سيحصل – ربط اكثر من عِلم – تعدد الحالات – فهم ما يحصل فعلاً داخل المنظومة – تحوي على كل العلوم التي قمت بدراستها واكثر بكثير.

3- المادة SolidWorks2014 / Computer Aided Design

- المحاضرات (تفاعلية) داخل المختبر اقل من 30 اسبوع وستكون هناك واجبات (Homework & Classwork)، وتُسمّى نشاط مختبري وتجمع وتقسّم على عدد الواجبات وتسلّم في كل اسبوع، ولا يُقبل التأخير الاّ بعذر رسمي، اما بدون عذر فيُصلّح الواجب البديل من 5 بدل الـ 10، وفي حال رغبة الطالب زيادة الدرجة فيُكلّف بواجب اضافي يُحدد من قبل الاستاذ المشرف.
- توزيع الدرجات:

الموضوع	الفصل الأول		الفصل الثاني		امتحان مختبر	السعي	الدرجة النهائية
	امتحان	نشاط مختبري	امتحان	نشاط مختبري			
CAD	10%	10%	10%	10%	10%	50%	50%

(دروس التقييم المستمر ذات المختبر)

- صيغة المحاضرة: Word, Power Point, PDF, Movies, Pictures, Lectures, Books, Examples, Solved Cases, Questions and Solutions, …etc.
- كل هذا موجود على حاسبات المختبر في الـ (C) مع اسئلة السنوات السابقة والواجبات الصفّية والبيتية وكذلك على موقع الخزن IDriveSync اما التبليغات والمستجدات فتكون على الـ Facebook.

4- ما مطلوب من الطالب:

- حاسبة شخصية جيدة وسريعة.
- فلاش رام لنقل اي شيء من حاسبة المختبر.
- تنصيب البرنامج على الحاسبة الشخصية (بدون ربط انترنيت مباشر).
- التواصل عبر الفيسبوك:

أ- ابحث عن الحساب Cae Cad Group من خلال ميزة البحث في فيسبوك وكتابة الايميل التالي:
caecadgroup@gmail.com

ب- اضف الحساب كصديق على الفيسبوك مع ارسال رسالة تذكر فيها المرحلة والاختصاص والشعبة واسمك الثلاثي كي يتم التعرف عليك من قائمة الاسماء.

ج- سوف يقوم احد تدريسيي المادة بقبول الاضافة وثم ضمك الى المجموعة.

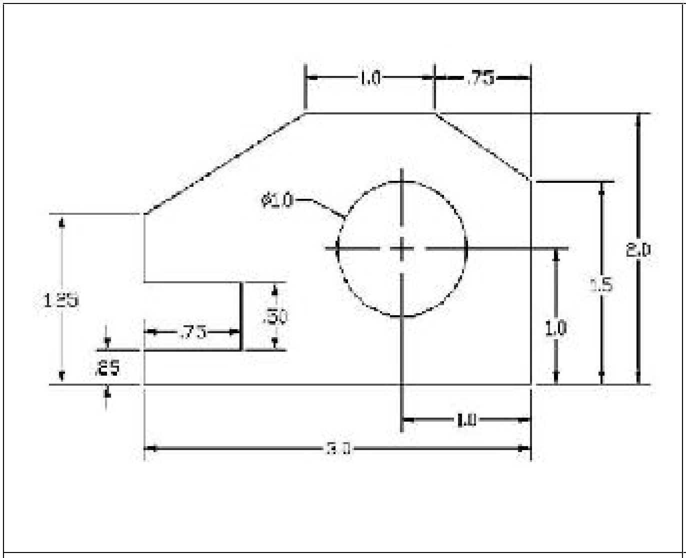
- خزن الواجبات الصفّية والبيتية لكل طالب على حاسبته في المختبر التي تحمل رقمه في قائمة الحضور بإسم project على الـ (D) داخل حافظة تحمل تسلسل واسم الطالب الثلاثي وتكون داخل حافظة بالمرحلة والتخصص والشعبة كلها باللغة الانكليزية ، ويكون هذا هو السياق في الامتحانات ايضاً. كما في المثال التالي:

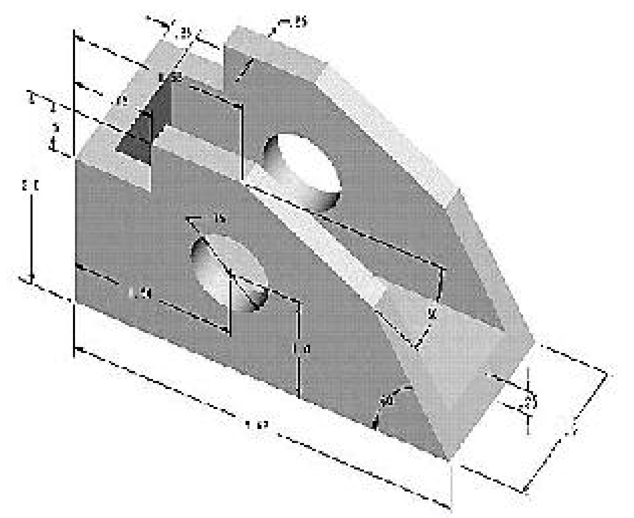
D:\ 3rd Year \ GE-A or -B (or AC-A or -B or AU or AE) \ 1Ahmed Mustafa Mohammed \ project1 (HW or CW).

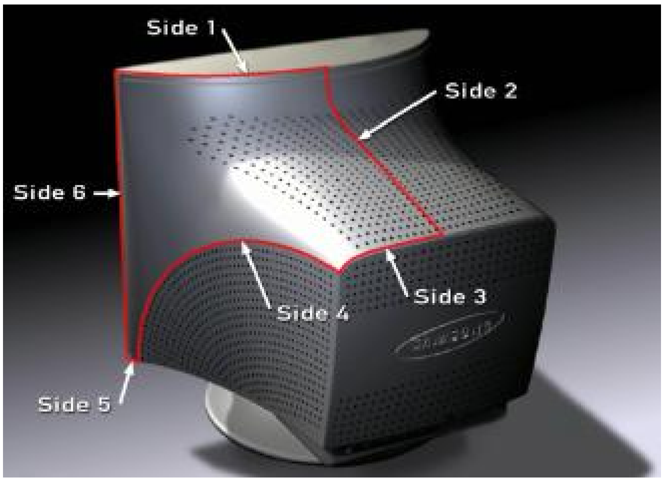
:CAD -5

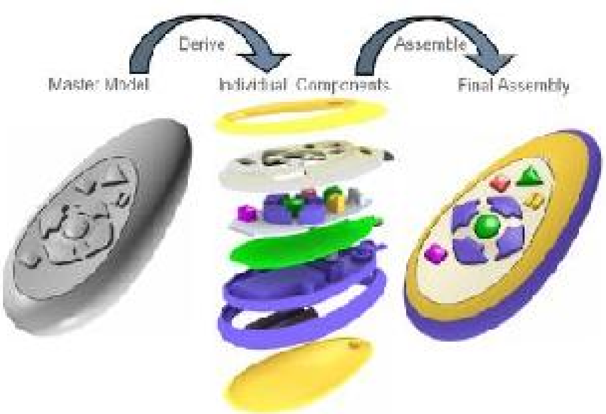
- **(Computer Aided Draughting/Design)**
- **• Creation of 3D 'virtual' models**
- **• Creation of 2D drawings**
- **• Creation of 2D drawings from 3D models**
- **• Simulation of behavior**
- **• Simulation of real life appearance**
- **• Assist in manufacture**
- **Advantages:**

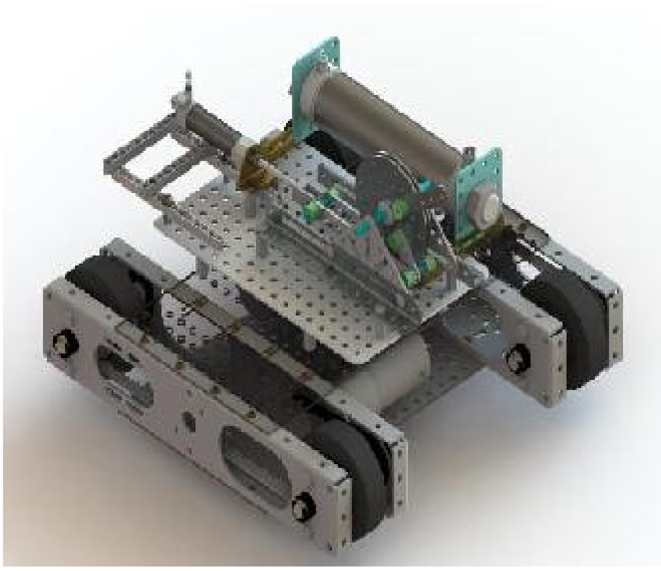
- **Easier creation and correction of drawings**
- **Better visualization of drawings**
- **Quick and convenient design analysis**
- **Simulation and testing of designs (stress)**
- **Increased accuracy**
- **Improved filing system of the drawing (Hard disk)**









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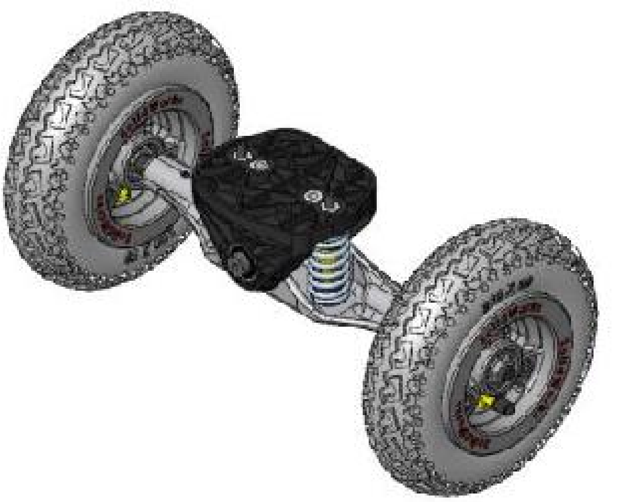
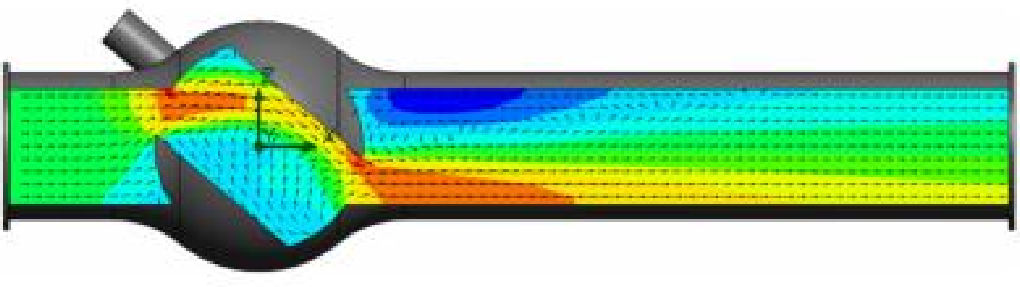
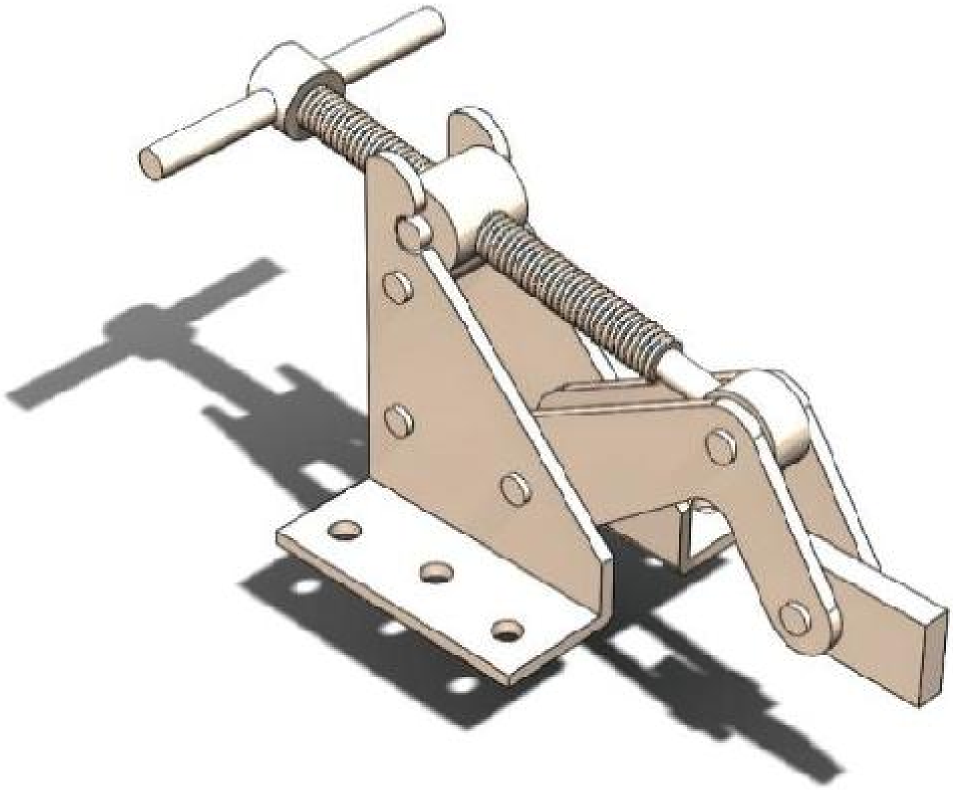






<u>**استعراض للحافظات**</u>

6- التنصيب:

خطوات تنصيب برنـامج الـ SolidWorks يعتمد على الاصدار ، وفي كل اصدار توجد تعليمات خاصة موضّحة بالتفصيل ، وهي عبارة عن صور في الحافظة solid work setup داخل حافظة المحاضرات.

7- المنـهاج:

1	**Introduction to CAD and parametric modeling** **- Basic Concepts - parts - assemblies - drawings**
2	**Sketching** **- Sketch Tools Toolbar - Edit Sketch**
3	**2D to 3D Conversion** **- 2D to 3D Conversion Overview - 2D to 3D Toolbar**
4	**Reference Geometry** **- Reference Geometry Overview - planes - axes - coordinate systems**
5	**3D curves** **- Projected Curve- Composite Curve- Helix and Spiral**
6, 7, 8, 9, 10, 11, and 12	**Features** **- Base/Boss, and Cut - Extrude - Revolve - Sweep and Loft - Fillet/Round - Chamfer and Draft - An application - Pattern and Mirror - Hole Wizard - Shell - Rib - Dome** **- An application**
13	**Part properties** **- Editing - moving copying, - color**
14	**Equations** **- Applying equations**
15	**Dependency** **- Geometric Dependency**
16, 17, 18, and 19	**Assemblies** **- Adding assembly components - Assembly mating - Working with sub-assemblies - Smart Fasteners - An application**
20, 21, 22, and 23	**Drawings** **- Creating a Drawing - 2D sketching in drawings** **- Creating standard views (named views and standard 3 views)** **- An application**
24	**Detailing** **- Detailing tools**
25	**Files** **- Importing and Exporting Files**
26 and 27	**Analysis** **- Basics and Cosmos Express - Stress analysis**
28, 29, and 30	**Design project** **- Machine design project for each student**

8- مقدمة:

- **What is SolidWorks؟**

•SolidWorks is design automation software.

•In SolidWorks, you sketch ideas and experiment with different designs to create 3D models.

•SolidWorks is used by students, designers, engineers, and other professionals to produce simple and complex parts, assemblies، and drawings.

- **Benefits of solid modelling:**

•Solid modelling enjoys many benefits not offered by 2D design methods.

•The solid model has a volume and surfaces.

•The solid model can easily be analyzed for its physical properties, such as volume, mass, surface area, cross sectional areas, location of center of mass, moments of inertia, etc.

•The 3D environment offers excellent visualization of the design as a shaded solid, with texture and color, or as a wire framed representation.

هناك نوعان من برامج الرسم الهندسي:

a. **Parametric (Geometry drives dimensions)**
b. **Non-Parametric (Dimensions drives geometry)**

- **SolidWorks is a fully parametric CAD program.**

•This means that when a part is designed and modeled dimensions are assigned which define the part. If, at a later time, these dimensions are found to be unsuitable they can be easily changed and the modification will filter through the system wherever the part appears.

•This is particularly helpful when dealing with an assembly since, if a modification is made to a single part, the modification is carried throughout the assembly.

•A designer can also define relationships between parts. For example, in an engine, if the diameter of the piston is increased or decreased, the corresponding engine block can be defined such that it is automatically modified to match the specifications of the modified piston.

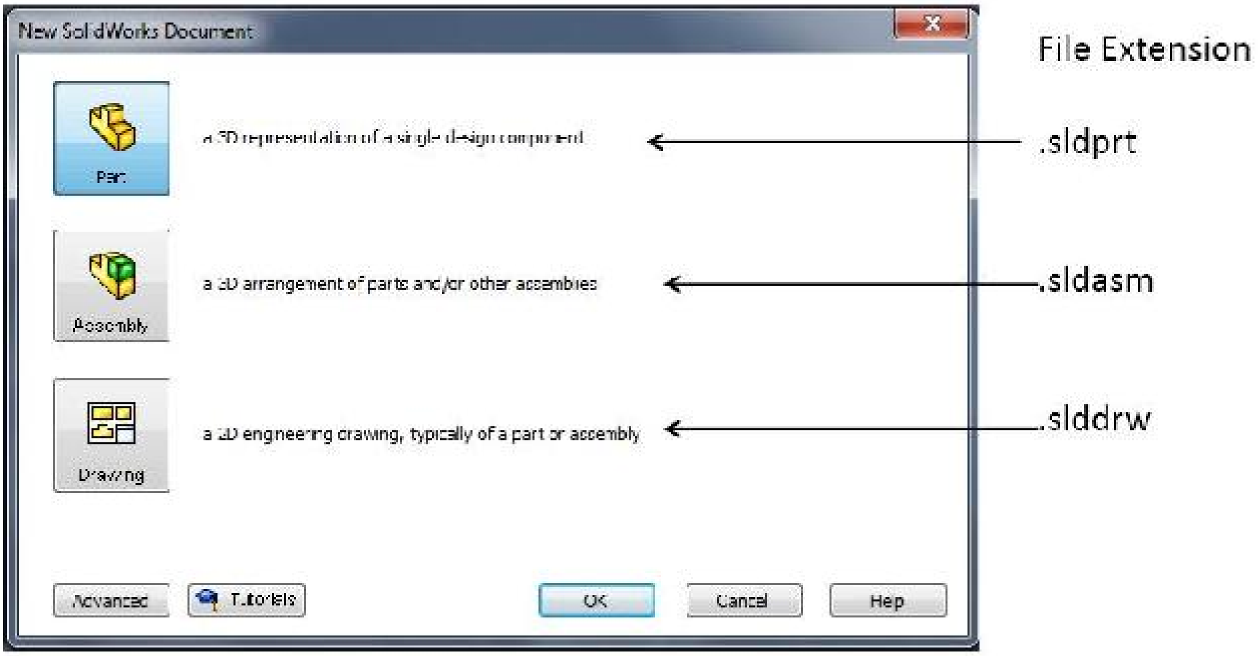
- **The SolidWorks Model:**

The SolidWorks model is made up of:

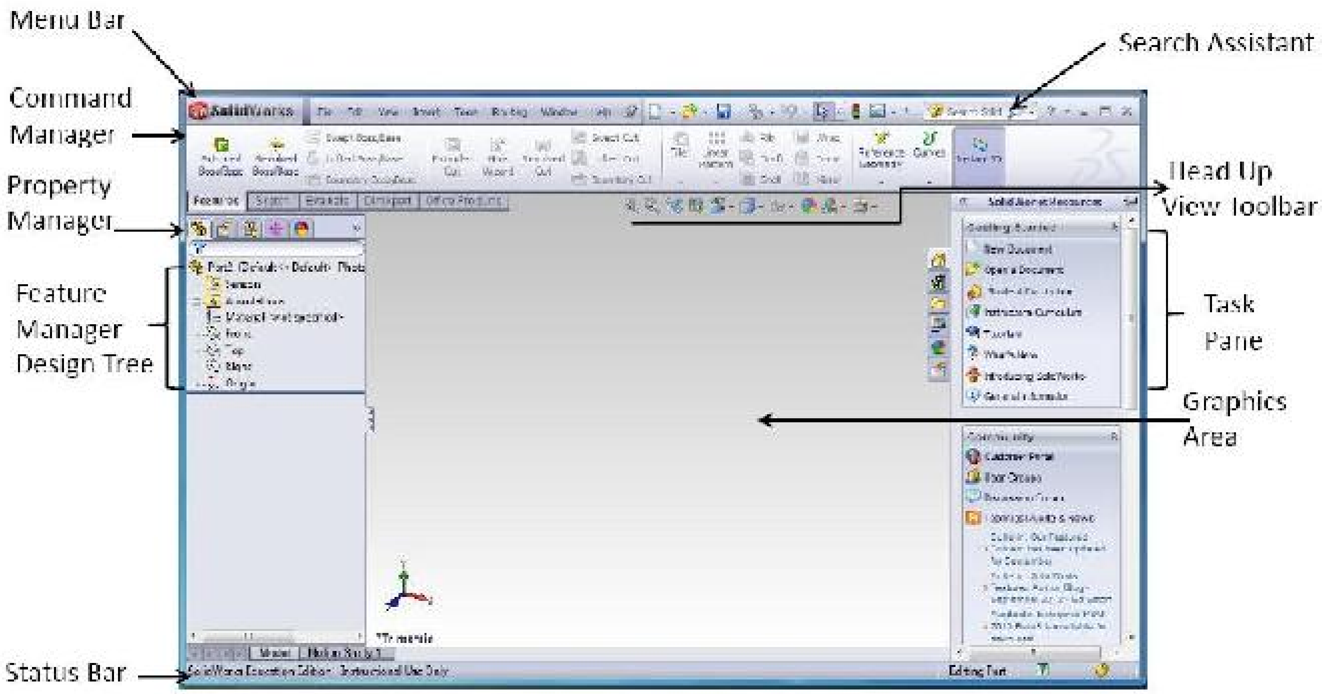
•Parts

•Assemblies

•Drawings



وتوجد اكثر من طريقة للبداية في الرسم: 1- <u>تجميع الخطوط</u> او 2- <u>رسم اشكال ثم مسح الزائد</u> او 3- <u>رسم جزء وعمل نسخ له اذا كان متناظر</u> ، وفي كثير من الاحيان نستخدم ثلاثتها معاً.



If you cannot see the *status bar* click *view* on the Menu bar and select *status Bar*

- **Solidworkds Menu**

- **CommandManager**

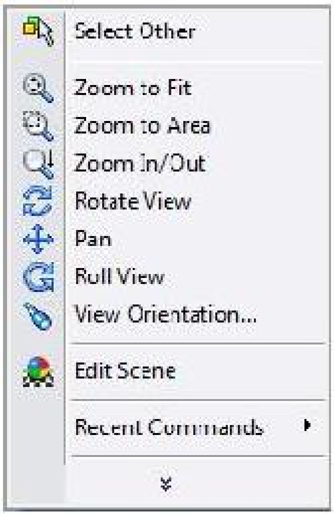
To toggle the descriptions and size of the buttons, right-click in the CommandManager and select or clear **Use Large Buttons with Text.**

- **Sketcher**

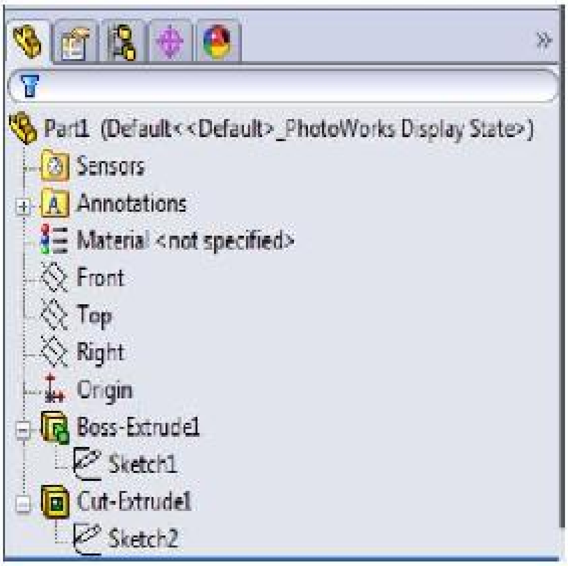
- **Features**

- **Head Up View Toolbar**

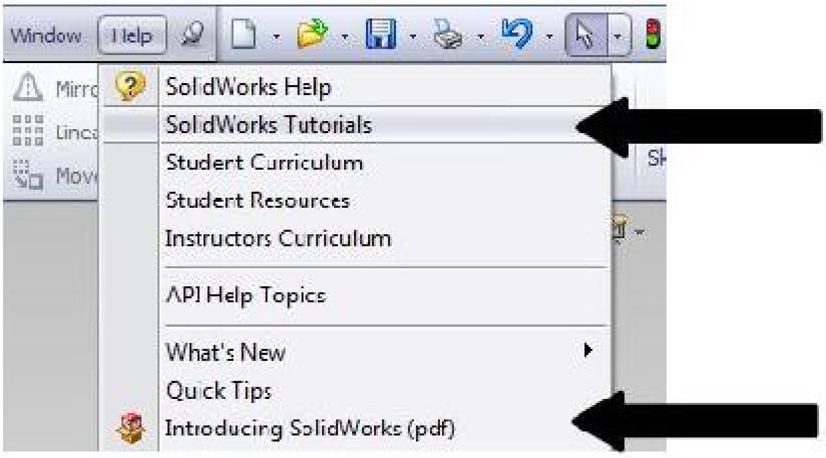
- **Manipulating the display using the Mouse**



- **FeatureManager Design tree**



- **SolidWorks provides tutorials**

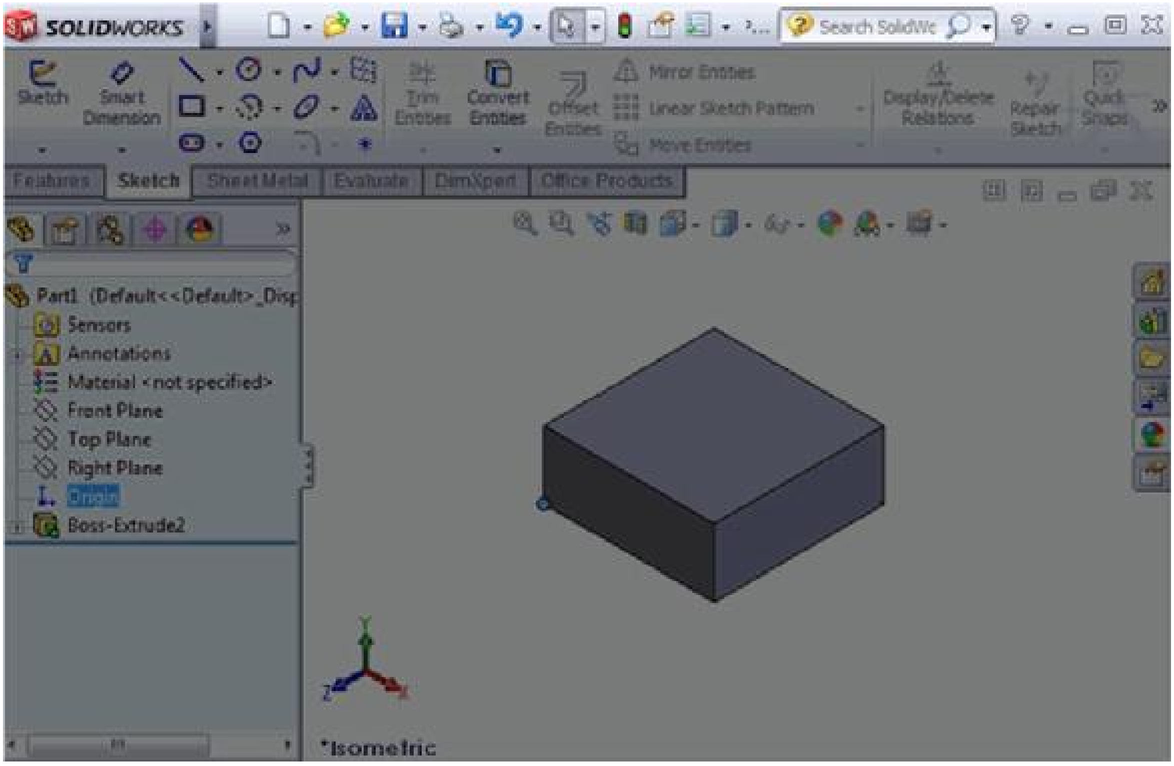


•http://www.solidworkslessons.info/#
•http://www.aboutsolidworks.com/solidworks_tutorials.htm
•YouTube

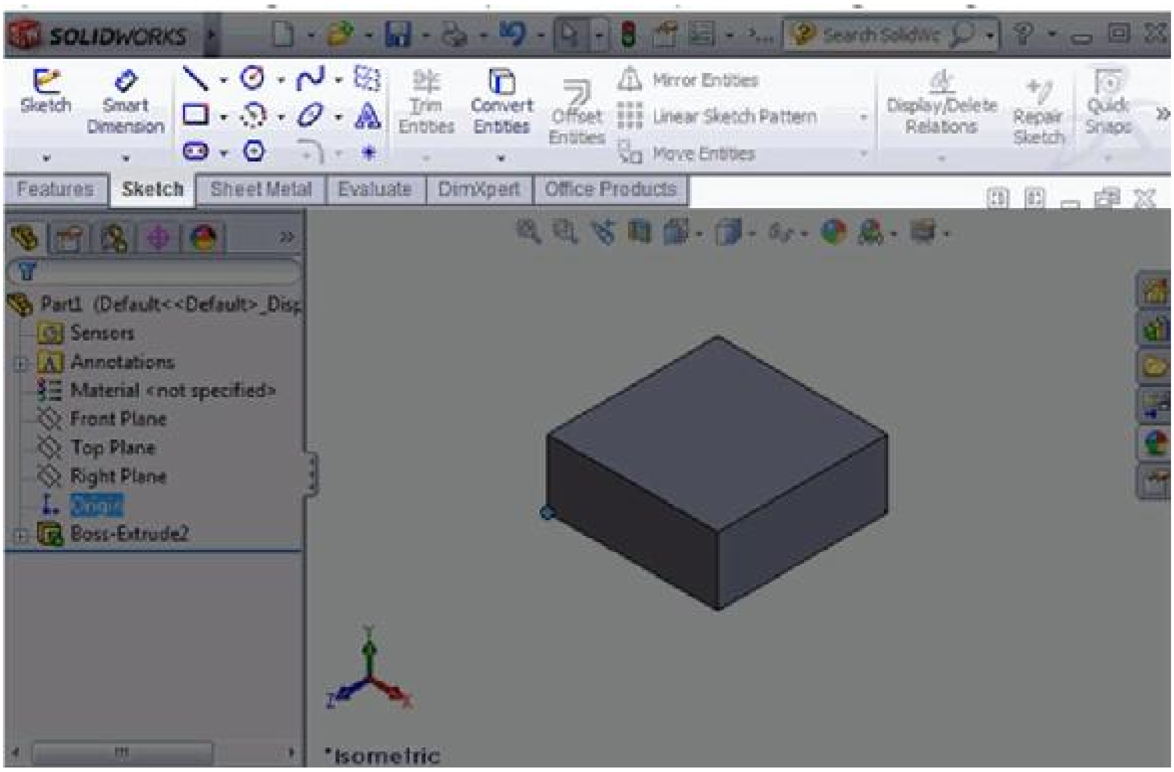
Tutorial 1: SolidWorks User Interface

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

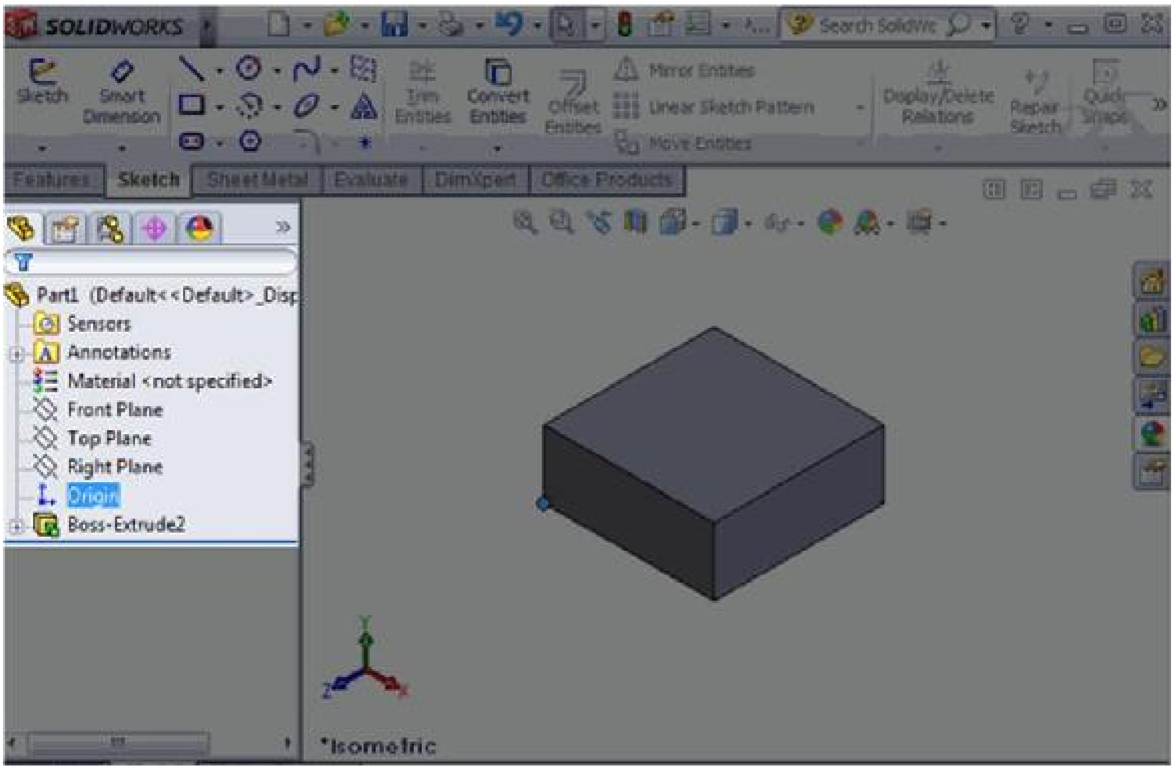
1) Menu Bar – Top most of the application, executing New File, Open File, Save, Print, Undo, Select, Rebuild, File Properties and Options.



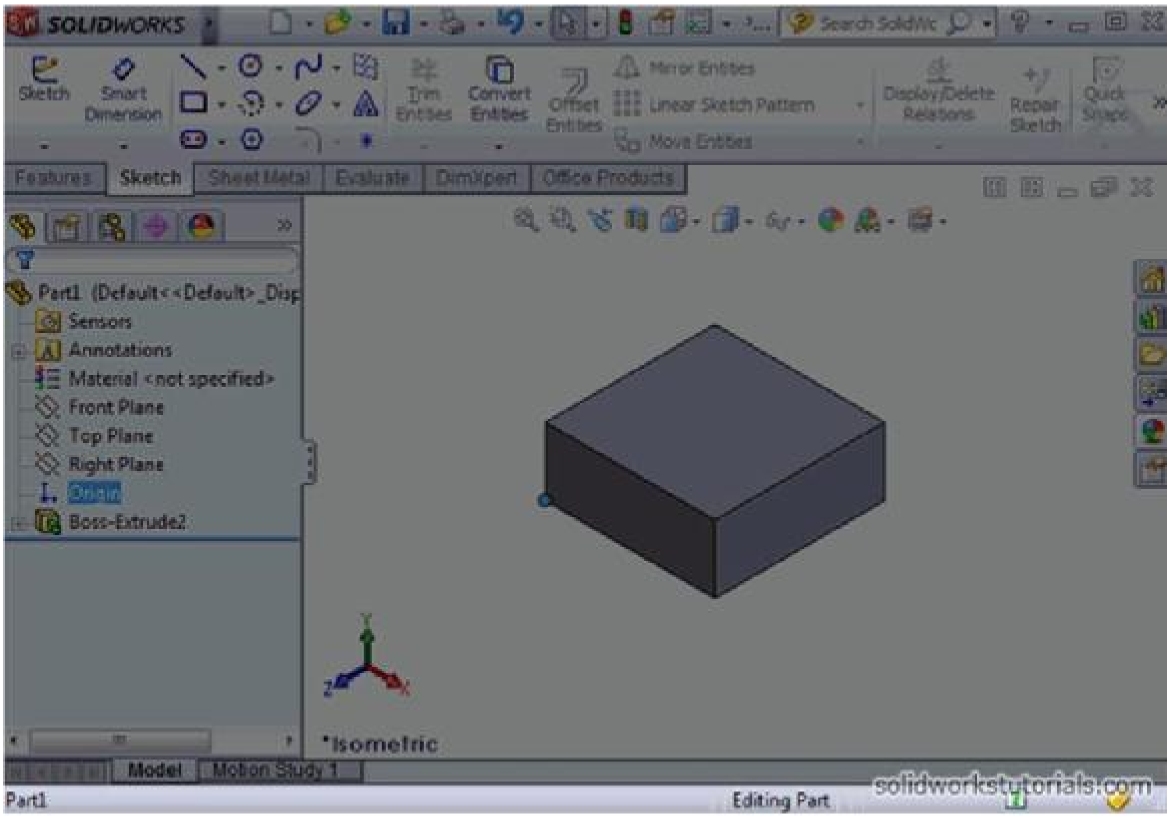
2) Command Manager – Access to part, assembly and drawing editting tools.



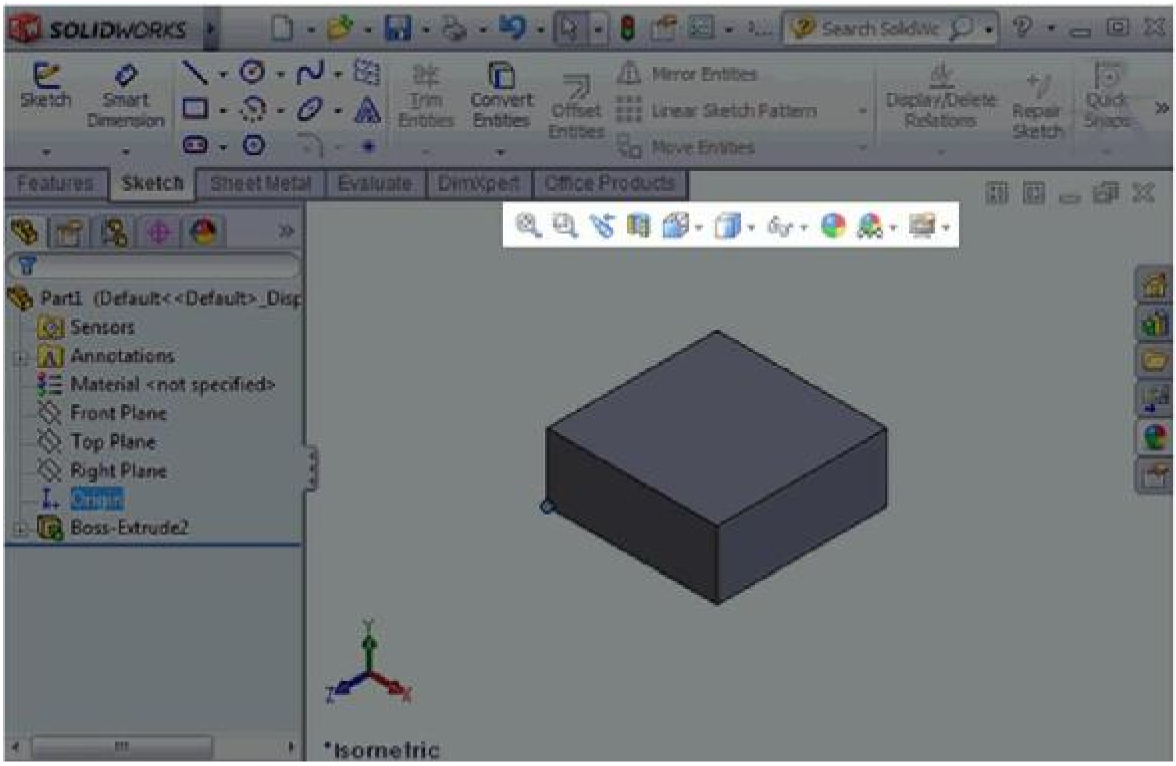
3) Feature Manager design tree – Outline overview how your part, assembly and drawing constructed.



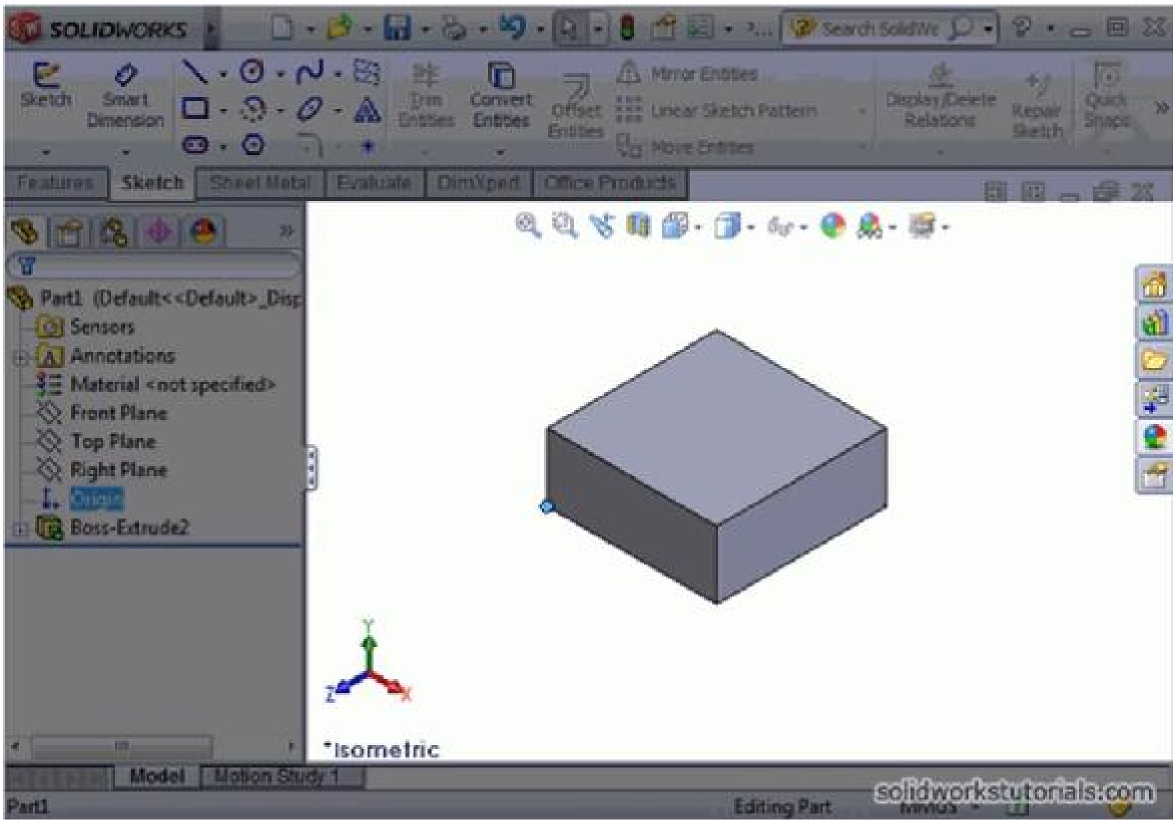
4) Status bar – Provide an information about your part, assembly and drawing.



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.

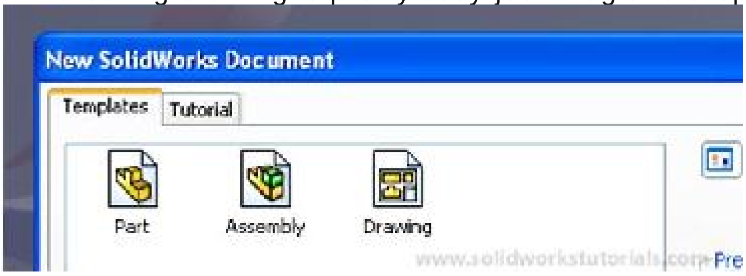


SOLIDWORKS
Sketch
Smart Dimension
Trim Entities
Convert Entities
Offset Entities
Mirror Entities
Linear Sketch Pattern
Move Entities
Display/Delete Relations
Repair Sketch
Quick Snaps
Features
Sketch
Sheet Metal
Evaluate
DimXpert
Office Products
Part1 (Default<<Default>_Disp
Sensors
Annotations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Boss-Extrude2
*Isometric
Model
Motion Study 1
Part1
Editing Part
solidworkstutorials.com

Introduction to SolidWorks

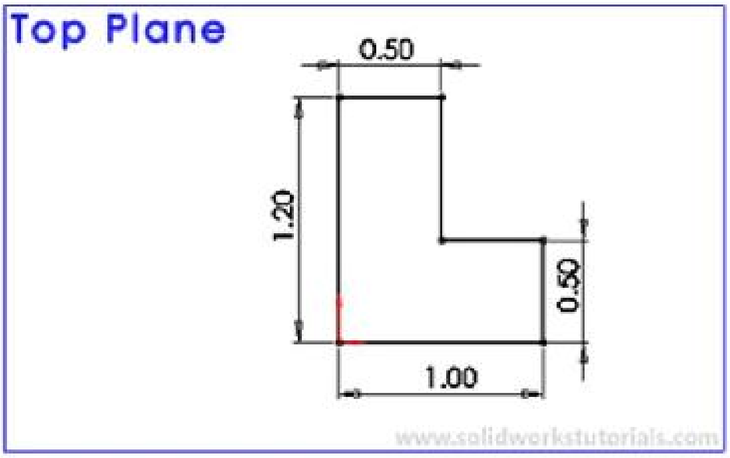
Solidworks Overview

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.

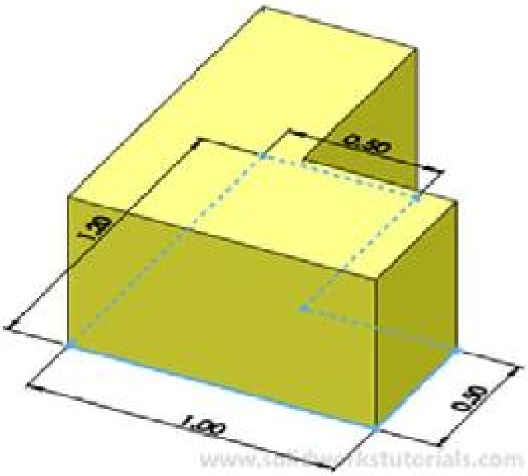


Part

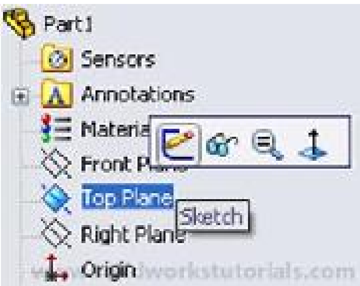
Part is created by sketch.

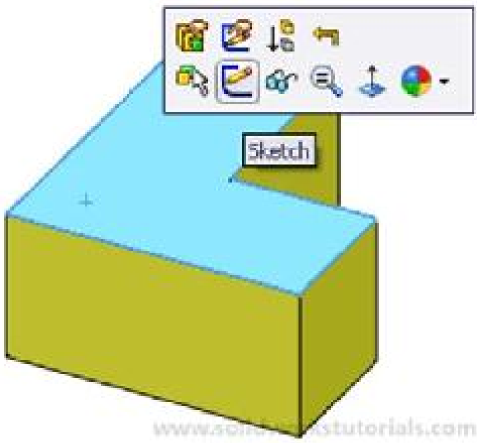


Sketch is the base to define your part, form and features.

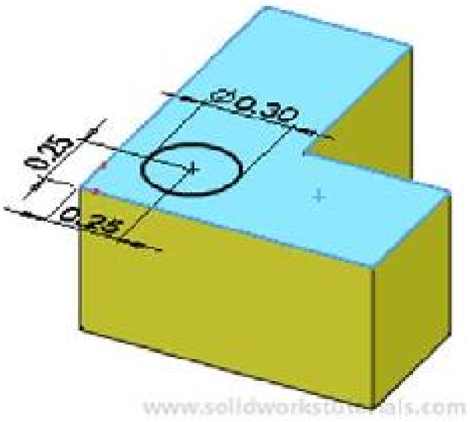


Before you start creating sketches you must select plane or face where the sketch will be place on.

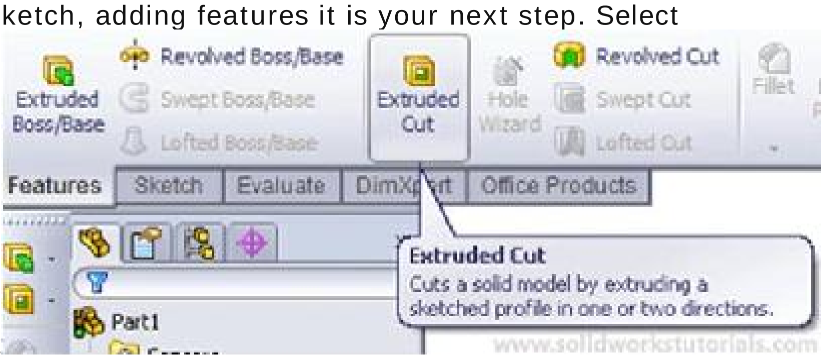




After select plane or face the sketch will be, sketch on it!

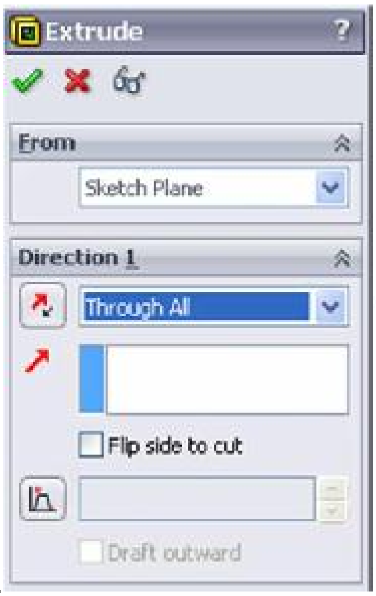


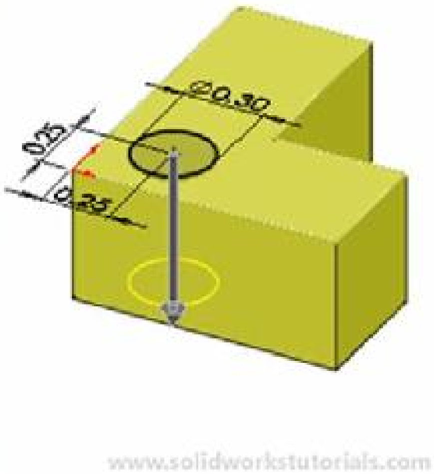
When you done with sketch, adding features it is your next step. Select

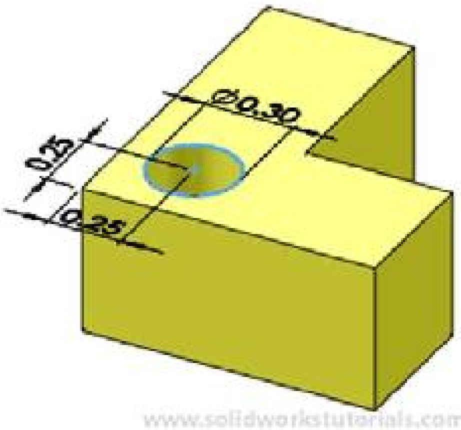


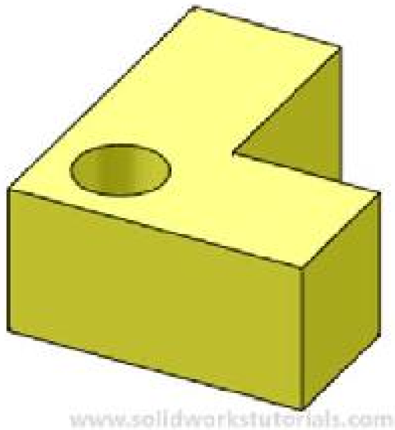
Feature>Extruded Cut

Select Through All and OK.



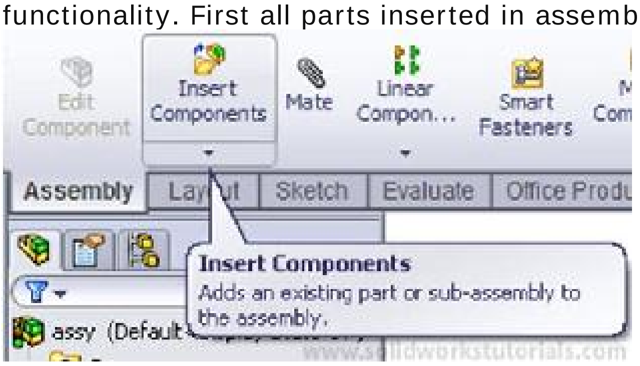




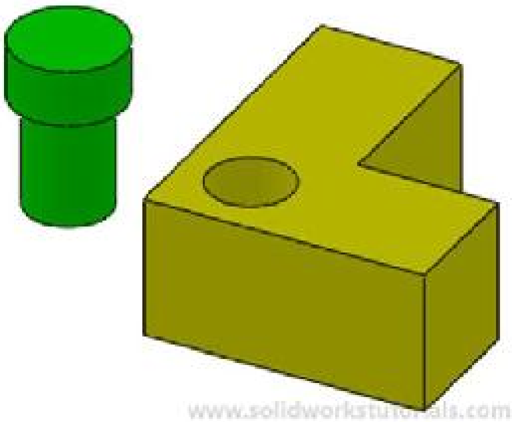


Assembly

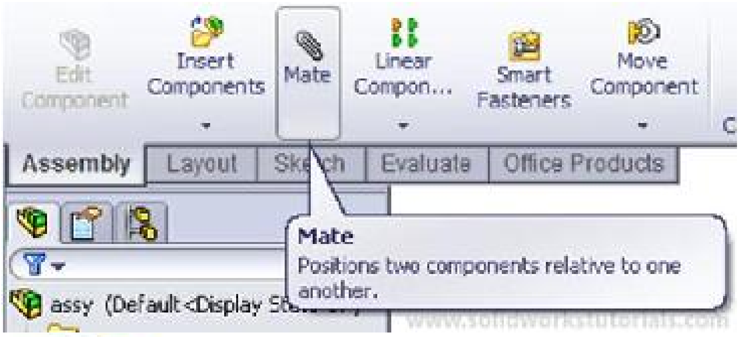
Assembly is how all parts works together in assembly, checking for clashes and it functionality. First all parts inserted in assembly by Insert Component tool.

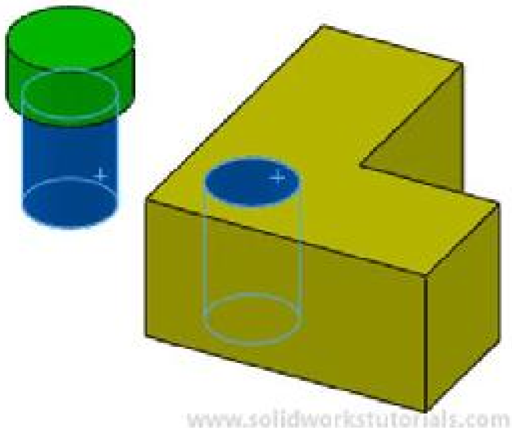


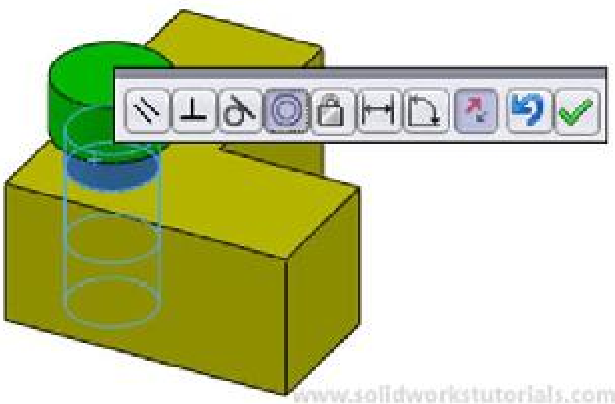
When all parts inserted into workspace, Mate is command to define how parts mate with each other.



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

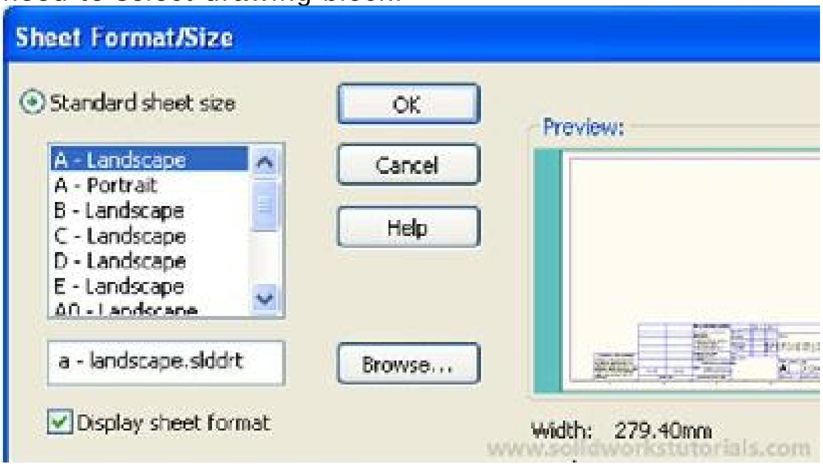






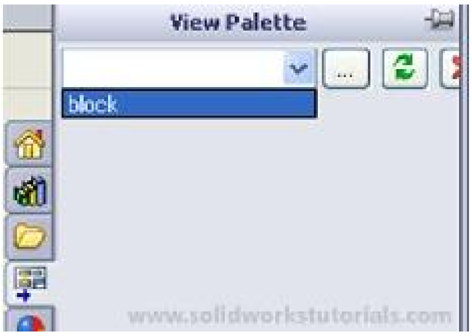
Drawing

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

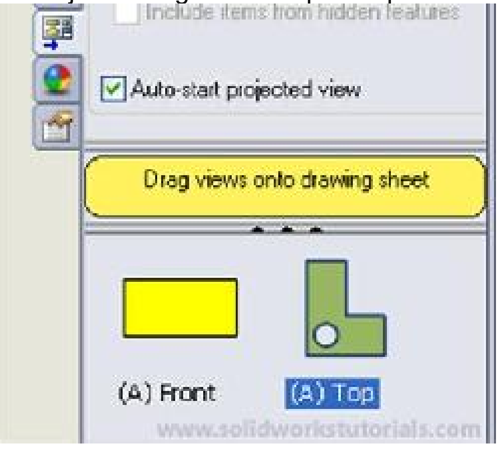


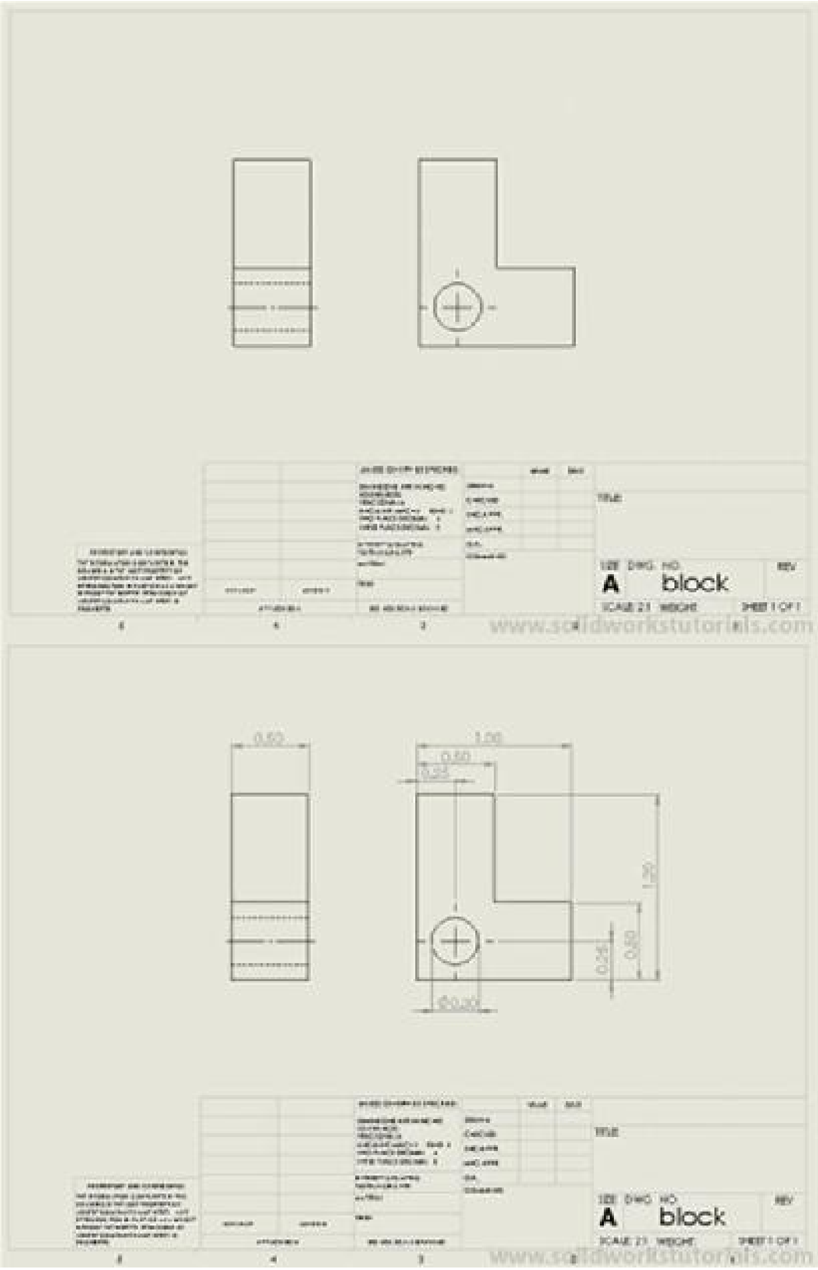
When block inserted, select click view palette to add drawing view.

Choose the part you wish to make drawing.



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

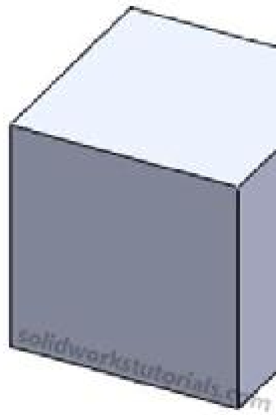




Summary

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

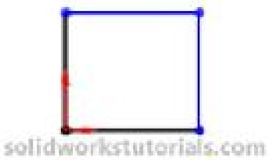
How to create simple box



1. Click New , Click Part 

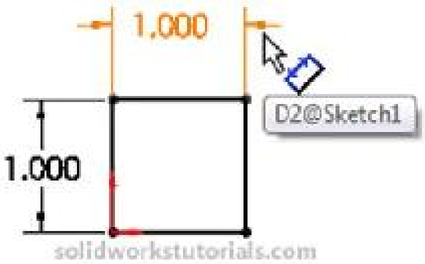
and OK.
2. Click on Top Plane and click Sketch.

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

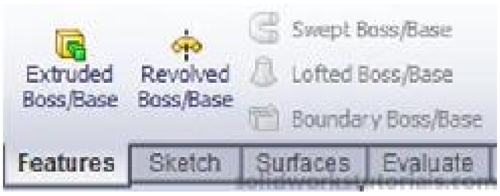


4. Click Smart Dimension 

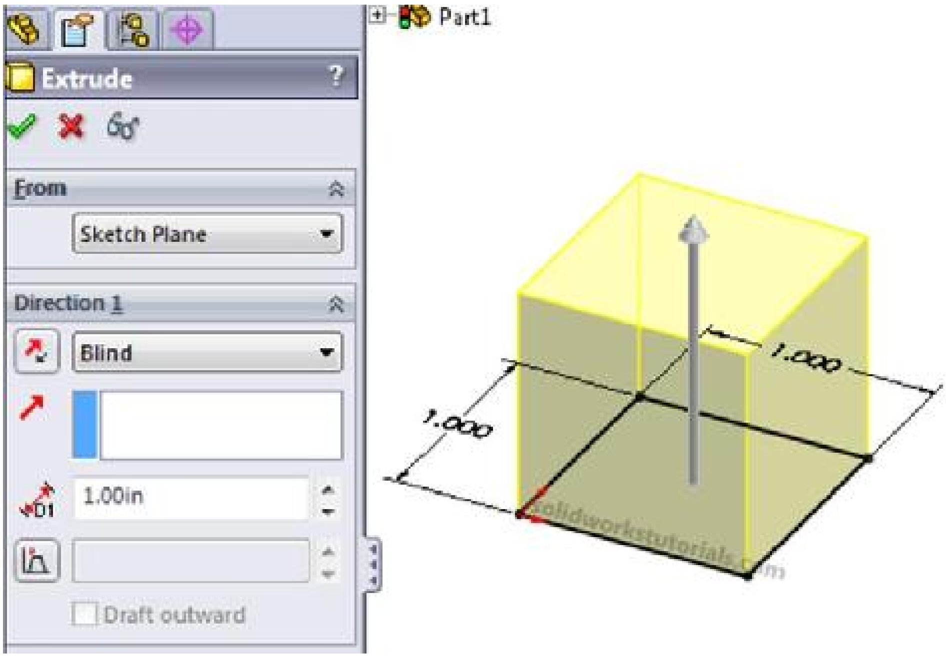
, click side edge and click top edge to dimension it as 1.0in x 1.0in.



5. Click Features>Extruded Boss/Base

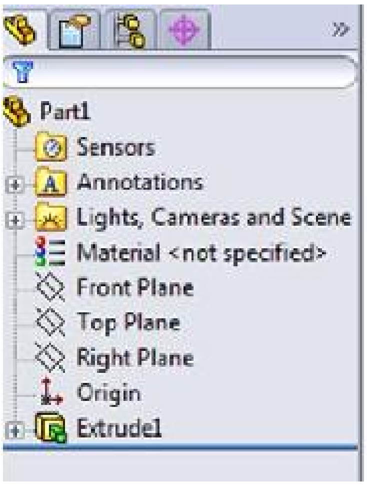


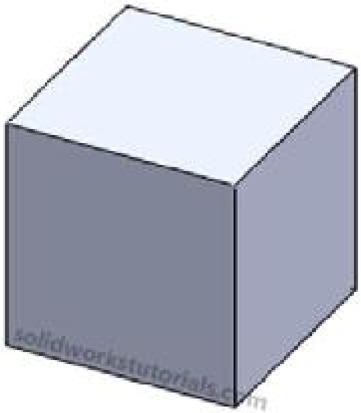
set D1 as 1.0in



and click .

6. It’s done.


Part1
Sensors
Annotations
Lights, Cameras and Scene
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Extrude1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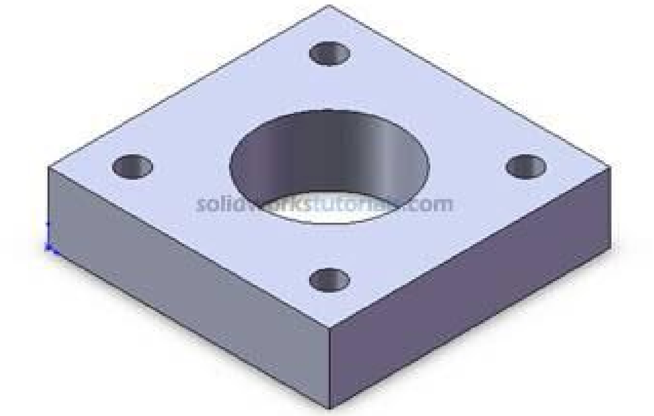
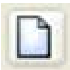
How to create simple plate



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

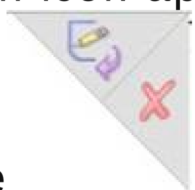
, OK.

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 

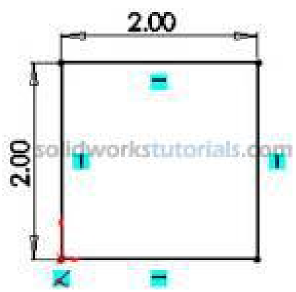
. As you can see on upper right corner sketch icon appear indicate that you're on sketch mode .

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

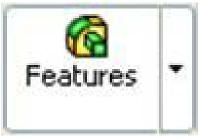
no need to be exact the size will define in later step. Press keyboard ESC to end rectangle sketch.

Note: There 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 2.00in , pick horizontal line and set to 2.00in 

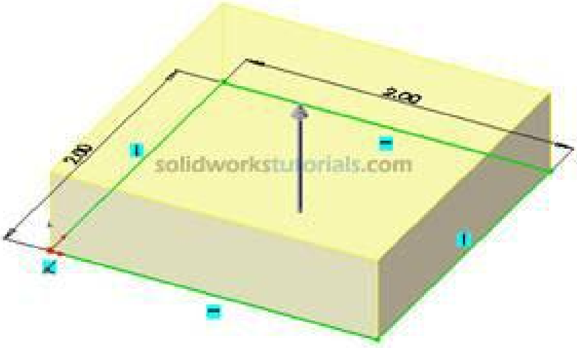
. Press keyboard ESC to end smart dimension

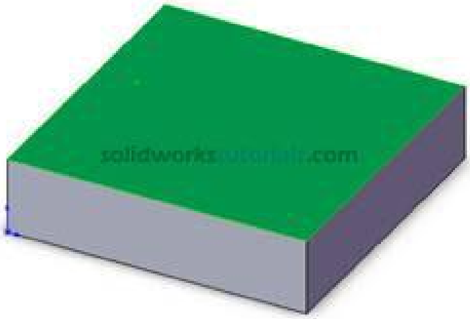
7. Build feature from sketch, click Features 

and activate features menu. Click Extruded Boss/Base 

and set D1 to 0.5in

and  .



8. Click front top face  , click Normal To 

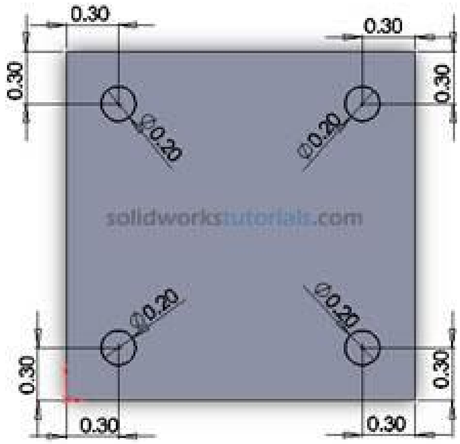
. Activate sketch menu by

click Sketch 

 and select Circle 

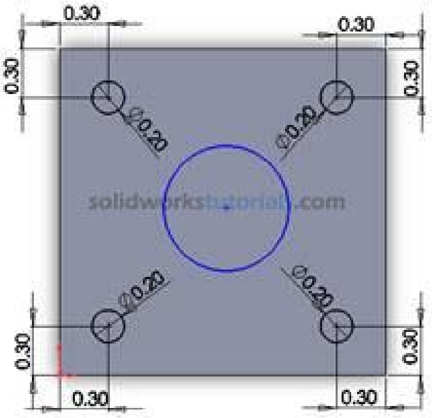
. Sketch 4 circle at four edges. 

9. Define new circle sketch, click Smart Dimension 

, set diameter circle to 0.2in . Select distance for edge set to 0.3in 

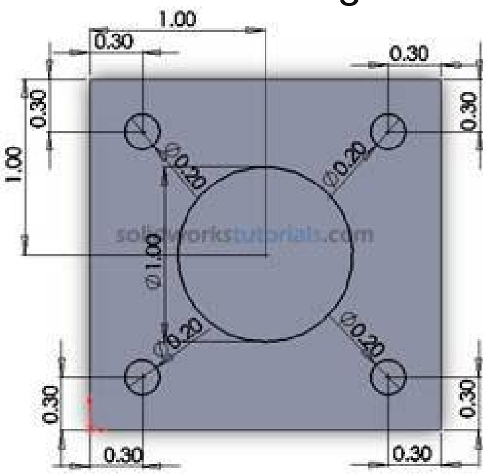
.

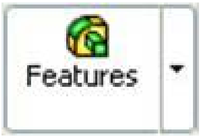
10. Click Circle  and sketch one circle at center



.

11. Define new circle sketch, click Smart Dimension , set diameter circle to 1.0in . Select distance for edge set to 1.0in.

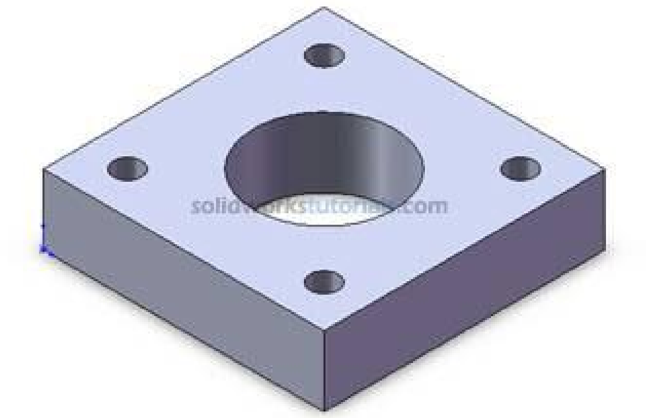


12. For cut click Features 

, click Extruded Cut 

, under Direction 1, Through All 

and .



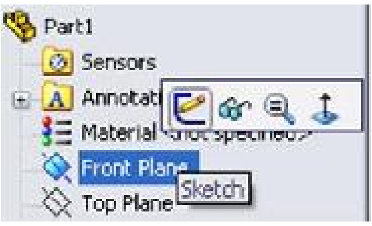
Done.

How to create Allen key

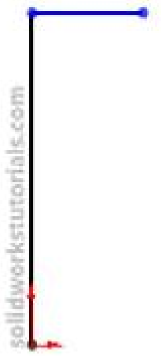
In this Solidworks tutorial, you will create simple allen key.

1. Click New.  Click Part, 

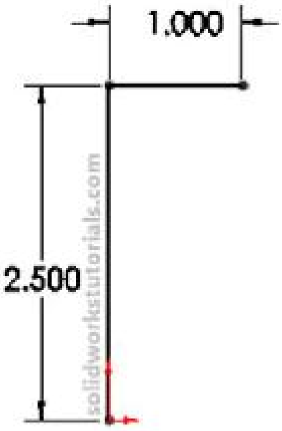
OK.
2. Click Front Plane and click on Sketch.



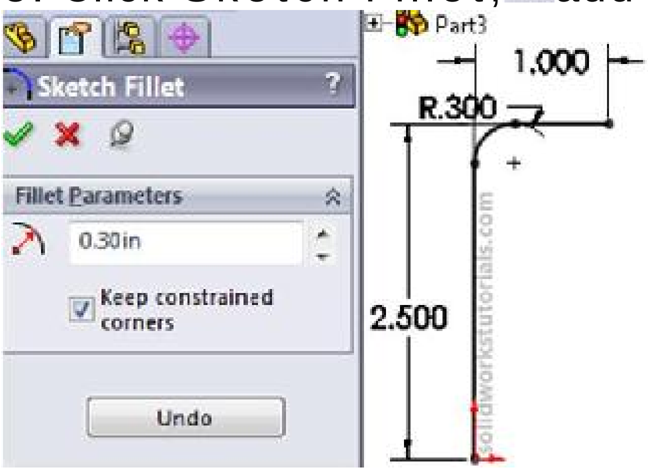
3. Click Line, skecth a L shape.



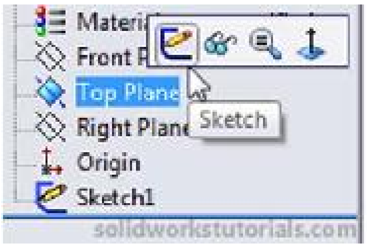
4. Click Smart Dimension,  and dimension sketch as 2.5″ and 1″.



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



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

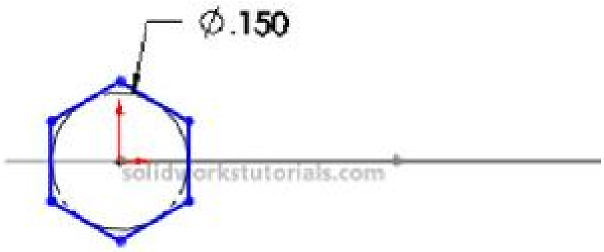


7. Click on Sketch2 and click Normal To.

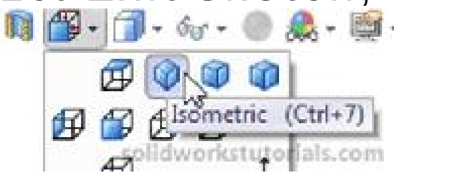
8. Click Polygon,  sketch a polygon at origin.

9. Click Smart Dimension, 

and dimension sketch diameter to 0.15″.

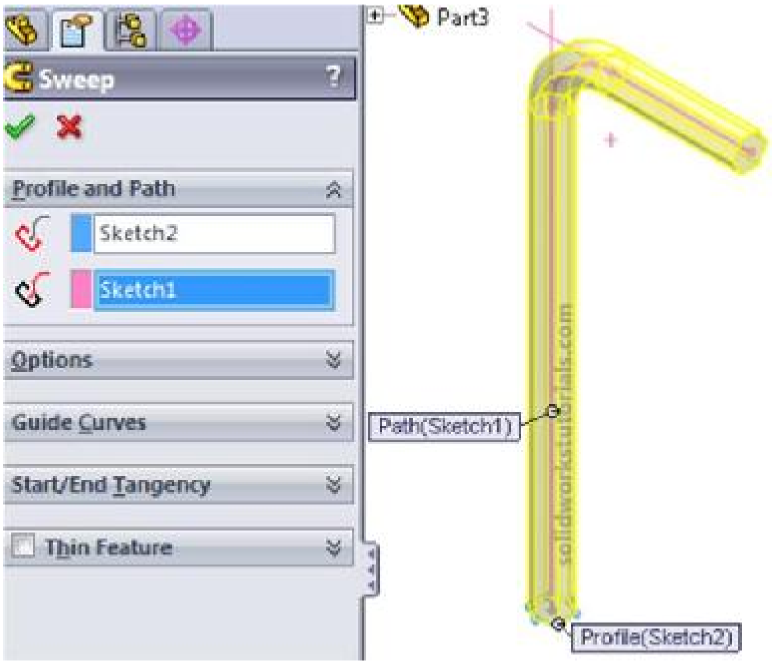


10. Exit sketch,  click on Isometric view.

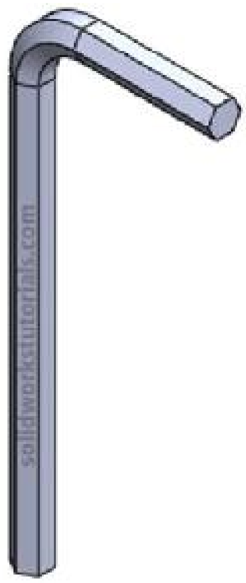


11. Click Features>Swept Boss/Base, 

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

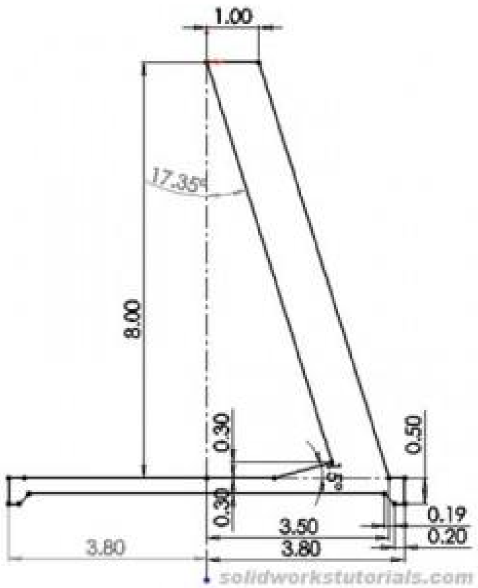

Part3
Sweep
Profile and Path
Sketch2
Sketch1
Options
Guide Curves
Start/End Tangency
Thin Feature
Path(Sketch1)
Profile(Sketch2)
solidworkstutorials.com

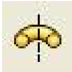
You're done!.


solidworkstutorials.com

How to create 17 inch car wheel

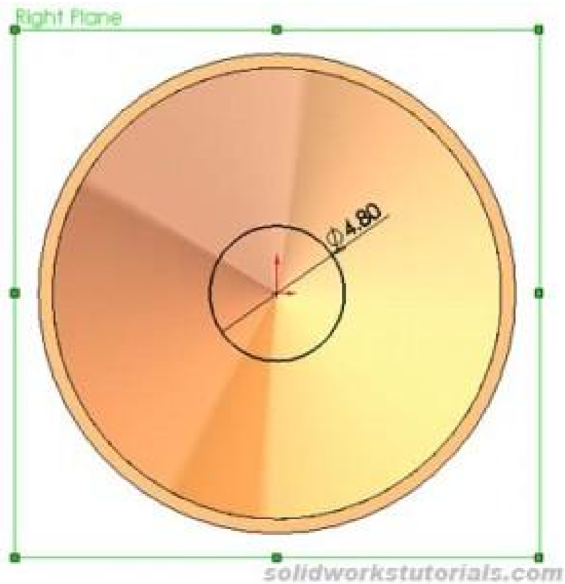
1. Create a skecth as show on Front Plane.



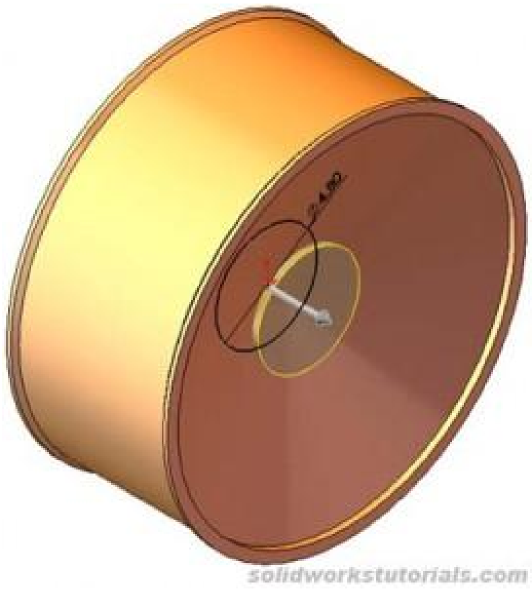
2. Revolve  sketch, 360 degree on top sketched line

. OK.

3. Create circle skecth, on right plane 4.8in

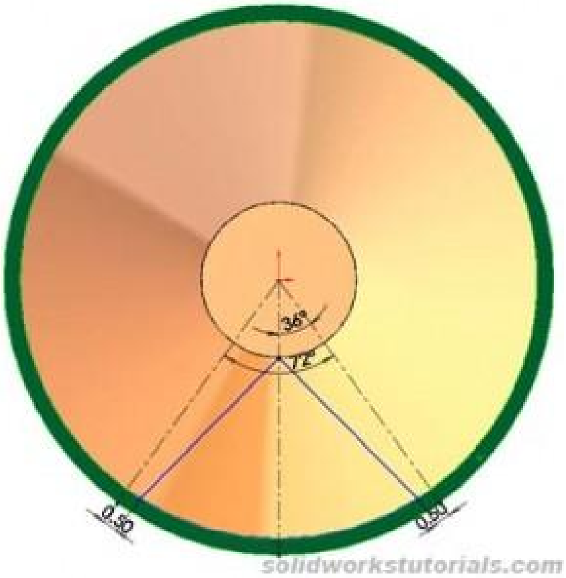


, extrude  2in

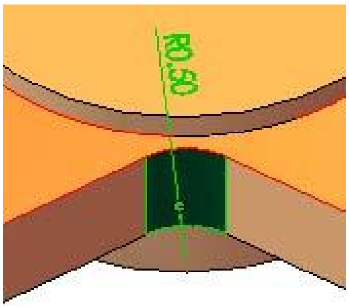


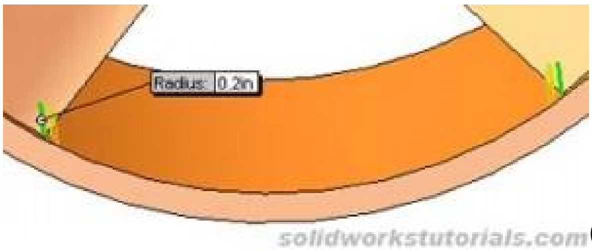
OK.

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

hole 

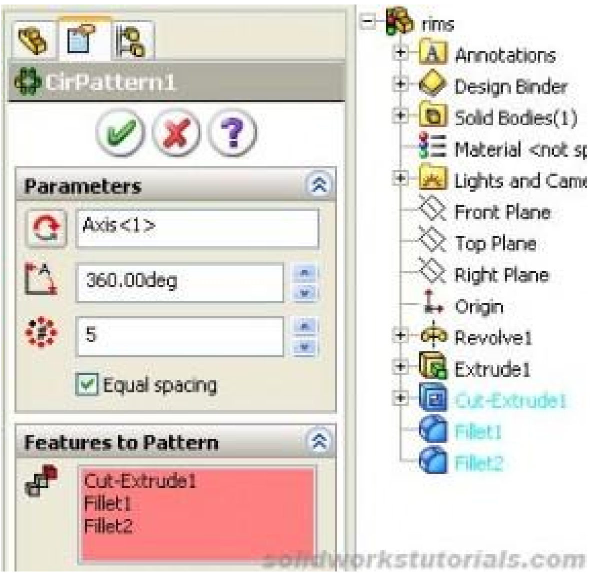
, extruded cut , through all, OK.

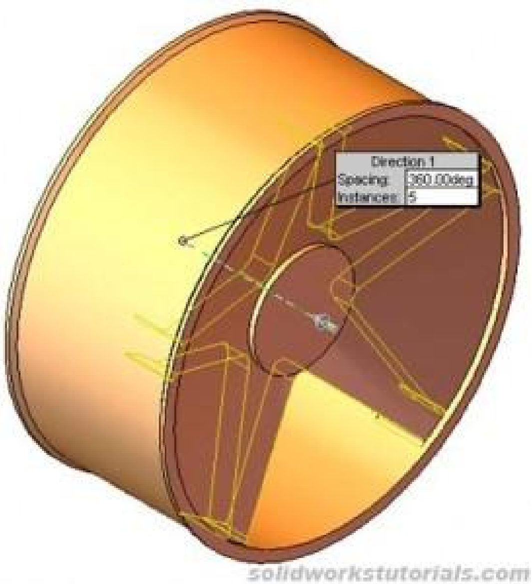
5. Add fillet R0.5in inner 

, add fillet 0.2in 

OK.

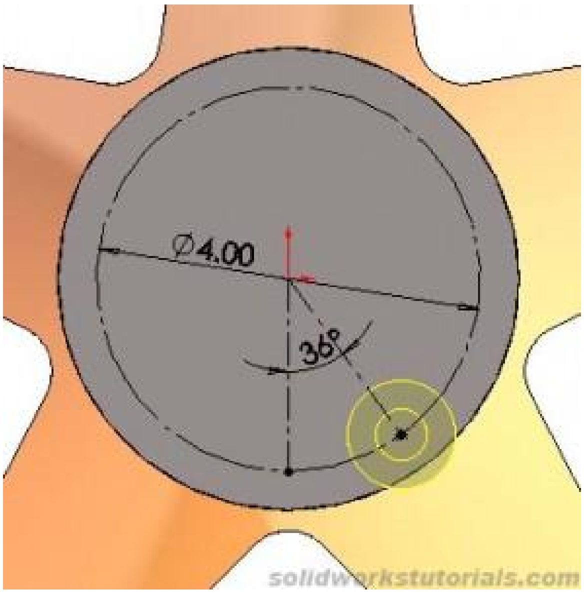
6. Click Circular Pattern , click View>Temporary Axes, select center axis as rotation axis. 360 degree

and #5 equal spacing 



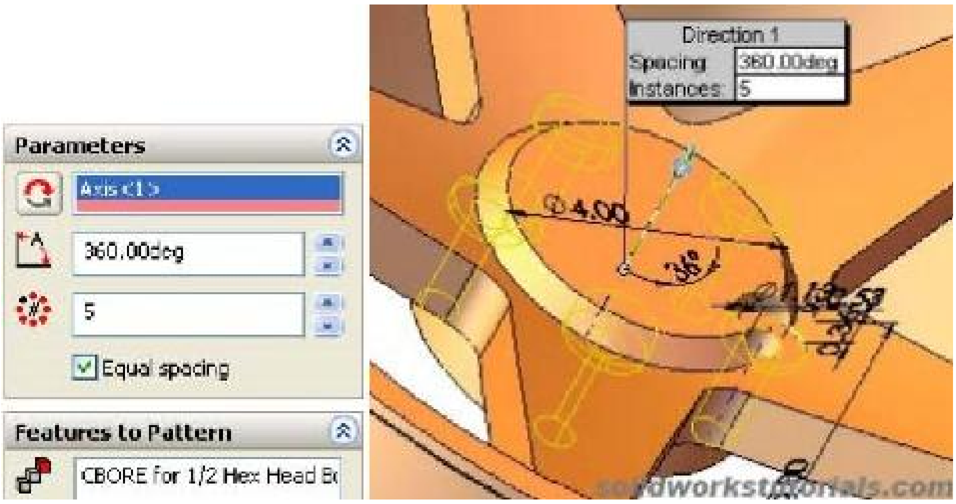
. Select Cut-Extrude1, Fillet1 and Fillet2 as a Features to Pattern. OK.

7. Select hub face, click Hole Wizard , select Ansi Inch, Hex Bolt, size 1/2, through all. Position point at

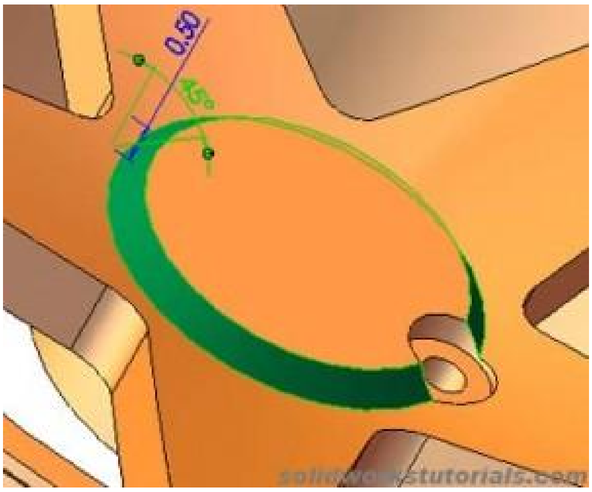
diameter 4in and 36 degree 

. OK.

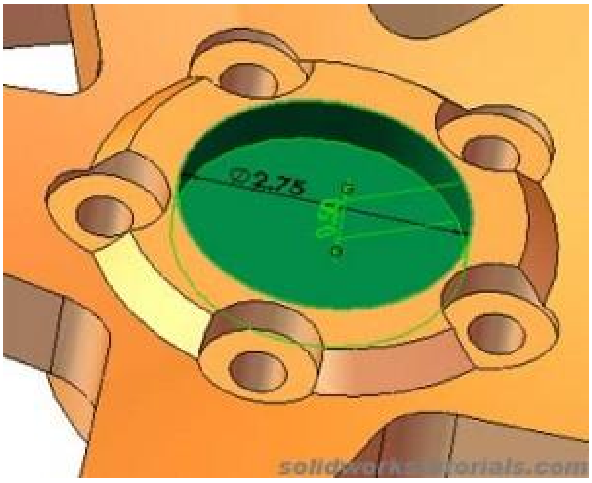
8. Click Circular Pattern  , select center temporary axis, 360 degree and #5 equal spacing. Select CBORE for 1/2 Hex Head Bolt as Features to Pattern. OK.

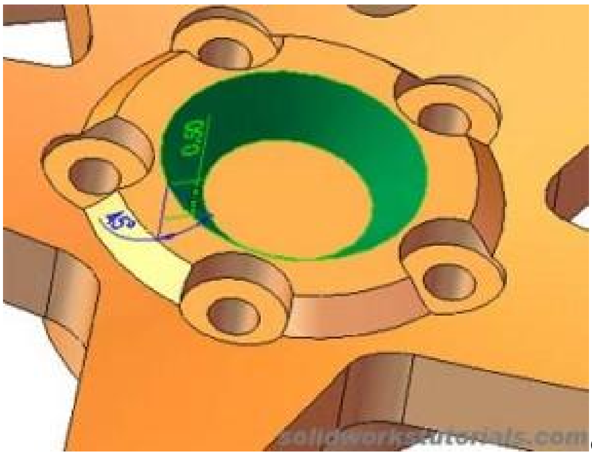


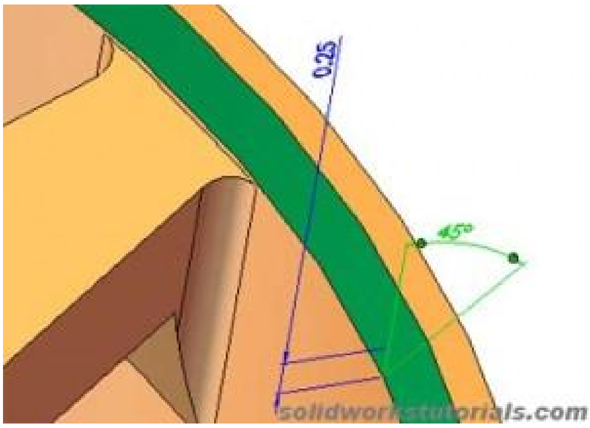
9. Add chamfer 0.5in to hub side.



10. Click on hub face, insert skecth, sketch circle diameter 2.75in. Extrude Cut  to 0.5in deep.

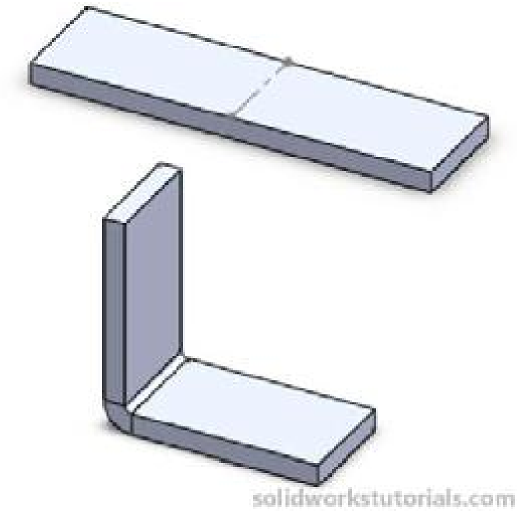


11. Add chamfer 0.5in to inner cut 

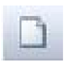
and add chamfer 0.25in to wheel edge 

, OK. Done.

How to create simple sheet metal bend



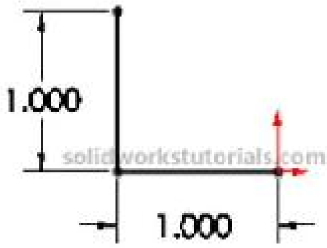
In this tutorials you will learn how to utilize sheetmetal tool such insert bend and flaten.

1. Click New.Click Part, 

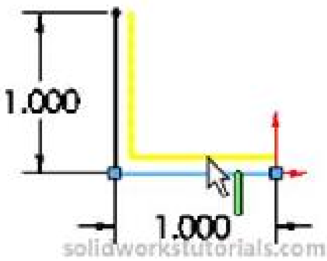
OK.
2. Click Front Plane and click on Sketch.

Use Line, sketch L shape. Dimension sketch with Smart Dimension

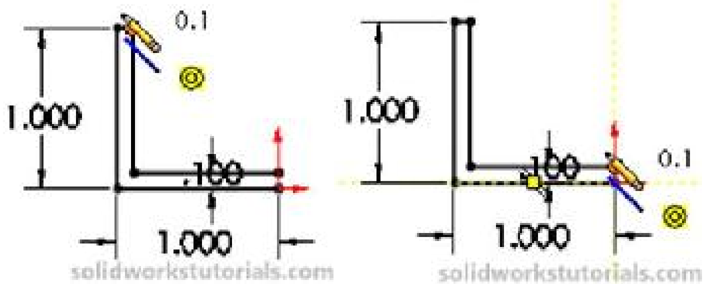
as 1in x 1in.

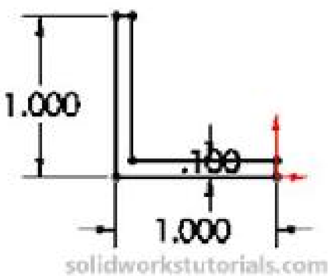


3. Click Offset Entities  and click L sketch. Set offset distance as 0.1in.



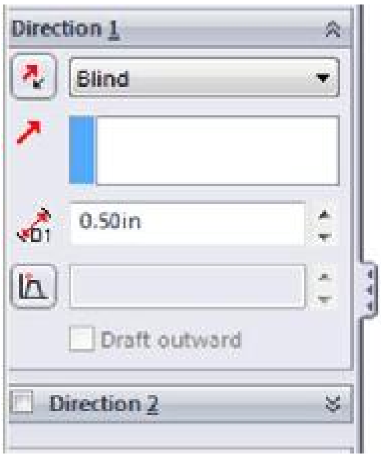
4. Use Line , sketch and connected open end of this sketch and make it close both end.

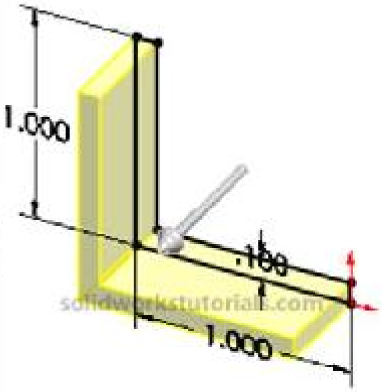




5. Click Features>Extruded Boss/Base 

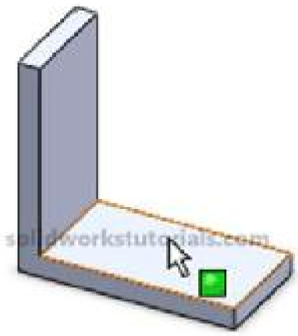
set D1 to 0.5in and OK.



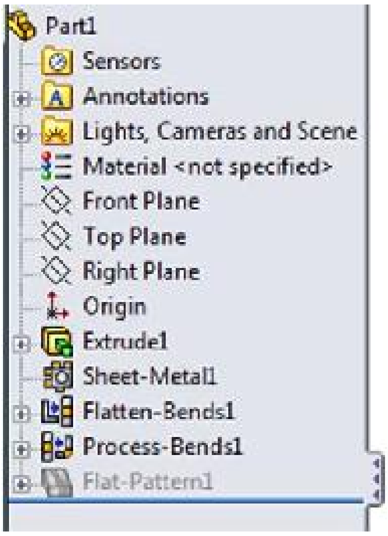


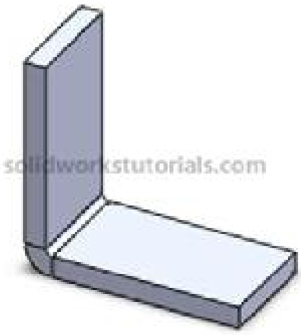
6. Click Sheetmetal>Insert Bends, 

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



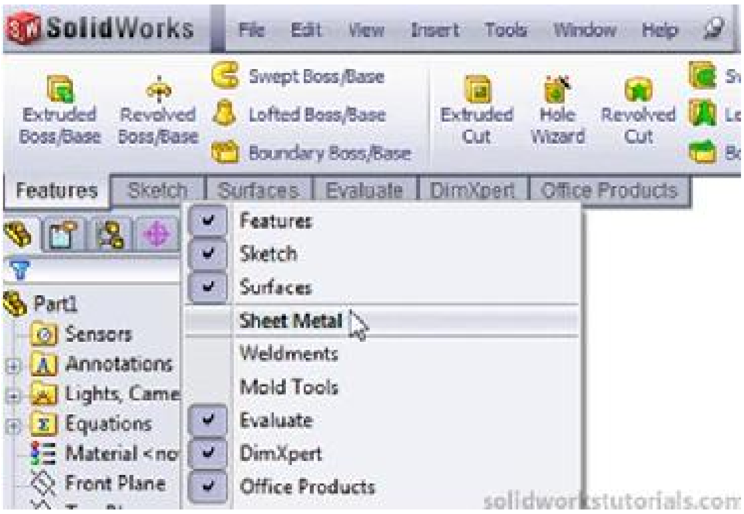
7. Your simple sheetmetal bend is ready. Look at part tree.



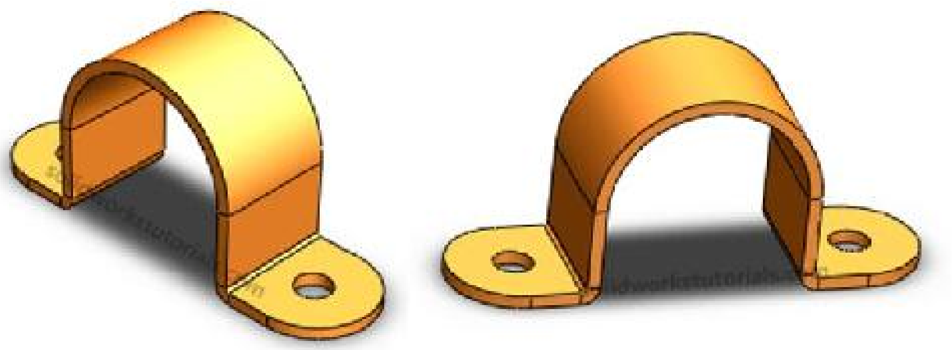


8. To view this part in flatten form
click Sheetmetal>Flatten.

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

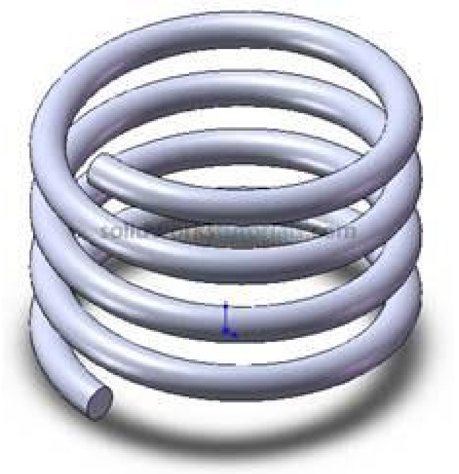


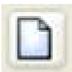
You know the basic, try model this bracket.



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

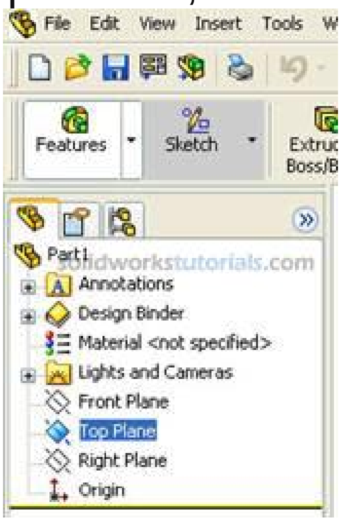
How to create spring



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

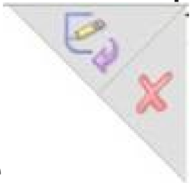
, OK .

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

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

4. Click Sketch 

in Command Manager, click Circle 

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

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

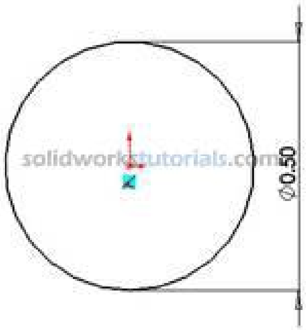
no need to be exact the size will define in later step. Press keyboard ESC to end circle sketch.

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

6. Define sketch with dimension. Click Smart Dimension 

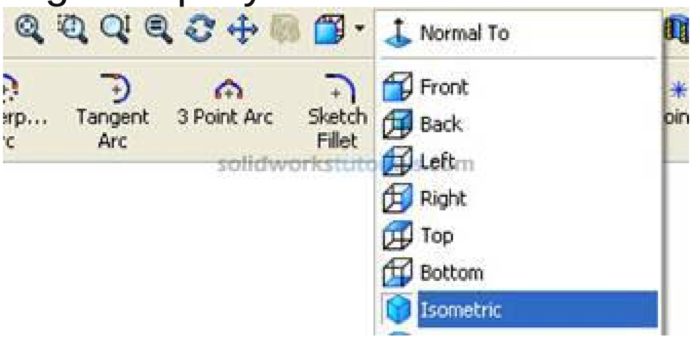
, and start dimensioning pick circle

edge and set to 0.50in



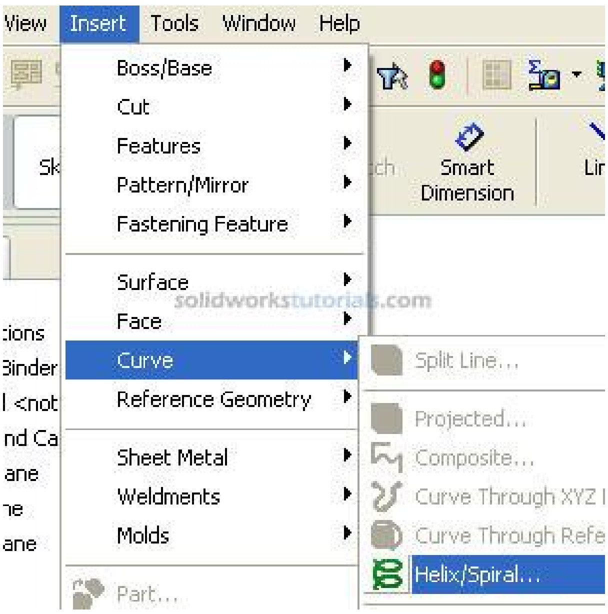
. Press keyboard ESC to end smart dimension.

7. Change display to Isometric



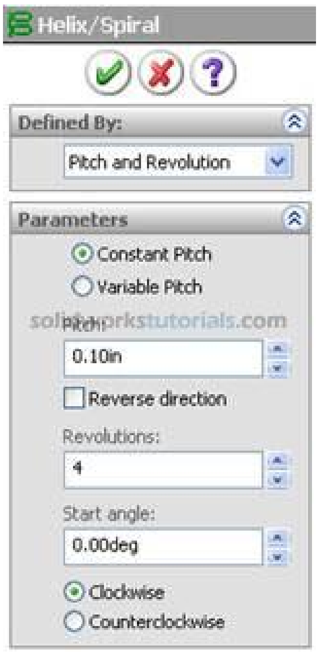
view.

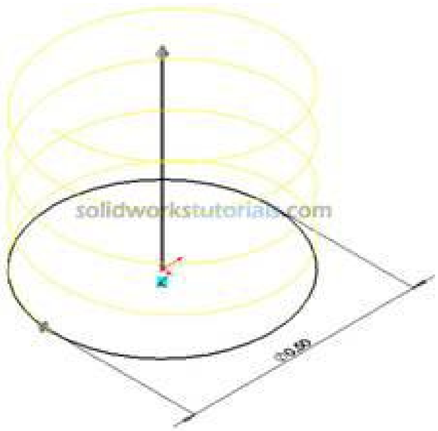
8. Insert coil, Click



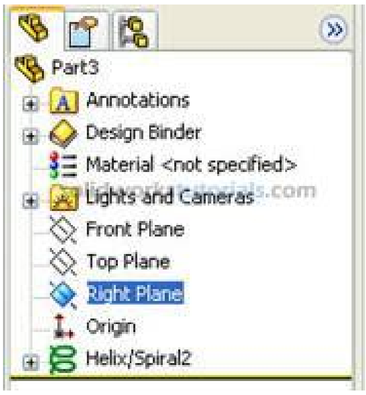
Insert>Curve>Helix/Spiral
.

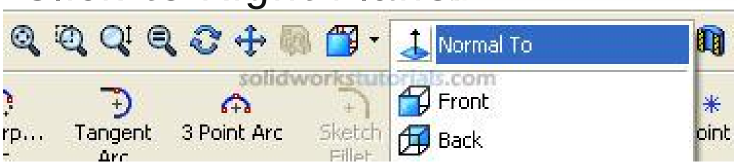
9. Press F to zoom fit, set Parameters Constant Pitch , Pitch 0.10in Revolutions 4 , Start angle 0.0deg





and  .

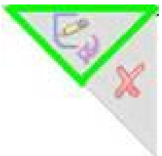
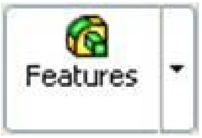
10. Click to Right Plane 

, click Normal To 

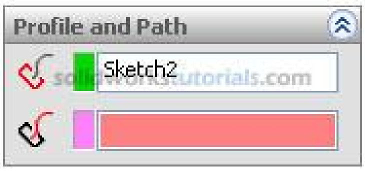
.

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

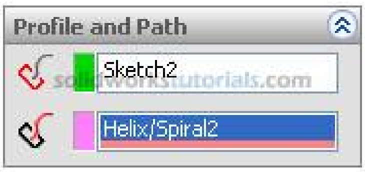
.

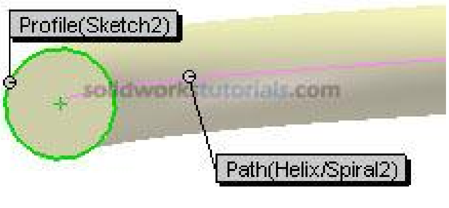
12. Click exit sketch . Click Features 

and activate features menu. Click Swept Boss/Base 

and set Profile to Sketch2 by click on circle sketch 

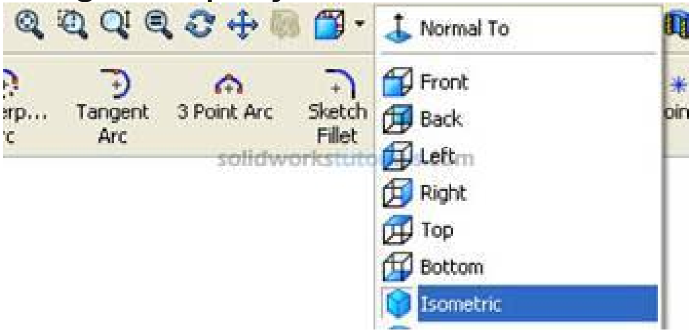
and set Path by click helix path





and .

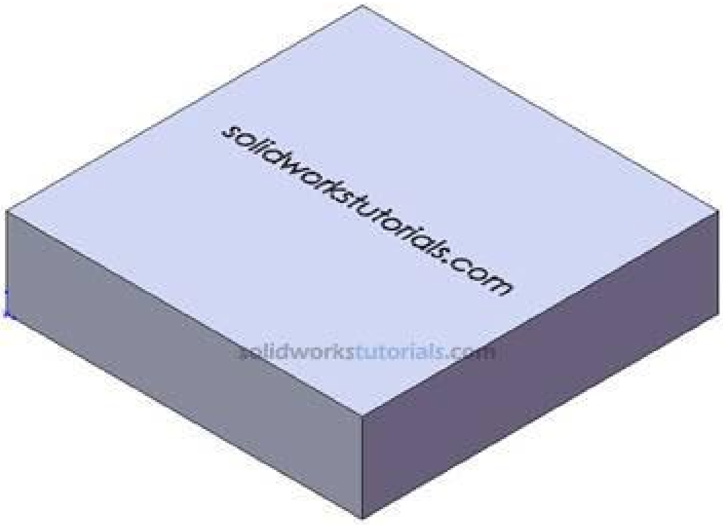
13. Change display to Isometric view.

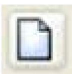


14. Press F to zoom fit.

Done. Pat yourself on back.

How to engrave text to part

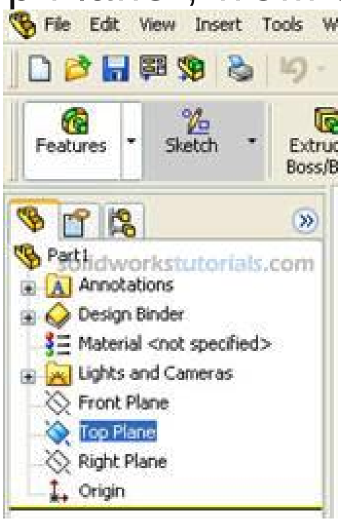


1. Click New  (File>New) , click Part 

, OK .

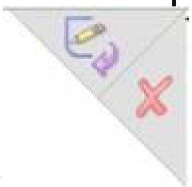
2. Click Option  (Tools>Option...) , select Document Properties tab. Select Units , under Unit System select IPS (inch, pound, second) OK .

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



4. Click Sketch 

in Command Manager, click Rectangle 

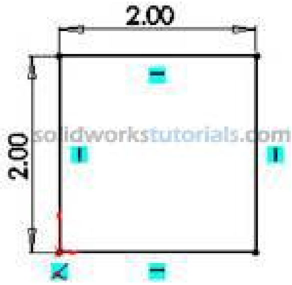
. As you can see on upper right corner sketch icon appear indicate that you’re on sketch mode .

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

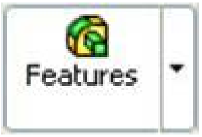
no need to be exact the size will define in later step. Press keyboard ESC to end rectangle sketch.

Note: There 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 2.00in , pick horizontal line and set to 2.00in 

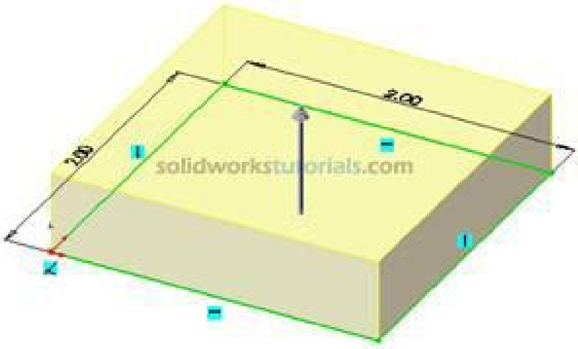
. Press keyboard ESC to end smart dimension.

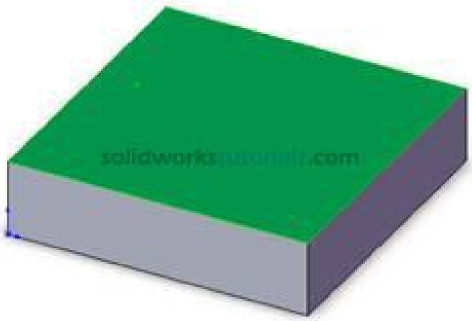
7. Build feature from sketch, click Features 

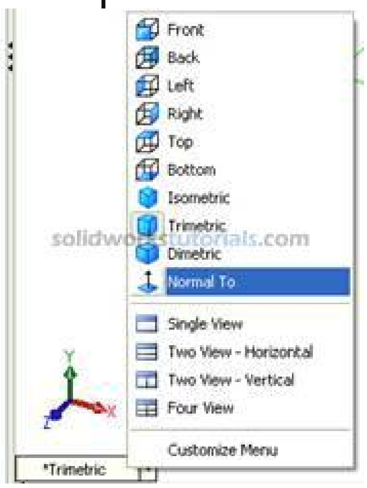
and activate features menu. Click Extruded Boss/Base 

and set D1 to 0.5in

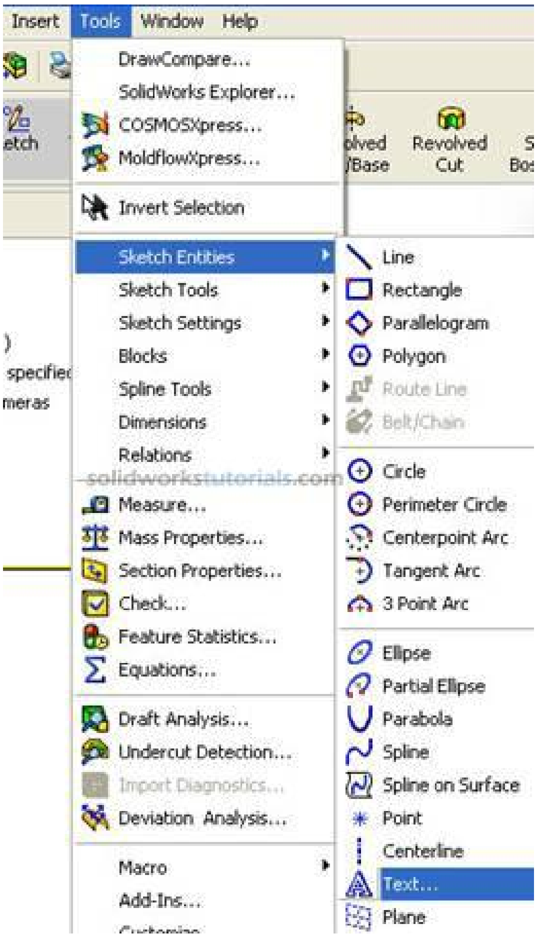
and .



8. Click front top face 

, click Normal To 

. Click Tools>Sketch



Entities>Text...

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

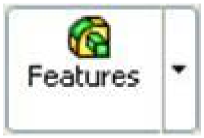
. Click

Font... 

set height to Points 10 OK.

10. Click to part face to relocate text to center

.

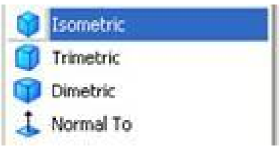
11. To engrave the text, click Features 

, click Extruded Cut 

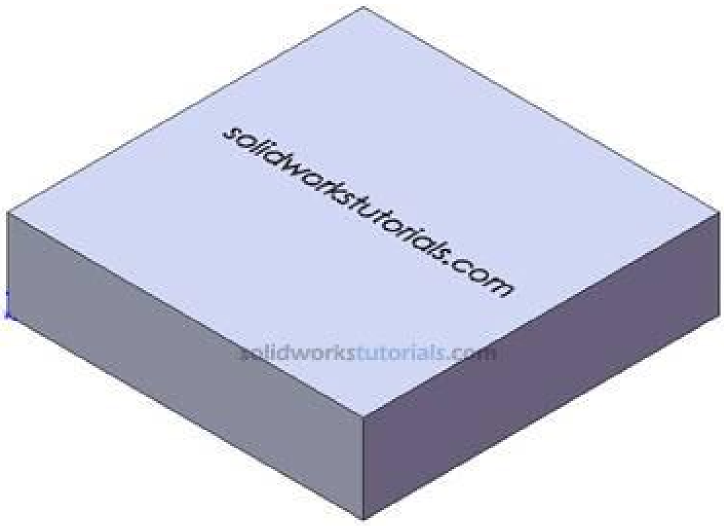
, under Direction 1 Blind, set D1

to 0.01in 

and . Click

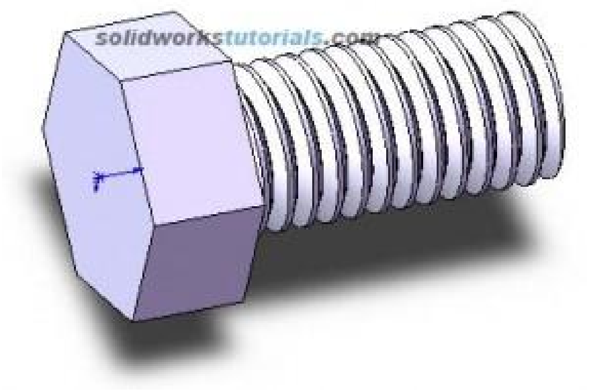
Isometric 

from lower left view menu.

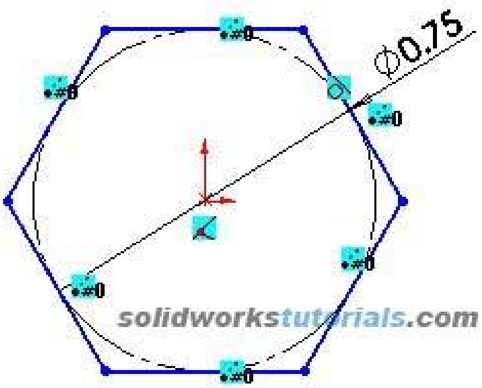


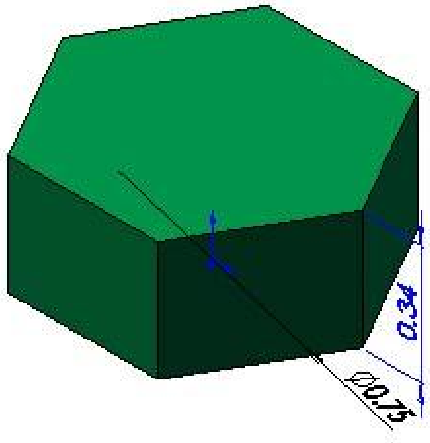
Done.

How to create hex bolt



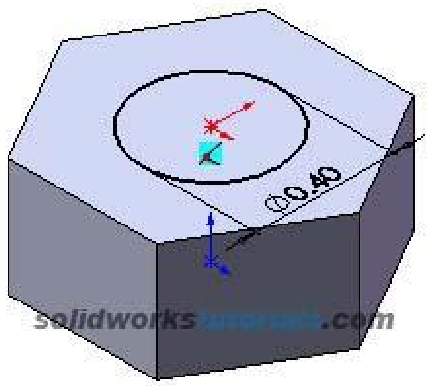
1. Sketch a polygon with 6 side, Tools>Sketch Entities>Polygon 

set diameter to 0.75in. 



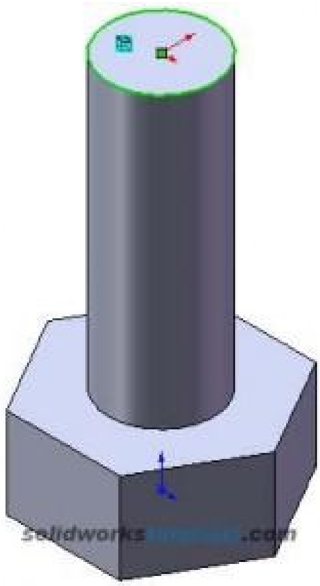
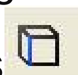
2. Extrude  sketch to 0.34in.

3. Create minor diameter for thread, sketch circle on



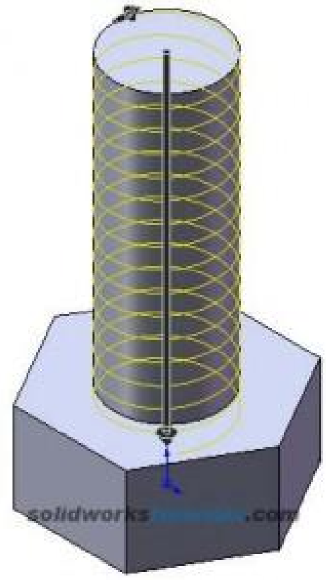
top face, set diameter to 0.4in.

4. Extrude sketch to 1.1in.

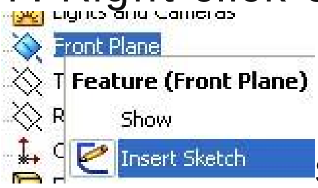
5. Click end edge of thread shaft,  click convert entities .

6. Select Helix/Spiral feature set height to 1.2in,

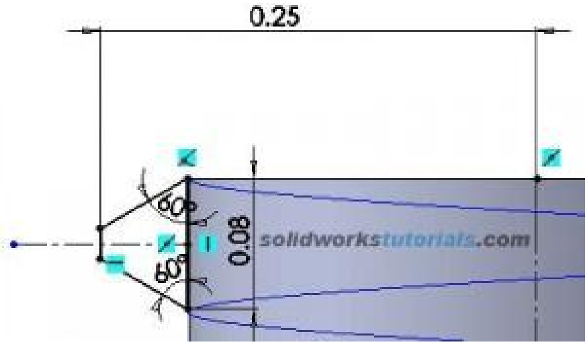
theap per inch=pitch 13/1in



Ok.

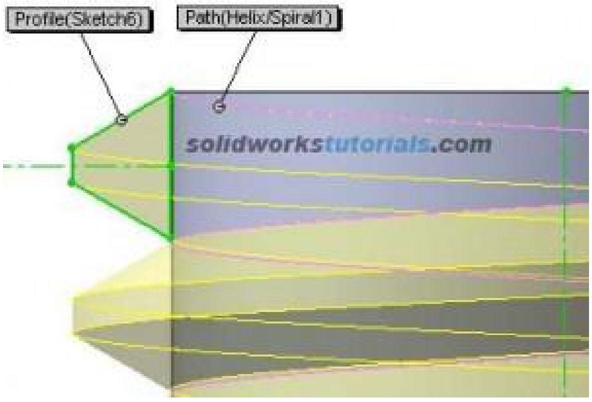
7. Right click on Front plane, Insert sketch 

sketch thead profile.

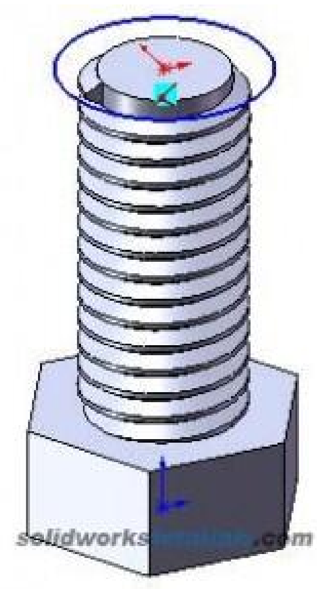


8. Click sweep feature , 

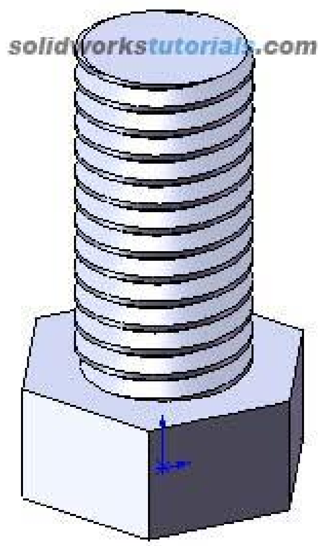
select sketch profuile as sketch and helix as a path, OK.



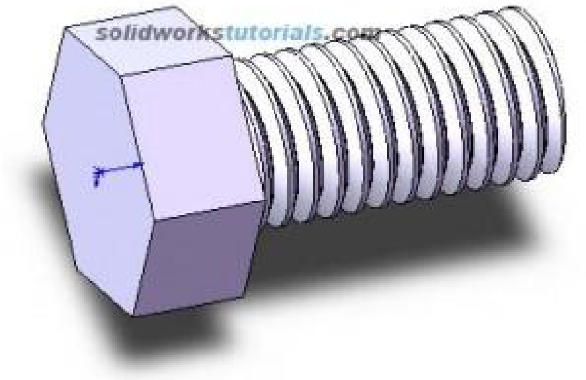
9. Create skecth a circle on the end shaft,



extrude cut 0.1in  .



10. Finish.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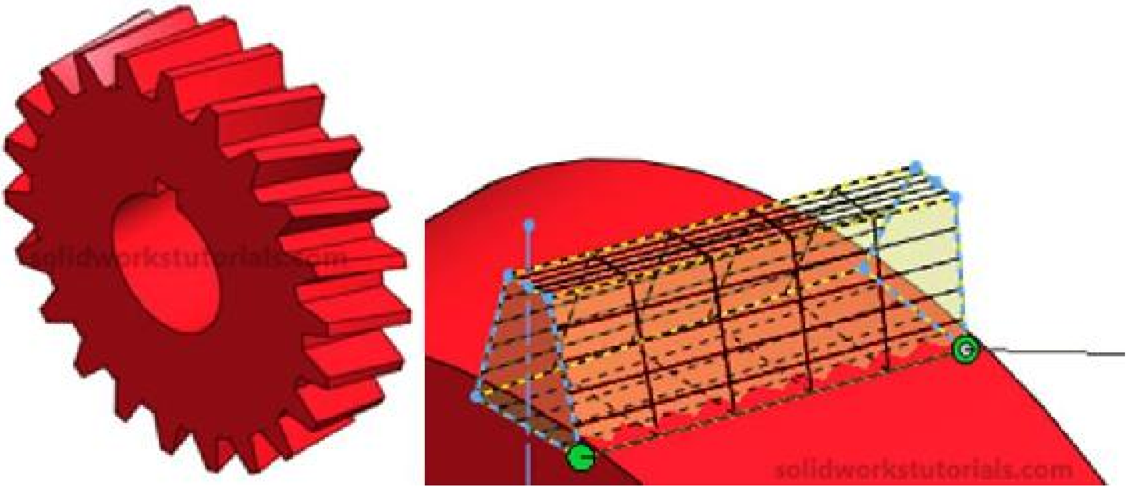
How to create helical gear



In this solidworks tutorial, you will create helical gear.

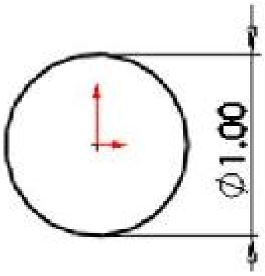
1. Click New.Click Part,

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

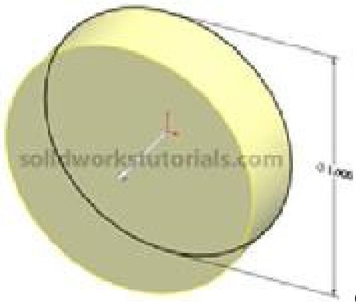
3. Click Circleand sketch a circle center at origin.

Click Smart Dimension,

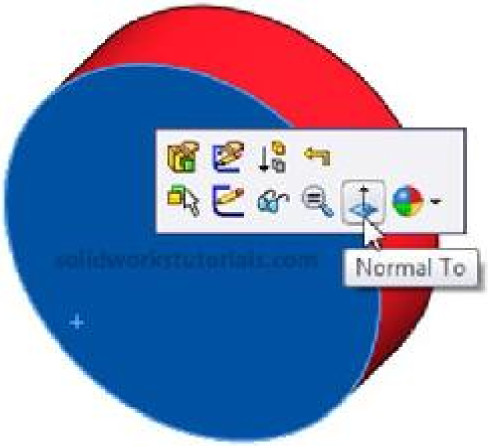
click sketched circle and set it diameter to 1.0in.



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

Set D1 to 0.3in  and .

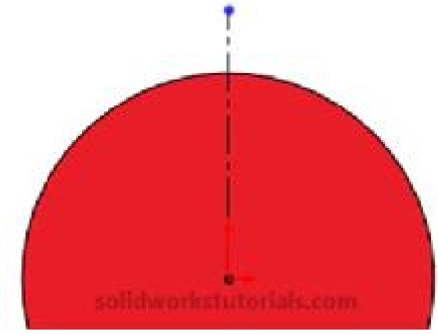
5. Click on front face and click Normal To.



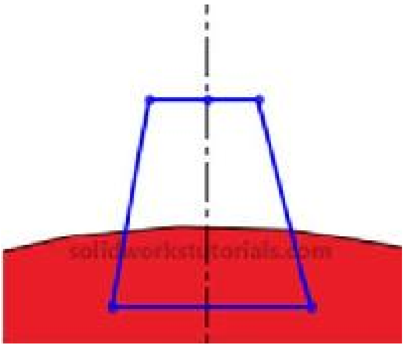
6. Click on front face and click Sketch.

7. Click on Centerline 

and sketch vertical Centerline.

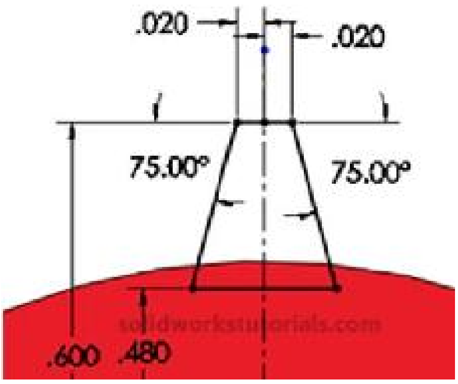


8. Click Line  and sketch gear teeth profile.

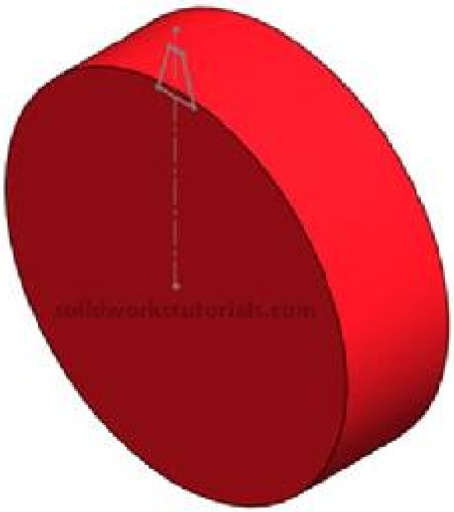


9. Click Smart Dimension, 

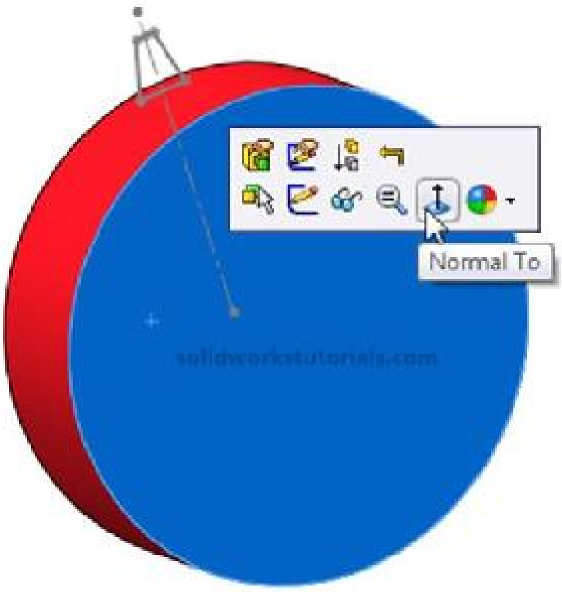
dimension sketch as sketched below.



10. Click Exit Sketch,  change view to Isometric.

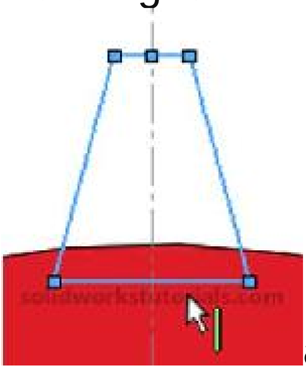


11. Click scroll mouse button and rotate the part to back side.



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

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



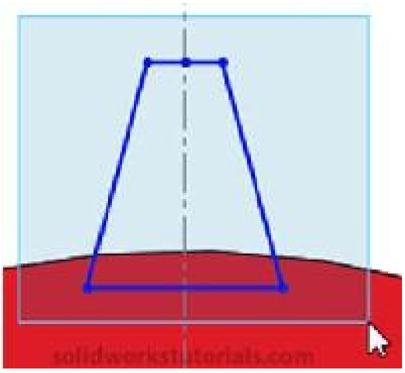
and click Convert Entities 

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

click Delete All

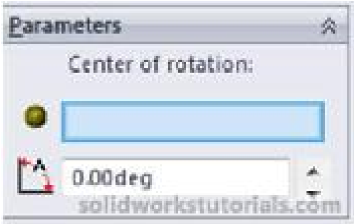
and  .

13. Click and drag select all the sketch line.

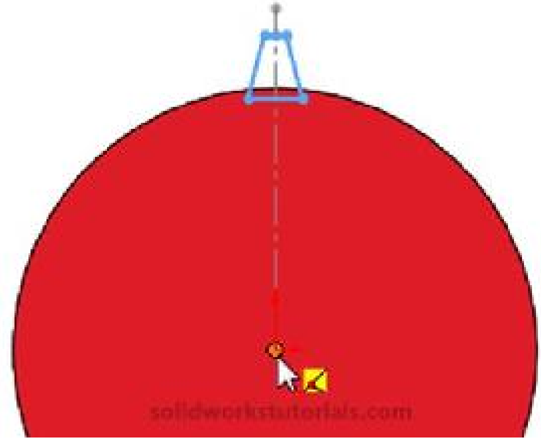


Click on Rotate Entities,

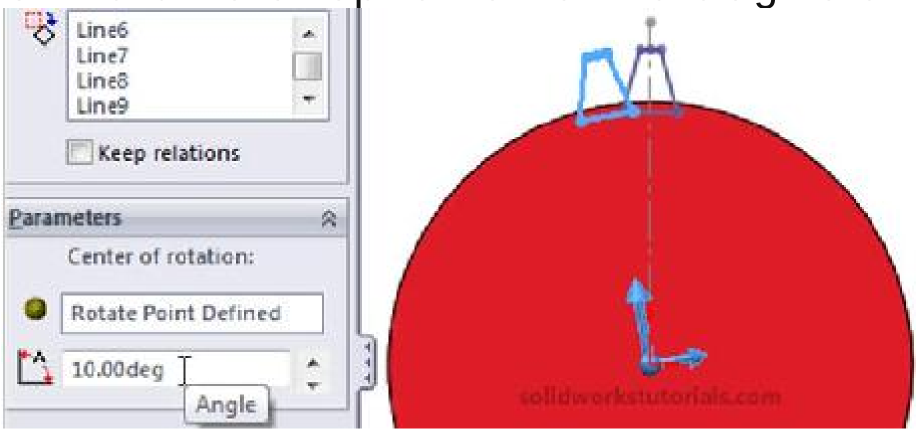
Click Center of Rotation box



and click origin (center part).

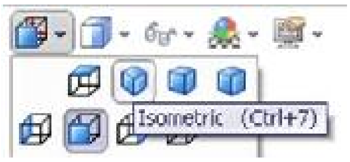


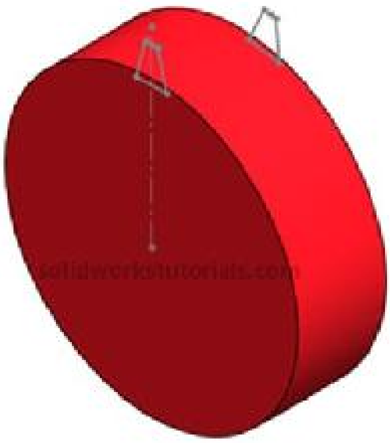
On Parameter option enter 10 deg rotation.



and  .

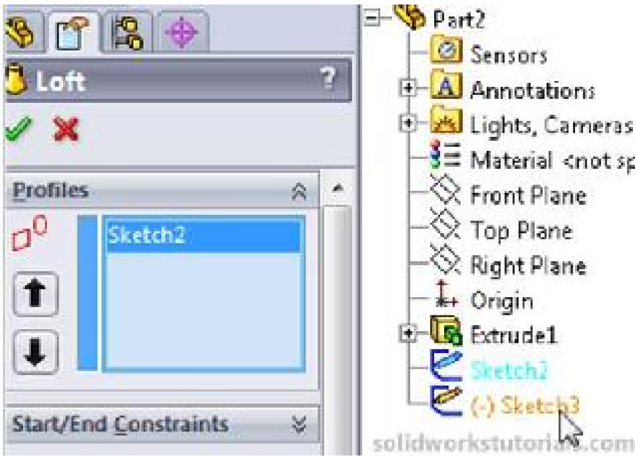
14. Click Exit Sketch,  change view to Isometric.

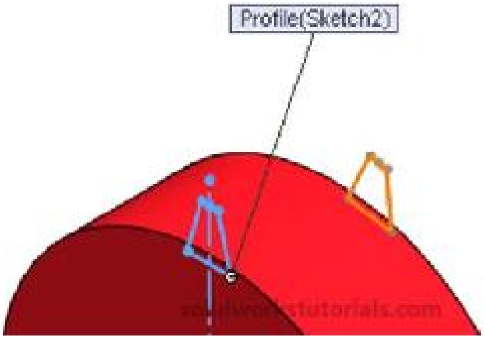




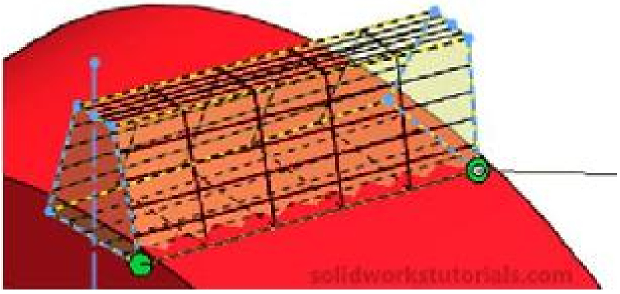
15. Click Features>Lofted Boos/Base,

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

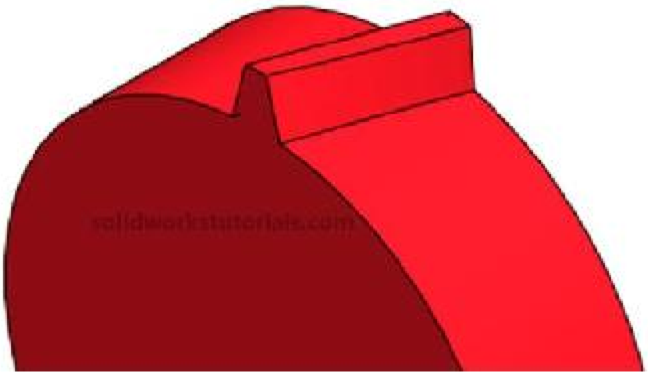


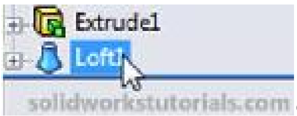


Make sure two green point is at the same edge as other sketch, if not drag and relocate it.

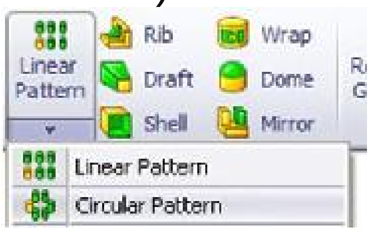


and  .

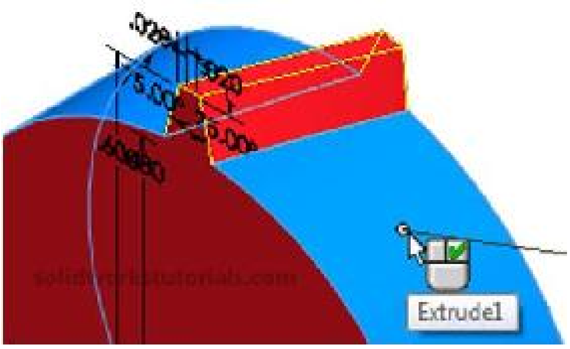


12. Click on Loft1 (gear teeth) 

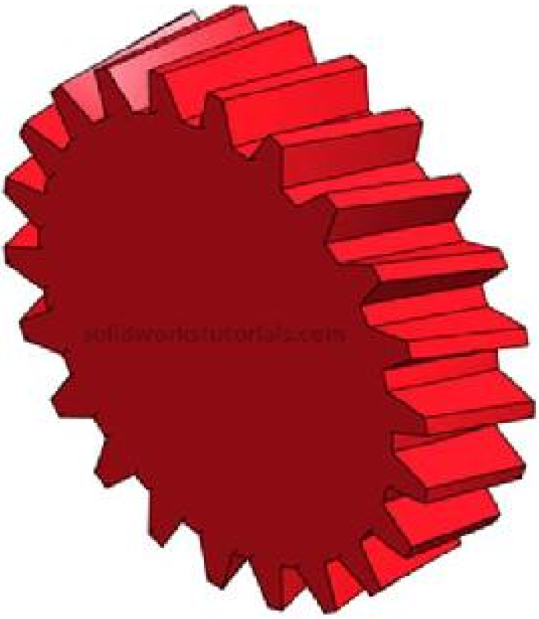
and

click Circular Pattern. 

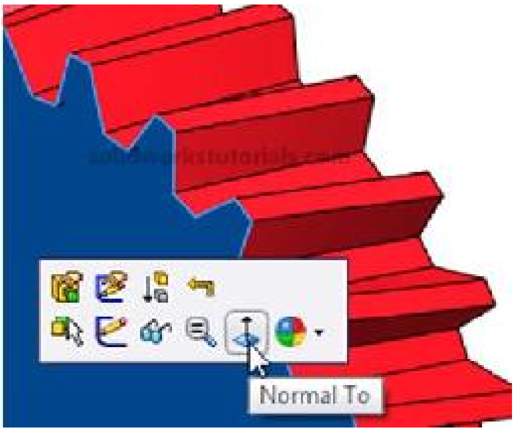
Click on the cylinder face as axis of rotation (or click on View>Temporary Axes select the temporary axis as axis of rotation).



Set Instances to 22 and .

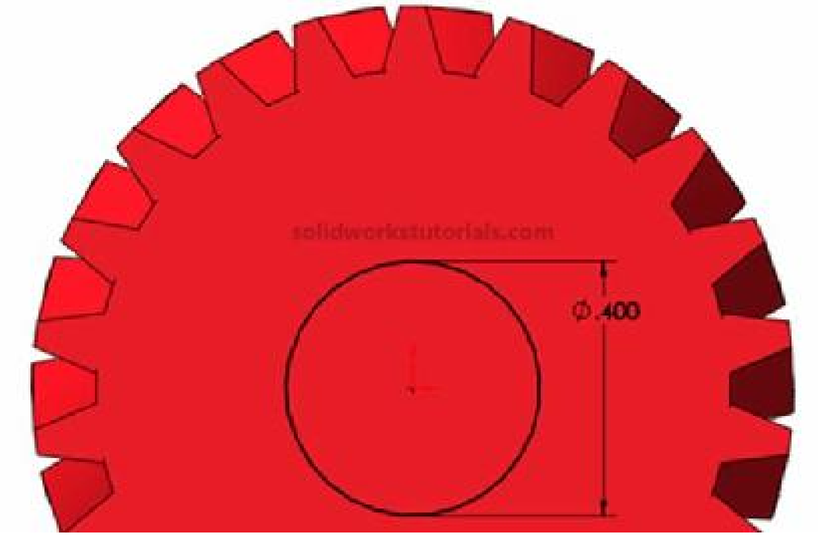


13. Click on Front face and select Normal To.



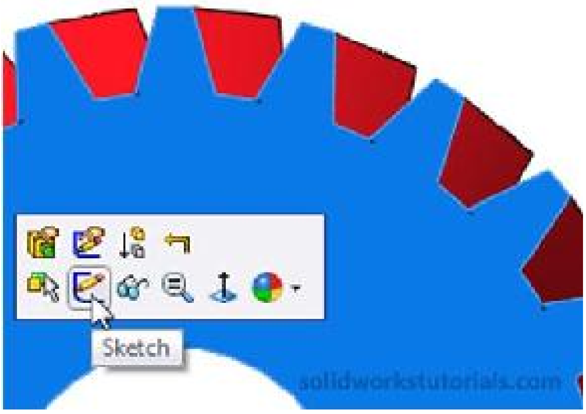
14. Click on front face and select Sketch.

15. Sketch a Circle and sketch a circle center at origin. Click Smart Dimension, dimension sketch as 0.40in circle.

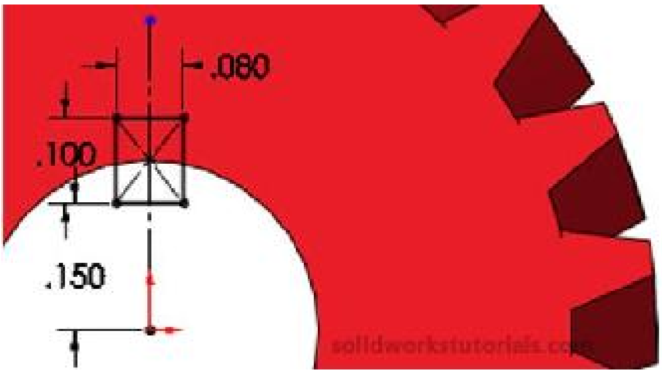


16. Click Features>Extruded Cut and set Direction to Through All and .

17. Click on front face and select Sketch.



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



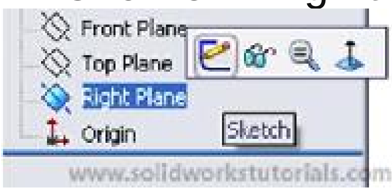
16. Click Features>Extruded Cut  and
set Direction to Through All and . You’re done!

How to create Airplane wings

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

1. Click New,  Part 

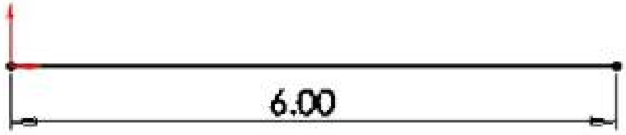
and OK.
2. Click on Right Plane and click Sketch.



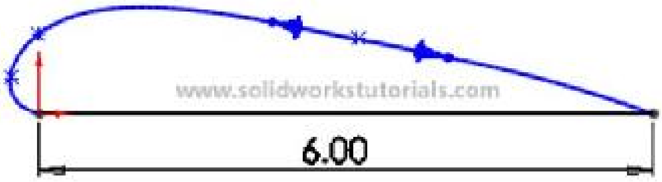
3. Sketch a center aerofoil profile at this plane.

Click Line,  sketch a horizontal line, click Smart Dimension 

and dimension the line as 6in.



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



Exit the sketch. 

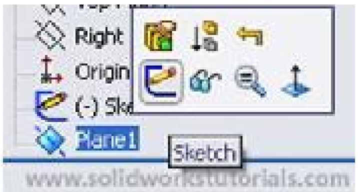
5. For another aerofoil profile at wing tip, you need to create another plane. Click onRight Plane

and click Reference Geometry>Plane

set distance between plane as 10in

and .

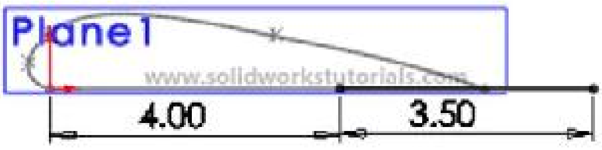
6. Click on Plane 1 and click Sketch.



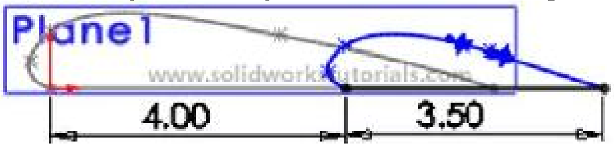
7. Click Line,  sketch a horizontal line on same level as first sketch a bit off set from origin,

clickSmart Dimension 

and dimension sketch as sketched below.

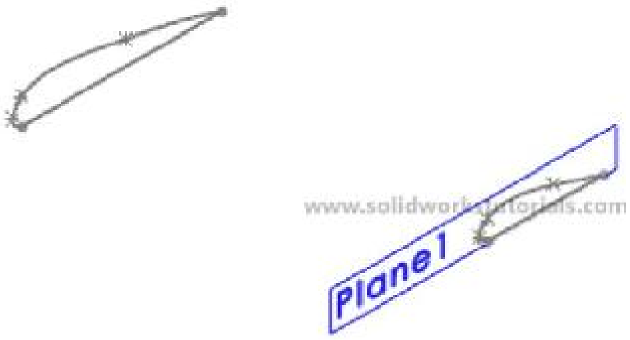


8. To create top curve of aerofoil, click Spline,  and sketch top curve as sketched below, to end Spline press Esc key.



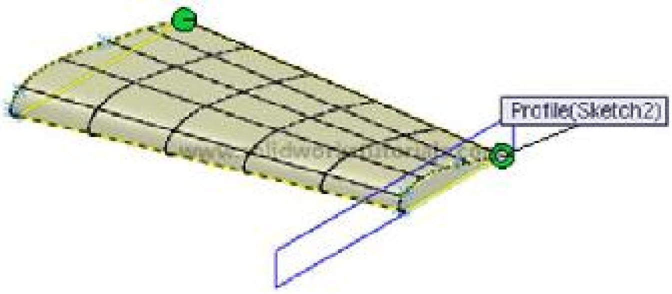
Exit the sketch. 

9. Click View Oreintation>Isometric.



10. Click Features>Lofted Boss/Base,

click Sketch1 and then Sketch2.



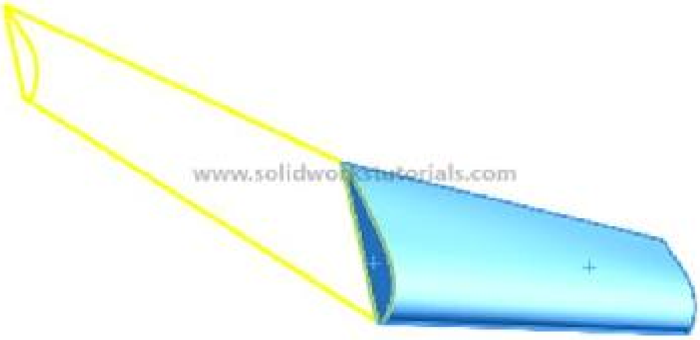
and  .

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

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

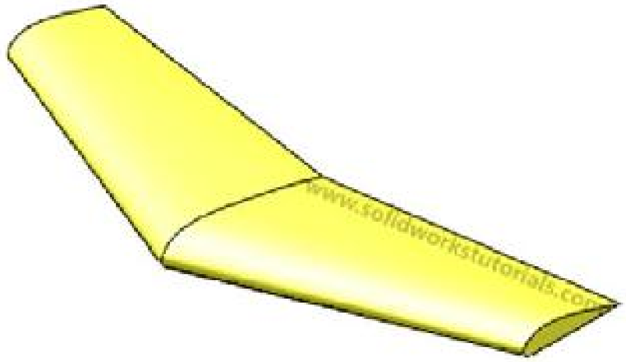
Turn the wings to right side and select center face as a Mirror Face/Plane.

Click on wing body as Features to Mirror

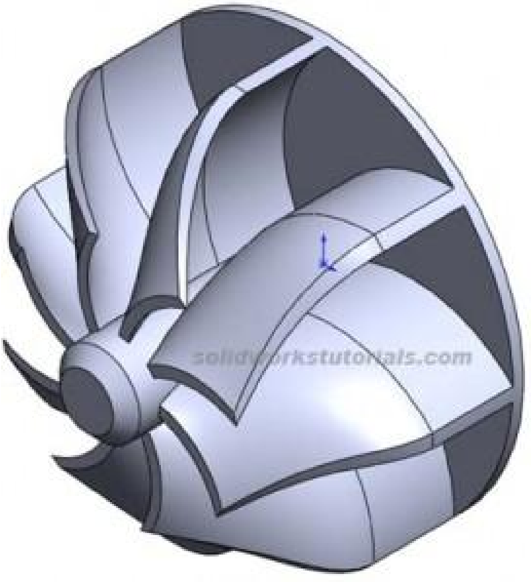


and .

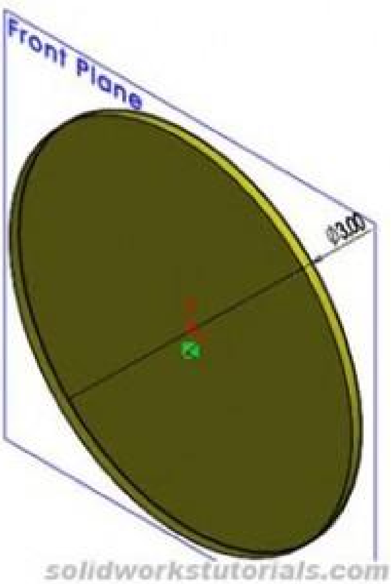
13. You're done.



How to create turbo fins

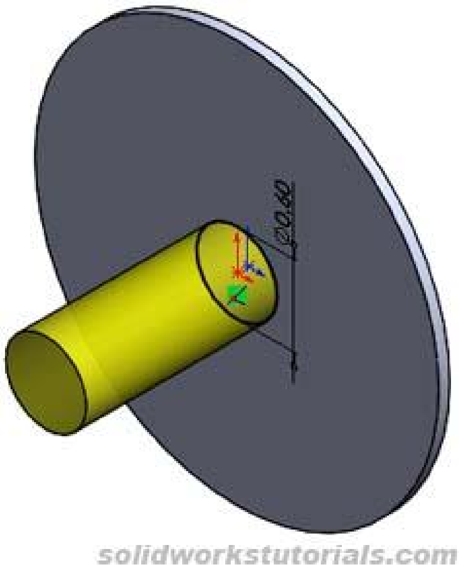


1. Skecth 3in circle and extrude to 0.08in on front



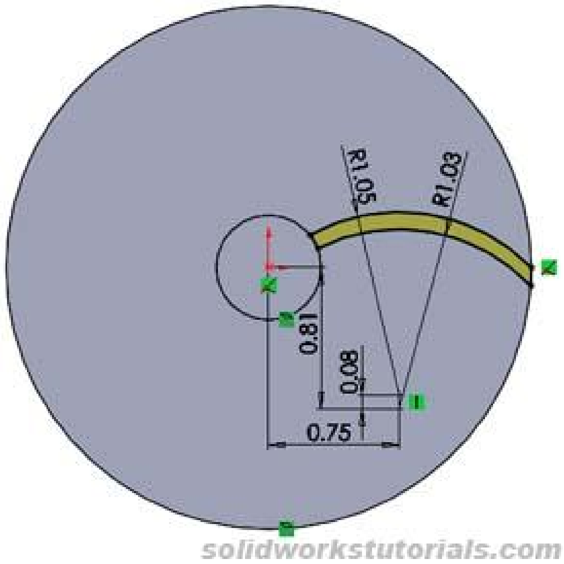
plane.

2. Skecth 0.6in circle on top extruded face and exrude

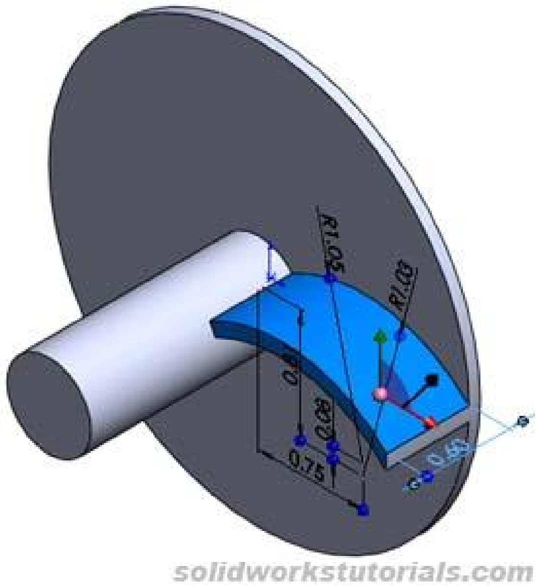


to 1.5in.

3. Sketch fin profile at extruded face as shown and

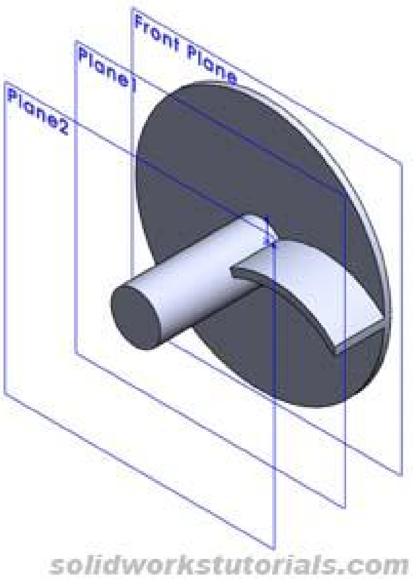


extrude to 0.6in.

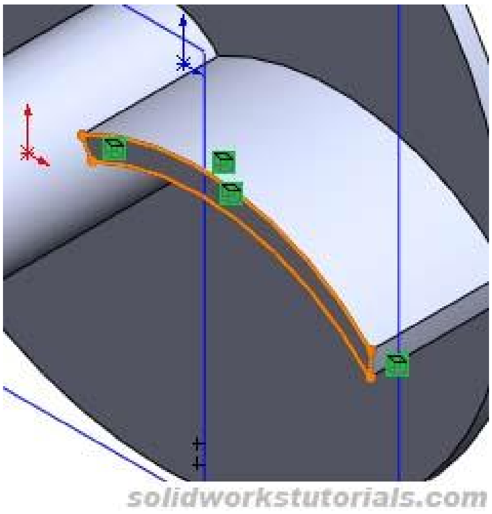


4. Add Plane 1 

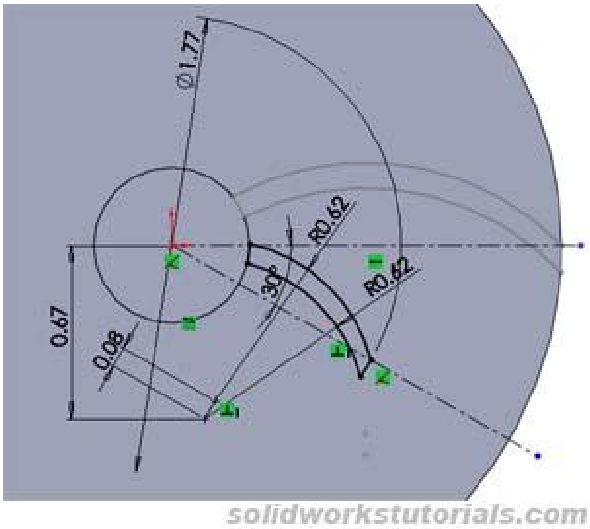
with 0.68in offset from Front plane and Plane 2 with 0.85in from Plane 1.



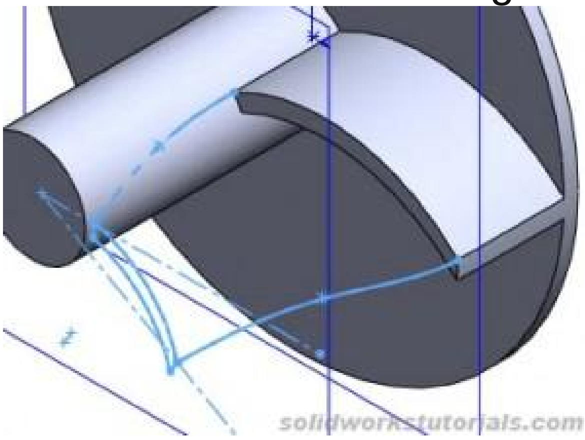
5. Insert sketch on Plane 1, select all edges to extruded fin and convert it to entities.



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



7. Sketch two curve line using 3D sketch tool, as

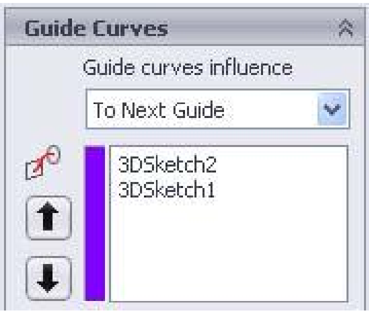


shown.

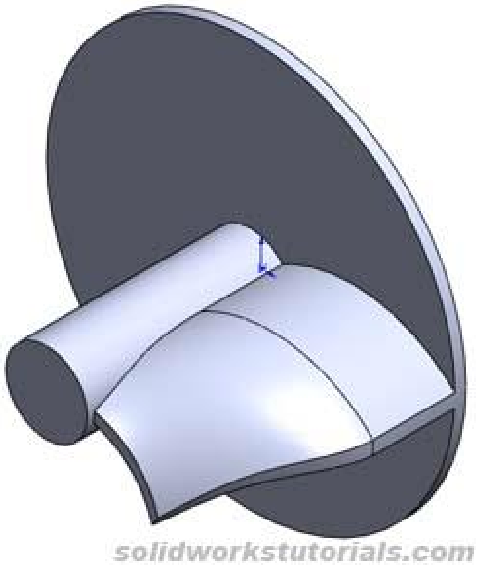
8. Click Lofted Boss/Base 

, select profile

Sketch 5 and sketch 6 and for guide curves select 3DSketch1 and 3DSketch2



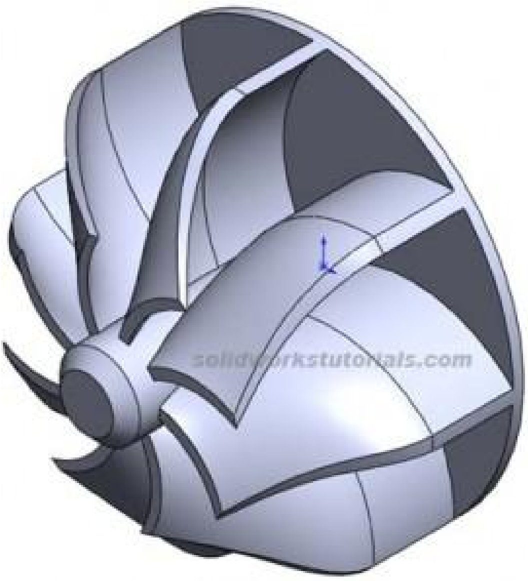
, OK.



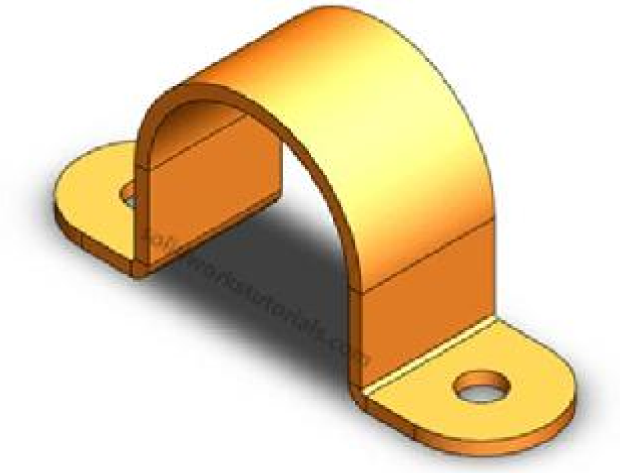
9. Click Circular Pattern 

, view temporary axis Tools>Temporary Axes. Select center axis, 360 degree, #8, Equal Spacing, OK

. Done!



How to create U bracket sheetmetal



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

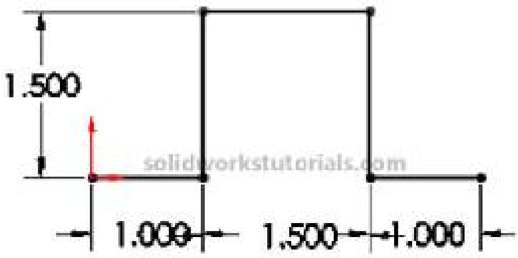
1. Click New.Click Part, 

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

Use Line, sketch U shape. Dimension sketch

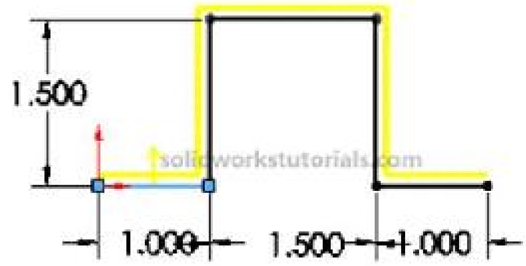
with Smart Dimension

as 1in x 1.5in x1in and 1.5in height.

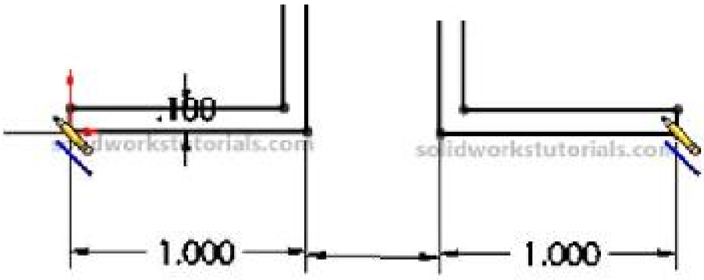


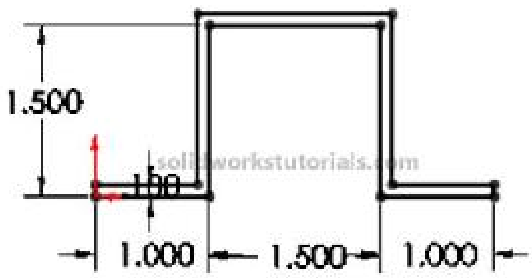
3. Click Offset Entities 

and click U sketch. Set offset distance as 0.1in, check Reverse box and OK.



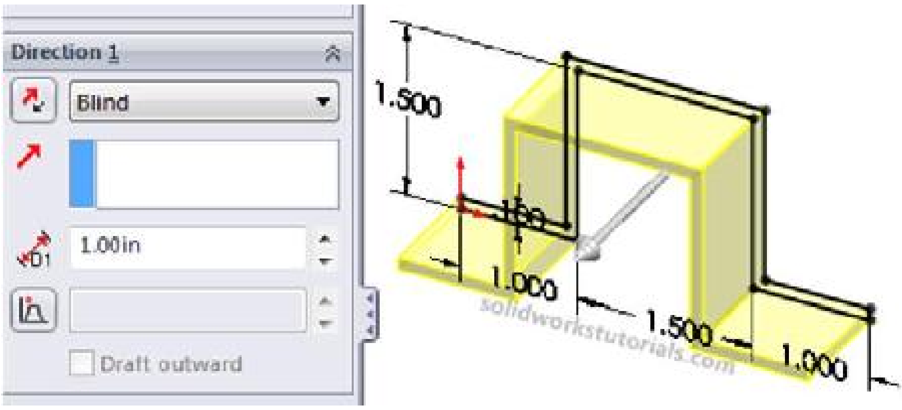
4. Use Line , sketch and connected open end of this sketch and make it close both end.

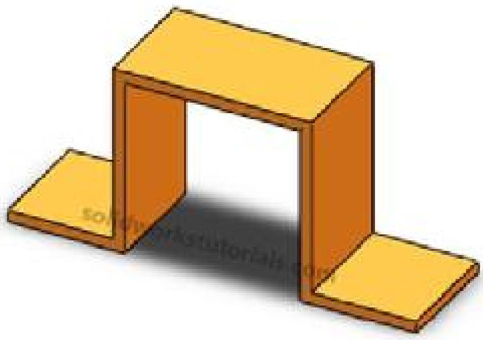




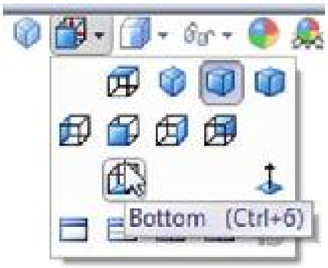
5. Click Features>Extruded Boss/Base 

set D1 to 1in and OK.





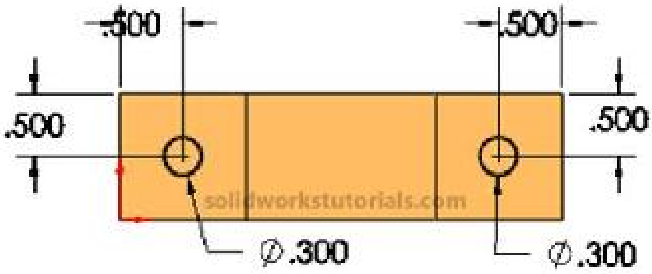
6. Click View>Bottom



click on bottom face and click Sketch.

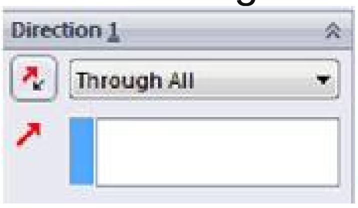
7. Click Circle  and sketch 2 circle on bottom face each side. Use Smart Dimension 

to dimension this sketch as sketched below.

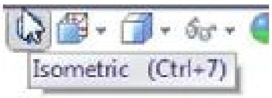


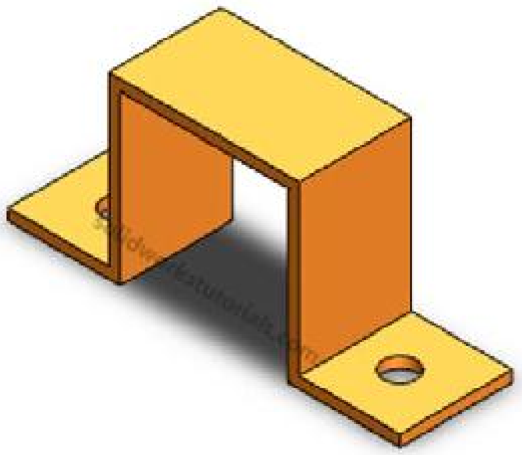
8. Click Features>Extruded Cut 

and cut Through All this circle.

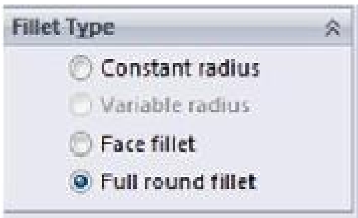


9. Click View>Isometric.

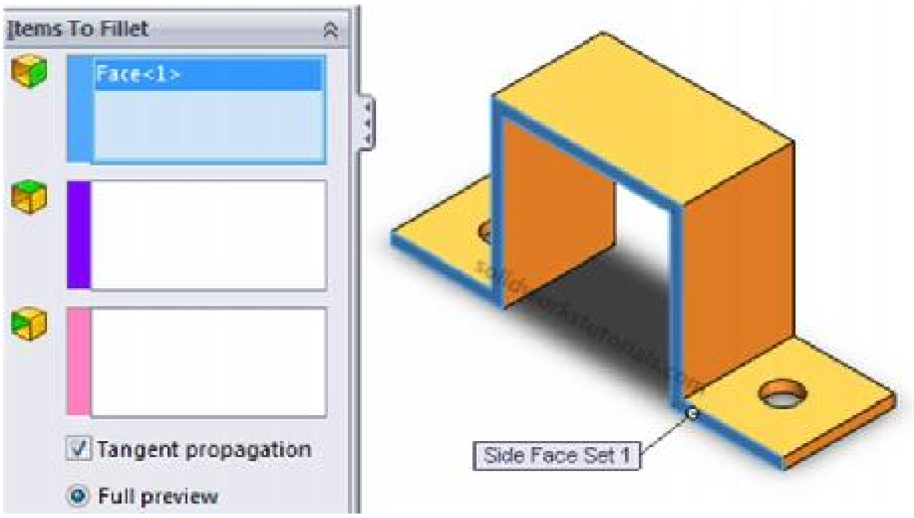




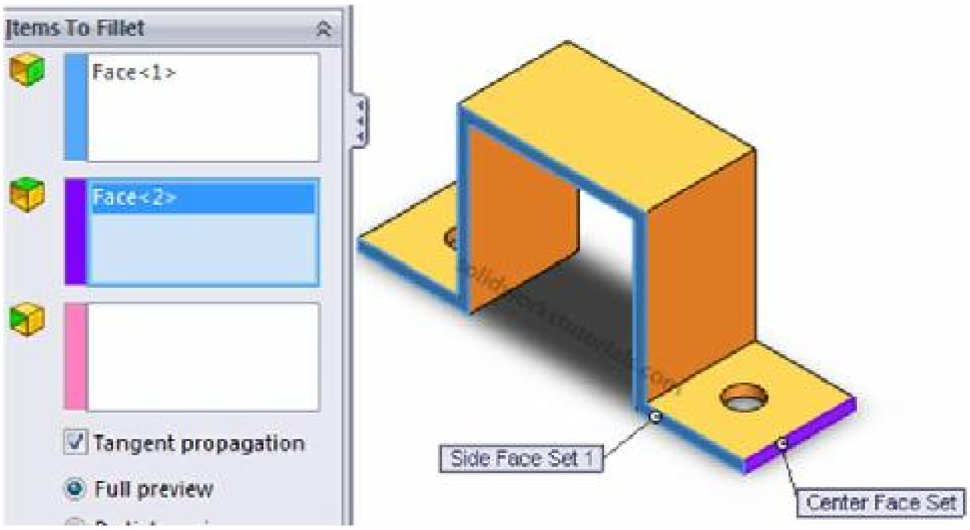
10. Click Fillet  , check box Full round fillet.



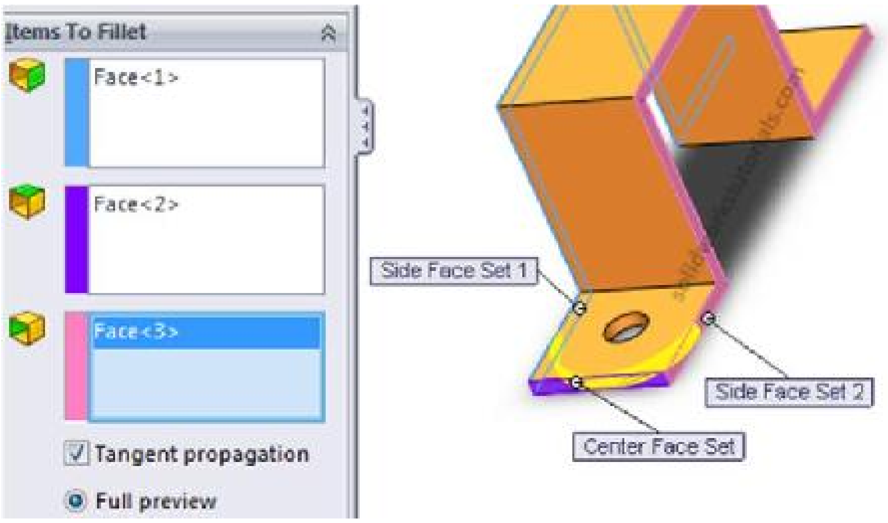
11. Click side left side face as Side Face 1.

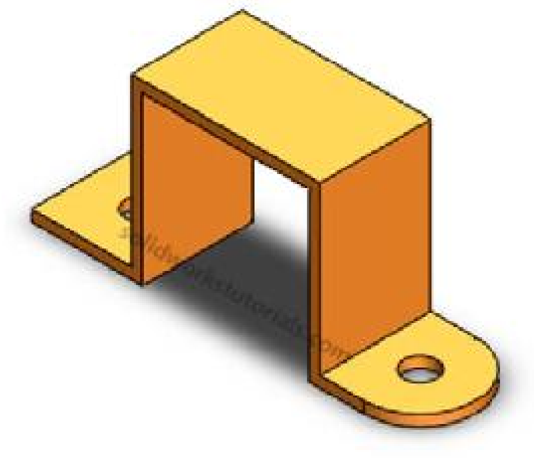


12. Click on purple box and click center face as Center Face Set.

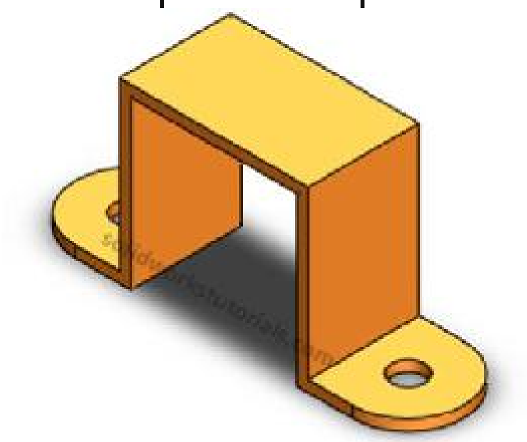


13. Click on pink box and click right side face as Side Face Set2 and OK.

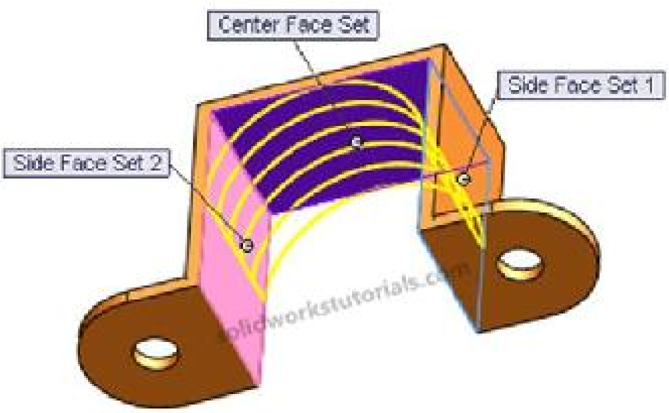


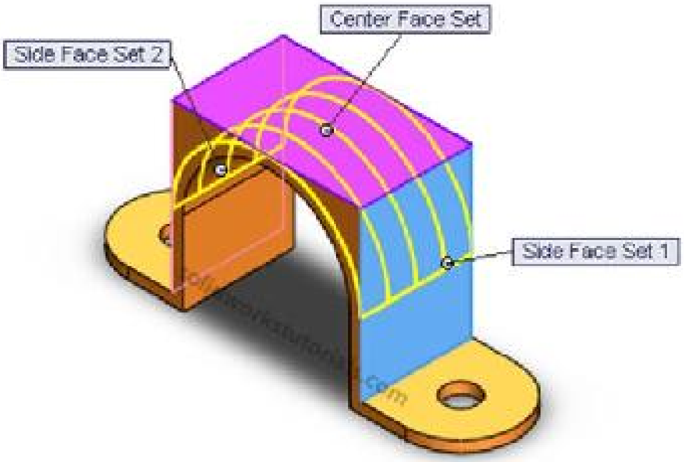


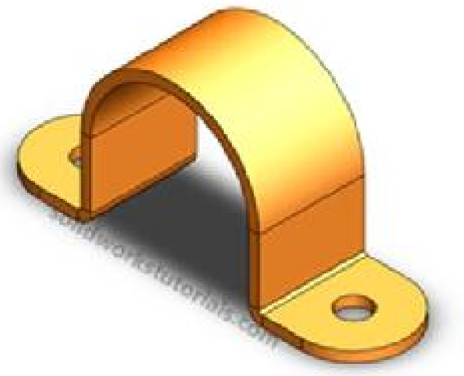
14. Repeat step 11 – 13 for the other side.



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

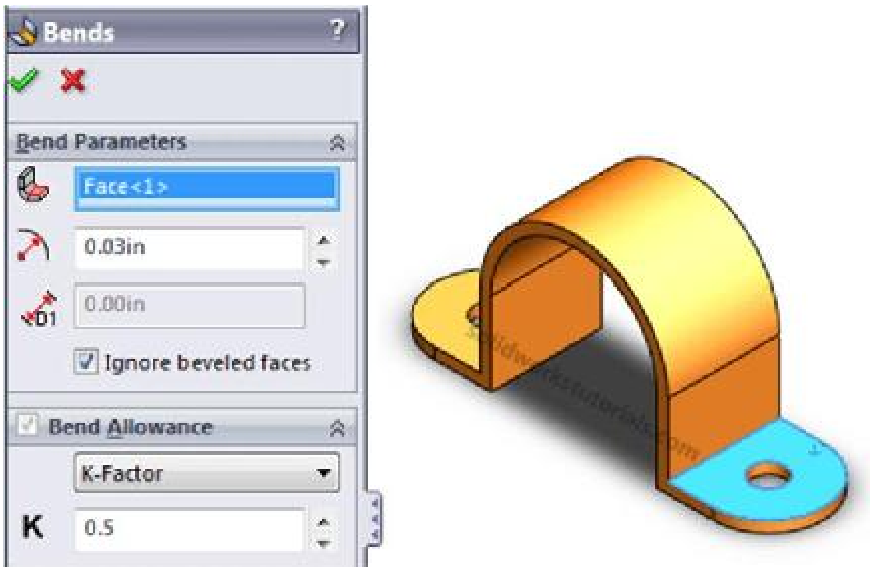




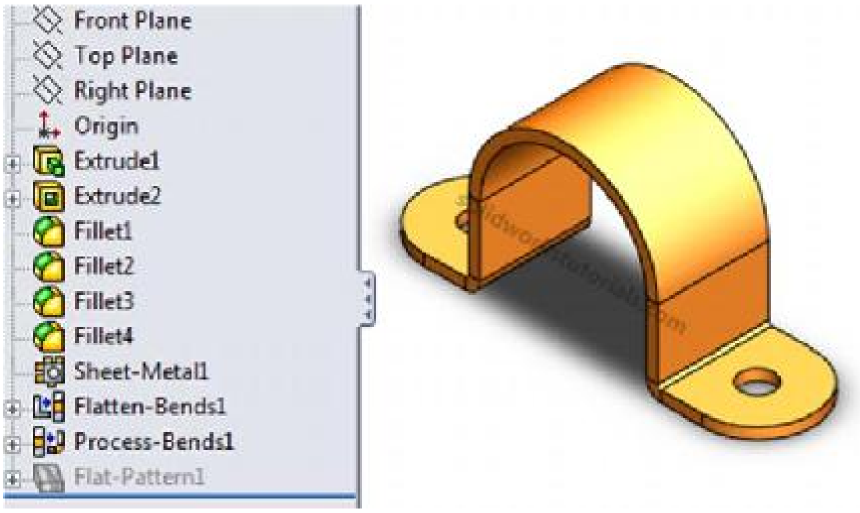


16. Click Sheetmetal>Insert Bends, 

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

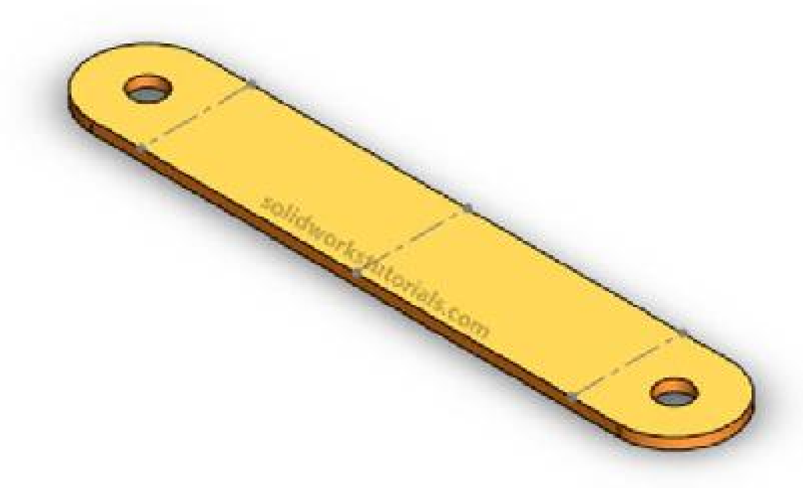


17. Your simple sheetmetal bend is ready. Look at part tree.

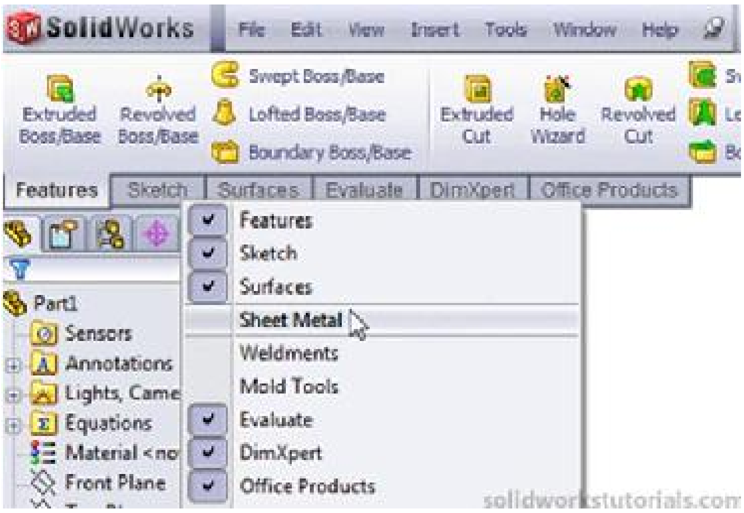


18. To view this part in flatten form click Sheetmetal>Flatten 

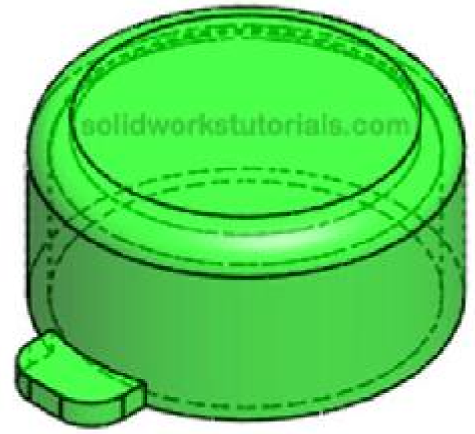
.



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



How to create bottle cap



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

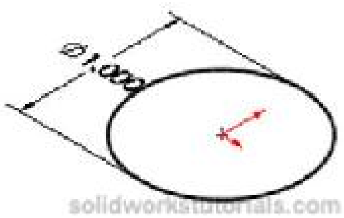
1. Click New , Click Part 

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

3. Click Circle  and sketch start at Origin,

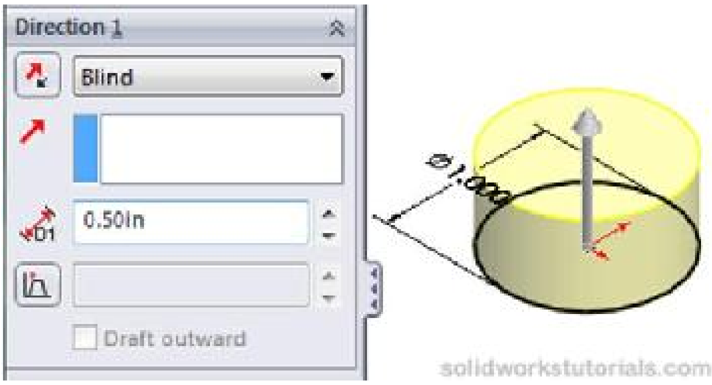
click Smart Dimension 

and dimension the circle as 1.0in diameter.



4. Click Features>Extrude Boss/Base 

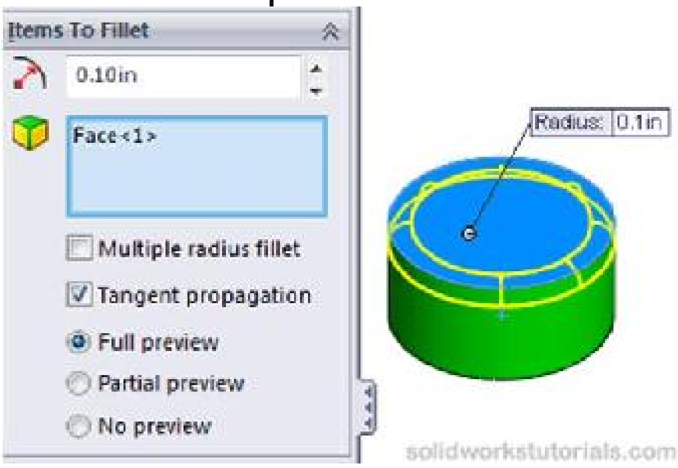
set the D1 to 0.5in



and .

5. Click Fillet 

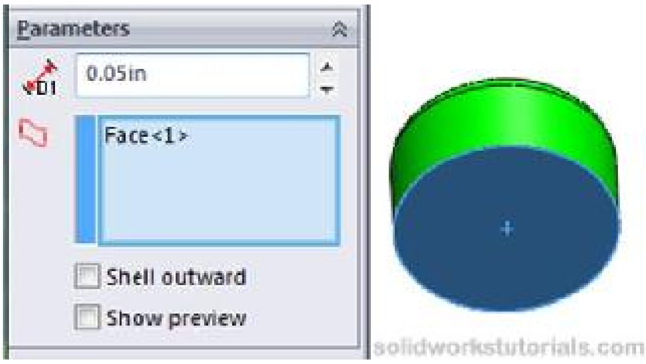
, set fillet size as 0.1in, select top face of the part



and .

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

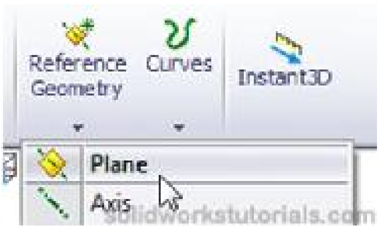
, select bottom face



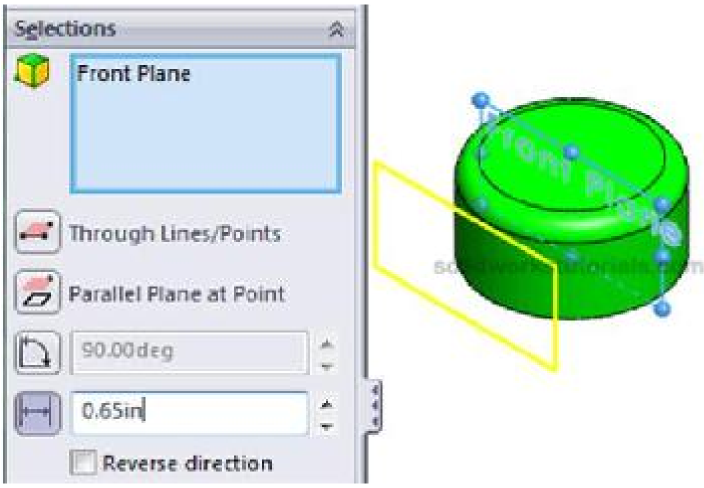
and .

7. Click Isometric View , click on Front Plane

and click on Reference Geometry>Plane.

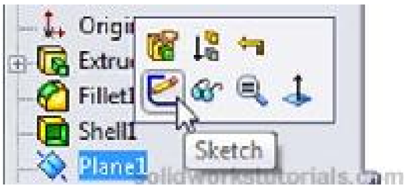


Set distance to 0.65in

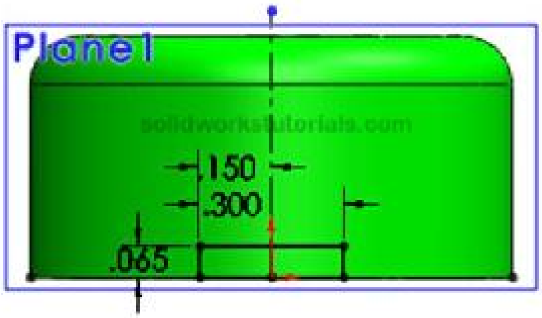


and  .

8. Click Plane1 and click Sketch.

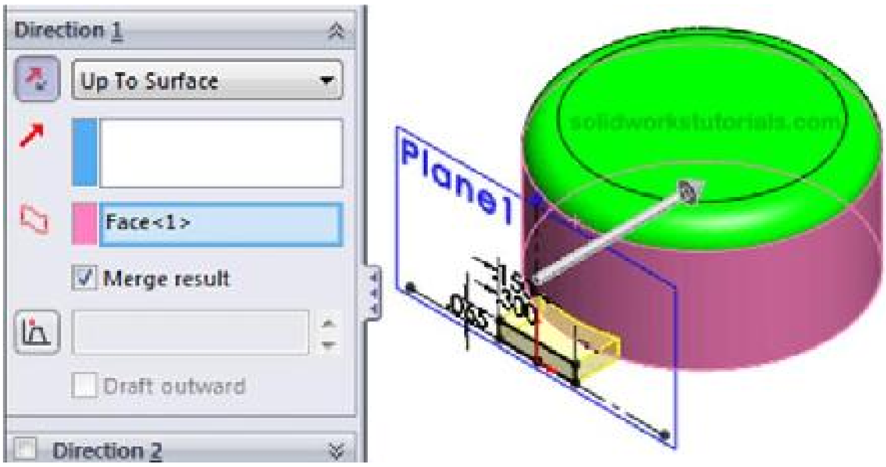


9. Click Rectangle  , sketch on Plane1 as sketched below and use Smart Dimension for your dimensioning.



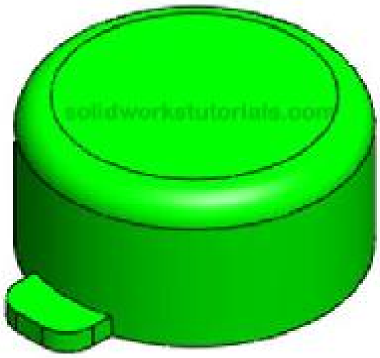
10. Click Features>Extrude Boss/Base 

set the Up To Surface



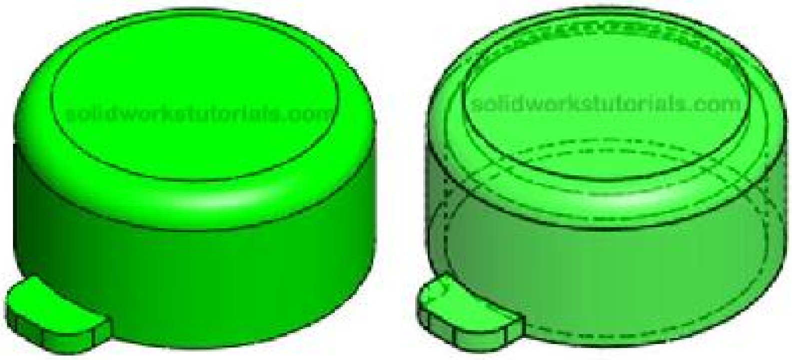
and .

11. Click Fillet , set fillet size as 0.1in, select side edge of the lid.



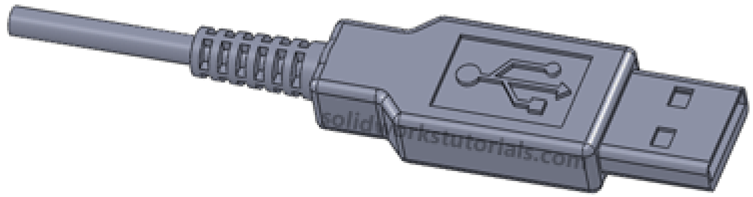
and .

12. And you're done!



Solidworks Tutorial #1: How to create USB head

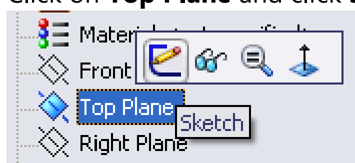
You can find almost all PC devices having USB feature, let's model one...



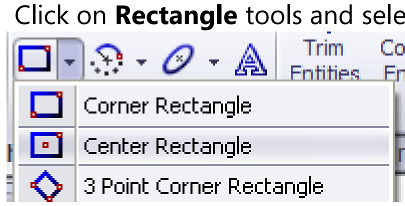
1. Click **New**,  click **Part**, 

OK.

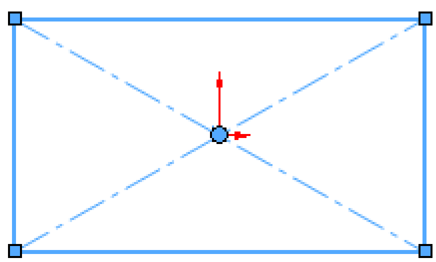
2. Click on **Top Plane** and click **Sketch**.



3. Click on **Rectangle** tools and select **Center Rectangle,**

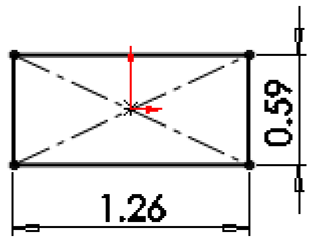


Click on **origin** as it center and **sketch a rectangle**.

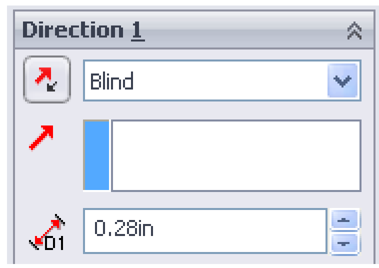


4. Click on **Smart Dimension** 

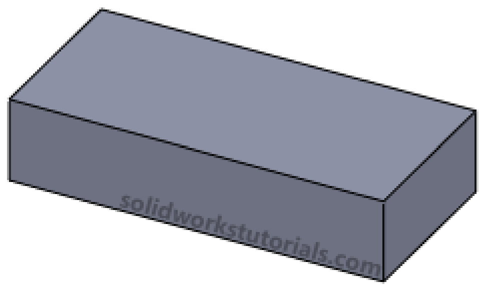
and click on side edges of rectangle to give dimension to the rectangle as **0.59" x 1.26"**.



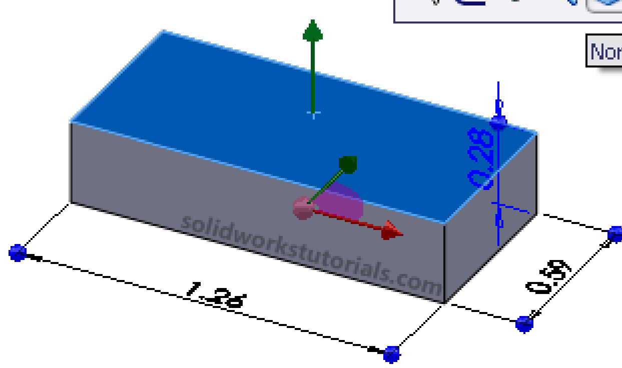
5. Click **Features**>**Extruded Boss/Base**,  on **Direction 1** set **D1** to **0.28".**



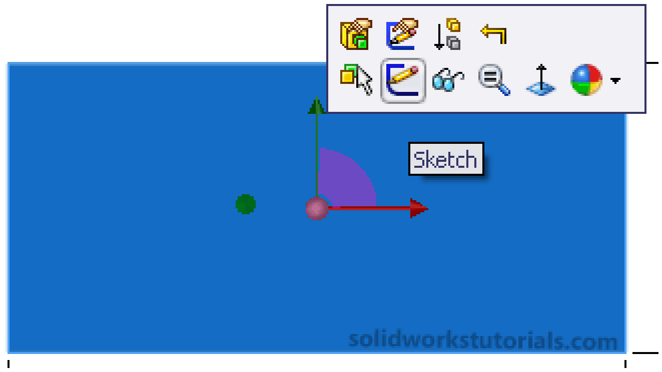
and .



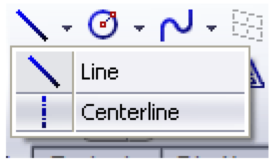
6. Click on **top face** of this block and click **Normal To**.



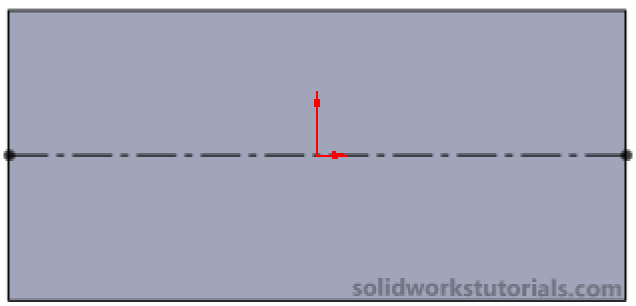
7. Click on **top face** again and click **Sketch.**



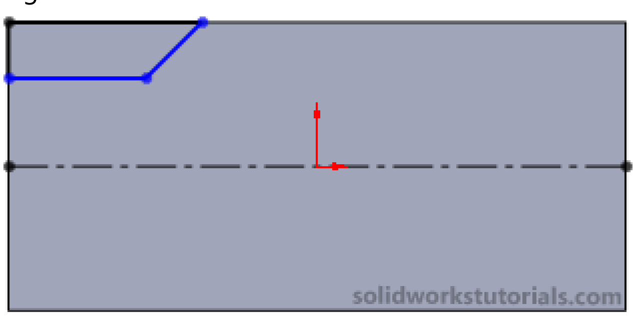
8. Click on **Line** tools and select **Centerline**,

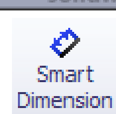


sketch a centerline through **left edge midpoint** to **right edge midpoint**.

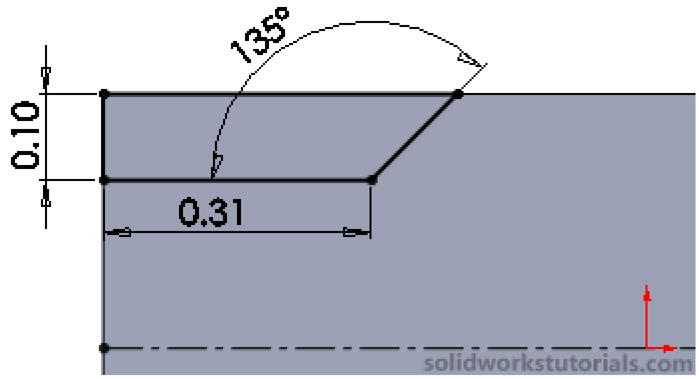


9. Click on **Line**  and sketch a cut section on upper left edge as sketched below.

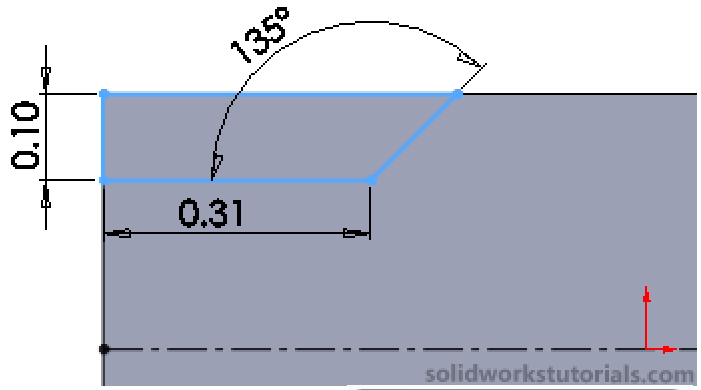


Click on **Smart Dimension** 

and dimension the sketch as sketched below.

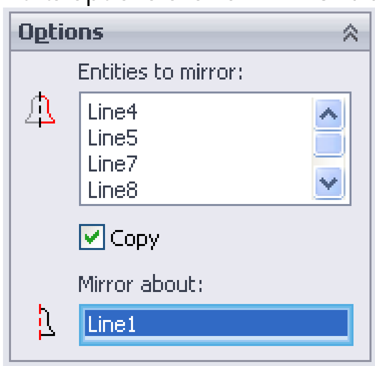


10. Select **cut section sketch**

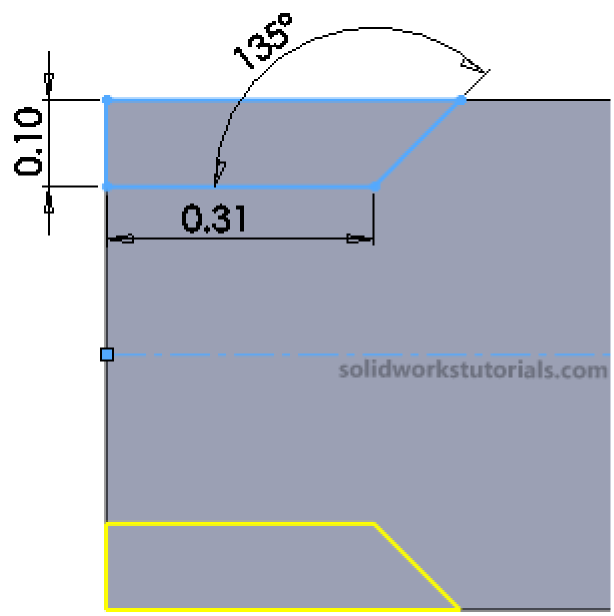


and click on **Mirror Entities**. 

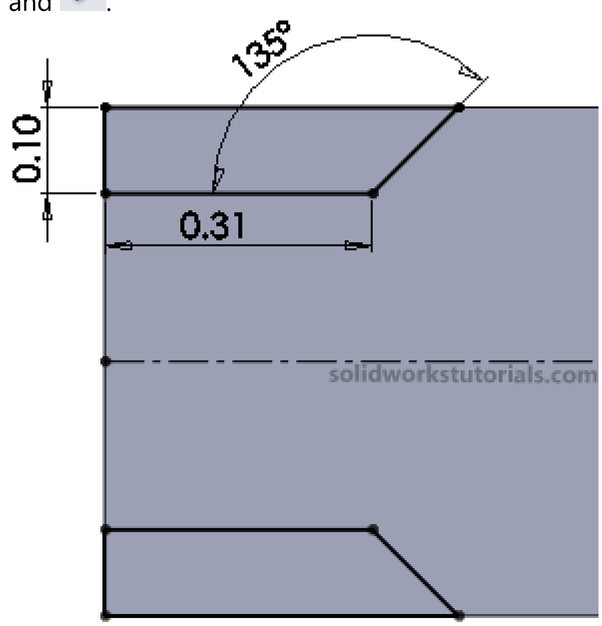
At its options click on **Mirror about:** box



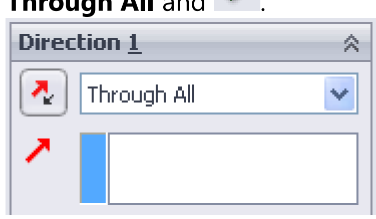
and click on **centerline**.

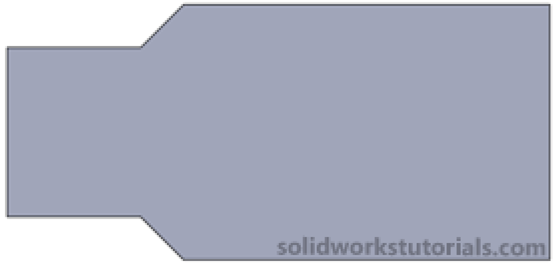


and .

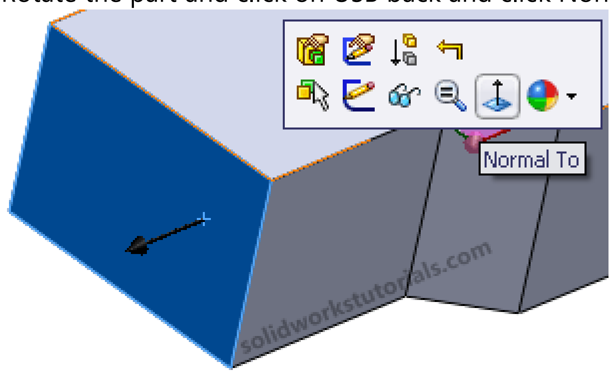


11. Click on **Features**>**Extruded Cut**  and set **Direction 1** to **Through All** and .

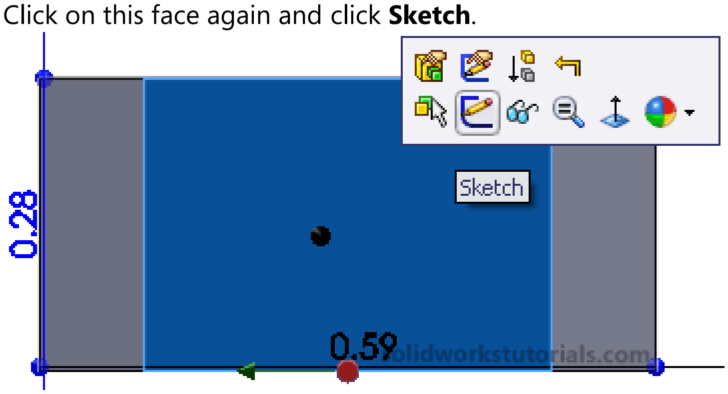




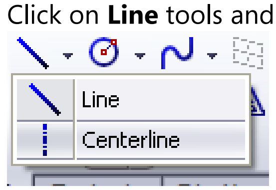
12. Rotate the part and click on USB back and click Normal To.



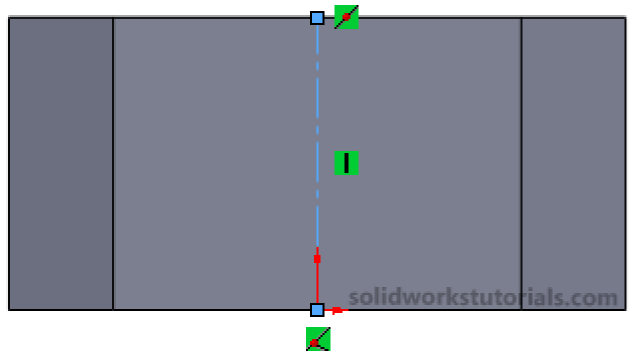
13. Click on this face again and click **Sketch**.



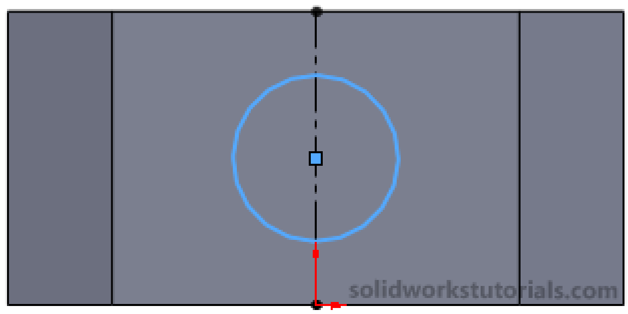
14. Click on **Line** tools and select **Centerline**,



sketch a centerline through bottom edge midpoint to top edge midpoint.

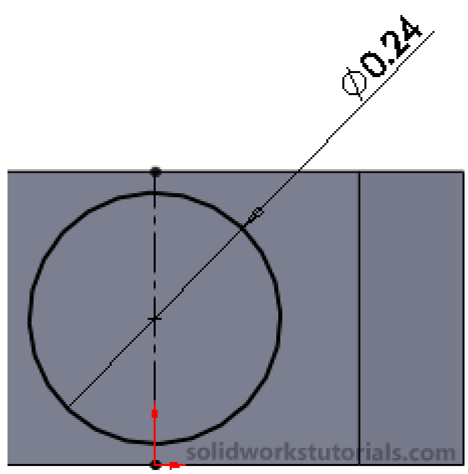


Click **Circle**  and click on midpoint of centerline as its center and sketch a circle.

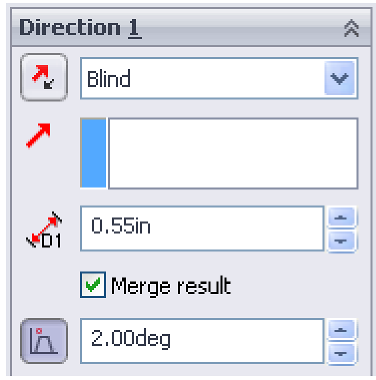


Click on **Smart Dimension** 

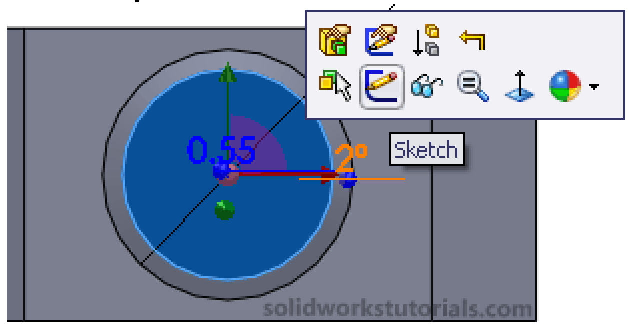
and dimension the circle as **0.24"**.



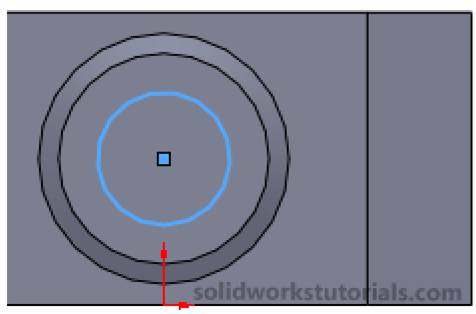
15. Click **Features**>**Extruded Boss/Base**,  on **Direction 1** set **D1** to **0.55"** and click on **taper icon** and add draft to **2deg** and .

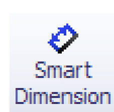


16. Click on **top face** of draft and click **Sketch**.

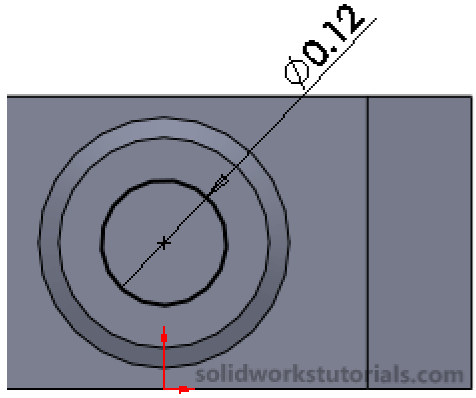


Click on **Circle** and sketch circle on the face, to get center align to center base, hover your cursor a moment at base circle edge, when center of it appear choose it as your new circle sketch center.

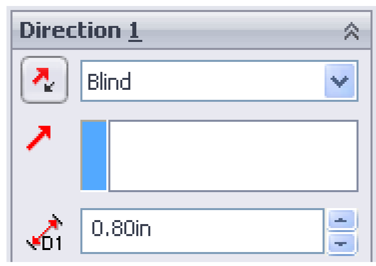


17. Click on **Smart Dimension** 

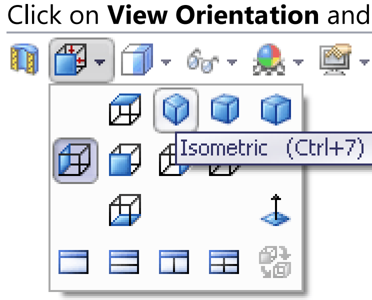
and dimension the circle as **0.12"**.



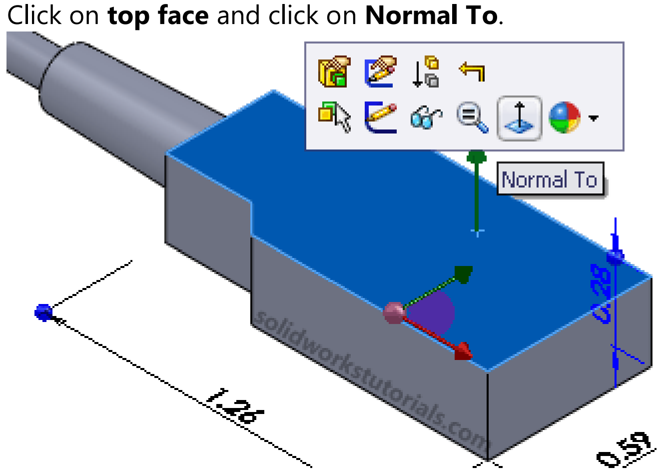
18. Click **Features**>**Extruded Boss/Base**,  on **Direction 1** set **D1** to **0.80"** and .



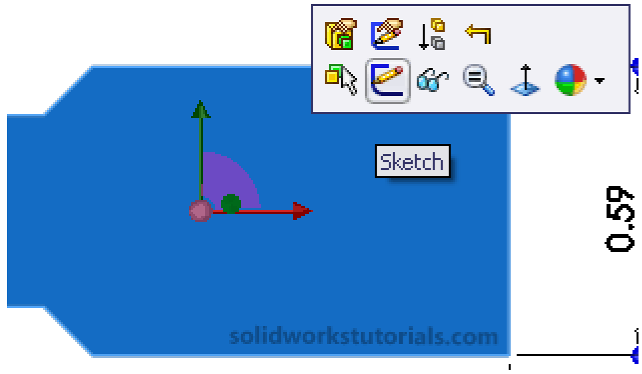
19. Click on **View Orientation** and click on **Isometric**.



20. Click on **top face** and click on **Normal To**.

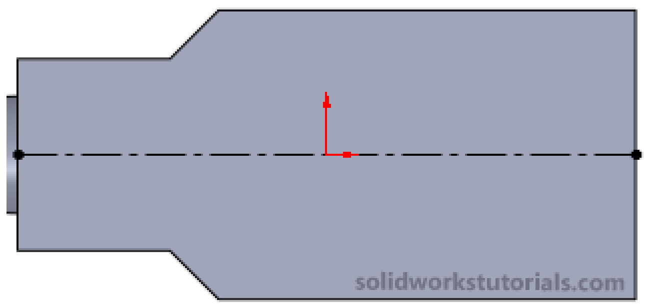


21. Click on **top face** again and click **Sketch**.

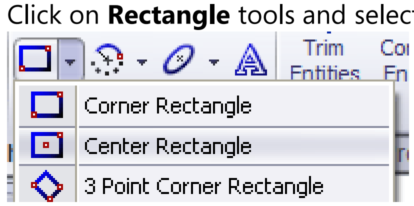


22. Click on **Line** tools and select **Centerline**,

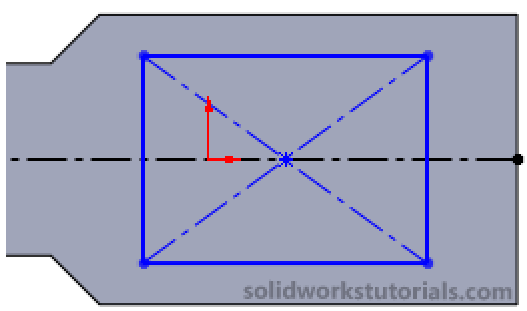
sketch a centerline through left edge midpoint to right edge midpoint.

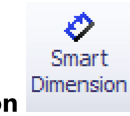


23. Click on **Rectangle** tools and select **Center Rectangle,**

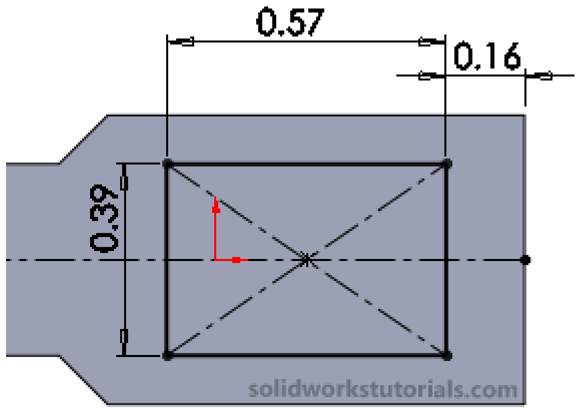


Click on **centerline** as it center and **sketch a rectangle.**

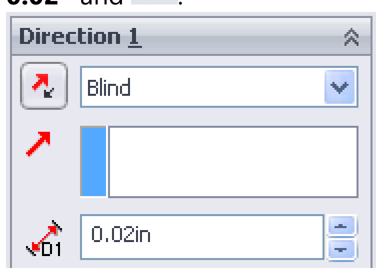


24. Click on **Smart Dimension** 

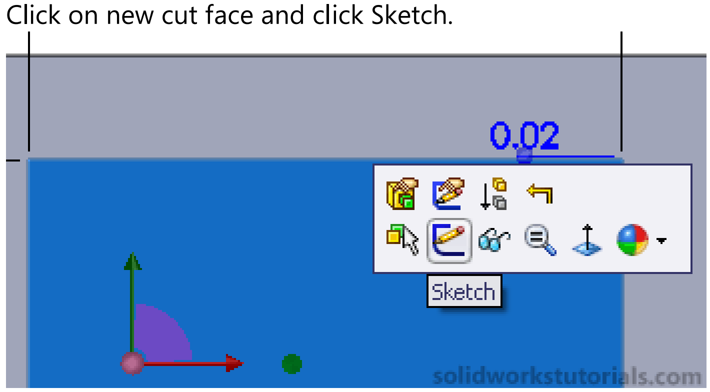
and click on side edges of rectangle to give dimension to the rectangle as **0.57" x 0.39"** and **0.16"** from right edge.



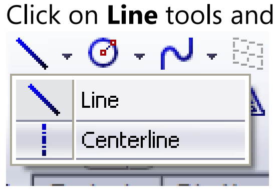
25. Click on **Features**>**Extruded Cut**  and set **Direction 1** to **0.02"** and .



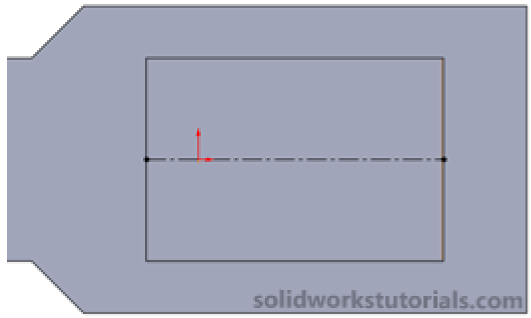
26. Click on new cut face and click Sketch.



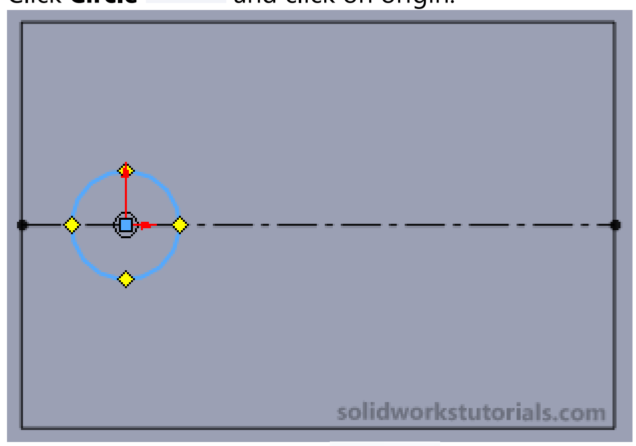
27. Click on **Line** tools and select **Centerline**,

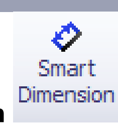


sketch a centerline through left edge midpoint to right edge midpoint.

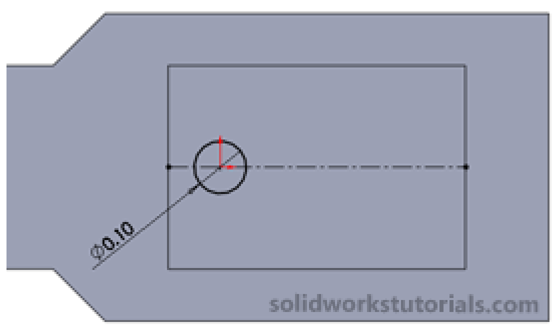


28. Click **Circle**  and click on origin.



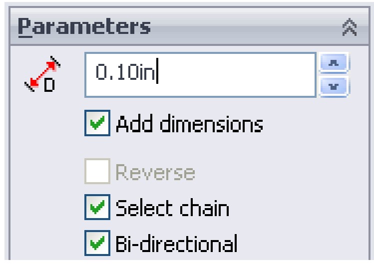
Click on **Smart Dimension** 

and dimension the circle as **0.1”**.

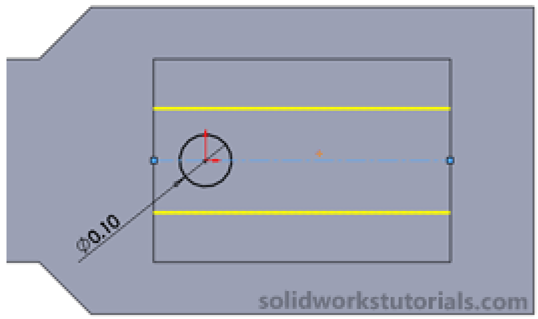


29. Click on **Offset Entities** 

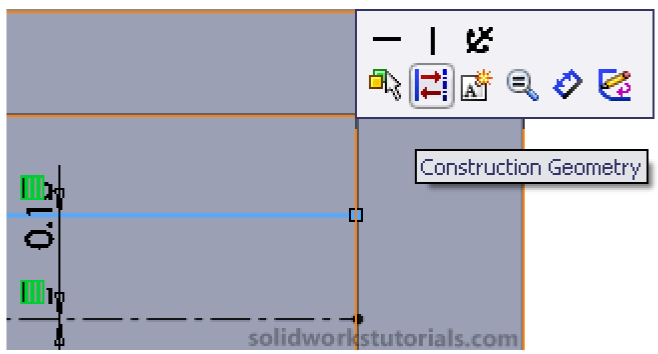
, click on centerline,



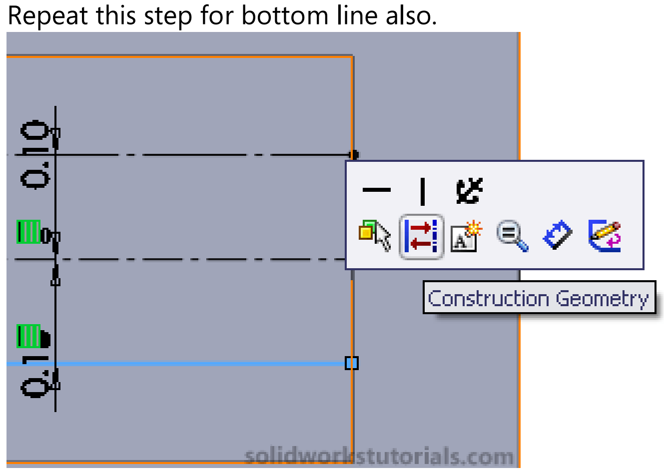
set **D** to **0.1”** and check **Bi-directional** option and .



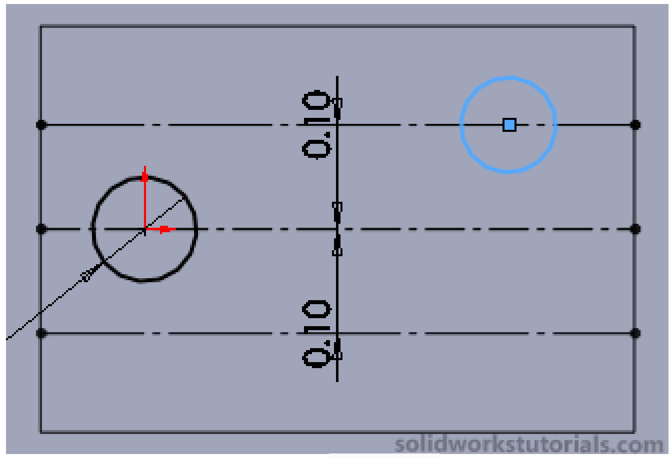
30. Click on **top line** and click **Construction Geometry**.



Repeat this step for bottom line also.

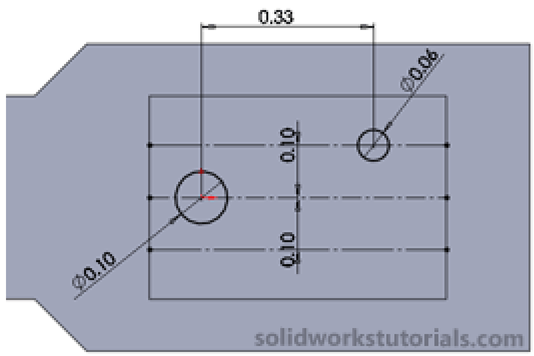


31. Click **Circle**  and click on top centerline and sketch a circle.

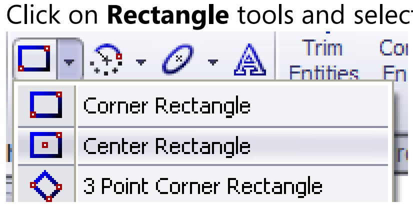


Click on **Smart Dimension** 

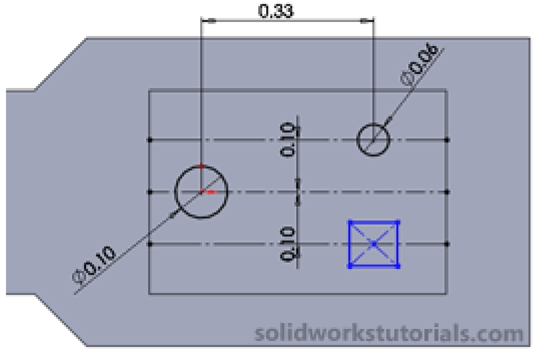
and dimension the circle as **0.06”** and **0.33”** distance from left circle.



32. Click on **Rectangle** tools and select **Center Rectangle,**

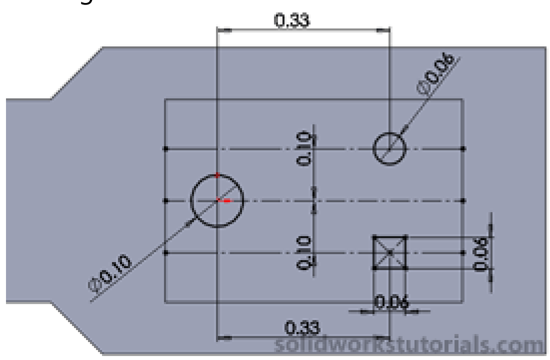


Click on **bottom centerline** as it center and **sketch a rectangle.**

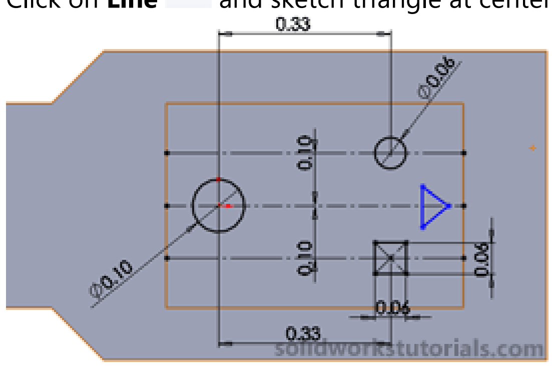


Click on **Smart Dimension** 

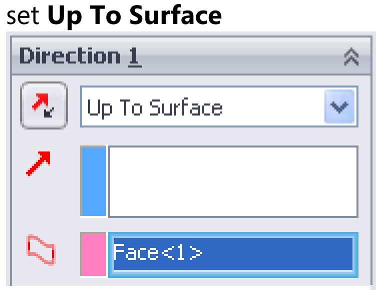
and dimension the rectangle as **0.06"x0.06"** and **0.33"** distance from left circle.



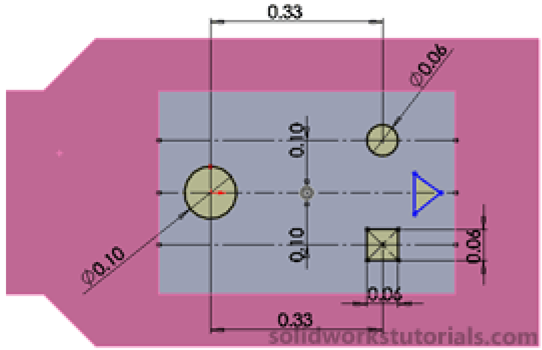
33. Click on **Line**  and sketch triangle at centerline.



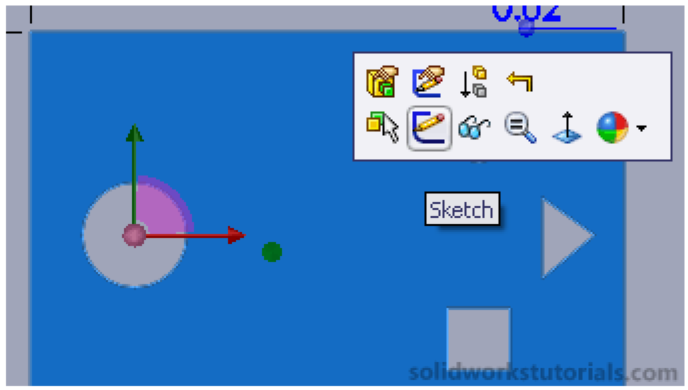
Click **Features**>**Extruded Boss/Base**,  on **Direction 1** set **Up To Surface**



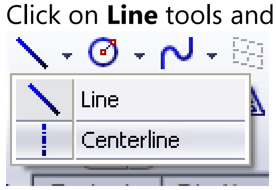
select **top face** (pink face) and .



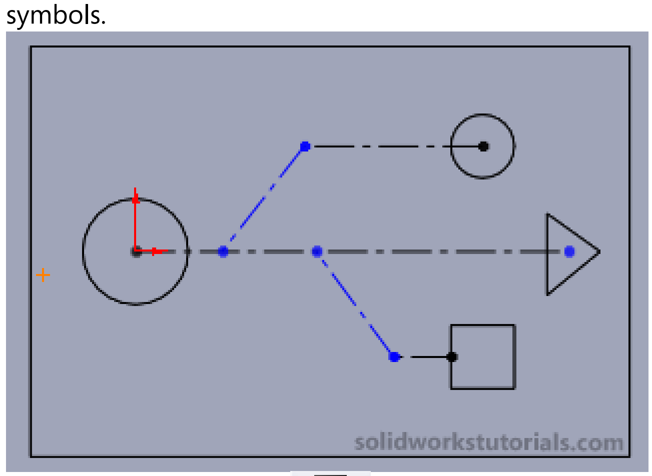
34. Click on **bottom cut face** and click **Sketch**.



Click on **Line** tools and select **Centerline**,

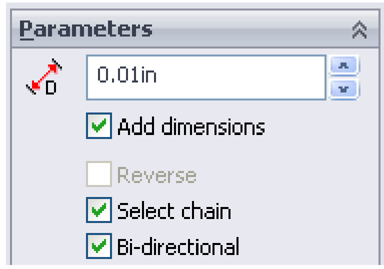


sketch a centerline connecting all the symbols.

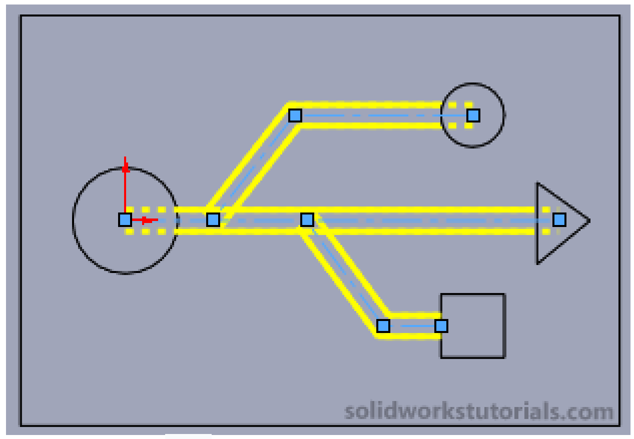


Click on **Offset Entities** 

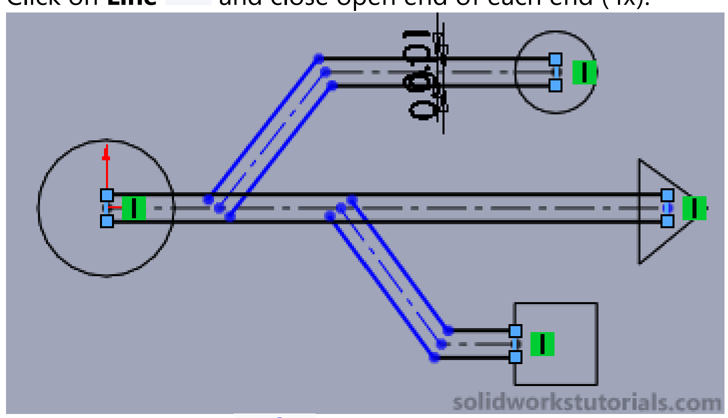
, set D to **0.01"** and check the **Bi-directional** option.



click on all connected centerline and .

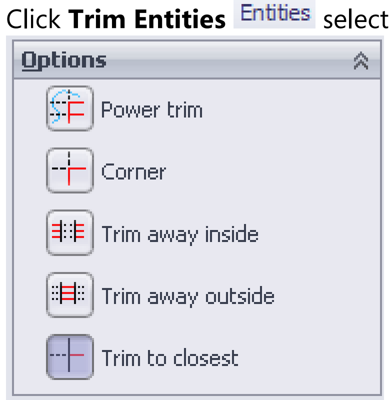


Click on **Line**  and close open end of each end (4x).

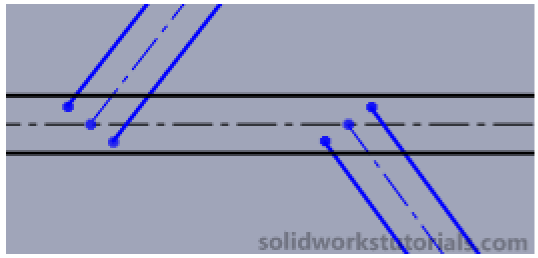


Click **Trim Entities** 

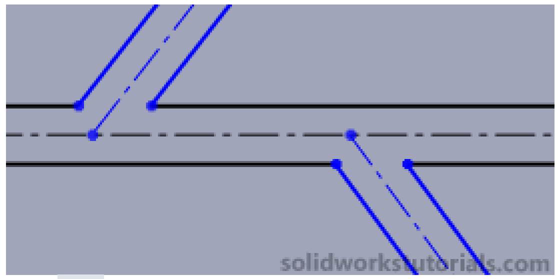
select **Trim to closest**



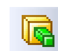
and trim off excess line from this

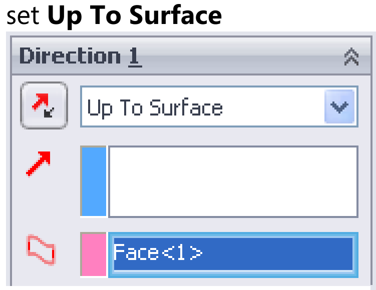


to this

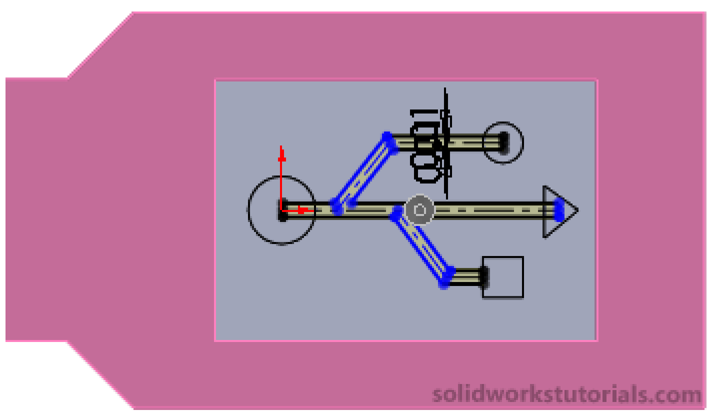


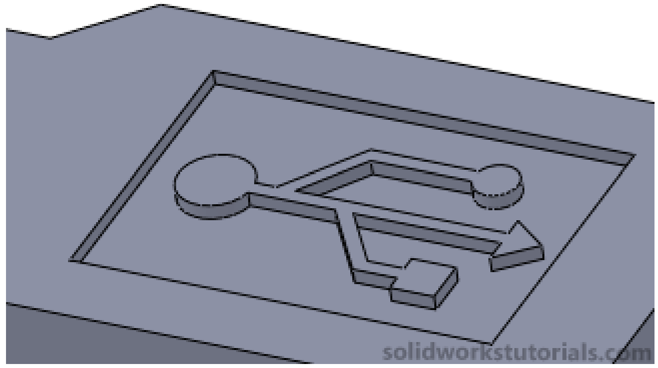
and .

35. Click **Features**>**Extruded Boss/Base**,  on **Direction 1** set **Up To Surface**

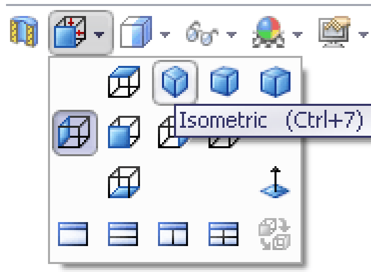


select **top face** (pink face) and .

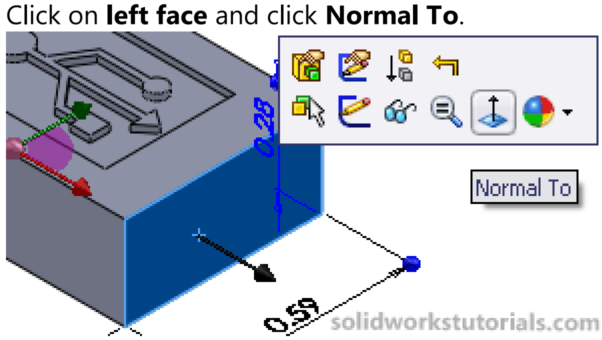




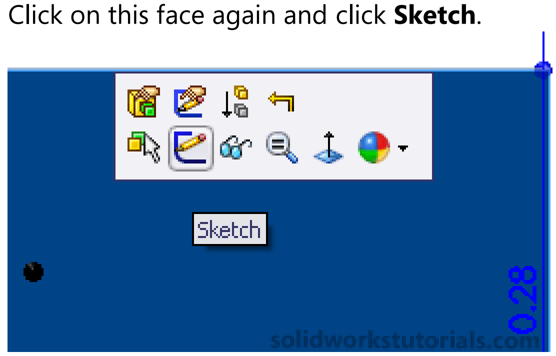
36. Click on **View Orientation** and click on **Isometric**.



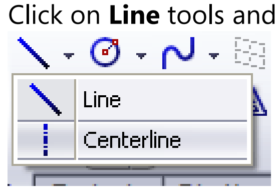
37. Click on **left face** and click **Normal To**.



Click on this face again and click **Sketch**.

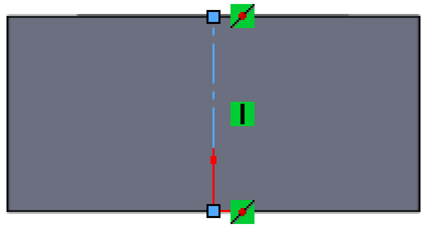


38. Click on **Line** tools and select **Centerline**,

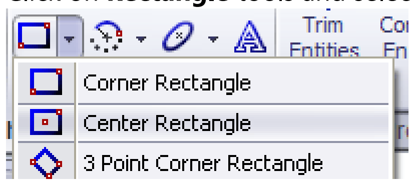


sketch a centerline from bottom edge

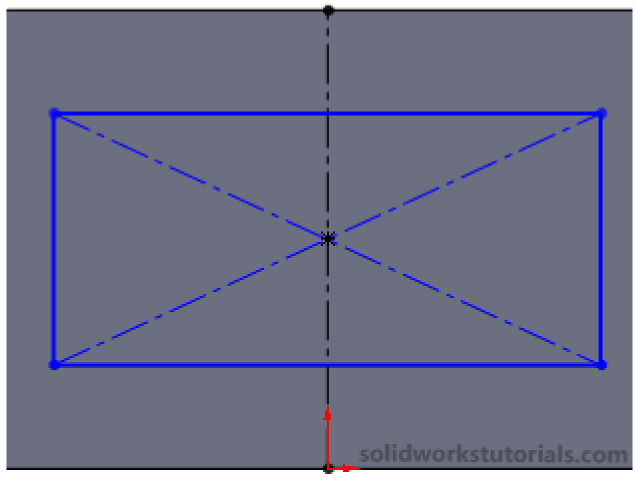
midpoint top edge midpoint.



Click on **Rectangle** tools and select **Center Rectangle,**

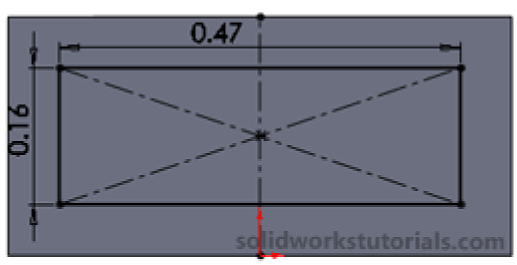


sketch a rectangle from midpoint of centerline.



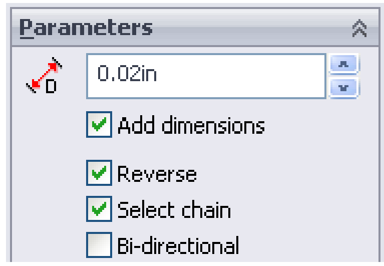
39. Click on **Smart Dimension** 

and dimension the rectangle as **0.16”x0.47”**

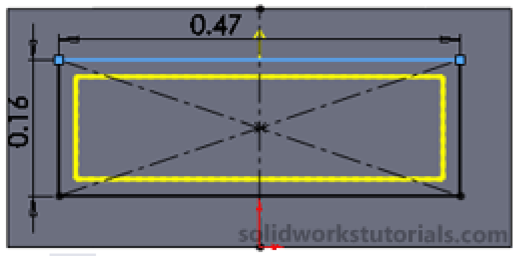


and  . Click on **Offset Entities** 

, set D to **0.02”**

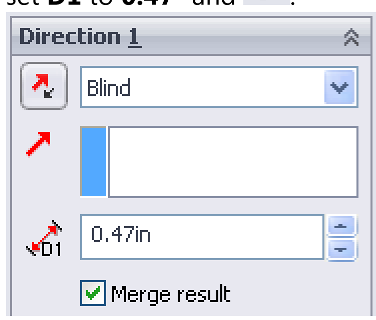


click on rectangle edge.

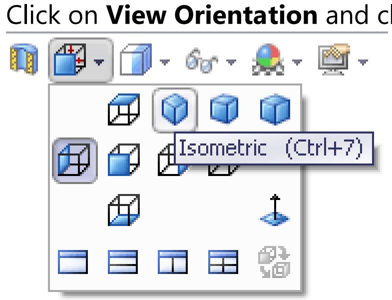


and .

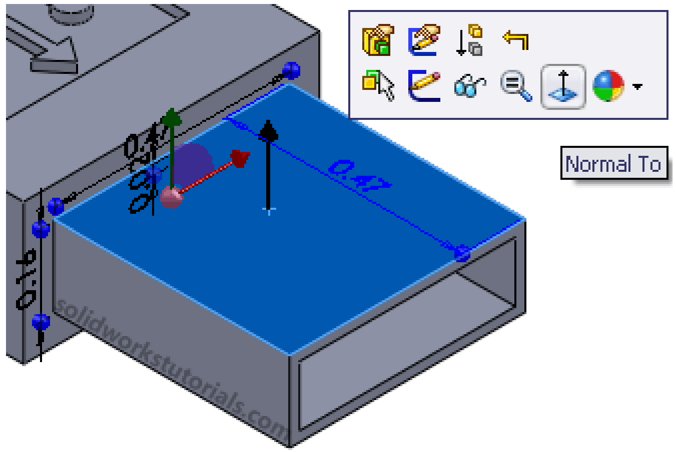
40. Click **Features**>**Extruded Boss/Base**,  on **Direction 1** set **D1** to **0.47**" and .



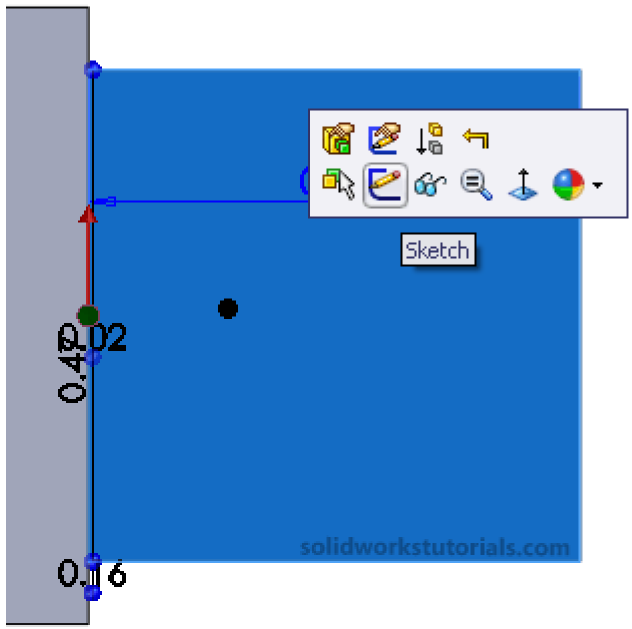
41. Click on **View Orientation** and click on **Isometric**.



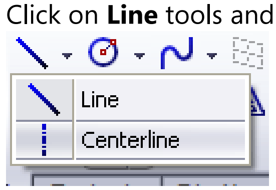
Click on **top head left face** and click **Normal To**.



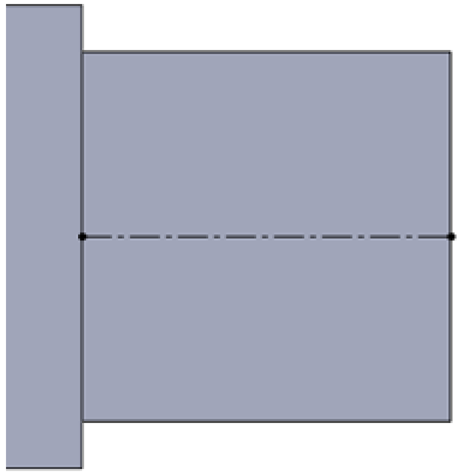
Click on this face again and click **Sketch**.



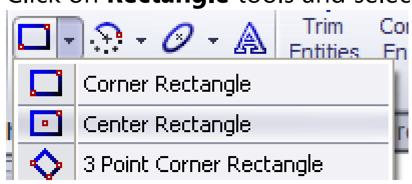
42. Click on **Line** tools and select **Centerline**,



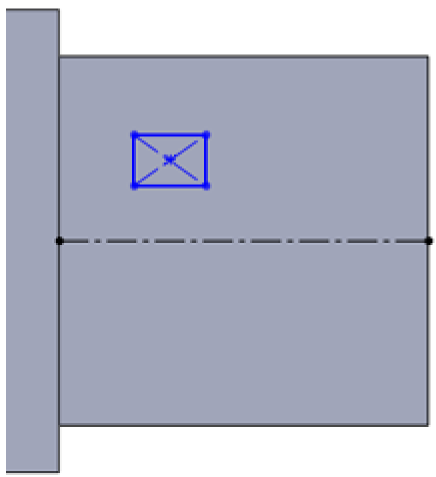
sketch a centerline from left midpoint edge to right midpoint edge.



Click on **Rectangle** tools and select **Center Rectangle,**

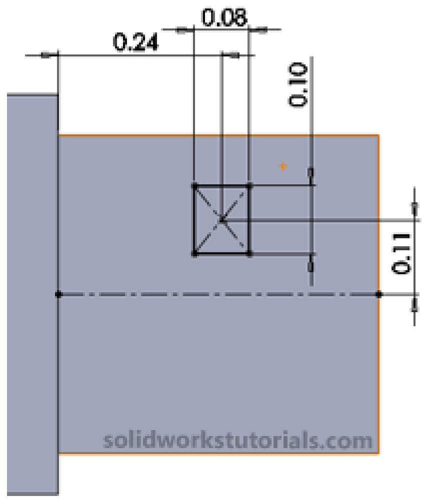


sketch a rectangle on the face.

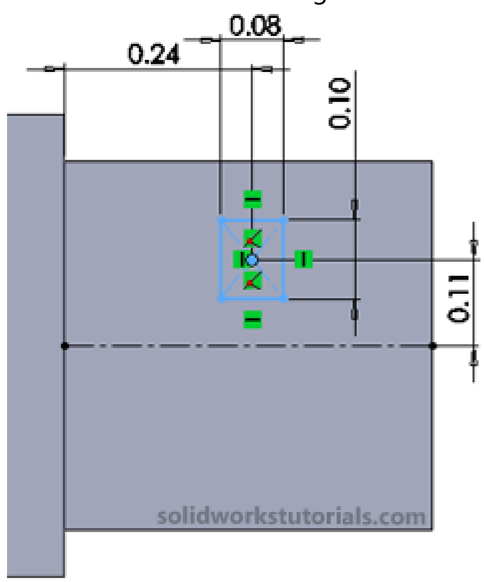


Click on **Smart Dimension** 

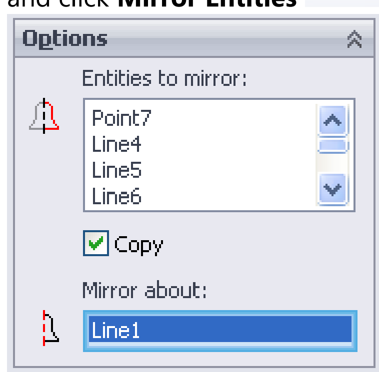
and dimension the rectangle as **0.10”x0.08”** and for its position **0.24”** from left edge and **0.11”** from centerline.



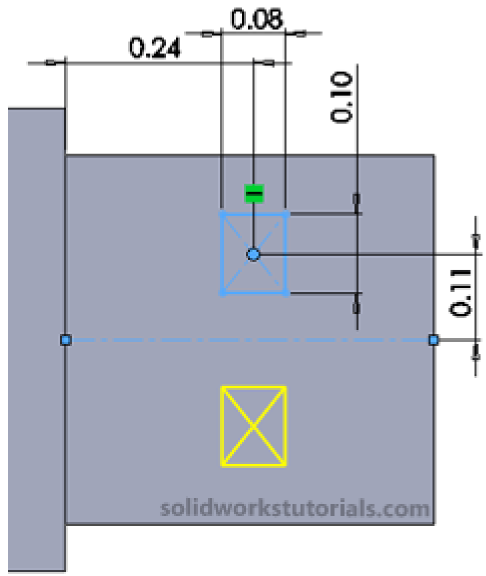
43. Select all entities of rectangle



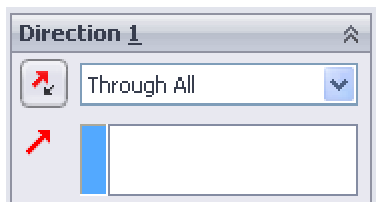
and click **Mirror Entities** 



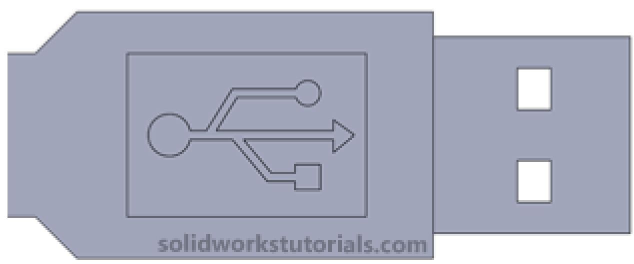
and on **Mirror about:** box select **centerline** and .



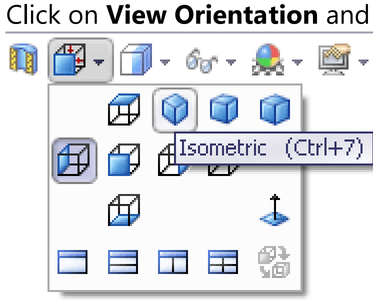
44. Click on **Features**>**Extruded Cut**  and set **Through All**



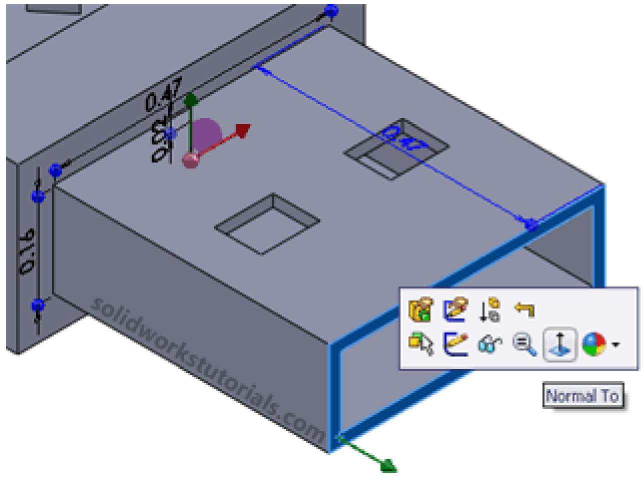
and .



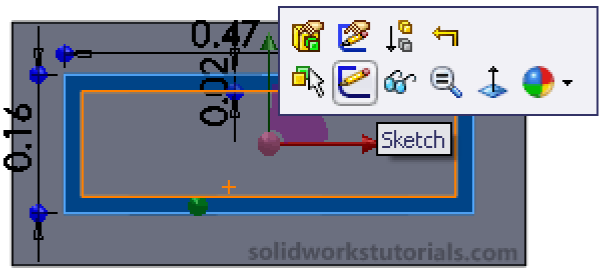
45. Click on **View Orientation** and click on **Isometric**.

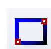


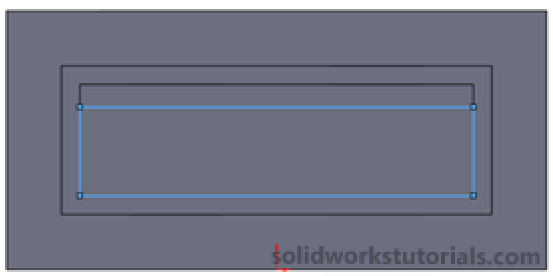
Click on **front head left face** and click **Normal To**.



Click on this face again and click **Sketch**.

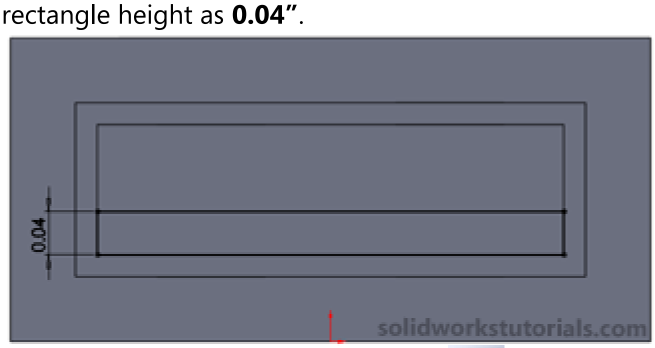


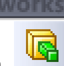
46. Click **Rectangle**  and sketch a rectangle on this face.

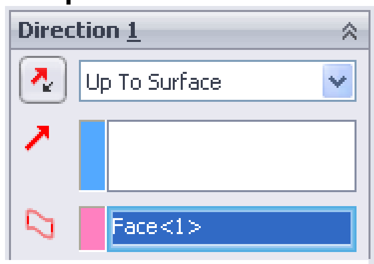


Click on **Smart Dimension** 

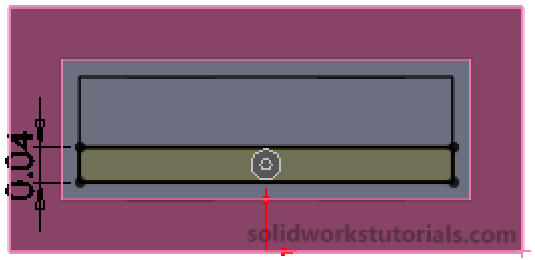
and dimension the rectangle height as **0.04"**.



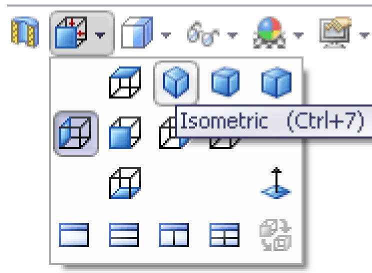
Click **Features**>**Extruded Boss/Base**,  on **Direction 1** set **Up To Surface**

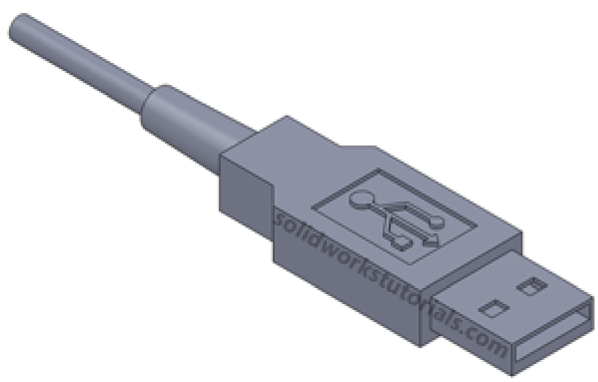


select **top face** (pink face) and .



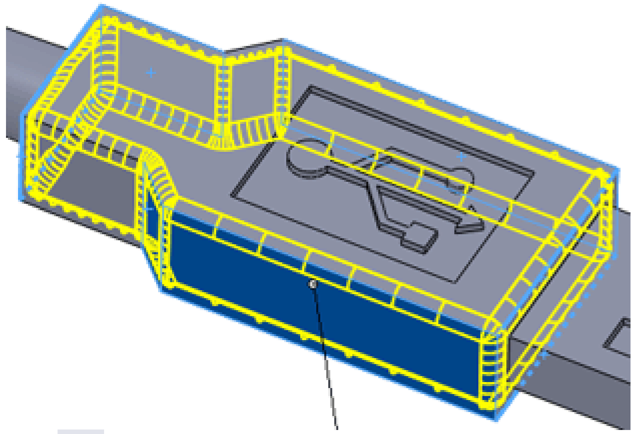
47. Click on **View Orientation** and click on **Isometric**.



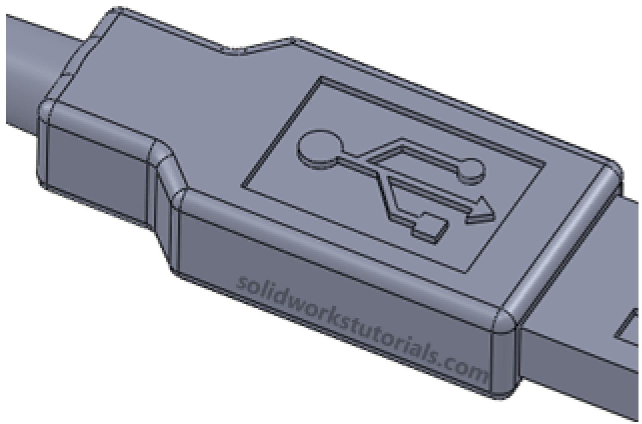


48. Click on **Fillet** , set fillet radius to **0.04”**

and select main body edge

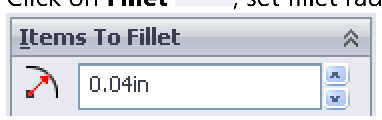


and .

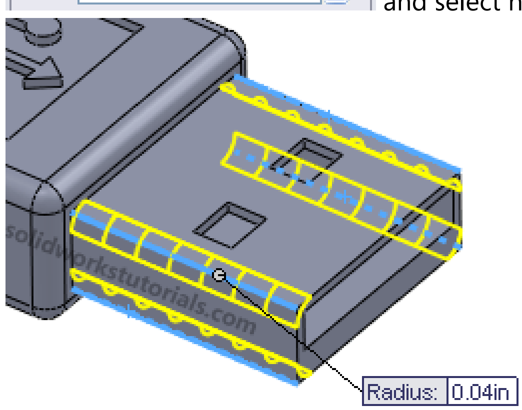


49. Click on **Fillet** 

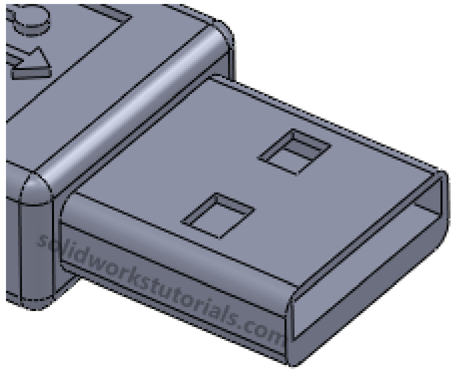
, set fillet radius to **0.04"**



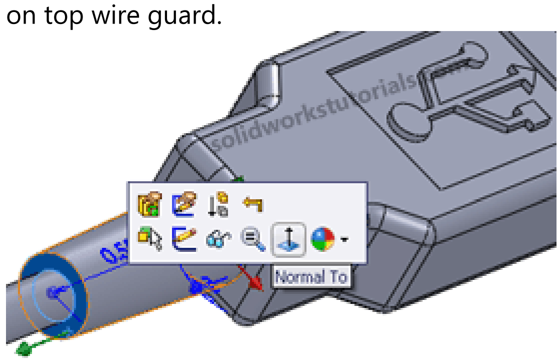
and select head edges



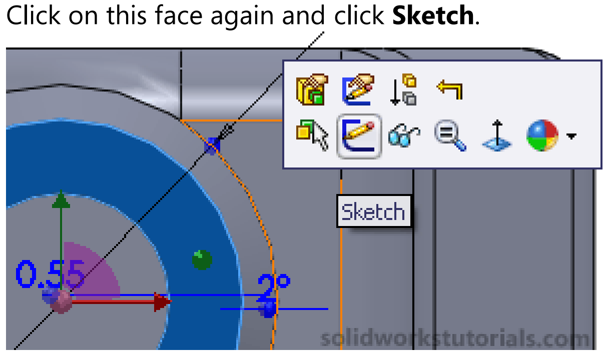
and .



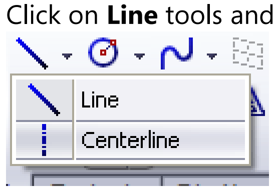
50. Rotate the pack to view the back end and click **Normal To** on top wire guard.



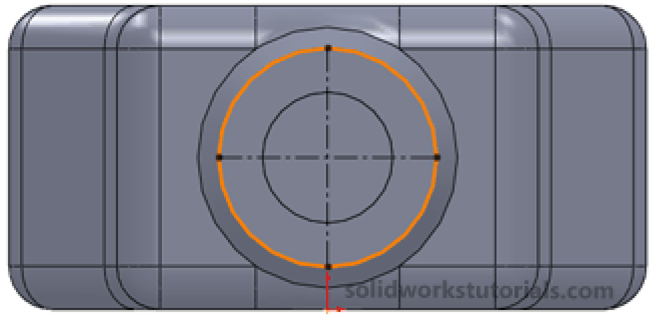
Click on this face again and click **Sketch**.

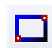


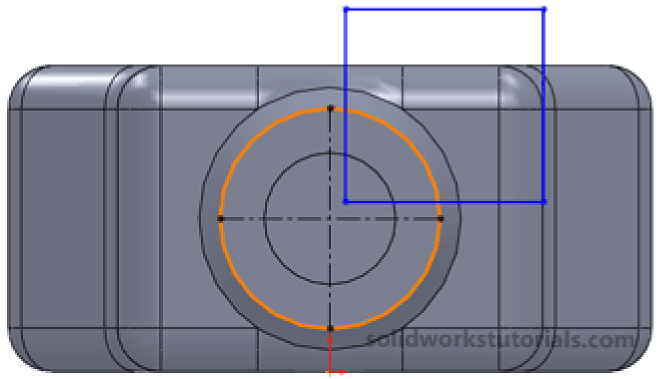
Click on **Line** tools and select **Centerline**,



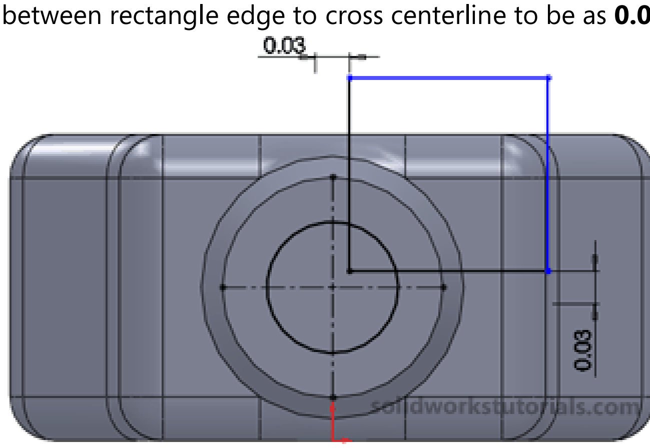
sketch a centerline from top to bottom and left to right as sketched.



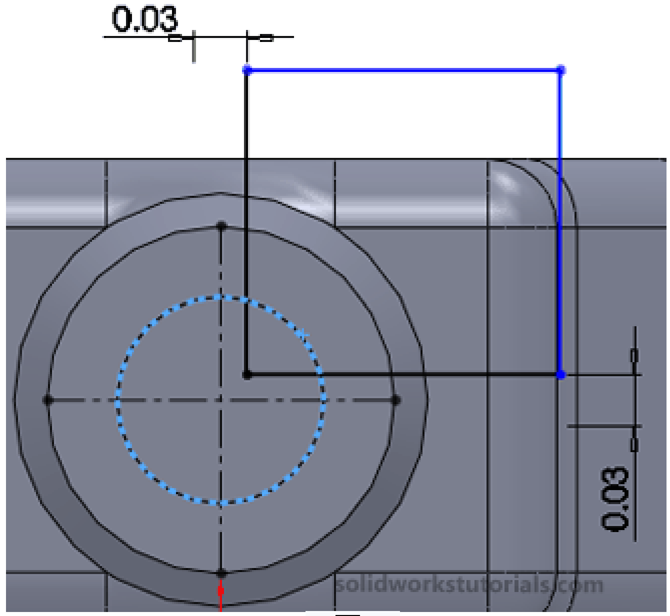
51. Click **Rectangle**  and sketch a rectangle on this face.



Click on **Smart Dimension** and dimension distance between rectangle edge to cross centerline to be as **0.03”**.

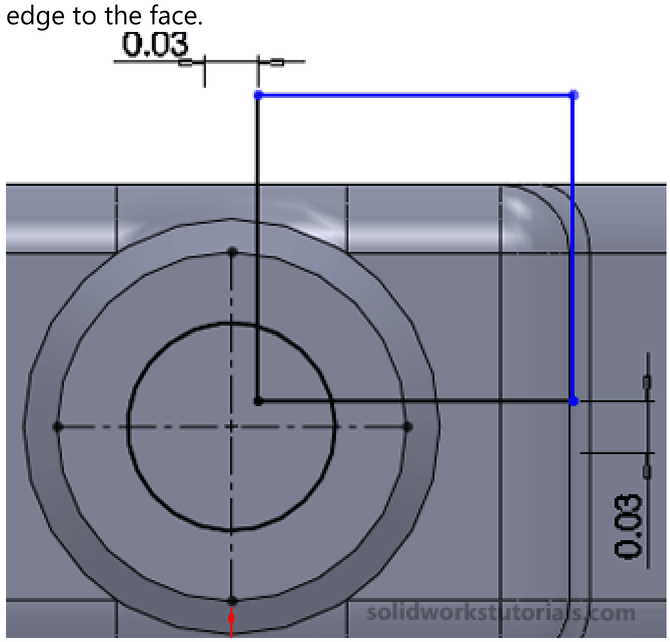


52. Click on wire edge,



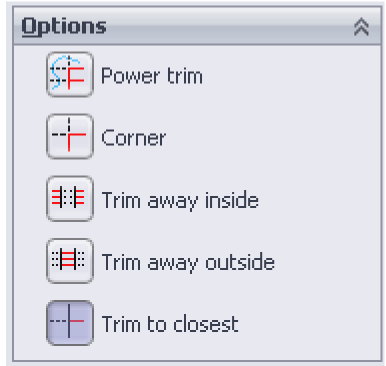
and click **Covert Entities** 

, this will project outer edge to the face.

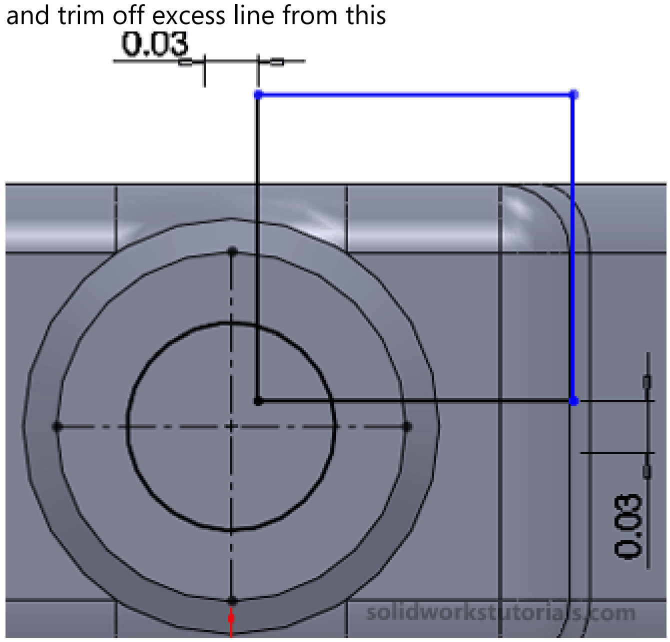


53. Click **Trim Entities** 

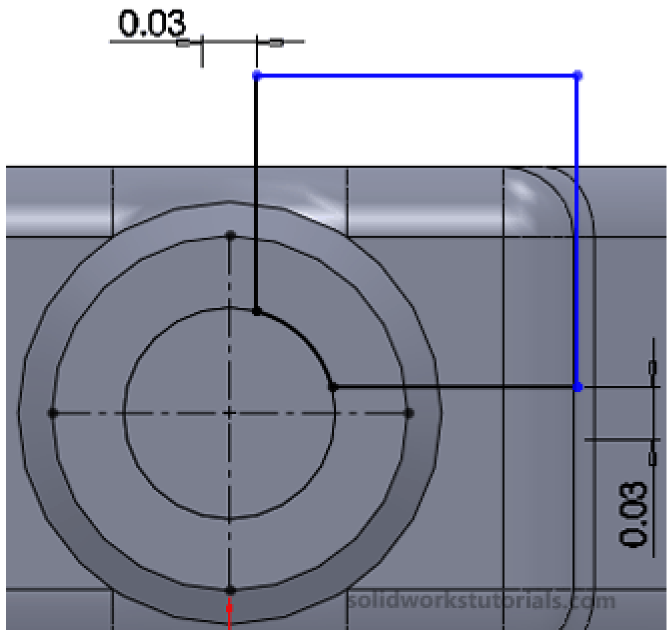
select **Trim to closest**



and trim off excess line from this

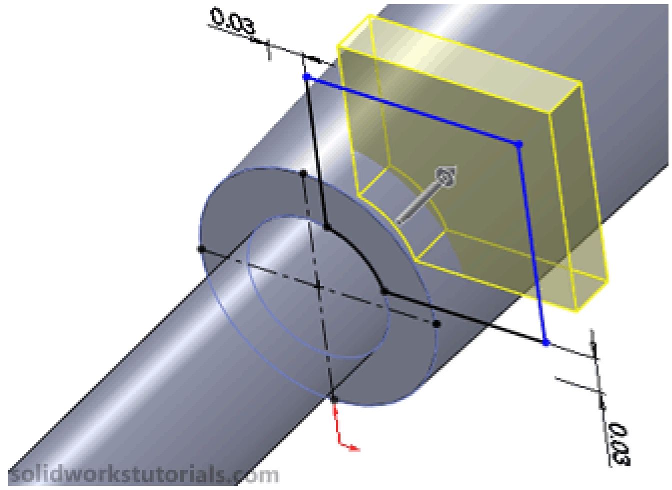


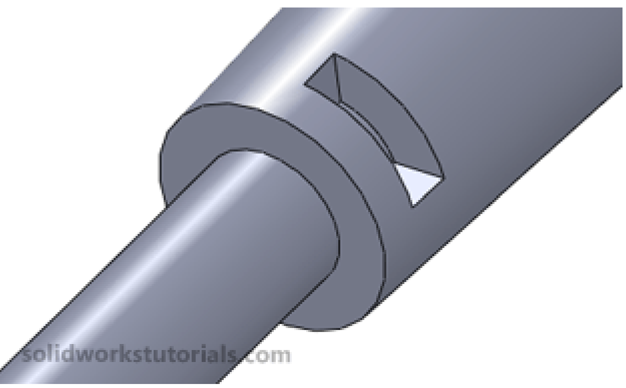
to like this



54. Click on **Features**>**Extruded Cut**  and set From: **Offset** and distance to **0.04"** and for Direction 1 set **Blind** and D1 to **0.04"** and .

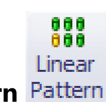
The cut should look like this;



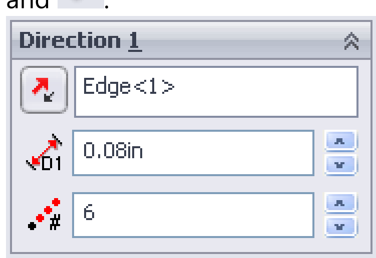


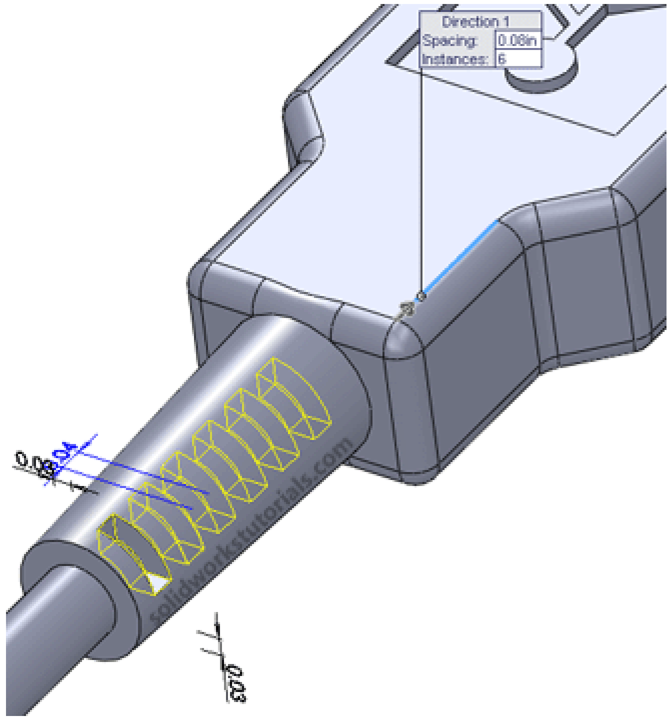
If your cut not in correct orientation you can try toggle direction of cut by clicking this button. 

55. Click on last extruded cut feature icon tree

and click on **Linear Pattern** 

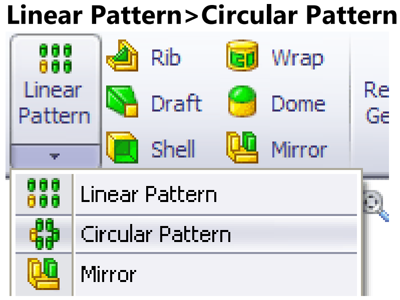
, for **Direction 1** click on body edge, set **D1** to **0.08"** and **#** to **6** and .



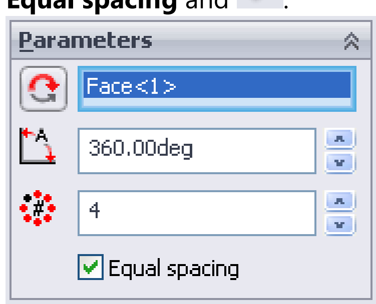


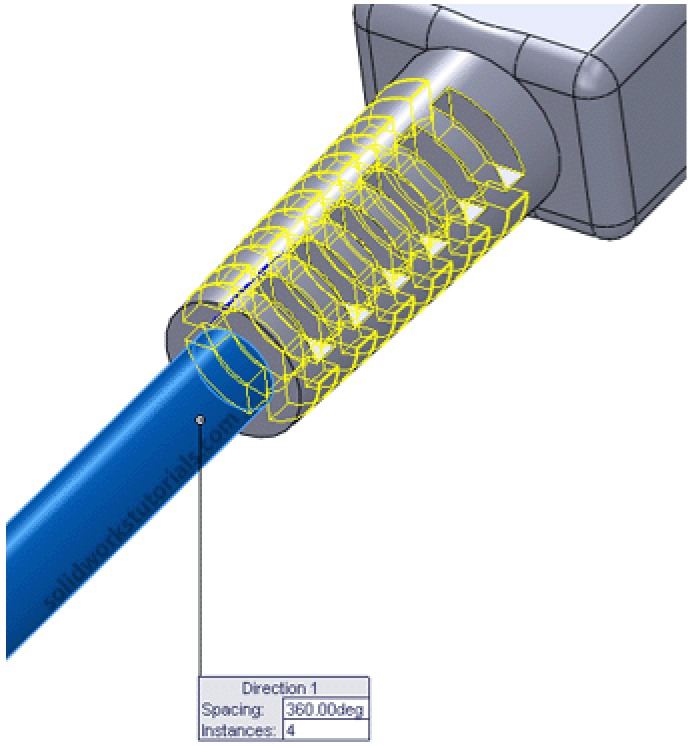
56. Click on **LPattern** icon tree 

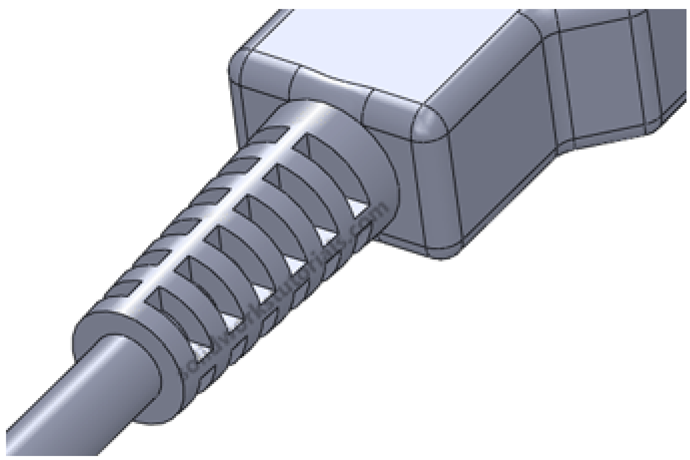
and click on **Linear Pattern>Circular Pattern**



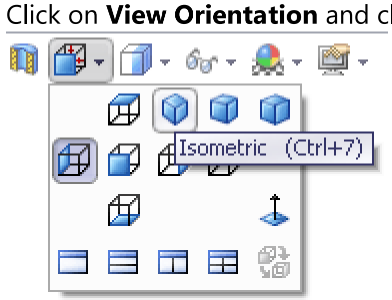
on **Parameters** click wire surface as axis of rotation, **#** to **4**, **Equal spacing** and .

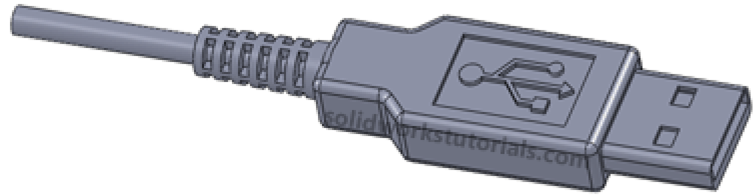






57. Click on **View Orientation** and click on **Isometric**.

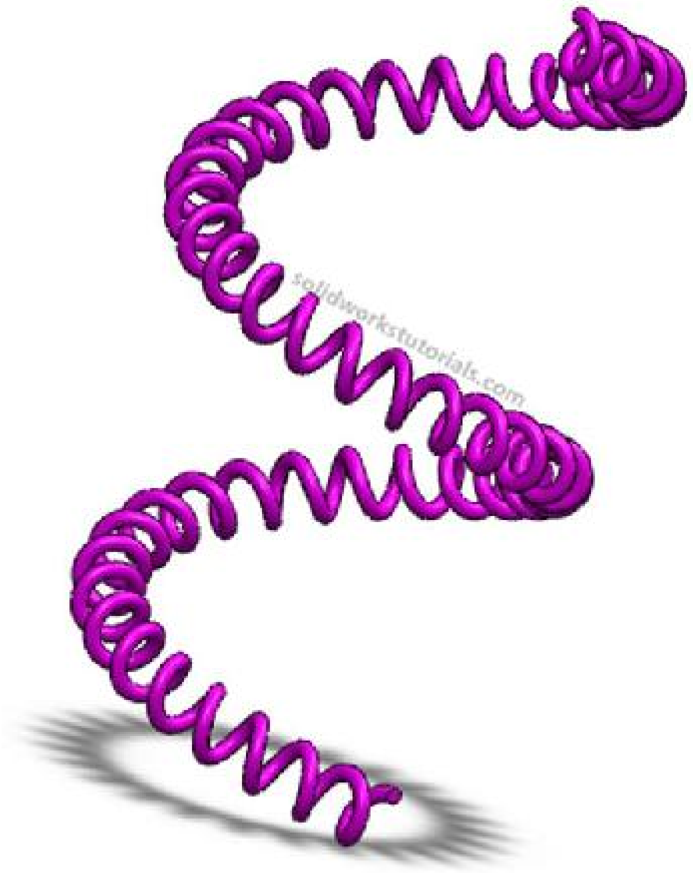


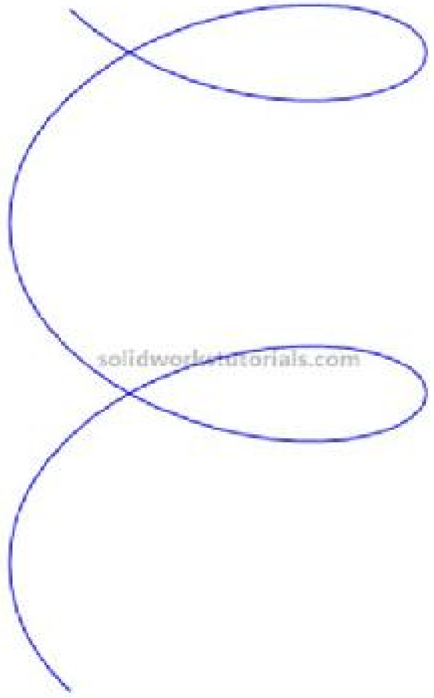


58. Save the part as **USB head** and you're done! Simple isn't it?

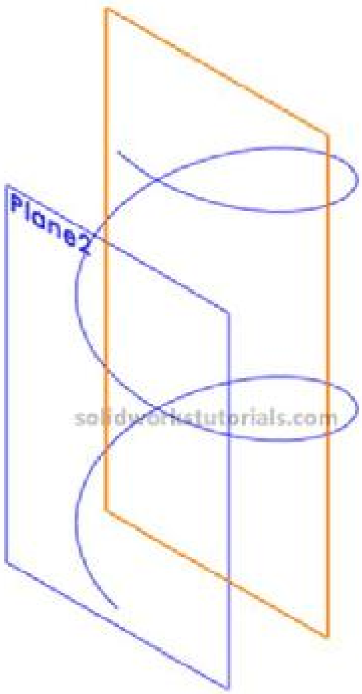
How to twist phone cord

Learn how to create twist phone cord… with you mouse

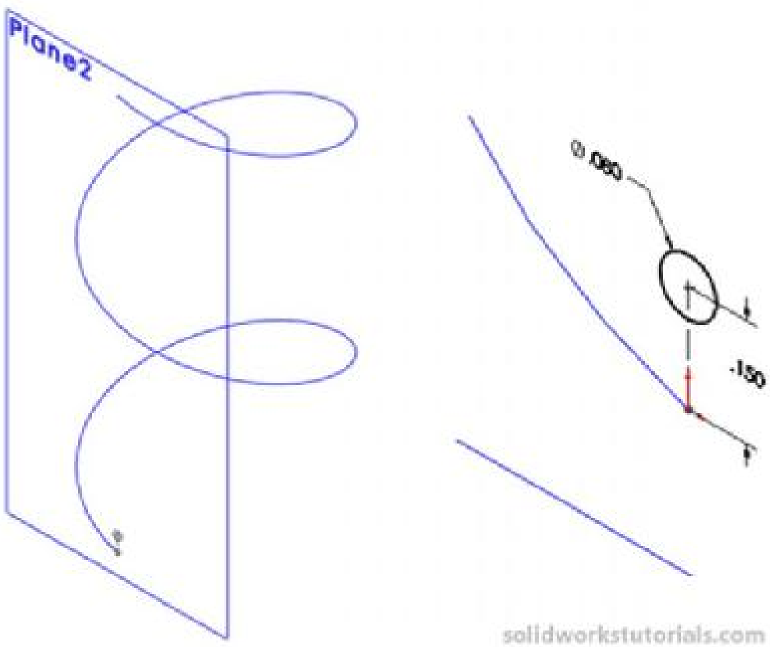




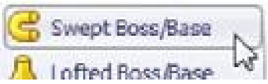
First you need to have spiral, with circle base 2″ , 2 revolution and 2 pitch. Don’t know how? Refer this tutorial; Tutorial #2: How to create simple spring



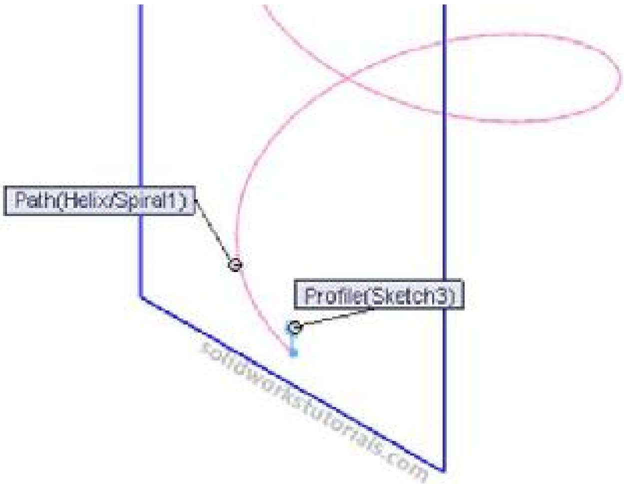
Now add a plane at end of spiral, select parallel to front plane.



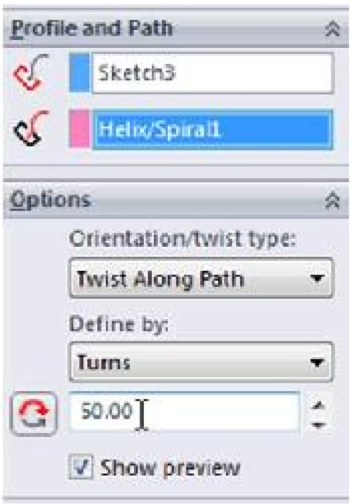
Sketch a circle on Plane2, 0.08″ and 0.15 height.

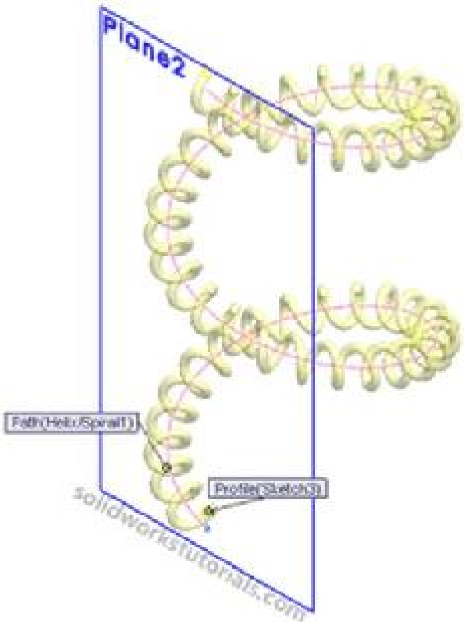
Click Swept Boss/Base. 

Select Sketch3 as profile and Helix/Spiral1 as path.

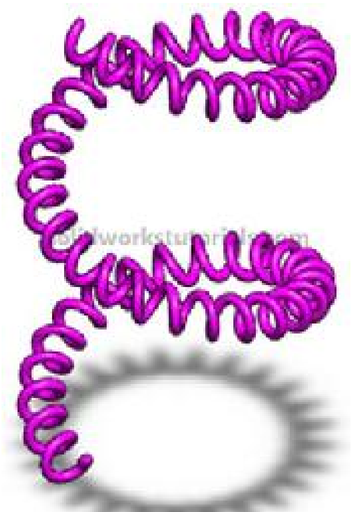


Open up Options and set Twist Along Path, define by Turns and 50 turns.





And OK you're done!



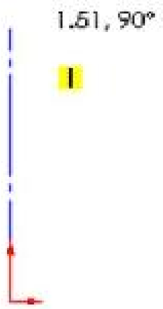
How to use Revolved Boss/Base

In this tutorial you will create this part.

1. Click New.  Click Part, 

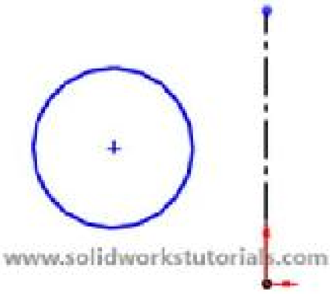
OK.
2. Click Front Plane and click on Sketch.

3. Select centerline, 

sketch vertical line start from origin, roughly 1.5in 

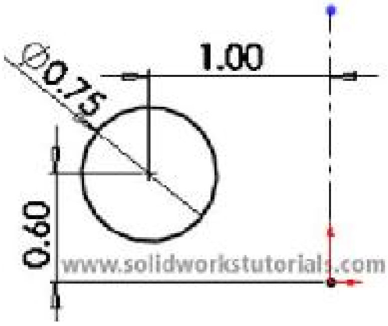
and OK. 

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



5. Click Smart Dimension,

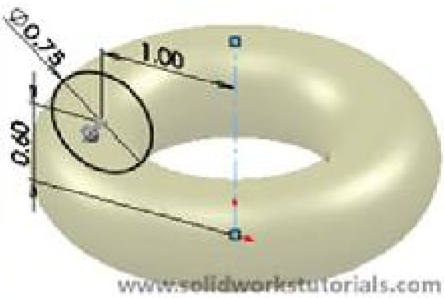
click sketched circle and set it diameter to 0.75in and add dimension for it location as below sketch



and OK.

6. You just completed your sketch, let's build feature from it. ClickFeature>Revolved Boss/Base

7. Click centerline as axis

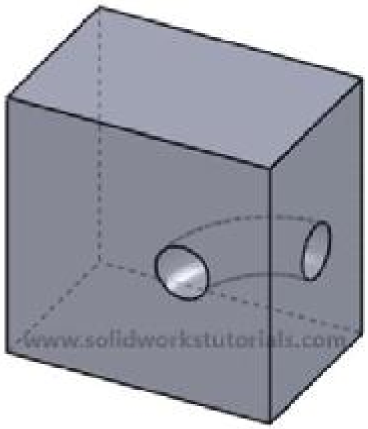


OK.

8. You're done!

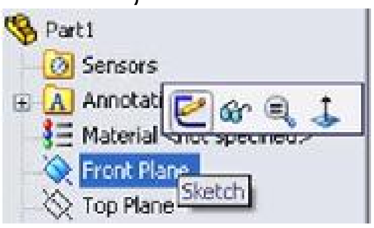
How to use Revolved Cut

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

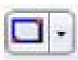


1.Click New.Click Part, 

OK.

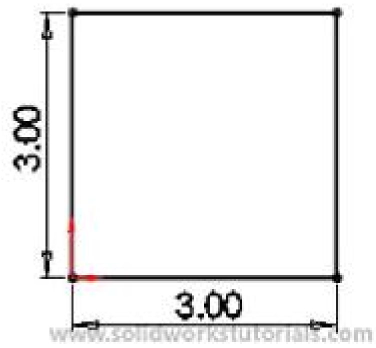
2.Click Front Plane 

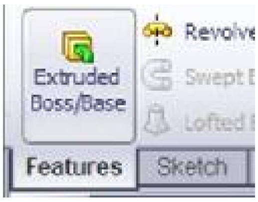
and click on Sketch.

3.Click Rectangle, sketch rectangular.

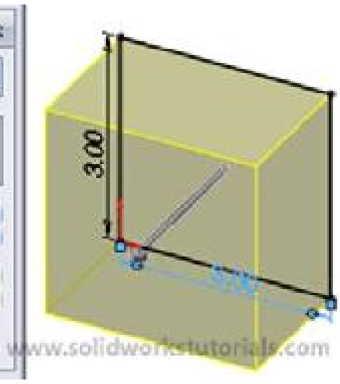
Click Smart Dimension, 

dimension rectangular 3in x 3in.



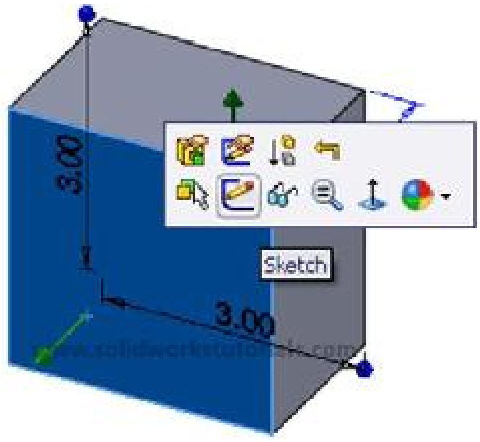
4.Click Feature>Extruded Boss/Base, 

set D1 to 2in

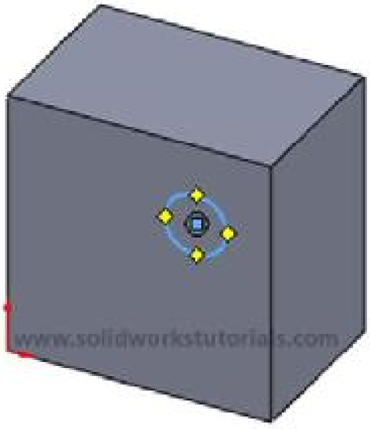


and  .

5.Click on front face and click Sketch.

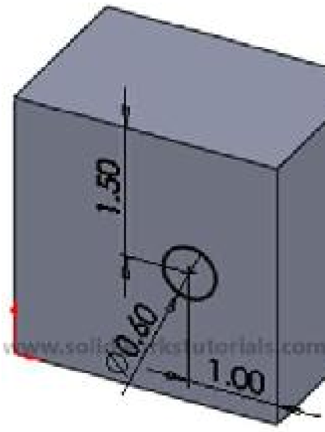


6. Click Circle,  and sketch a circle on front face.



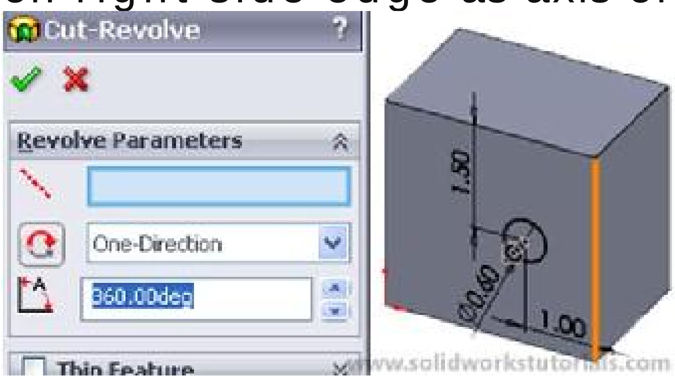
7. Click Smart Dimension, 

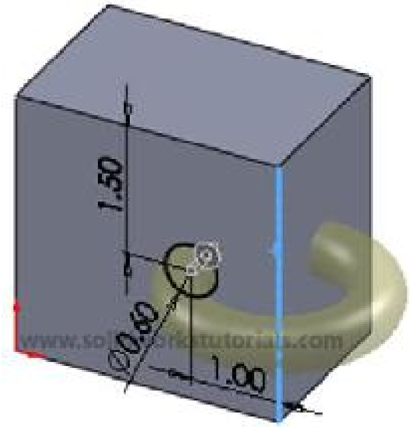
dimension sketch as below sketched.



8.Click Features>Revolved Cut 

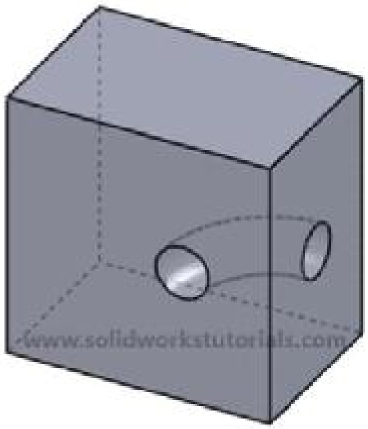
click on right side edge as axis of revolution,





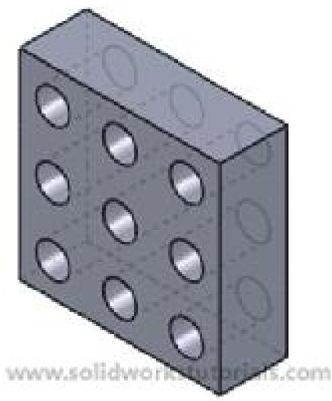
and  .

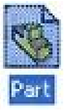
9. You’re done!



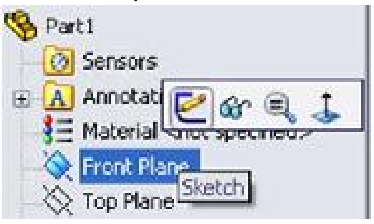
How to use Linear Pattern

In this tutorial, you will create this part.

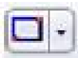


1.Click New.Click Part,

OK.

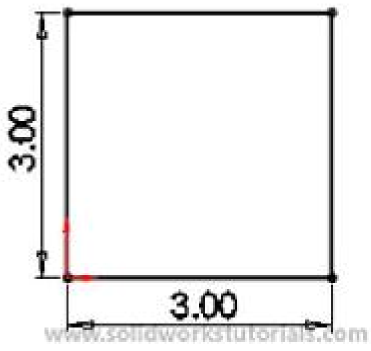
2.Click Front Plane

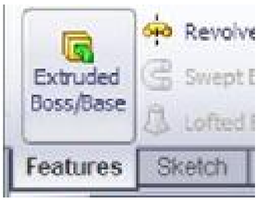
and click on Sketch.

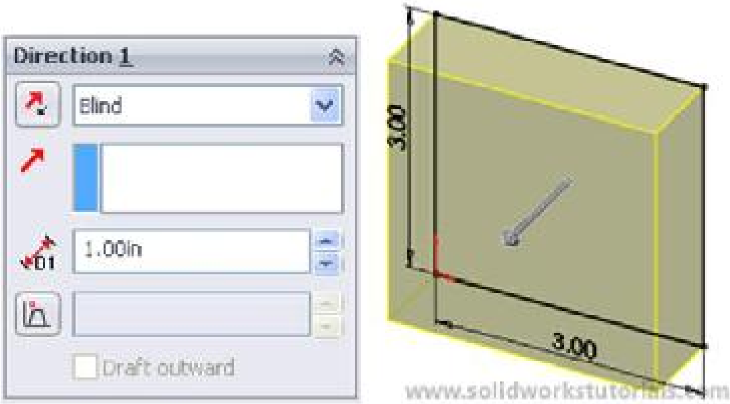
3.Click Rectangle,sketch rectangular.

Click Smart Dimension,

dimension rectangular 3in x 3in.

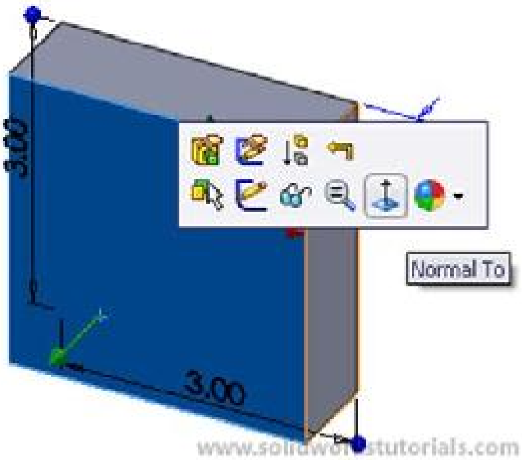


4.Click Feature>Extruded Boss/Base, 

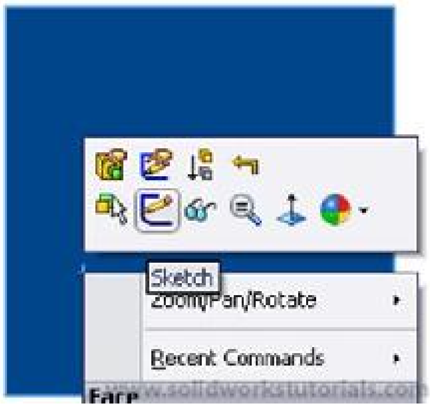


set D1 to 1.0in and OK. 

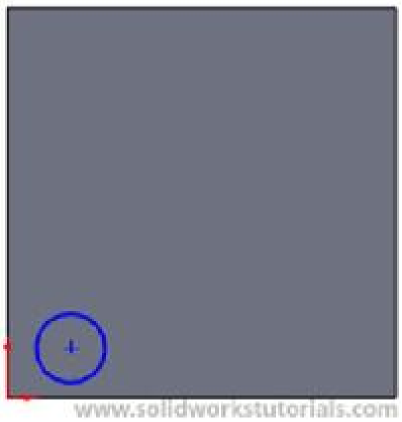
5. Click on front face and select Normal to.



6.Click front face and Insert Sketch.

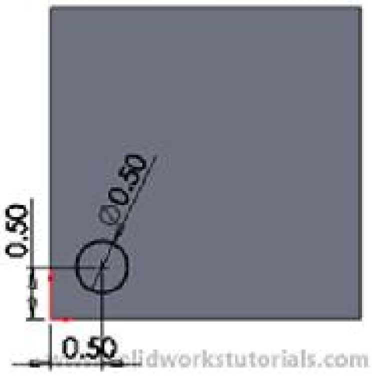


7.Click Circle,  sketch circle at one edge.



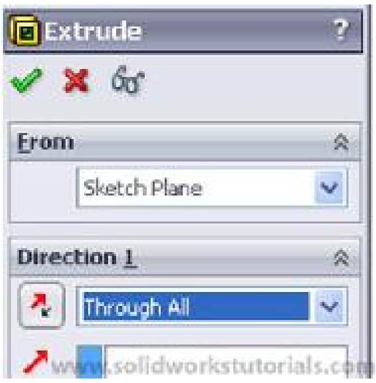
8.Click Smart Dimension, 

dimension circle as below sketch.

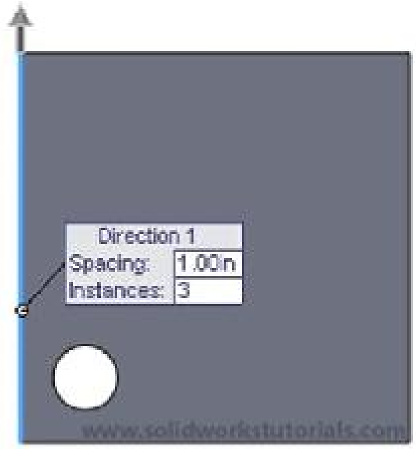


9.Click Features>Extruded Cut,

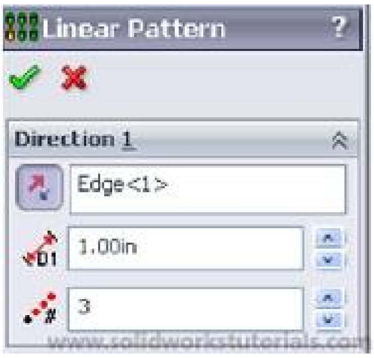
set Direction 1, Through All and OK. 



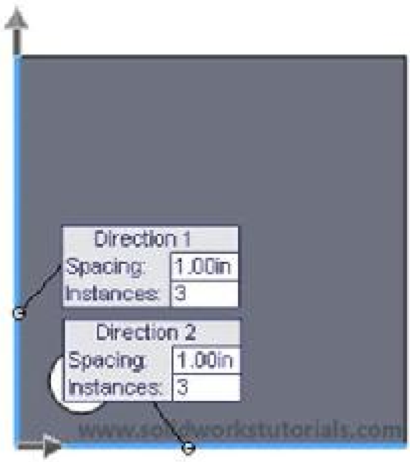
10.Click Linear Pattern,  click left edge,



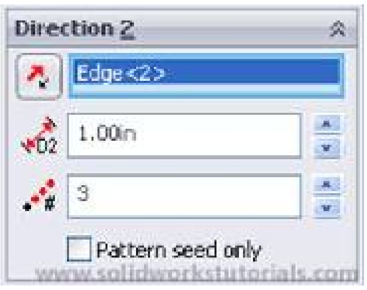
on Direction 1 set spacing D1 to 1.0in and Instances # to 3.



11.Click bottom edge,

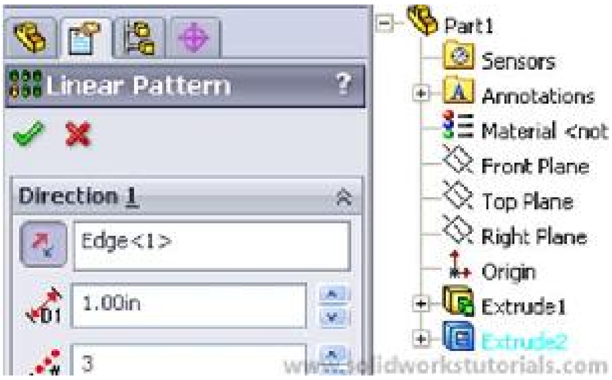


on Direction 2 set spacing D2 to 1.0in and Instances # to 3.



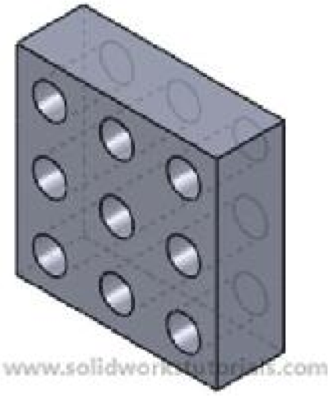
12.Click inside white box Features to Pattern.

Open up part tree, select Extrude 2



and OK. 

13.You’re done!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