



Sri Lanka Institute of Information and Technology

Department of Mechanical Engineering

Engineering Drawing – ME2031

SolidWorks Laboratory - 1

Mr. Thilina Weerakkody
Mr. Kulunu Samarakrama

Time: 2 hours
Date :

Targeted out comes of this lab

- Introduction to SolidWorks Environment
- Sketching in SolidWorks
- Implementation of Basic features
 - Extrude Boss/Base
 - Extrude cut
 - Revolve cut & Mirror
 - Linear Pattern & Fillet
 - Mass properties

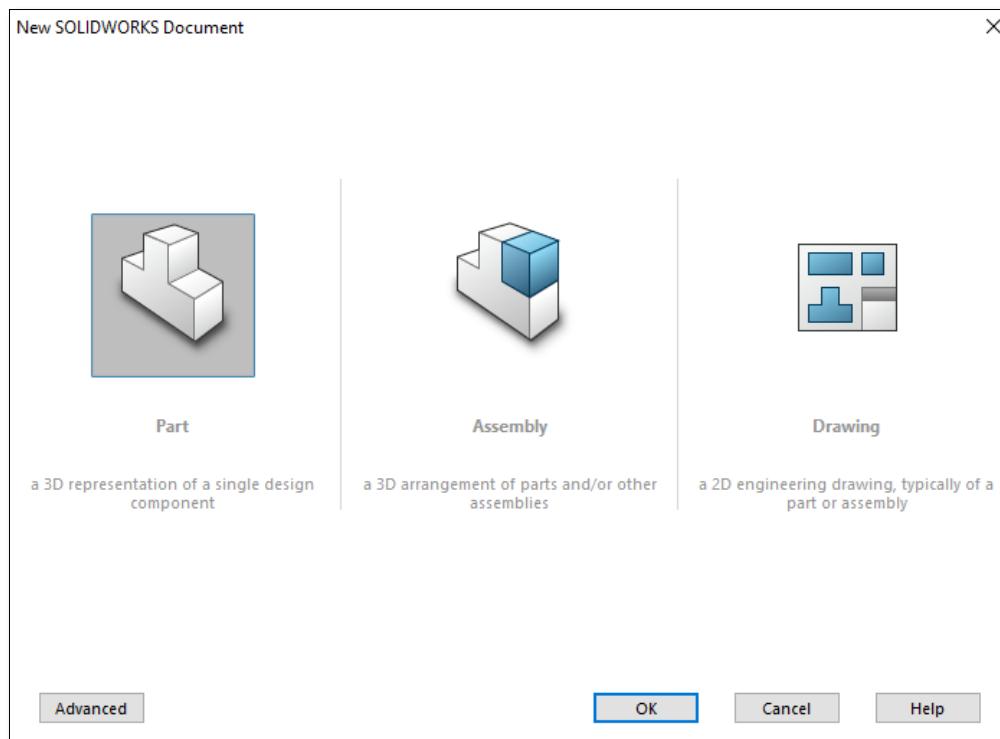
1. Getting Started

Open SolidWorks Application

On your desktop **Navigate: >> SolidWorks (Version)**

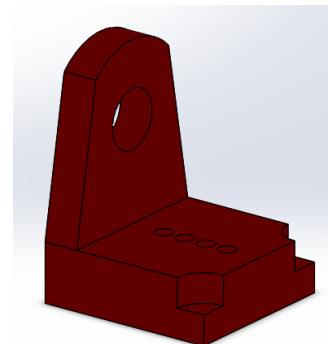
Wait for the Application to configure and open

Once SolidWorks interface opens **Navigate: New >> select “Part File”**

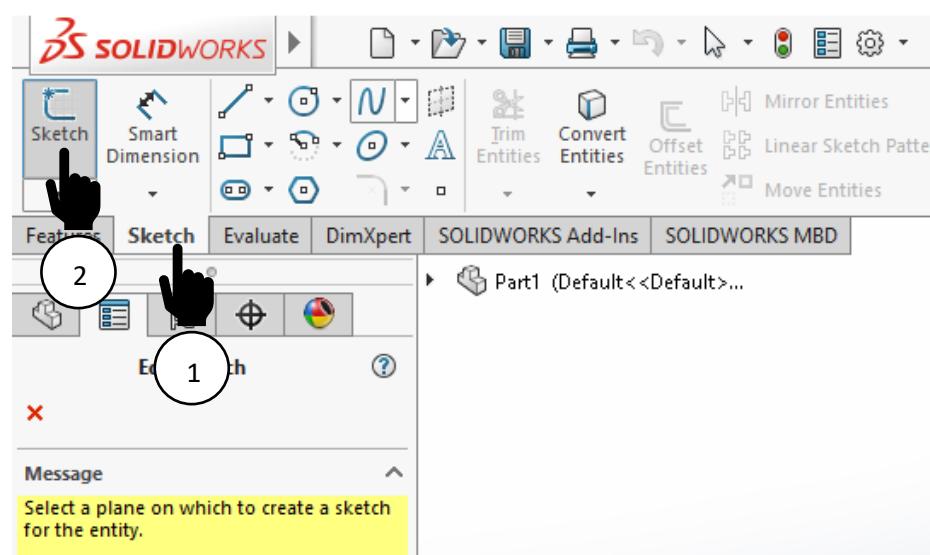


2. Sketching in SolidWorks

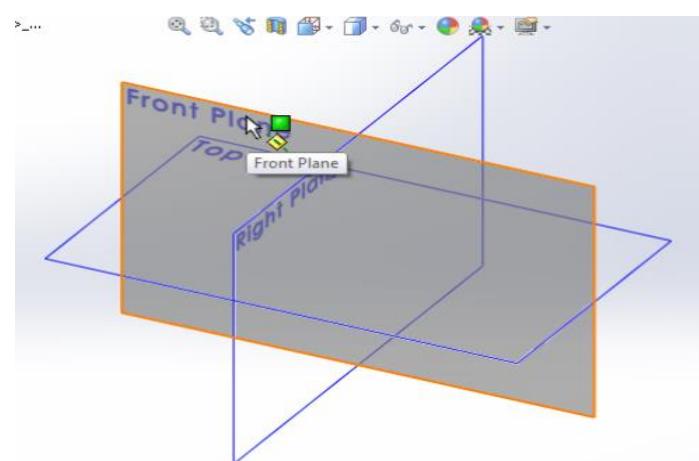
We are going to Sketch the 3D drawing of the following L shaped– Bracket



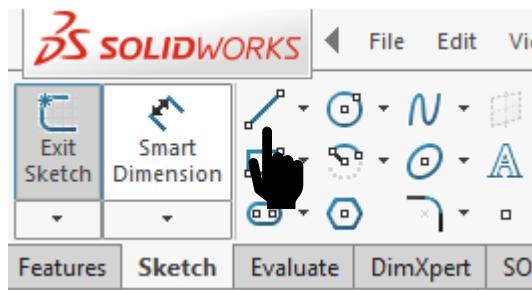
- a) Go to the Sketching environment



- b) Select the front plane to start the sketch



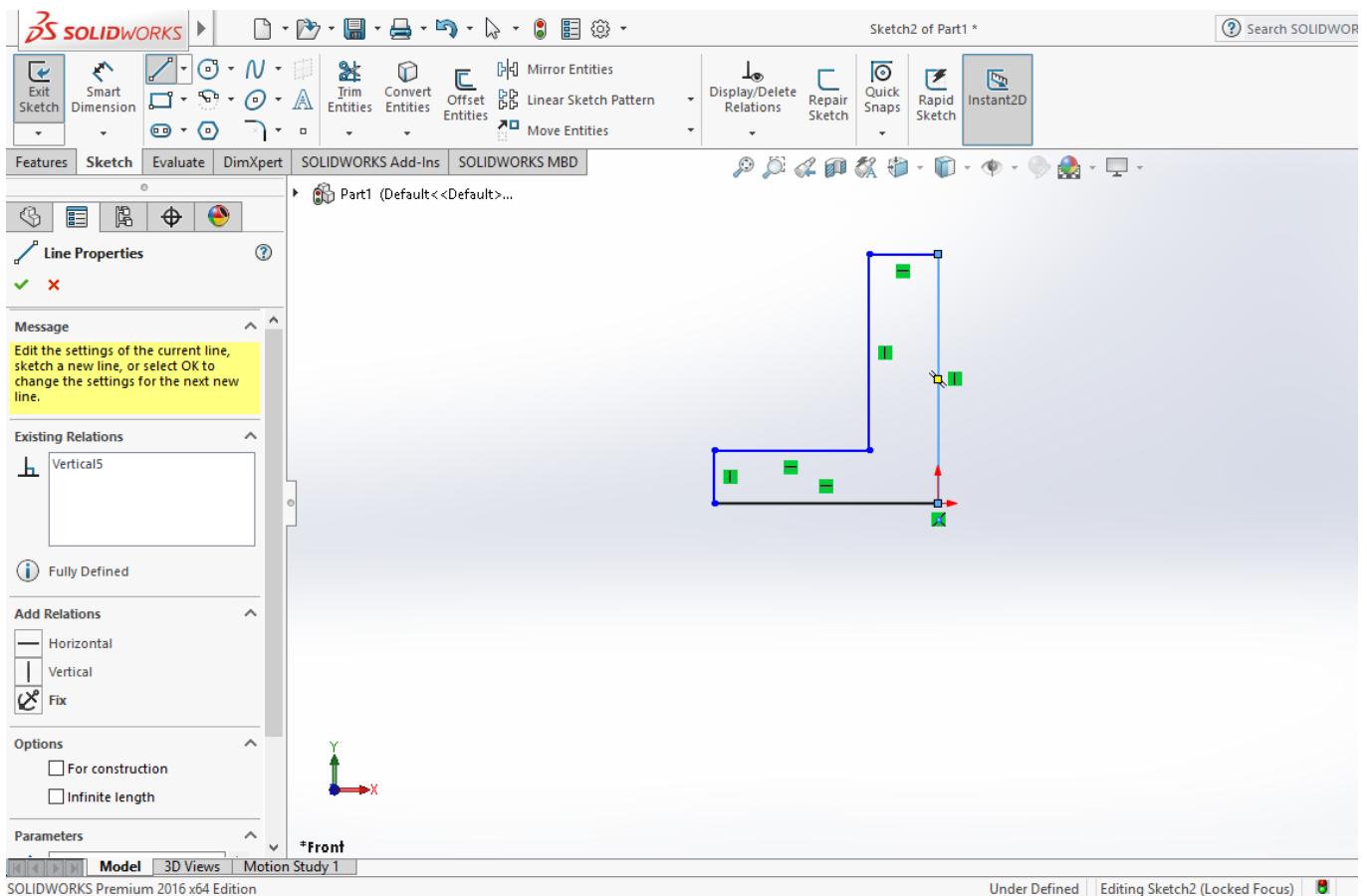
- c) Select the Line option as below



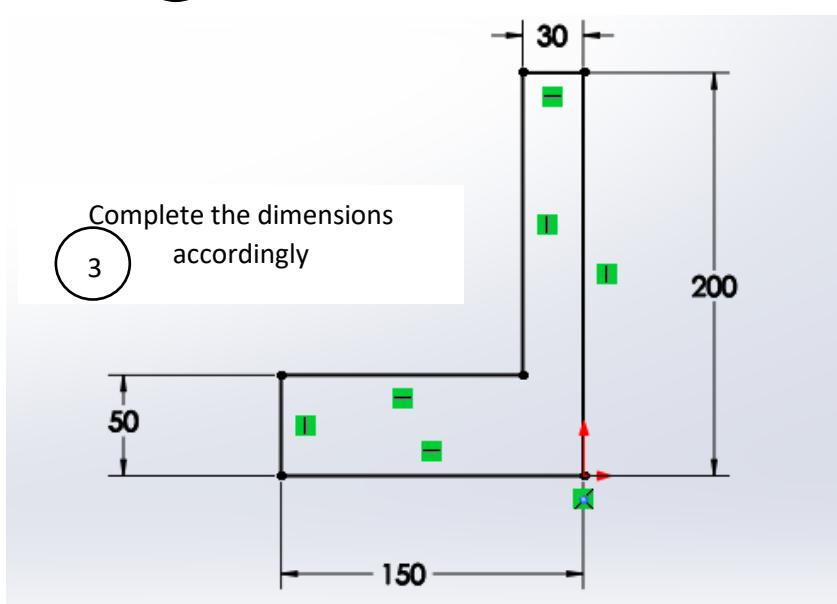
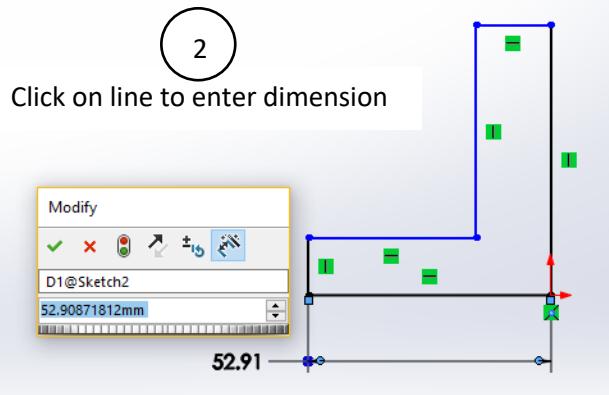
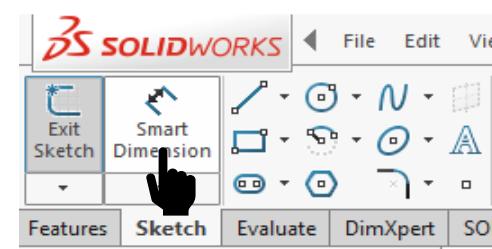
After selecting the Line option, move the cursor to front plane drawing environment.

To draw a line, start by left-clicking and line stops upon the next left-click. But a new line begins at the end point until you complete a closed figure. You can stop this continuing line by double left-clicking at this step.

- d) Start from the origin. Draw a L-Shape with arbitrary lengths as shown in the sketch. You can always change dimensions later. Importance is to sketch the “Shape” properly

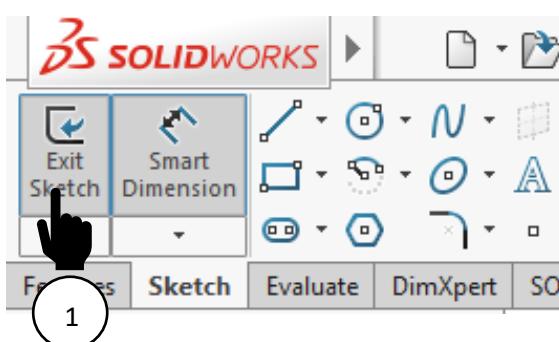


e) Dimensioning

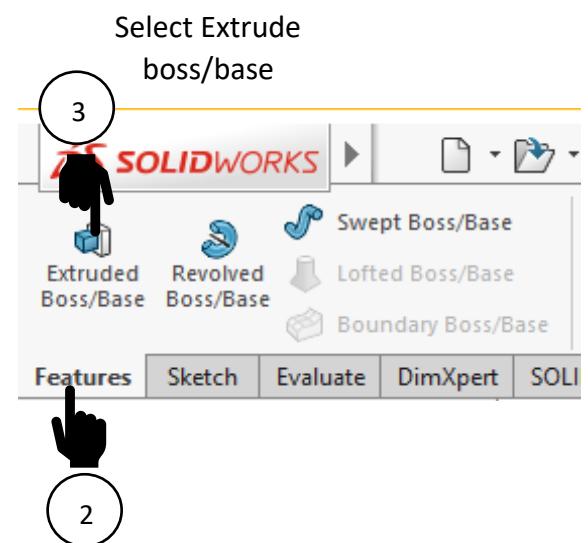


It is important to do dimensioning until the sketch is fully defined. When it's not fully defined, the lines which are not defined are indicated in blue color. Once the picture is fully defined all the lines turn black. It is also denoted in the bottom bar of the SolidWorks interface

3. Boss Extrude

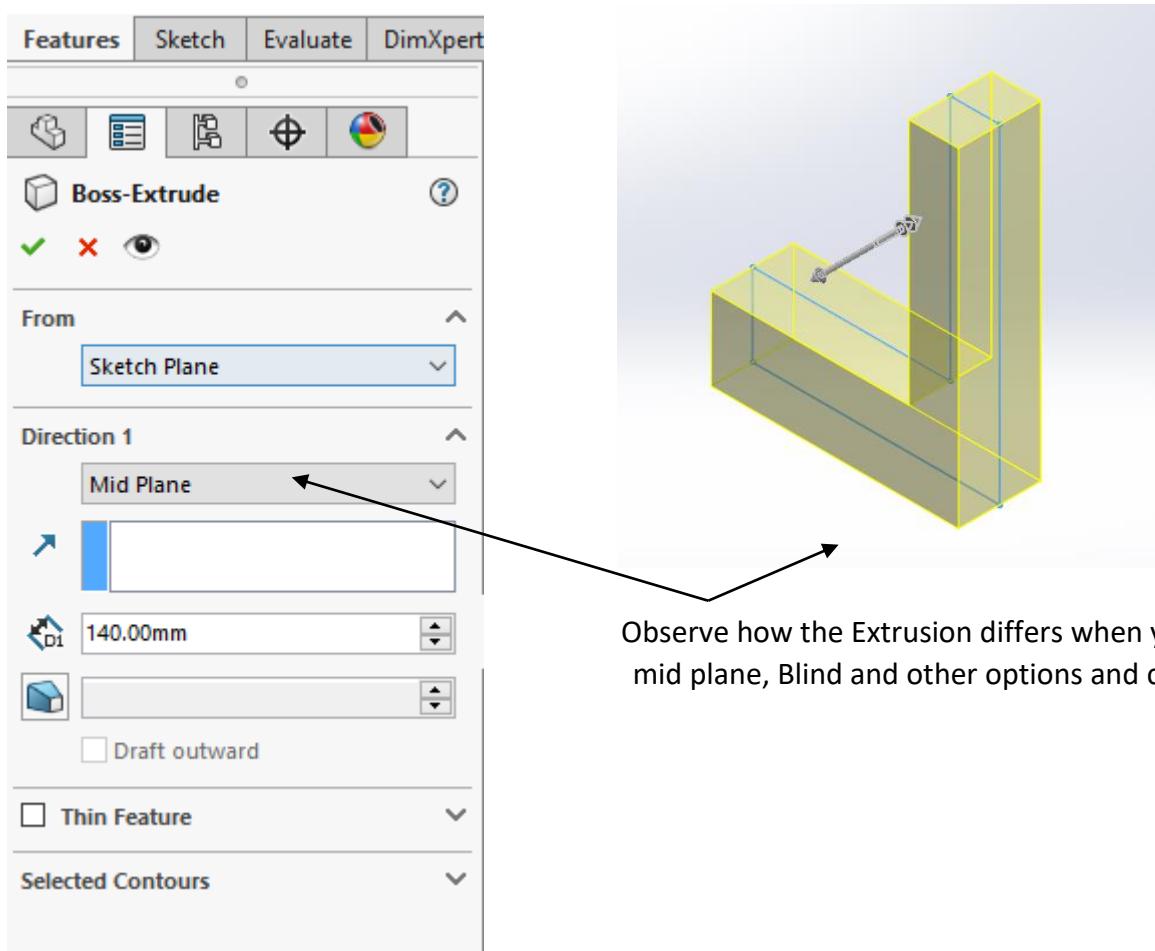


Exit Sketch first!



Click on Features

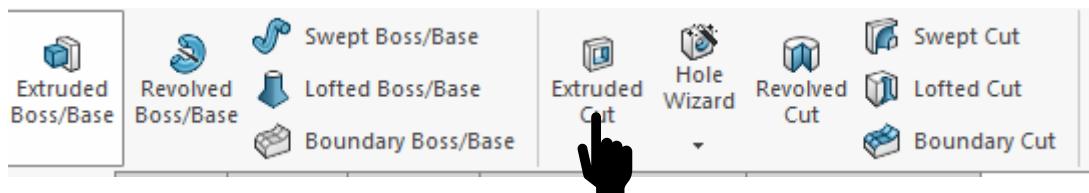
- f) Set the boss extrude settings as follows. Extrude the surface 50m from Mid plane.



Observe how the Extrusion differs when you select mid plane, Blind and other options and compare.

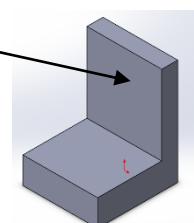
- g) Upon completion of step, click the green “” symbol,

4. Cut Extrude

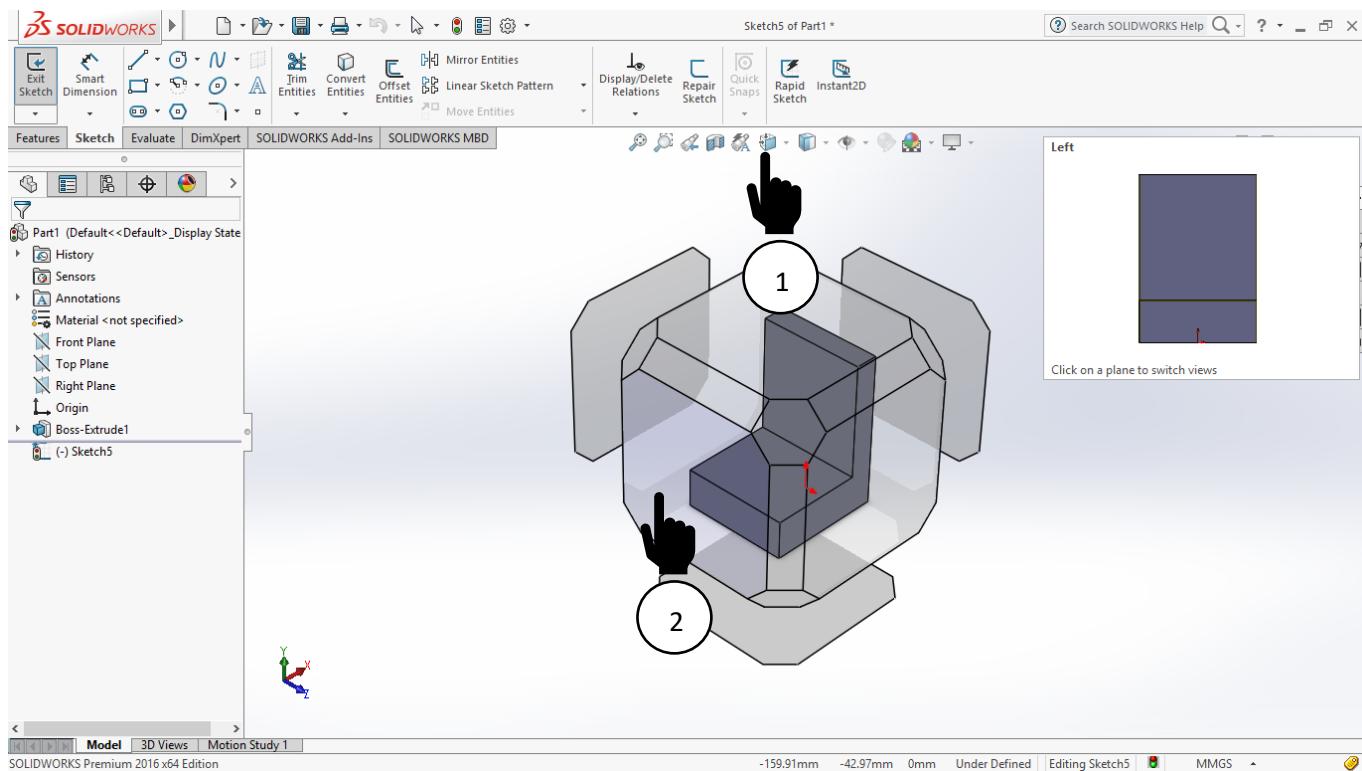


Click on Extruded cut

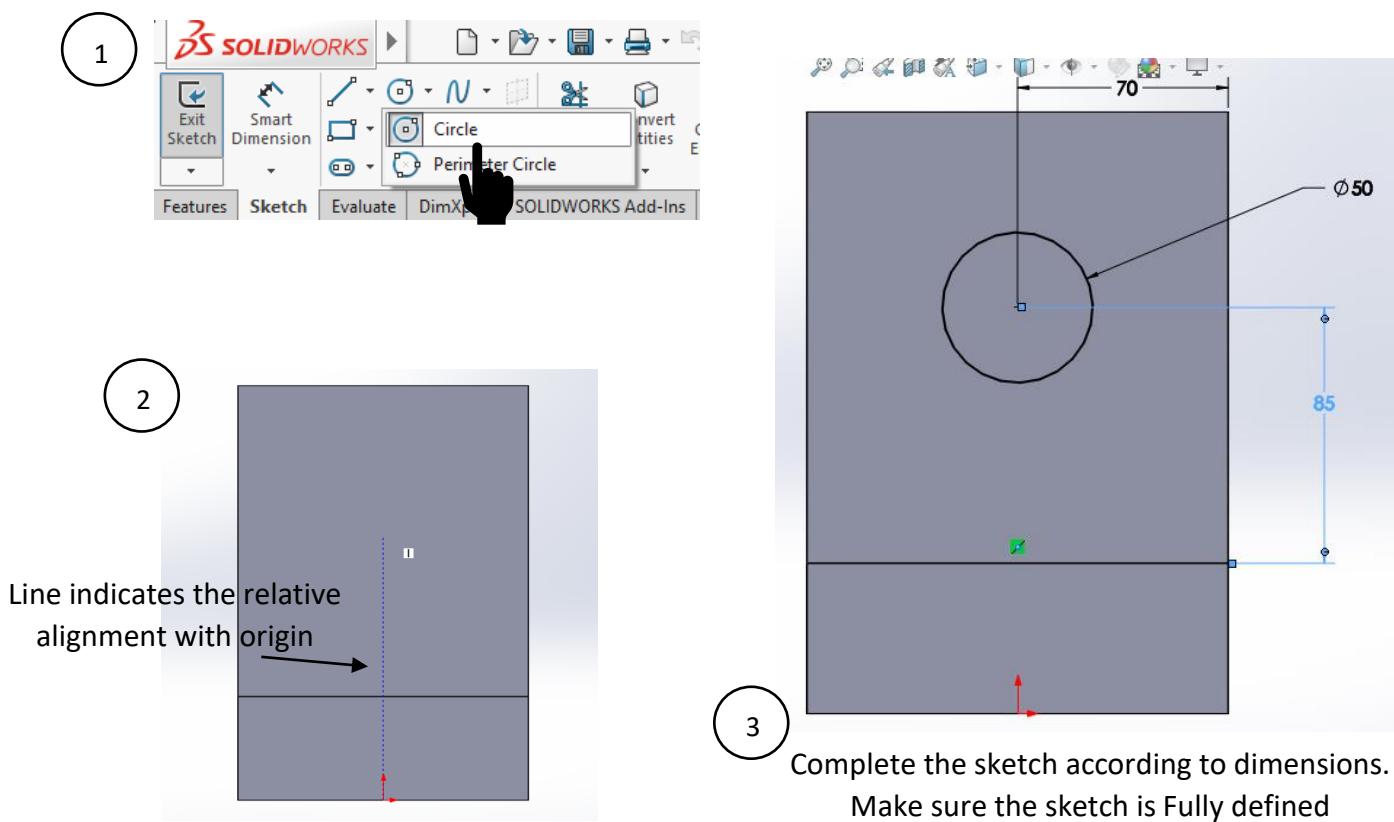
- a) You must select a surface to draw the shape you want to cut from the surface. Refer to the message displayed on the “Property Manager” Window
- b) Left Click and Select the surface as shown in the Diagram.

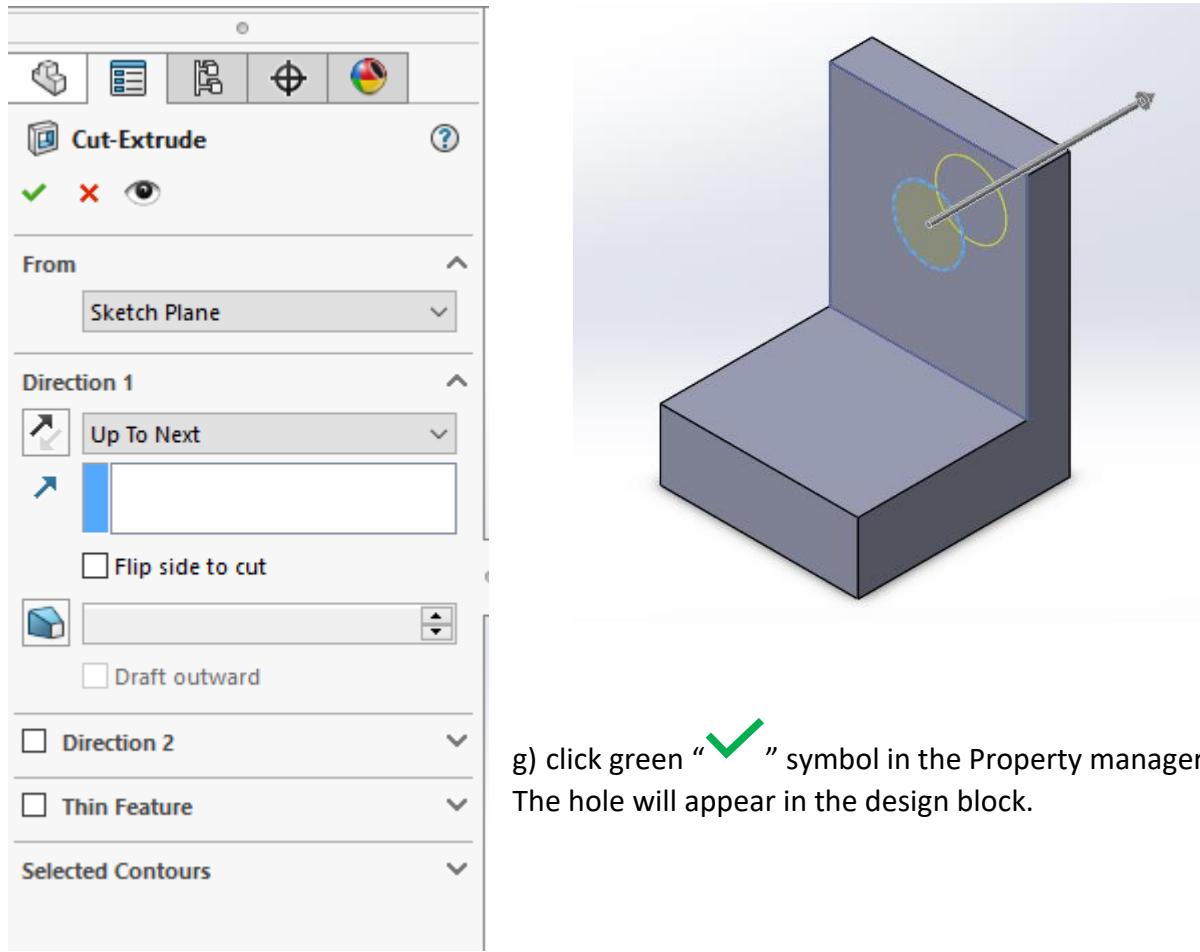


- c) Change view Orientation so that you get a clear view of drawing surface perpendicular to the screen



- d) Now draw a circle on the surface. The center of the circle must be vertically in line with the origin.

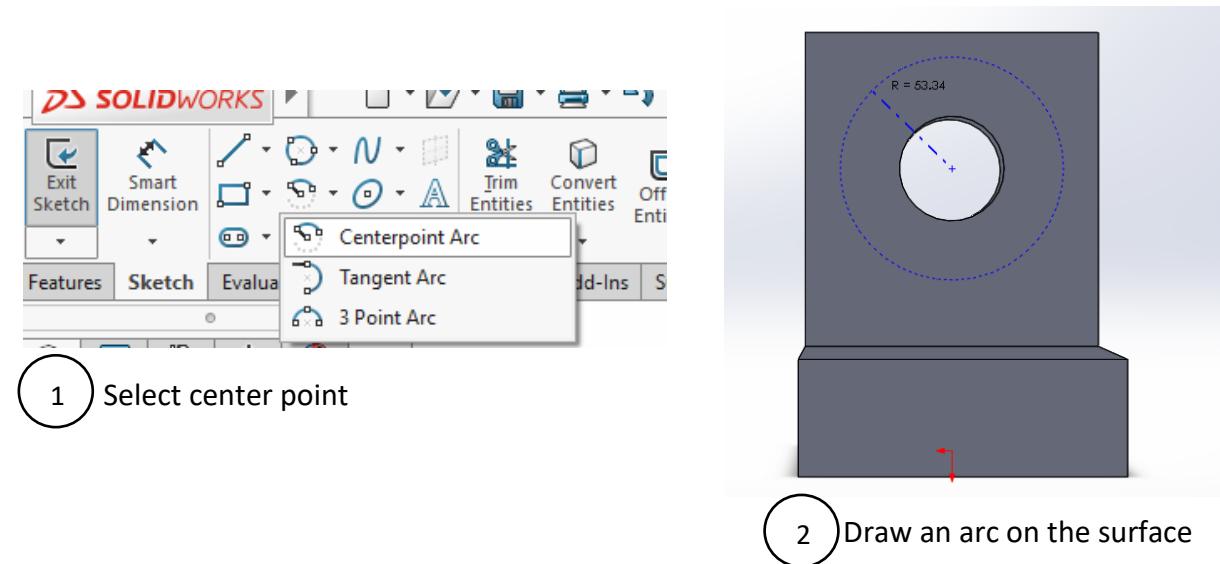


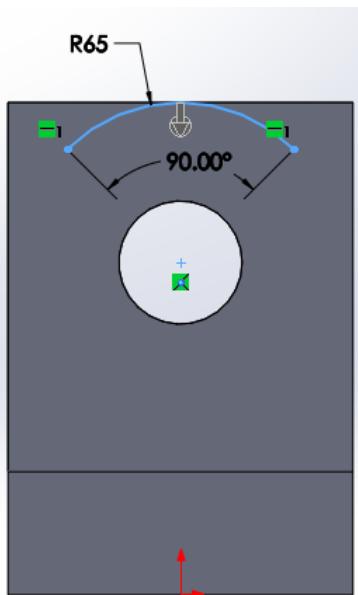
- e) Exit the sketch. Remember click green “


The screenshot shows the 'Cut-Extrude' Property Manager in SolidWorks. The 'From' section is set to 'Sketch Plane'. Under 'Direction 1', the 'Up To Next' option is selected. There is a checkbox for 'Flip side to cut' which is unchecked. Below it is a 'Draft' section with a checkbox for 'Draft outward' which is also unchecked. There are sections for 'Direction 2', 'Thin Feature', and 'Selected Contours'. To the right of the Property Manager is a 3D view of a gray rectangular block with two circular holes. One hole is a solid blue circle, and the other is a yellow circle. A callout arrow points from the text below to the blue hole.

g) click green “

- h) Create another cut extrude and select the same surface.
- i) Draw an Arc using the Center point arc as shown

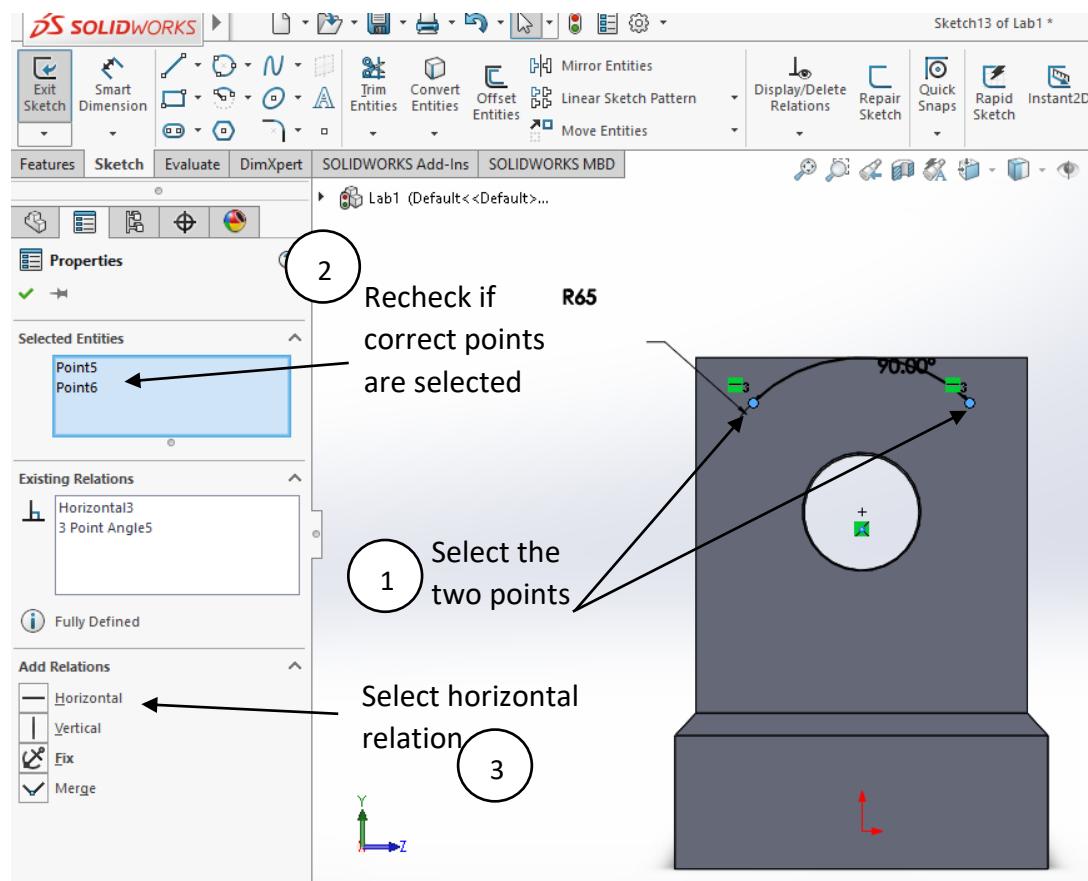




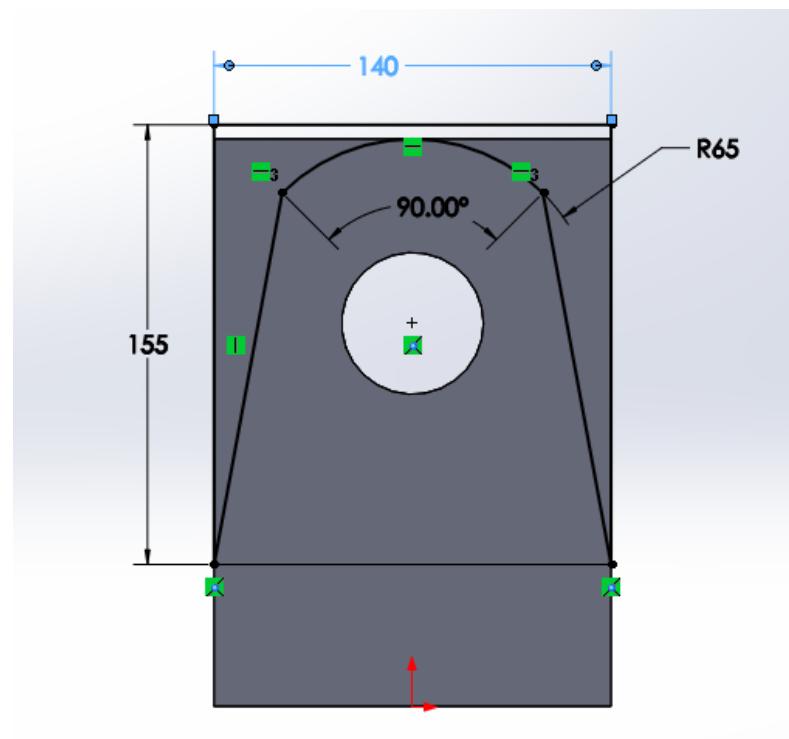
3

Use smart dimensions to set the radius to 65 units and angle to 90 degrees

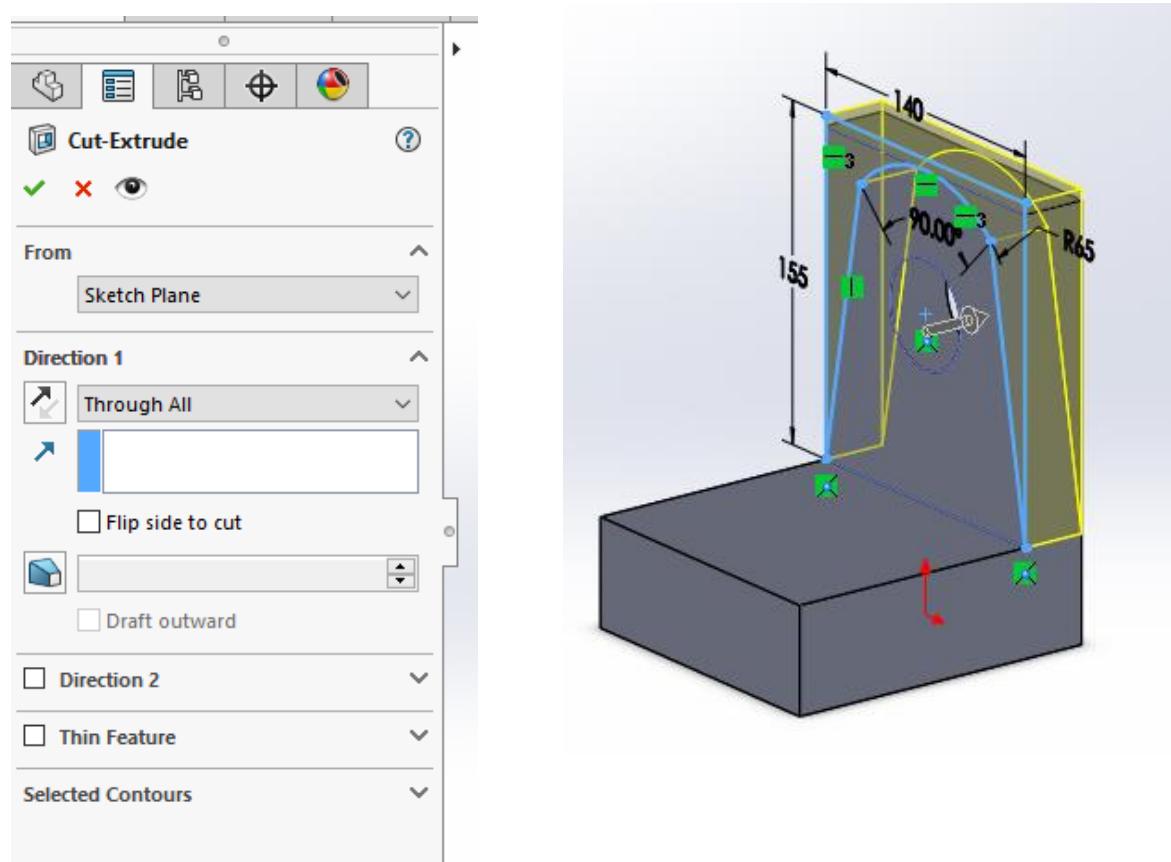
- j) Exit the Smart dimensioning. Select the two terminal points of the created arc at the same time (Control + Right click).
- k) Create a “horizontal” relation between the two points as shown below and click ✓



Complete the sketch until it is fully defined according to the dimensions displayed below

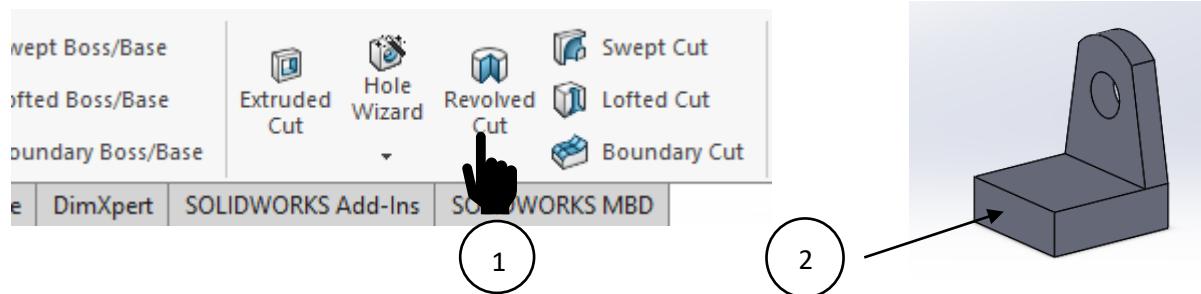


- I) Exit the sketch. You will enter the Extruded cut property manager. Select “through all” in direction menu. Select ok

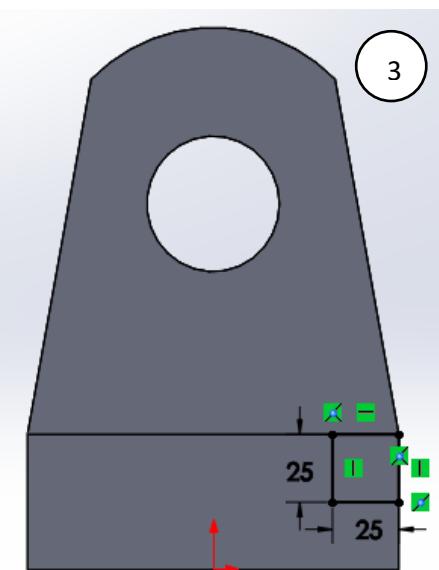


5. Revolved cut and Mirror

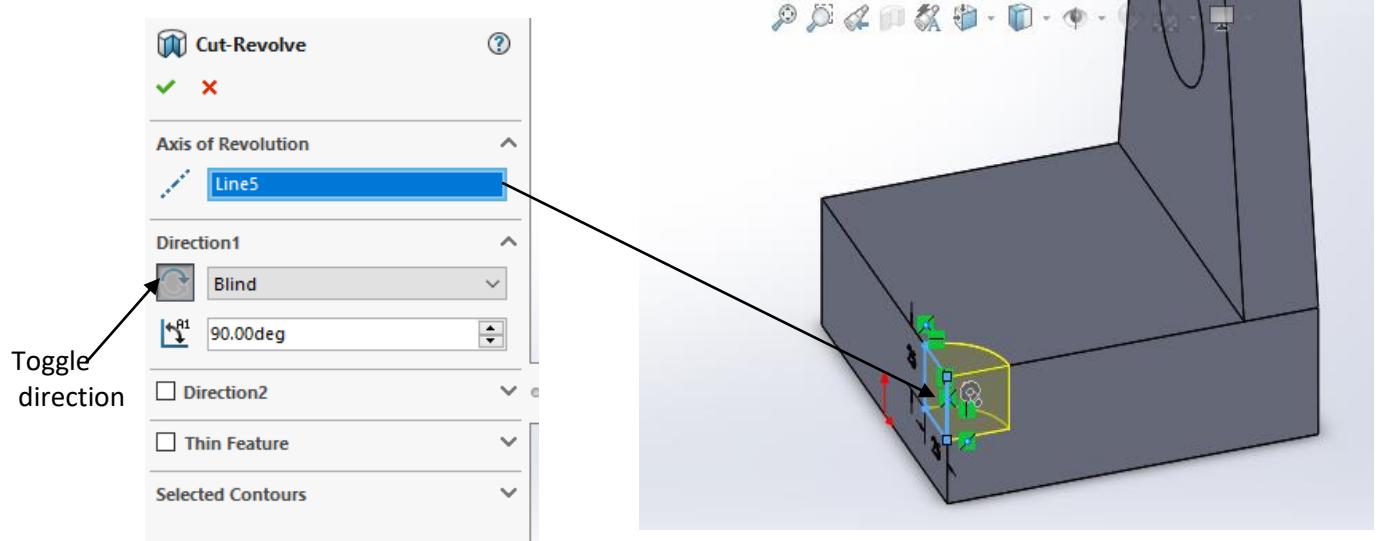
- a) Select Revolved cut feature. Select the given surface as sketch plane



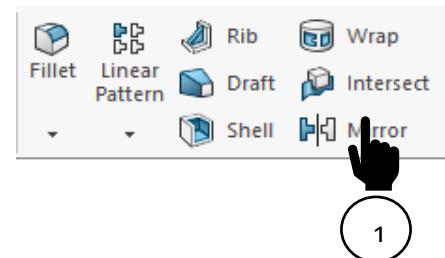
- b) Draw the sketch displayed below. (press Control + 3 to get the right plane view)



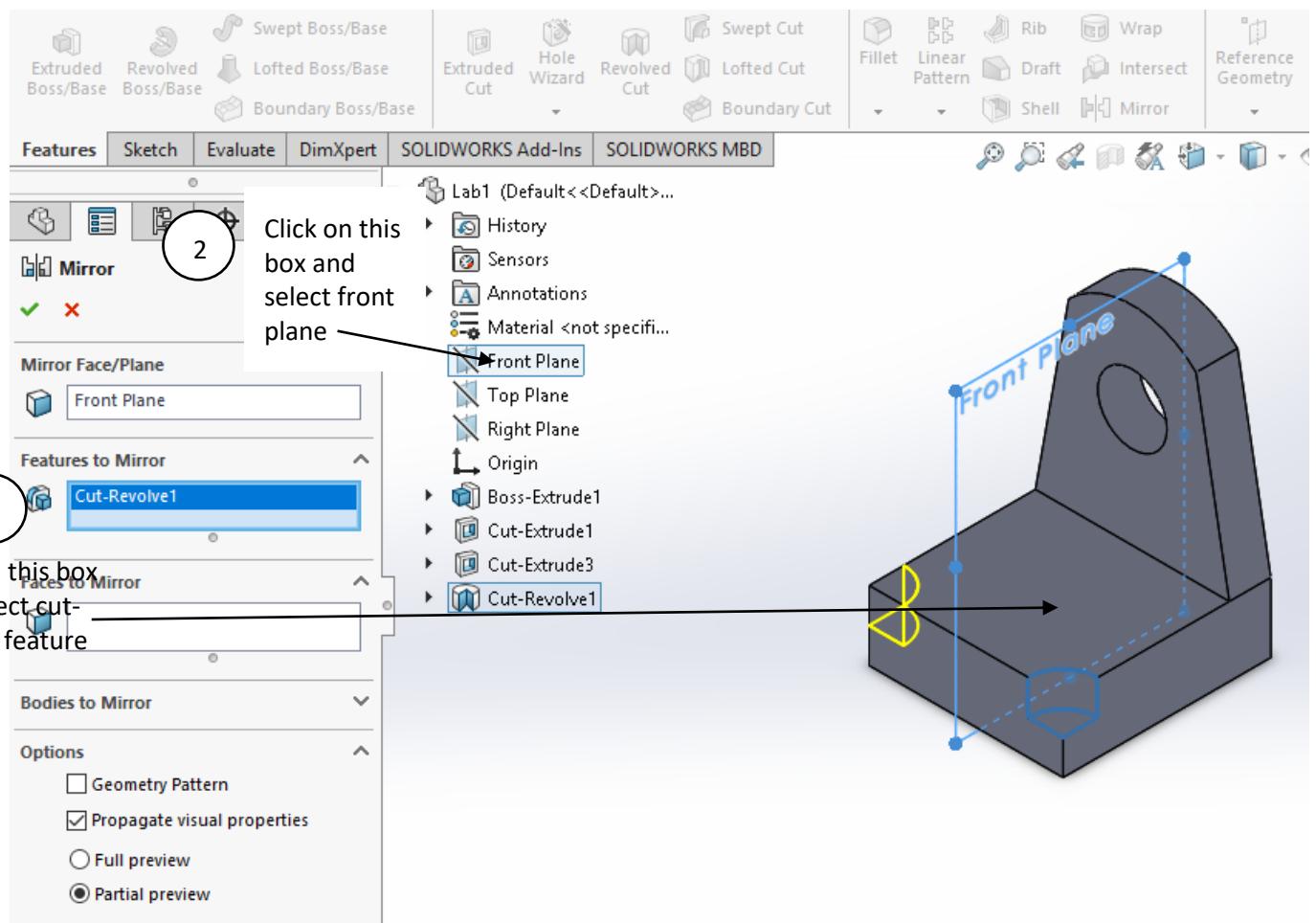
- c) Exit the sketch. You will be directed to Cut-Revolve property manager
d) Select the “line 5” the edge of the surface as displayed in figure as the axis of revolution.
Set the correct direction and set angle of 90 degrees. Click ✓ to proceed.



e) Select the Mirror feature from the toolbar

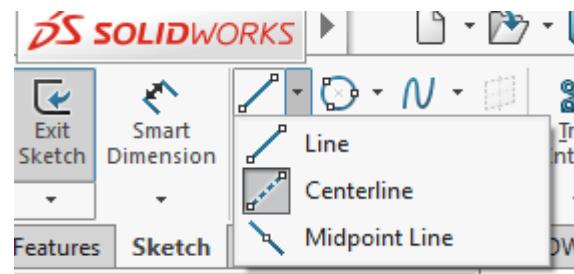
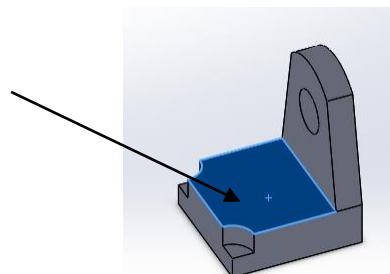


f) Then edit the Mirror property manager by selecting the relevant Mirror feature and the Mirror plane and click ✓ to continue

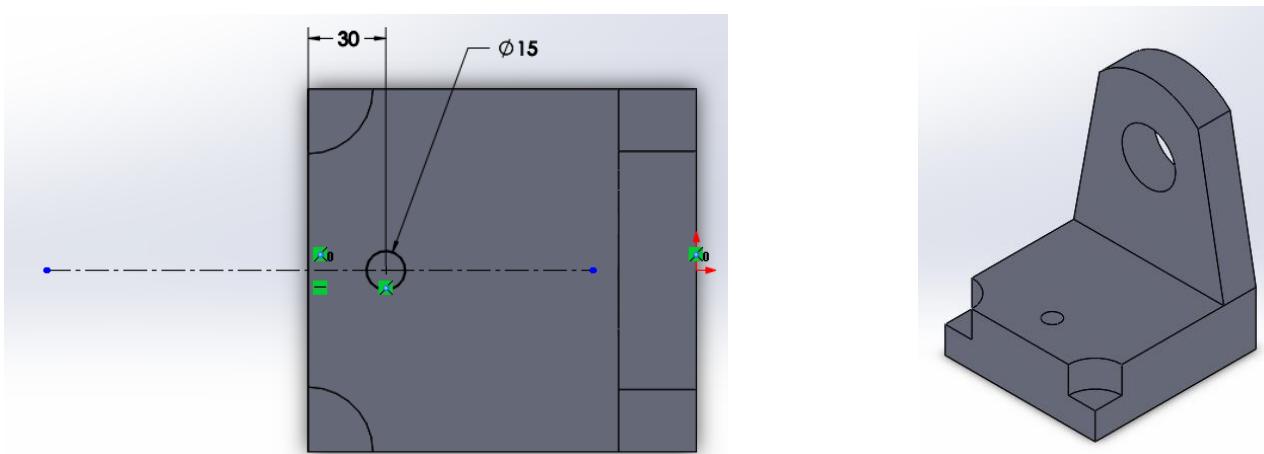


6. Linear pattern and Fillet

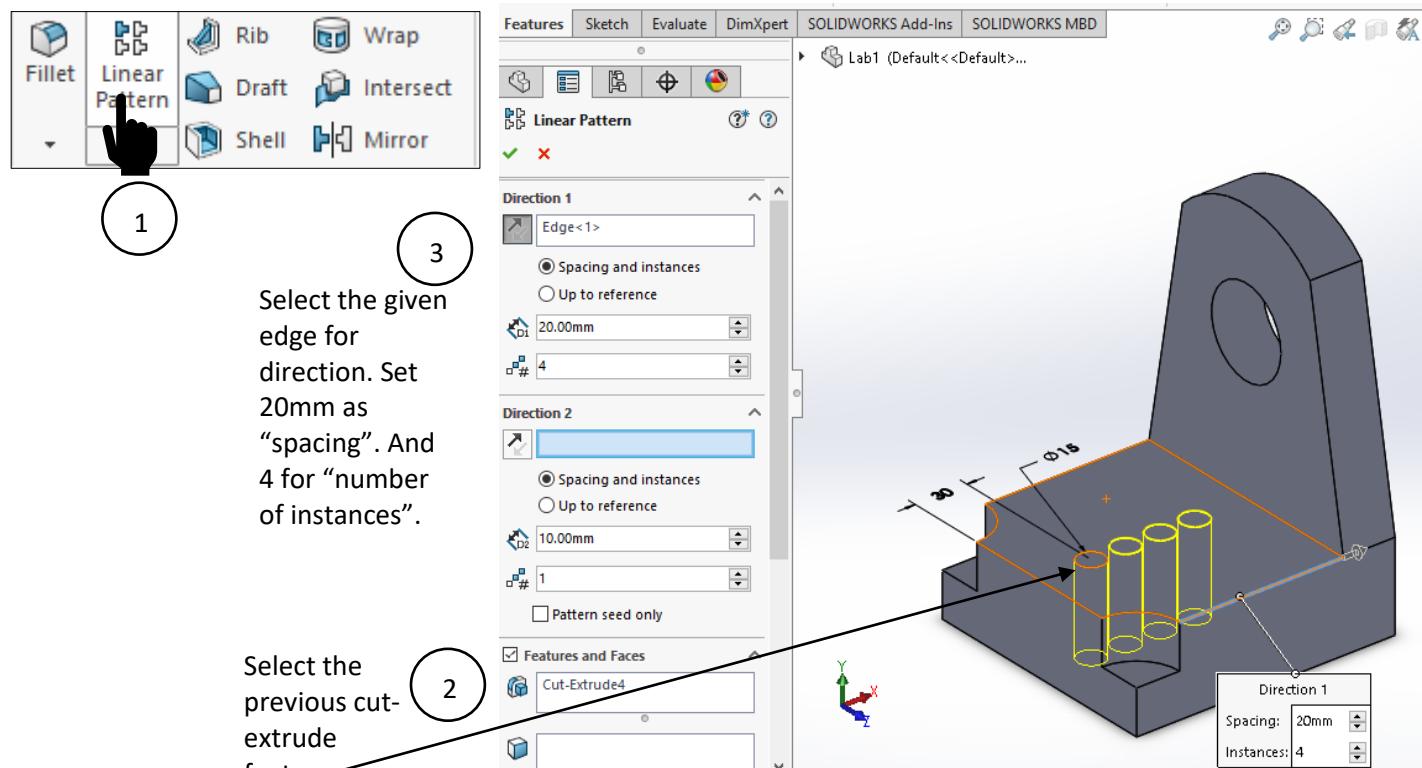
- Select the shown surface and use the previous knowledge in creating a sketch and cut extrude to create a hole.
- Draw a center line through the surface in the sketch to create the hole at middle .



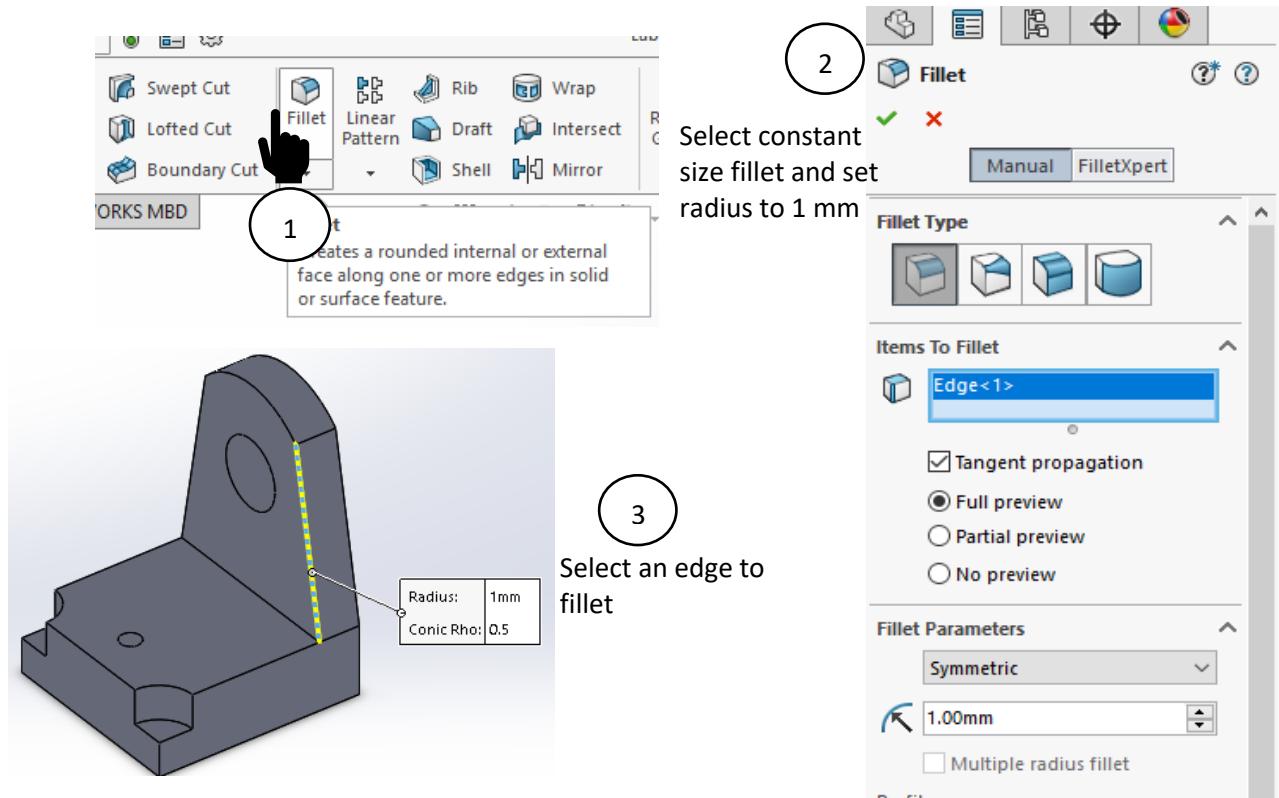
- Complete the drawing and cut-extrude. (cut extrude – select surface – sketch). Remember to cut extrude the hole “through all”. click ✓ to continue



- Click on “Linear Pattern” Feature



- e) click ✓ to continue
f) Use the **fillet** feature to smoothen the edges

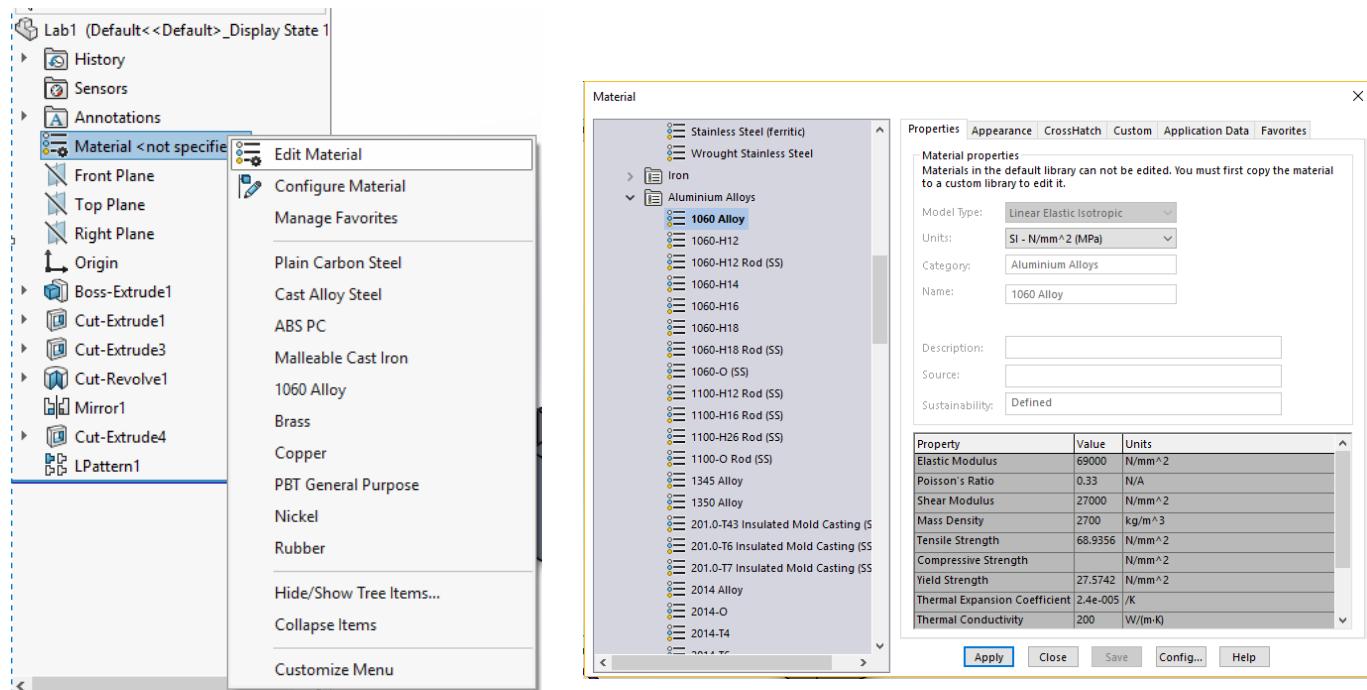


- g) Perform the fillet feature on all other edges and click ✓ to continue

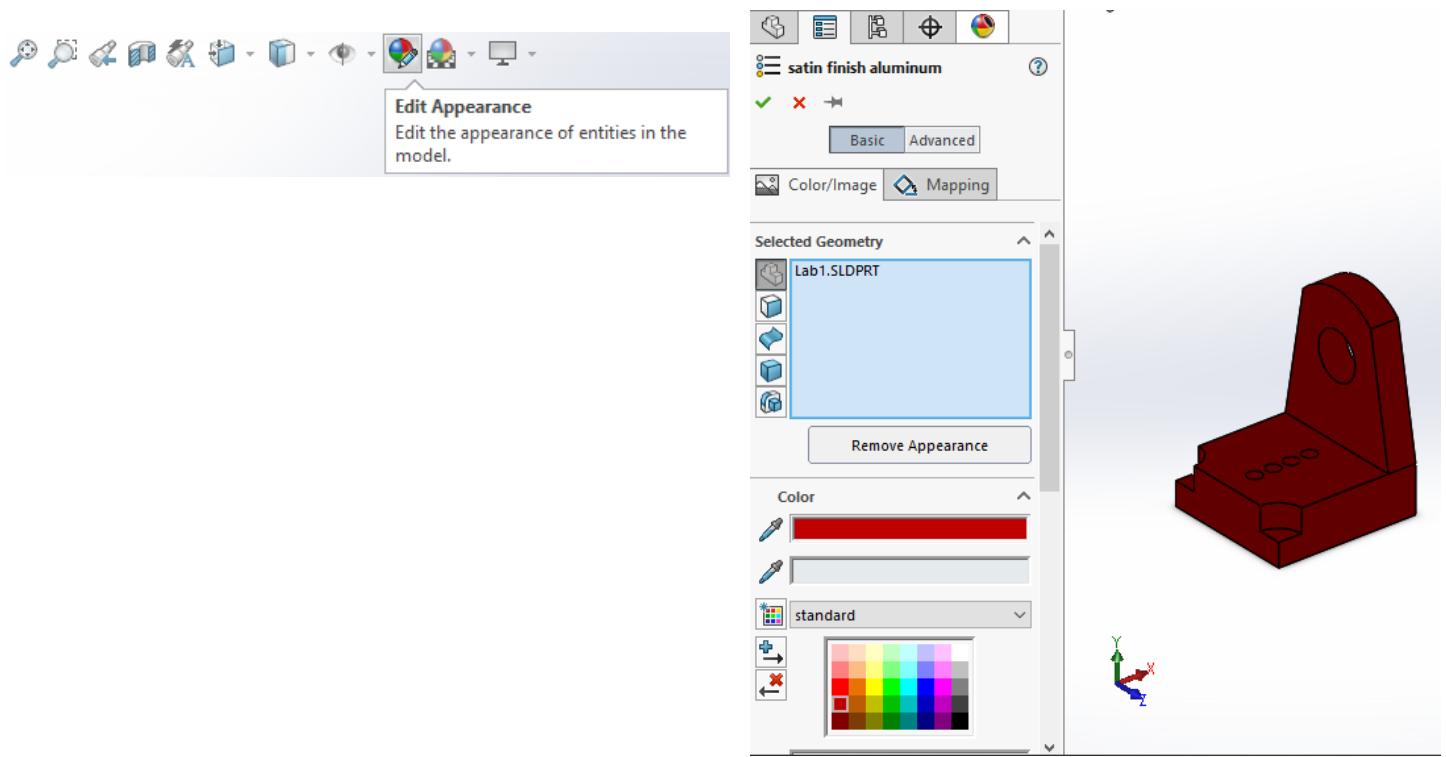
7. Mass properties

Follow the steps to change the material, color and view mass properties.

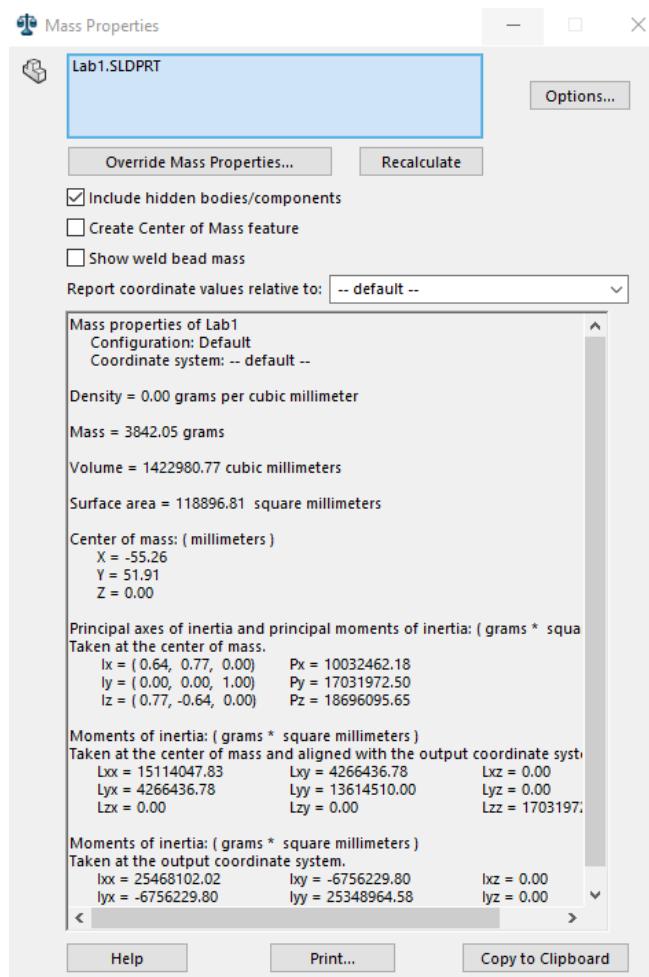
- a) Edit material from **Feature manager design tree** and select Al – 1060 alloy



- b) Click on Edit appearance. Select a color to apply to part



- c) To view Mass properties Navigate: Tools >> Evaluate >> Mass properties from main toolbar





Sri Lanka Institute of Information and Technology

Department of Mechanical Engineering

Engineering Drawing – ME2031

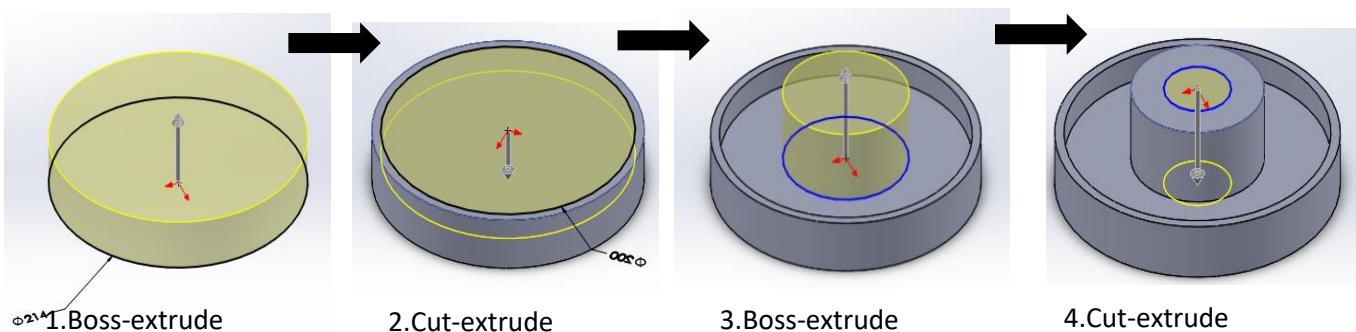
SolidWorks Laboratory - 2

Mr. Thilina Weerakkody
Mr. Kulunu Samarakrama

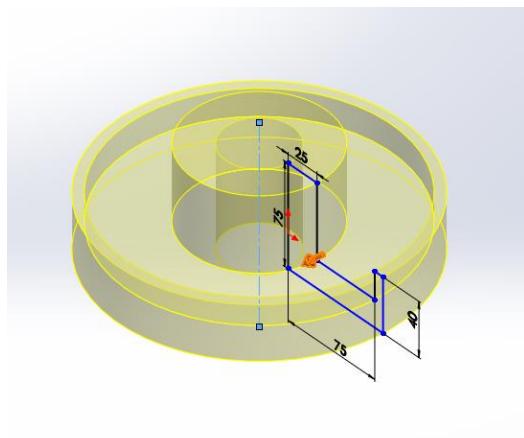
Time: 2 hours
Date : 26/02/2019

Targeted out comes of this lab

- Modelling approaches
 - Revolve feature vs Extrude Feature
 - Analyzing the drawing
 - Construction lines for constraints
 - SolidWorks assembly and Mate feature
- Observe the Drawing of the Flange described in the Drawing attached to the Annex of this Lab sheet (Lab – 2- Flange)
- The steps in modelling the given object can be stated as follows
- a. Modelling the shape of the body
 - b. Drilling holes in the body
 - c. Filleting the edges
- Modelling the shape of the body has two approaches. It is important to select the right approach to model your part accurately and efficiently. Look at the following modelling approaches explained

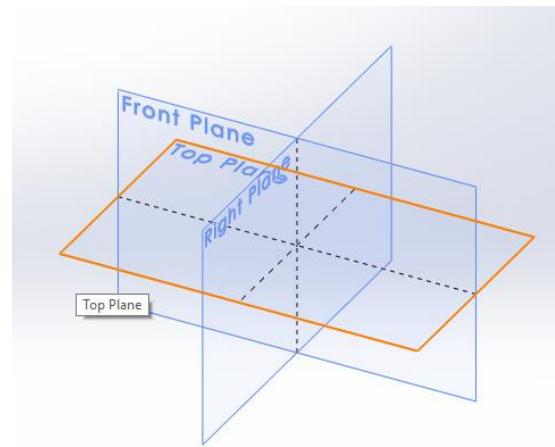


- Extrude-Boss/Extrude-cut approach
- Revolve approach

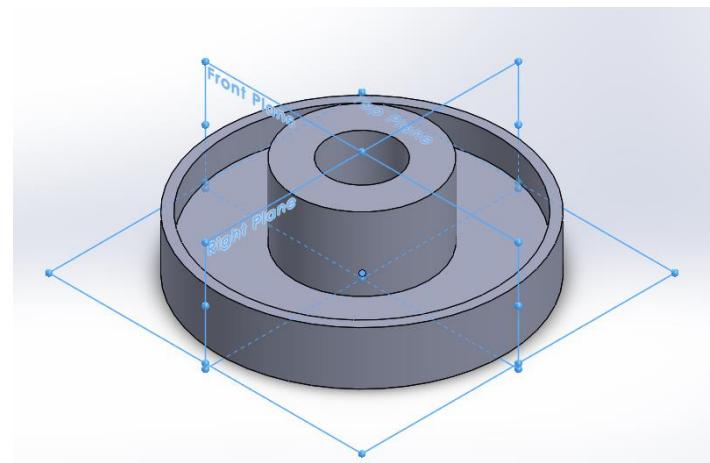


Approach 1

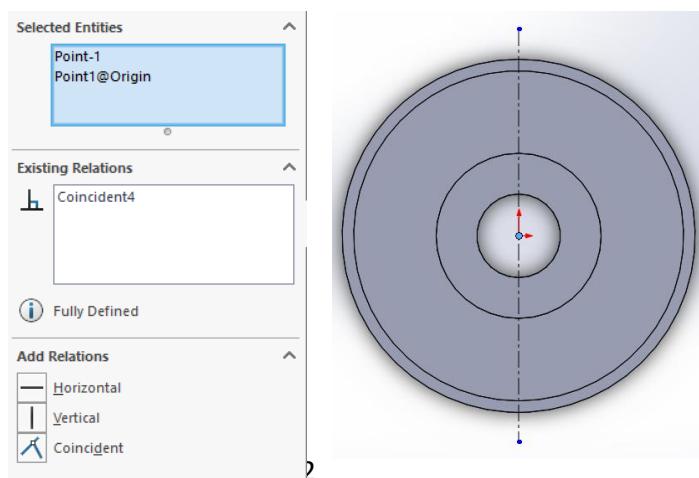
1. Create a new part file and start modelling the drawing given in the annex (Lab-2-Flange) using Boss-extrude/Cut-extrude approach. Use the knowledge from the previous lab when modelling. But follow the important steps given below as guidelines
 - a. Analyze the drawing and select the optimum drawing plane. Select the **Top-plane** in this scenario.



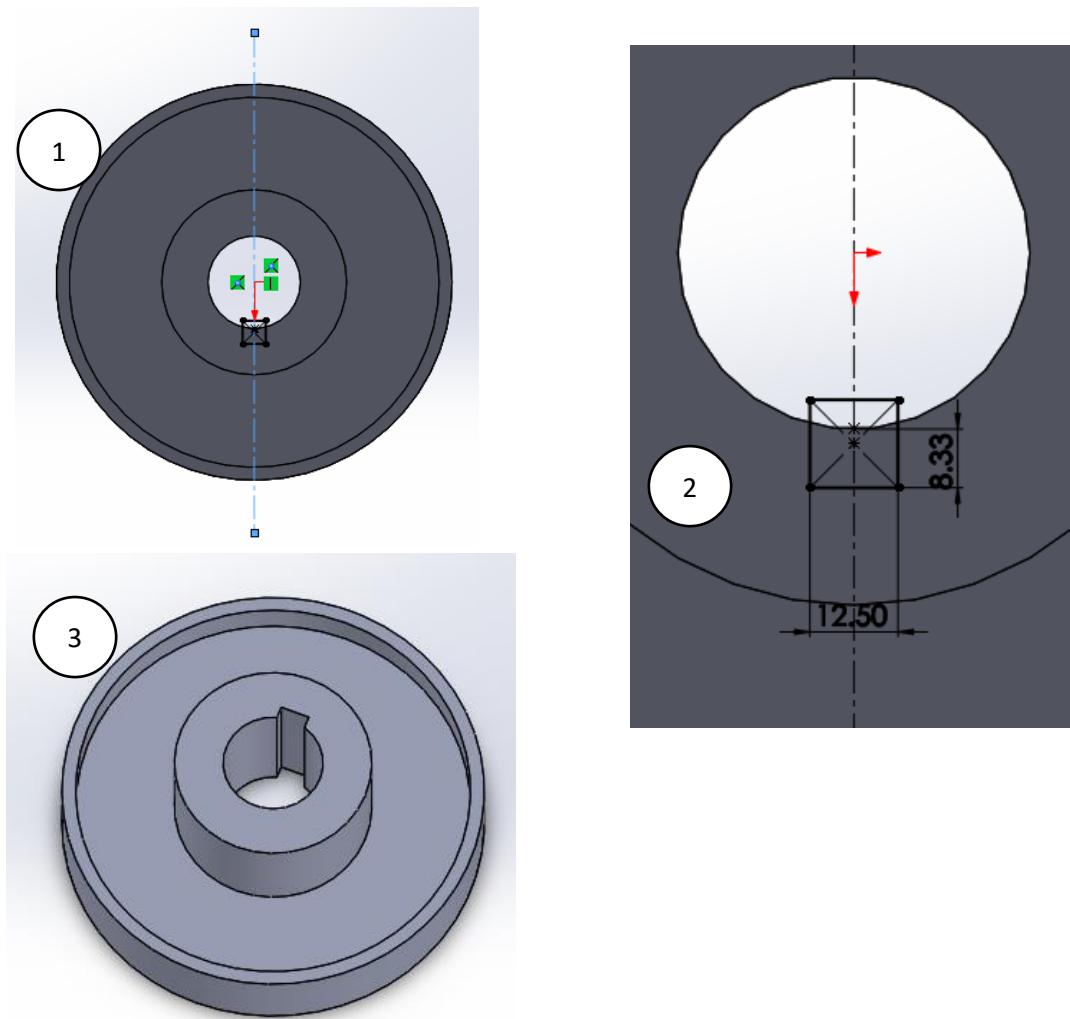
- b. Start modelling the base sketch from the origin. And continue with boss-extrude and cut extrude to obtain the following model. Make sure the sketches you draw are fully defined



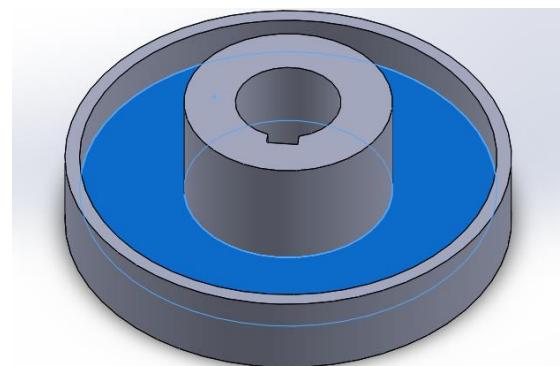
- c. To cut the guide in the inner surface of the flange draw “Center line” through the origin on top surface.



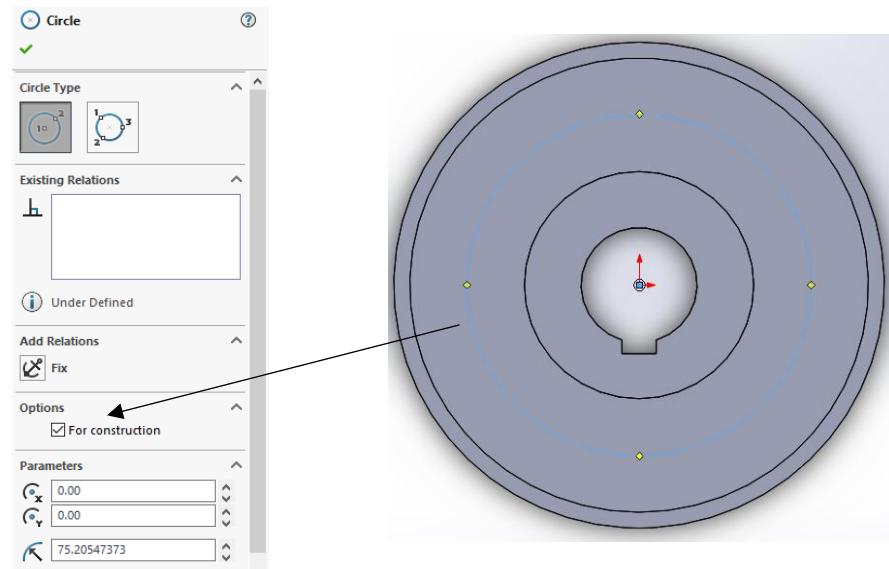
- d. Use the symmetry of the object to draw the sketch. Perform cut extrude through all.



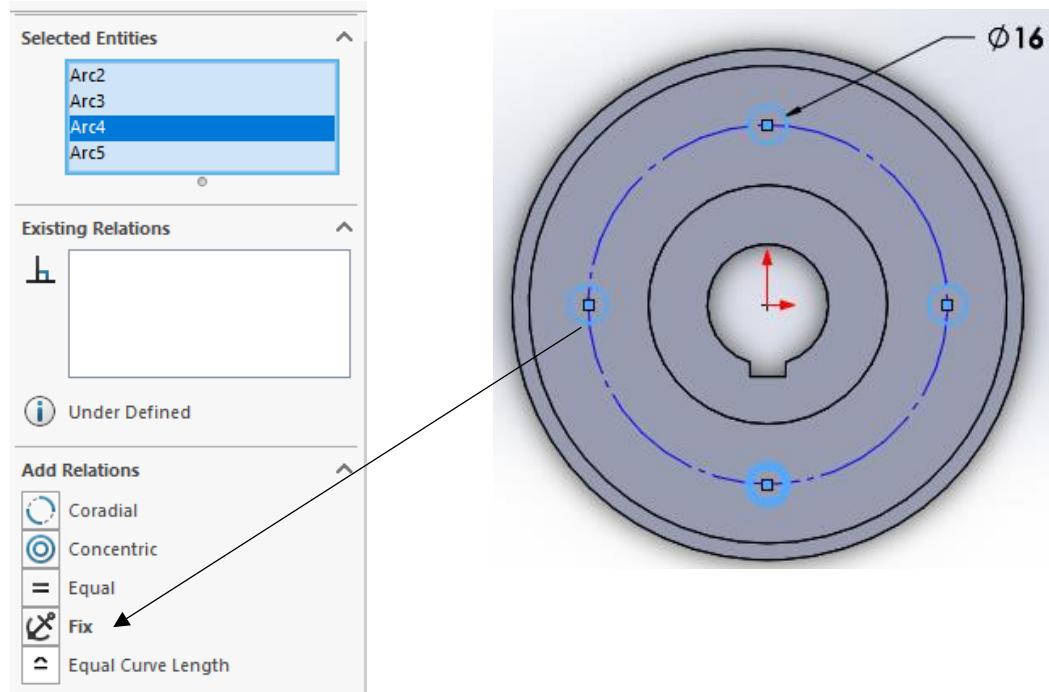
- e. Drawing the holes can be done in several ways. Either you can draw 4 individual holes and dimension them each. But since the all four holes lie symmetrically it is more convenient to draw four circles and create relationship to control them simultaneously. Select the following surface to start the sketch



- f. Draw a circle in the sketch and ark it as “For construction” in properties window



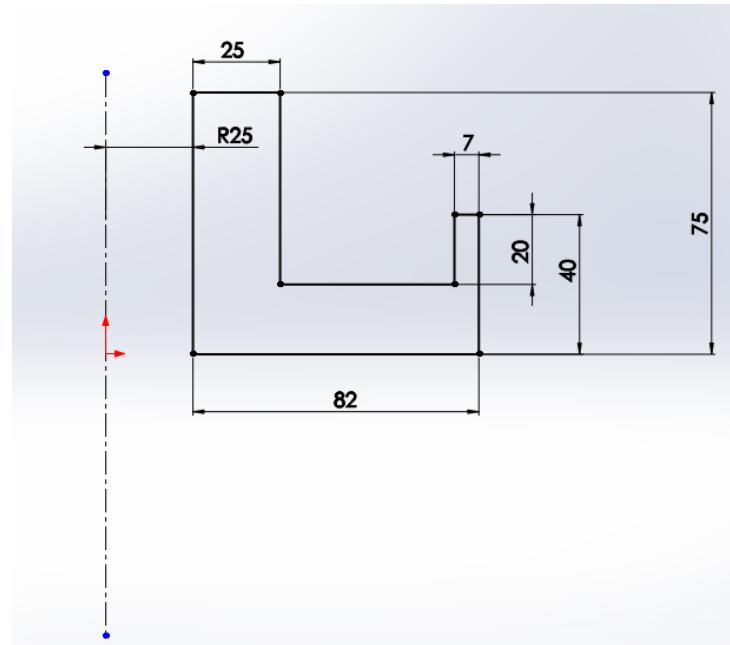
- g. Draw 4 circles of arbitrary radii on the 4 quadrants of the construction circle. And dimension one circle according to the drawing. Press control + Select the four circles and apply the relationship “Equal”. Now you can see when you drag the construction line the 4 circles move while maintaining their relationship. Now define the construction circle to fix the location of the circles so that the sketch is fully defined.



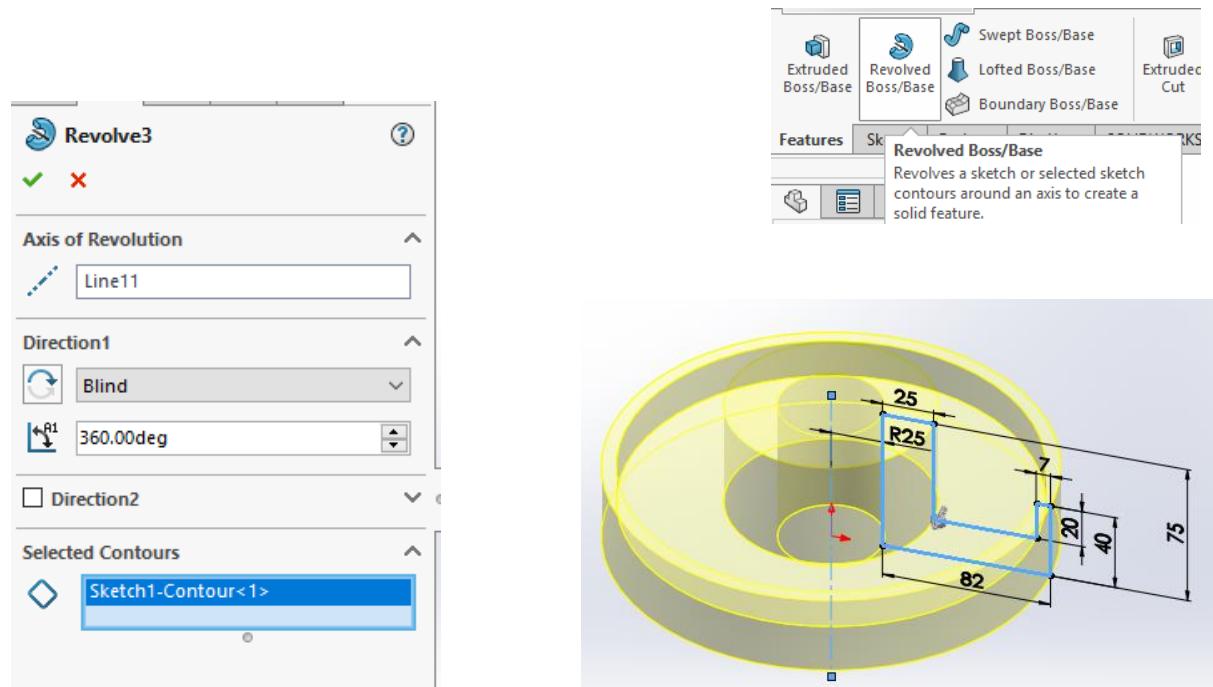
- h. Extrude cut through all to create holes
i. Do filleting accordingly by analyzing the drawing
j. Save the part file as Flange_01 in a Folder named Lab_02

Approach 2

2. Create a new part file and start modelling the drawing given in the annex using Boss-extrude/Cut-extrude approach.
 - a. Create a sketch in the Front plane as follow



- b. Click on Revolved Boss/Base from the feature tool bar

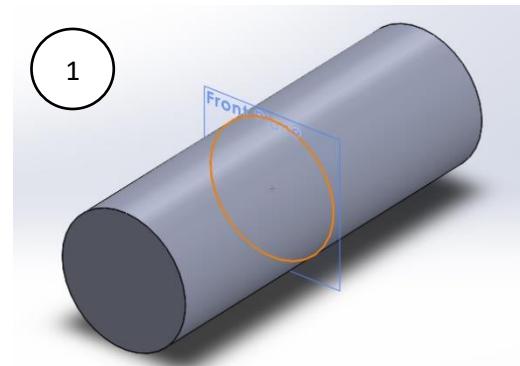


- c. Complete the model by drilling the holes and the rectangular key cut.
 - d. Perform the filleting.
 - e. Save the part file as Flange_02 in Folder Lab_02

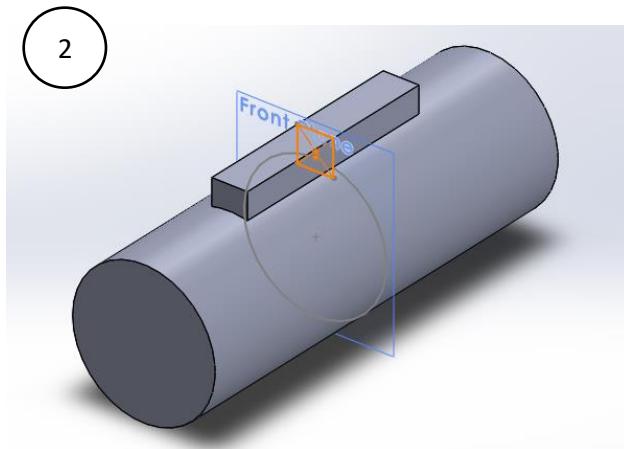
Modelling the shaft with key

1. Analyze the drawing of the shaft with key attached with annex (Lab -2- Shaft). Follow the important guidelines mentioned below in modelling the shaft key

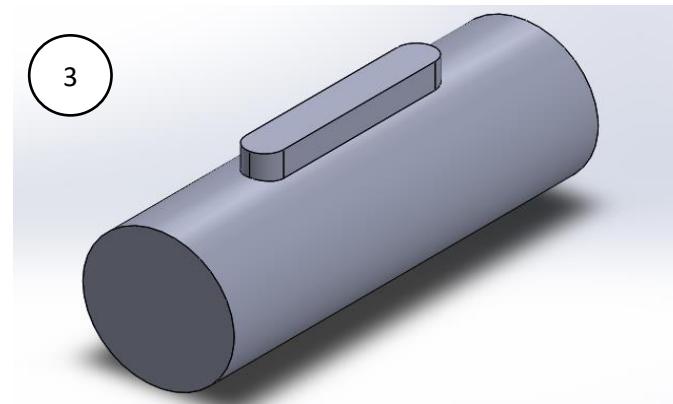
- a) Start the sketch in Front plane. Extrude the shaft first. Use extrude from midplane



- b) Draw the sketch of the key separately in the same plane and extrude from mid plane.



- c) Perform the filleting. And save the file as “shaft” in folder Lab_02.

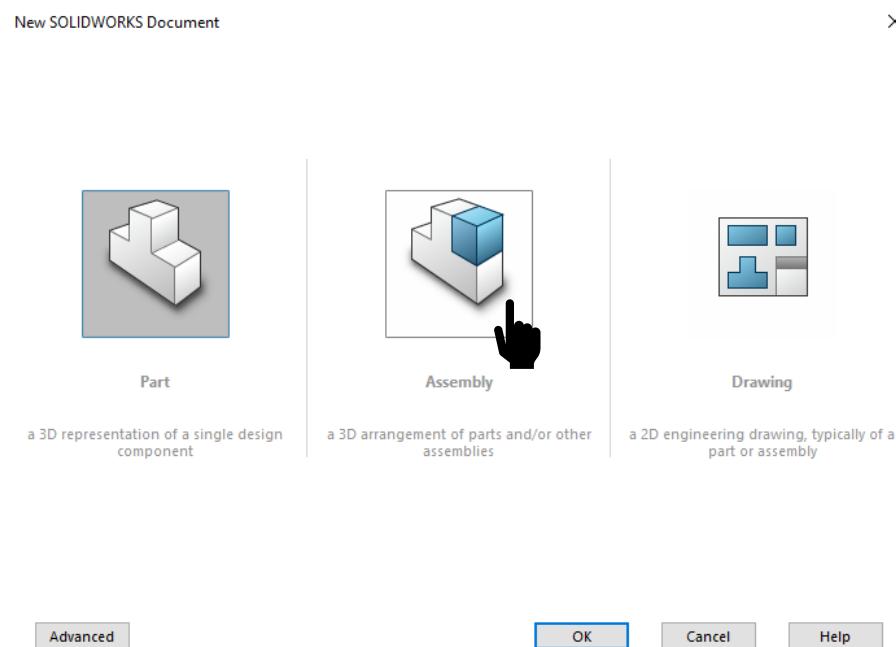


Your First Assembly!

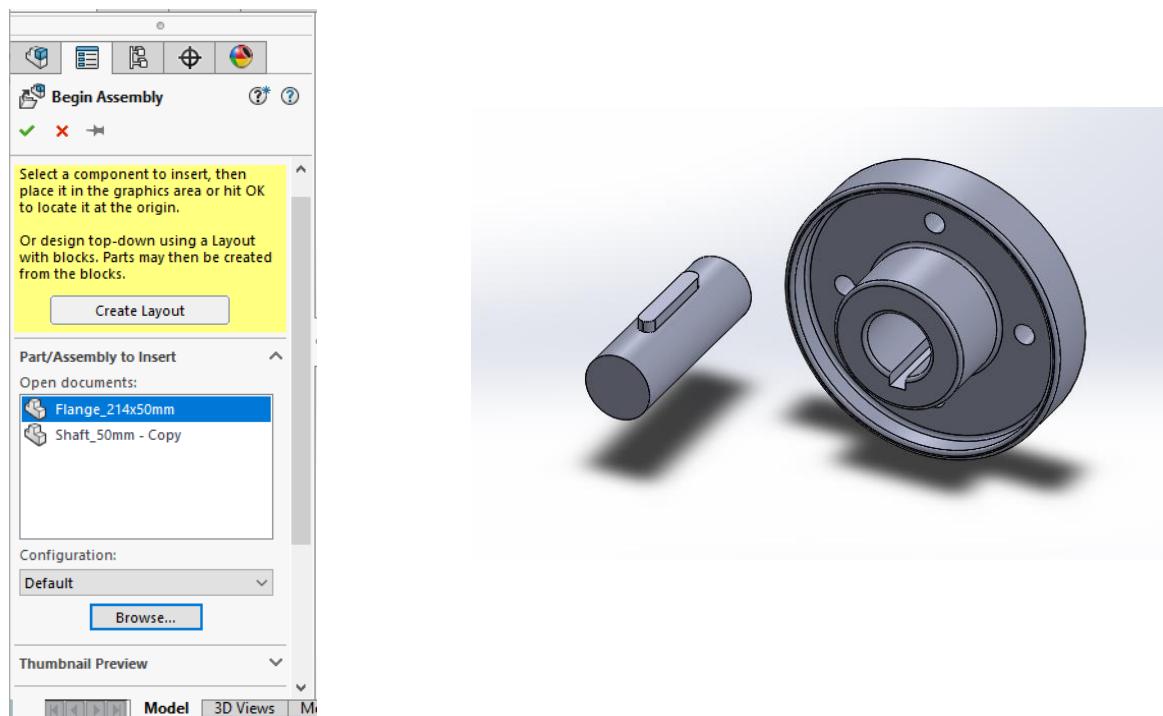
By now you must have 3-part files inside the folder named “Lab_02”

Follow the instructions given to create an Assembly.

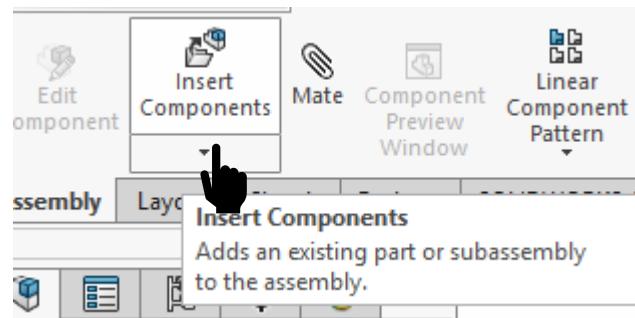
- a) File >> New >> Select Assembly



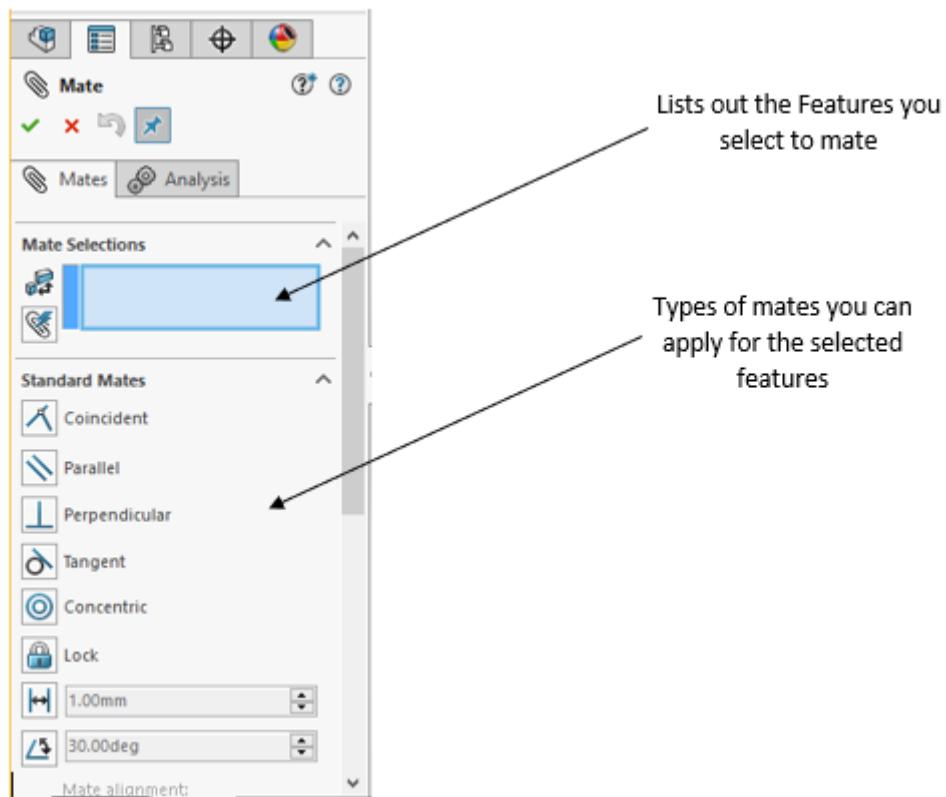
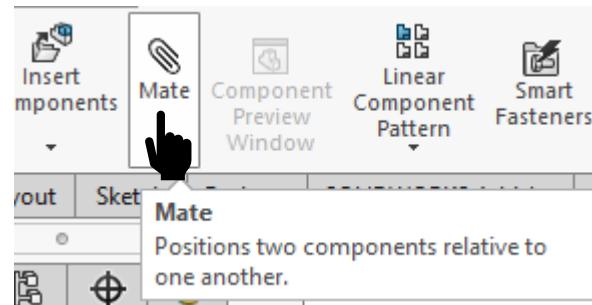
- b) As you open the assembly, a dialog box will appear on the left corner of the window. There your part files are listed. In case they are not listed, you can browse the folder and open the parts. Select each part and place them on modelling environment



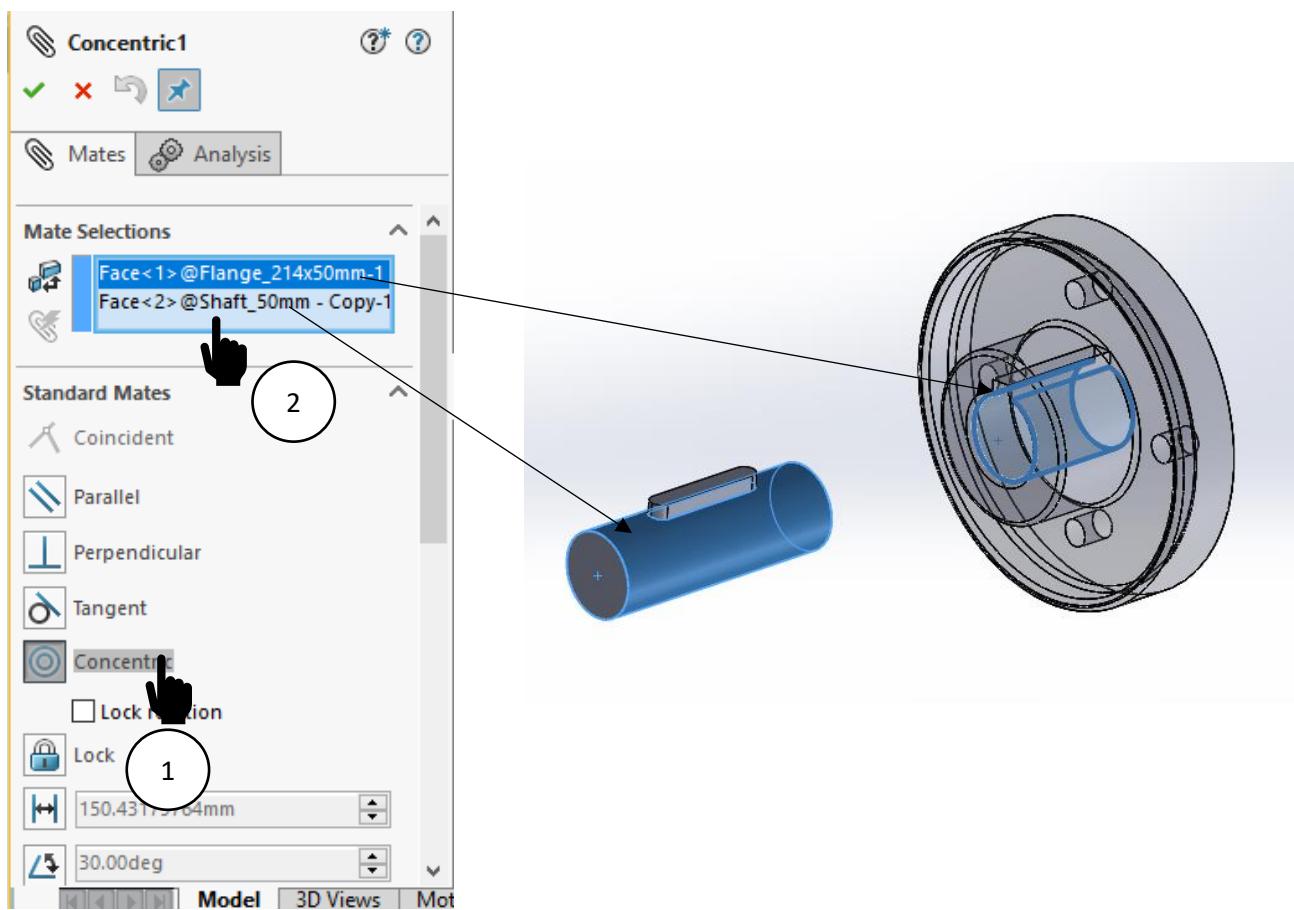
- c) Alternatively, you can insert part files from the tool bar



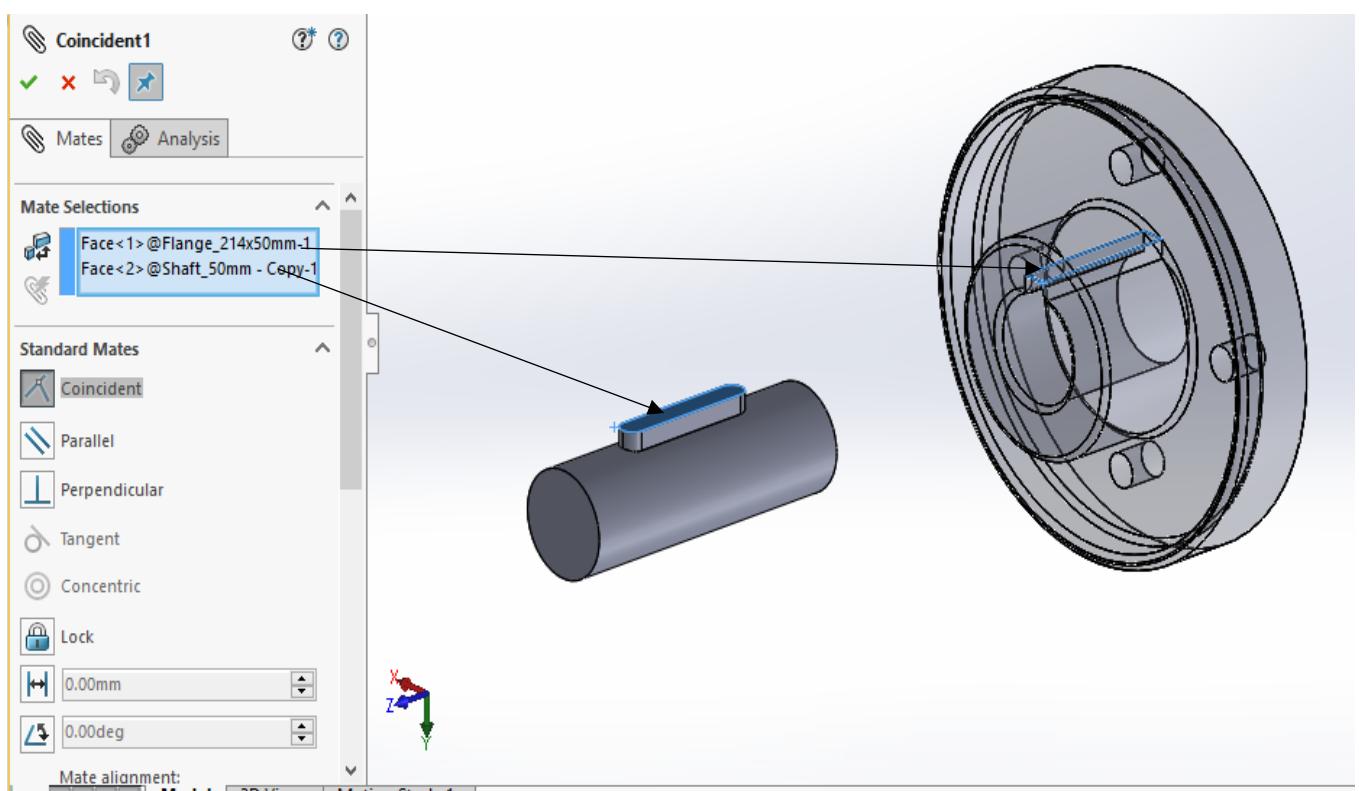
- d) Mate function: Mate function allows you to apply a set of rules that governs the physical appearance of part files relative to each other



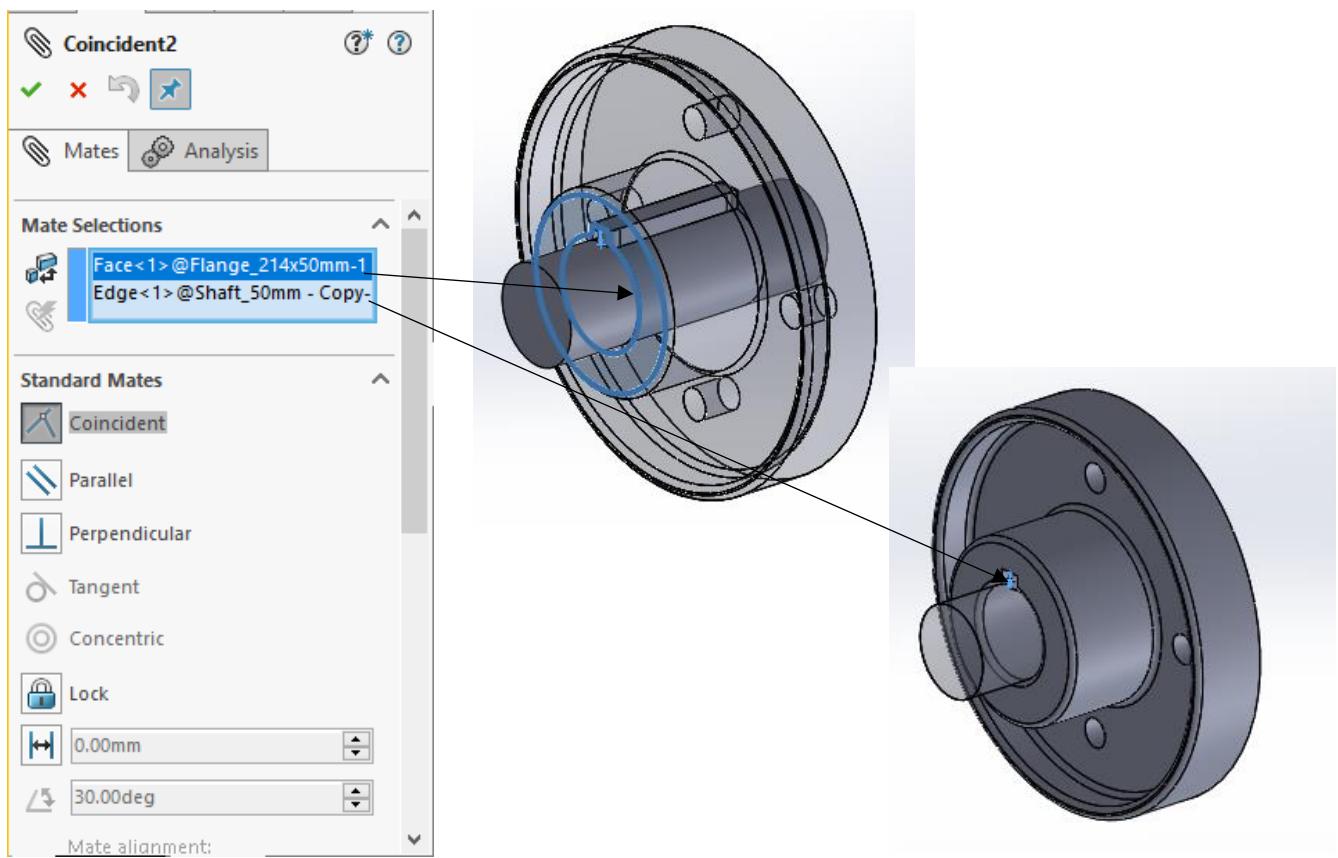
e) Apply the **concentric mate** for the following two curved surfaces as shown below



f) Apply coincidence mate for the key as shown below

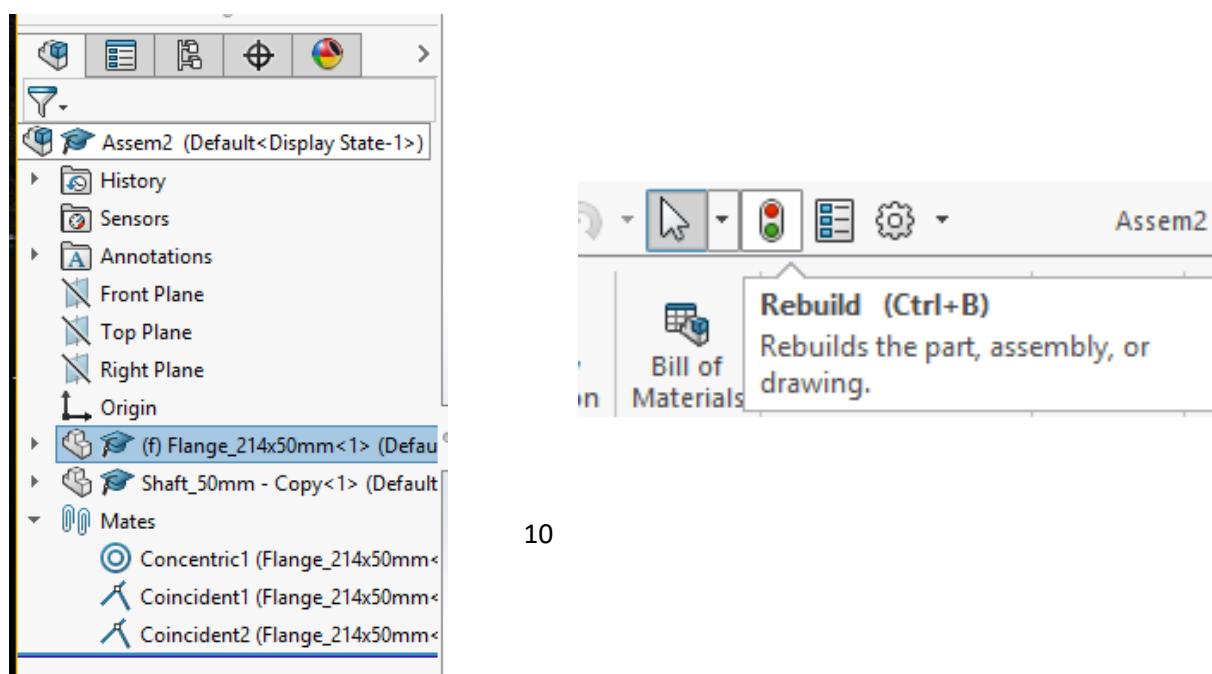


- g) Apply another coincidence mate for the following features as shown



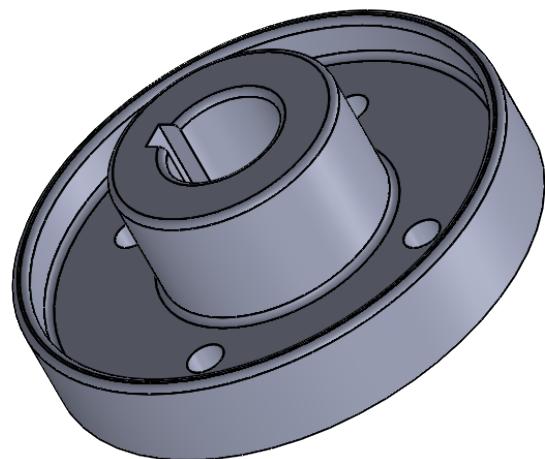
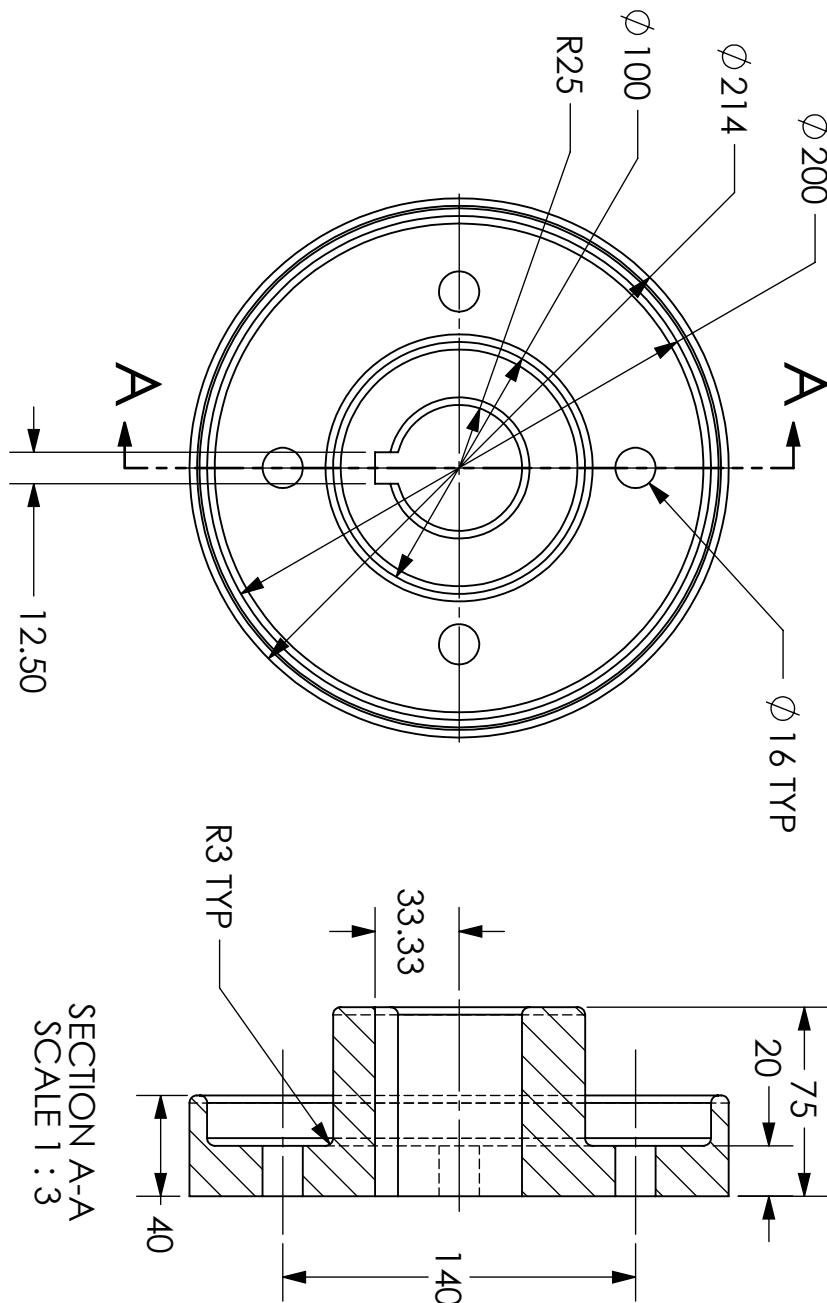
- **Feature manager design tree** lists out the parts, mates and many other details related to the assembly. To edit any part-file you must right click on the parts and open them separately.
- After editing part-files you must rebuild the assembly to update any changes. Save the assembly in the folder “Lab _02” as Assembly1
 - h) Edit the colors of the two parts to two different colors so that assembly is clear for the vision

Feature manager design tree





PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRI LANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND ANY REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
IS PROHIBITED.



Lab - 2 - Flange

APPLICATION	USED ON	FINISH	DO NOT SCALE DRAWING
TOOL	MATERIAL Steel	INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL	

UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MM
TOLERANCES:
FRACTIONAL MIA
ANGULAR: MACH $\pm 0.1^\circ$ BEND $\pm 1^\circ$
ONE PLACE DECIMAL
TWO PLACE DECIMAL

DRAWN	NAME	DATE
CHECKED		26/02/2019
ENG APPR.		
MFG APPR.		
Q.A.		

COMMENTS:

SIZE DWG. NO. **A** REV **A**
SCALE: 1:5 WEIGHT: SHEET 1 OF 1

PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND ANY REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
OF SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY IS PROHIBITED.



UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MM

TOLERANCES:

FRACTIONAL N/A

ANGULAR: MACH $\pm 0.1^\circ$

BEND $\pm 1^\circ$

ONE PLACE DECIMAL

± 0.1

TWO PLACE DECIMAL

± 0.01

MACH APPR.

BEND APPR.

ENG APPR.

MFG APPR.

Q.A.

CHECKED

DRAWN

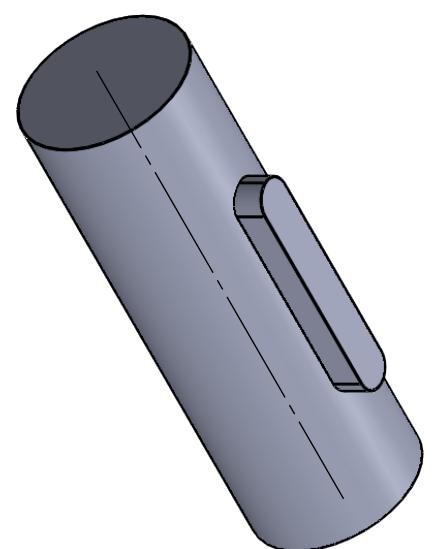
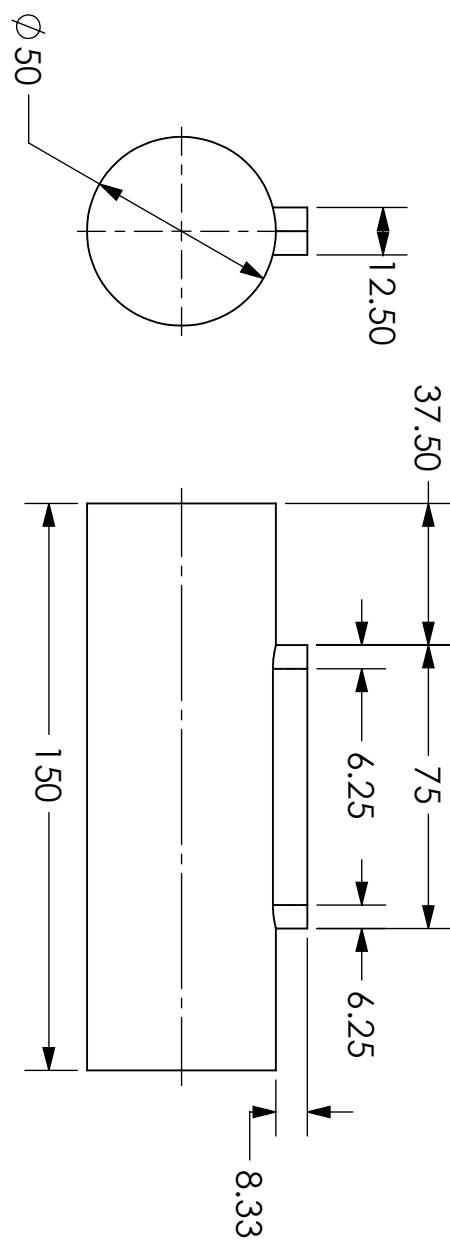
NAME

DATE

26/02/2019

TITLE:

Lab - 2 - Shaft



APPLICATION	USED ON	FINISH	DO NOT SCALE DRAWING	INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL Steel	COMMENTS:	SIZE A	DWG. NO. 2	REV A	SCALE: 1:2	WEIGHT:	SHEET 1 OF 1
-------------	---------	--------	----------------------	--	-----------	------------------	----------------------	-----------------	------------	---------	--------------



Sri Lanka Institute of Information and Technology

Department of Mechanical Engineering

Engineering Drawing – ME2031

SolidWorks Laboratory - 3

Mr. Thilina Weerakkody
Mr. Kulunu Samarakrama

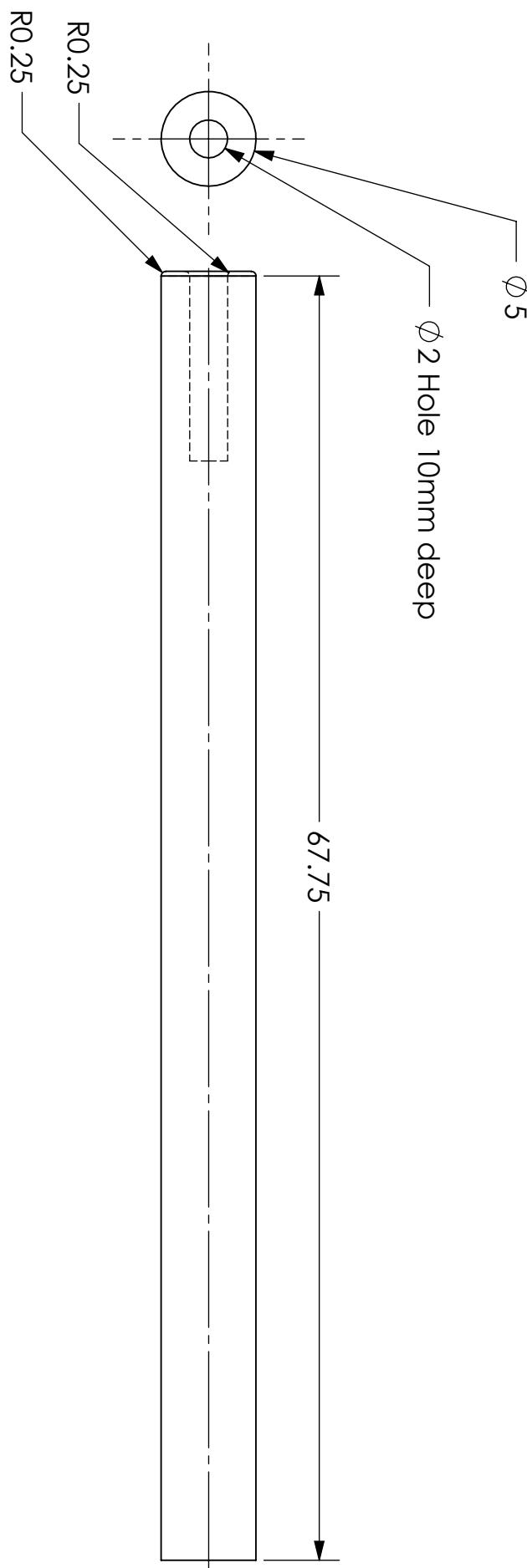
Time: 2 hours
Date : 05/03/2019

Targeted out comes of this lab

- Testing Modelling approaches
- Revolve feature
- Analyzing the drawing
- Assembling in SolidWorks
- Assembly Drawing

Follow the Instructions and complete the tasks explained exactly in each step:

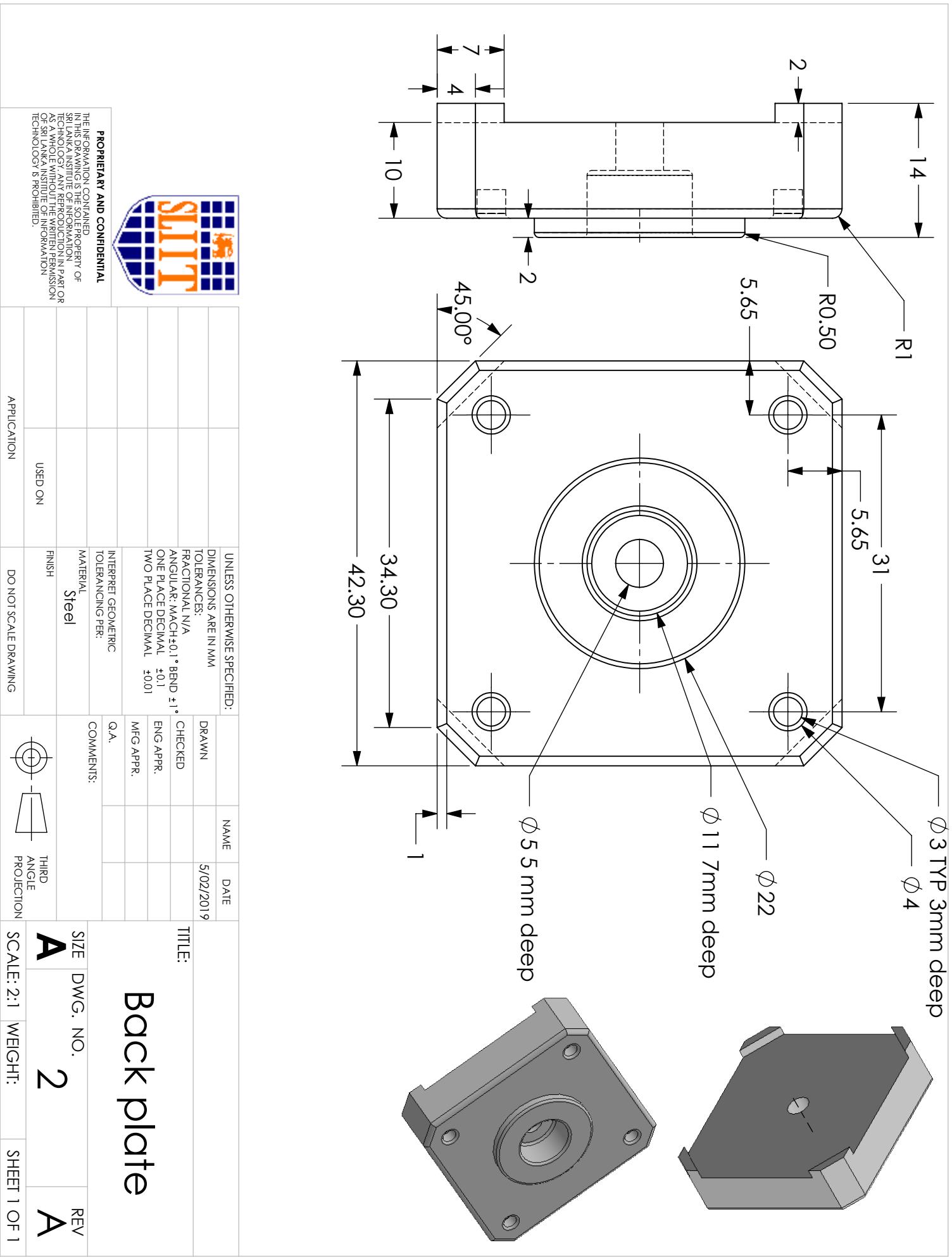
1. Analyze the exploded view of assembly drawing attached to this lab sheet titled “Assembly”.
This lab session is intended to assemble a Stepper motor– NEMA 17.
2. There are 5 Parts associated with the Assembly
 - a. Axial rod
 - b. Back plate
 - c. Base
 - d. Body
 - e. Pulley
3. Analyze the Drawings of the above parts attached in this Lab sheet
4. Model the SolidWorks part files for parts; “a” to “e” given above using the knowledge from previous lab sessions and assemblies
5. Save them inside a folder named with your student ID Ex: ENXXXXXXX_Lab03
6. Save the part files using their part names inside the folder as you create them (.SLDPRT files)
7. Create the NEMA-17 motor Assembly as detailed in the Assembly drawing. Use the knowledge from previous lab on SolidWorks Mate Features
8. Save the SolidWorks assembly File inside the folder as “NEMA”(.SLDASM file)
9. Listen to the **lecture** on detailed drawing creation for SolidWorks part files and assembly.
10. Create **SolidWorks drawings** for each part and Assembly and Save them inside the Folder.
11. Save them with the name of the part followed by “Drawing”. Ex: Axial rod_ Drawing



PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
OF SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY IS PROHIBITED.

UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MM TOLERANCES: FRACTIONAL N/A ANGULAR: MACH $\pm 0.1^\circ$ BEND $\pm 1^\circ$ ONE PLACE DECIMAL TWO PLACE DECIMAL		DRAWN CHECKED ± 0.1 ± 0.01	NAME DATE 05/02/2019	DATE 05/02/2019	TITLE: Axial rod
INTERPRET GEOMETRIC TOLERANCING PER:	COMMENTS:				
MATERIAL Steel	USED ON FINISH		THIRD ANGLE PROJECTION	SIZE A	DWG. NO. REV A
	DO NOT SCALE DRAWING			SCALE: 1:1	WEIGHT: SHEET 1 OF 1

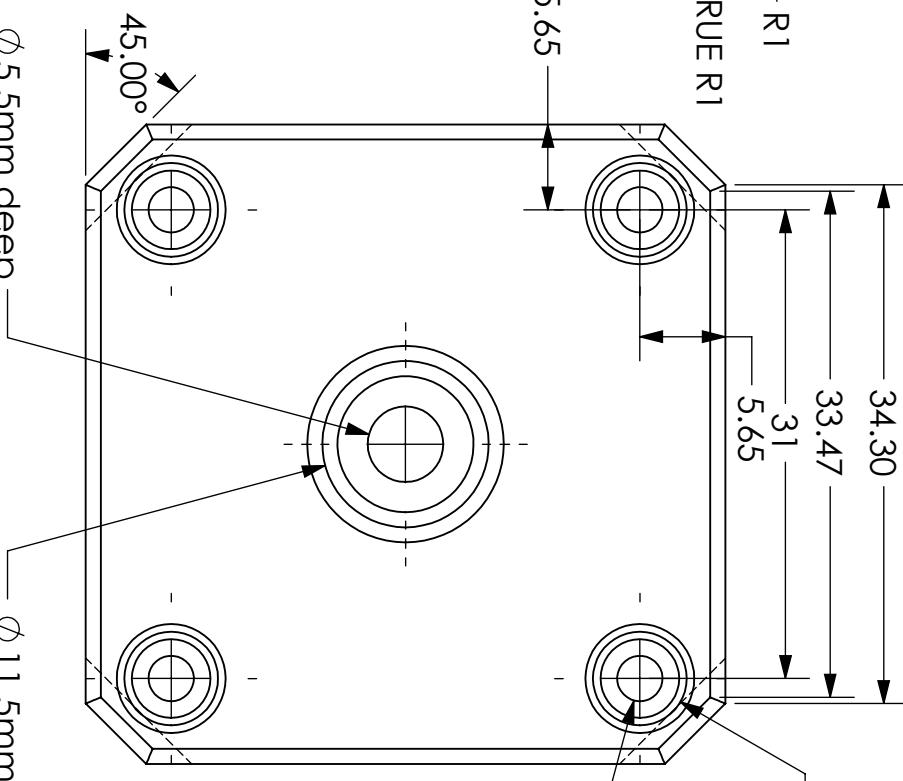




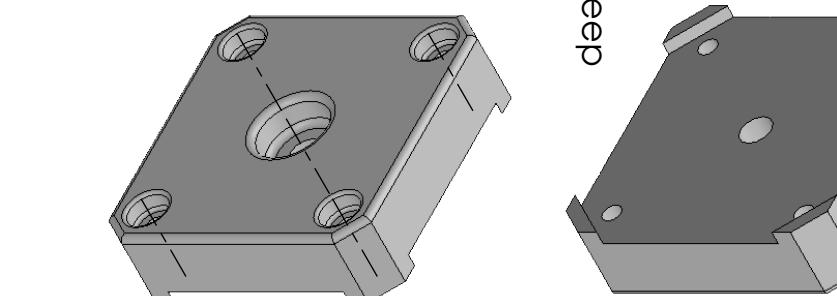
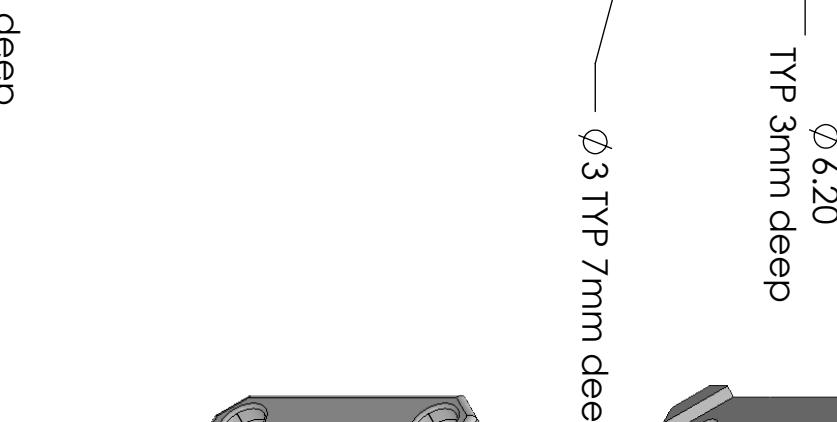
PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
IS PROHIBITED.

$\phi 5$ 5mm deep



$\phi 11$ 5mm deep



Base

SIZE

A

DWG. NO.

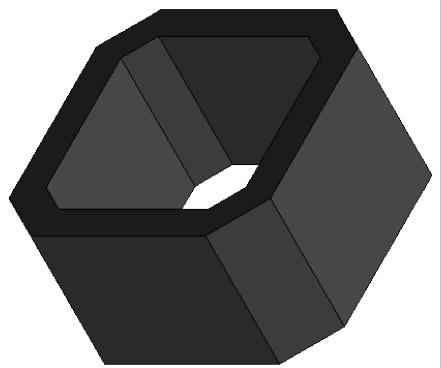
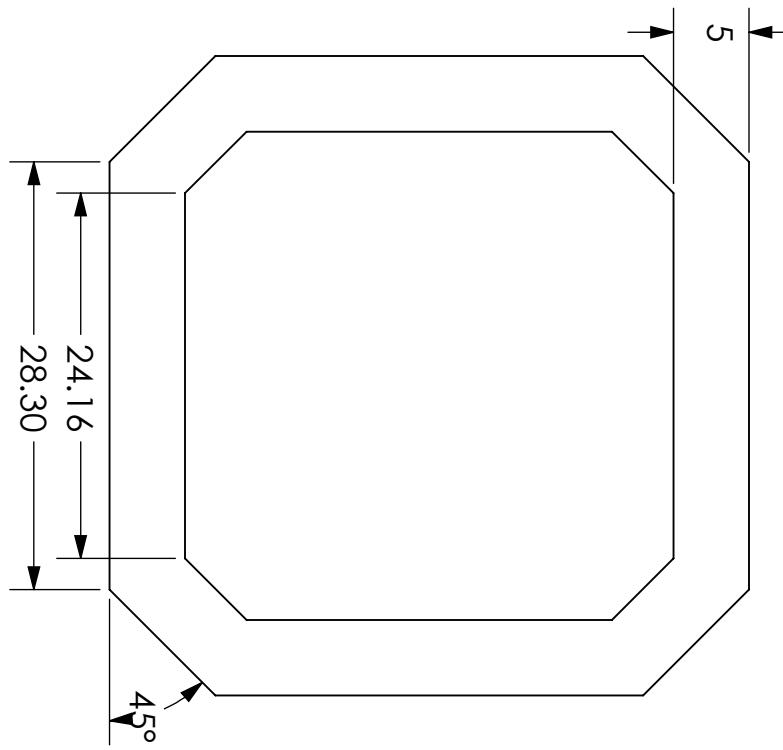
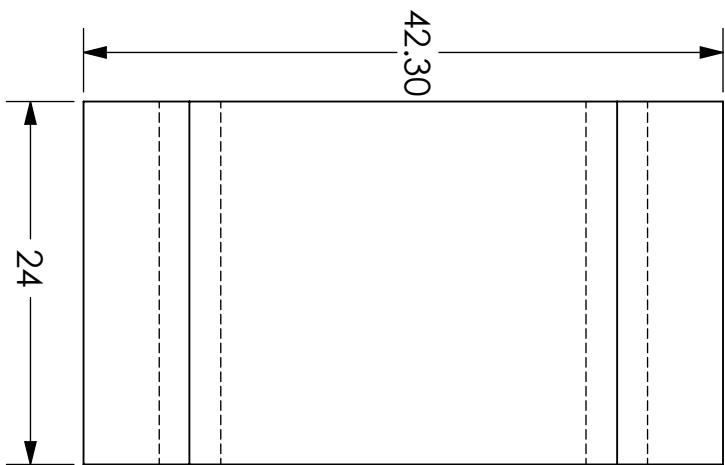
REV

A

SCALE: 2:1

WEIGHT:

SHEET 1 OF 1



PROPRIETARY AND CONFIDENTIAL

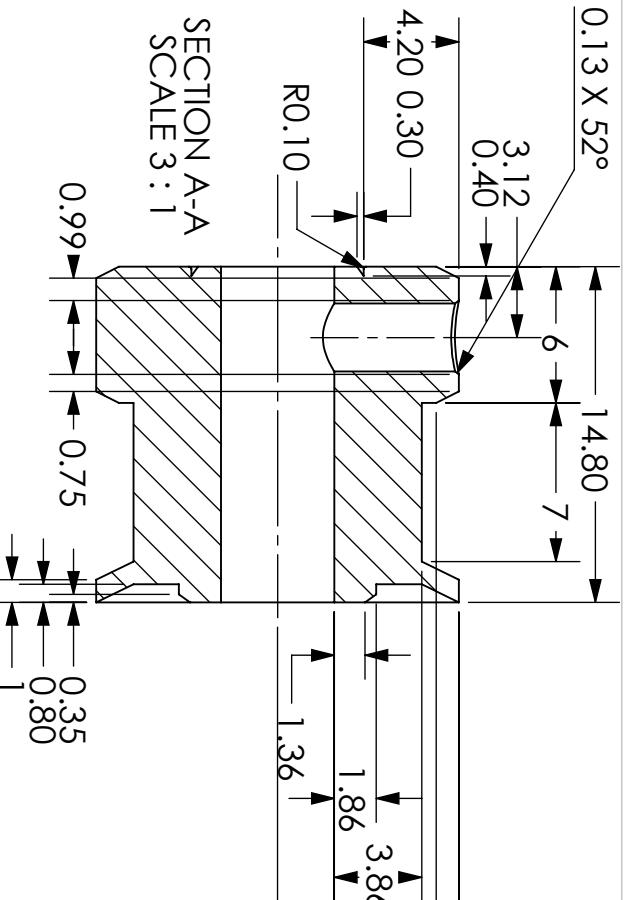
THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
OF SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY IS PROHIBITED.

UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MM TOLERANCES: FRACTIONAL N/A ANGULAR: MACH $\pm 1^\circ$ BEND $\pm 1^\circ$ ONE PLACE DECIMAL TWO PLACE DECIMAL				DRAWN	NAME	DATE	TITLE:
INTERPRET GEOMETRIC TOLERANCING PER:	MATERIAL Steel	FINISH	COMMENTS:	CHECKED		5/02/2019	
				ENG APPR.			
				MFG APPR.			
				Q.A.			

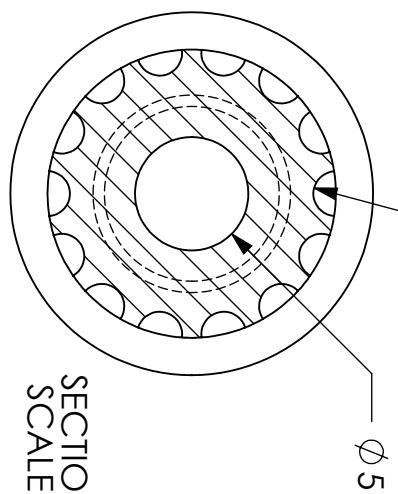
APPLICATION	USED ON	DO NOT SCALE DRAWING	SCALE: 1:1	REV A	SIZE A	DWG. NO. 3	WEIGHT:	SHEET 1 OF 1
					THIRD ANGLE PROJECTION			

Body

SECTION A-A
SCALE 3 : 1



SECTION B-B
SCALE 3 : 1



PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
OF SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY IS PROHIBITED.

UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MM

TOLERANCES:

FRACTIONAL MIA

ANGULAR: MACH $\pm 0.1^\circ$

BEND $\pm 1^\circ$

ONE PLACE DECIMAL

TWO PLACE DECIMAL

± 0.1

± 0.01

MFG APPR.

ENG APPR.

CHECKED

DRAWN

NAME

DATE

5/03/2019

TITLE:

Pulley

INTERPRET GEOMETRIC
TOLERANCING PER:
MATERIAL

COMMENTS:

FINISH

APPLICATION	USED ON	DO NOT SCALE DRAWING
-------------	---------	----------------------

SIZE

DWG. NO.

A

REV

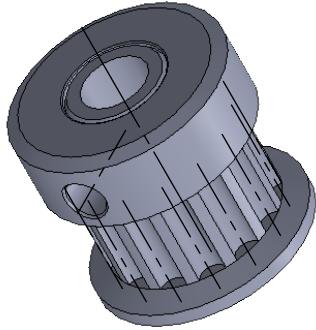
A

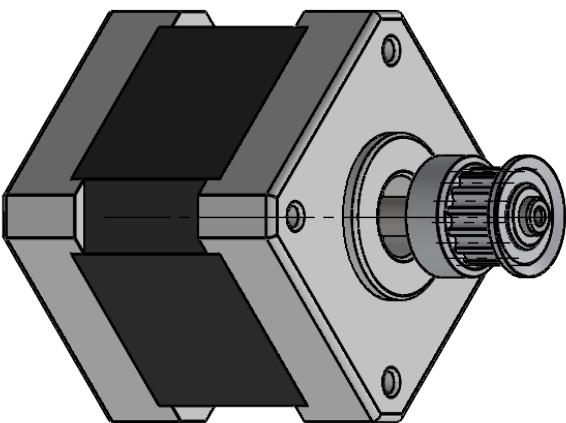
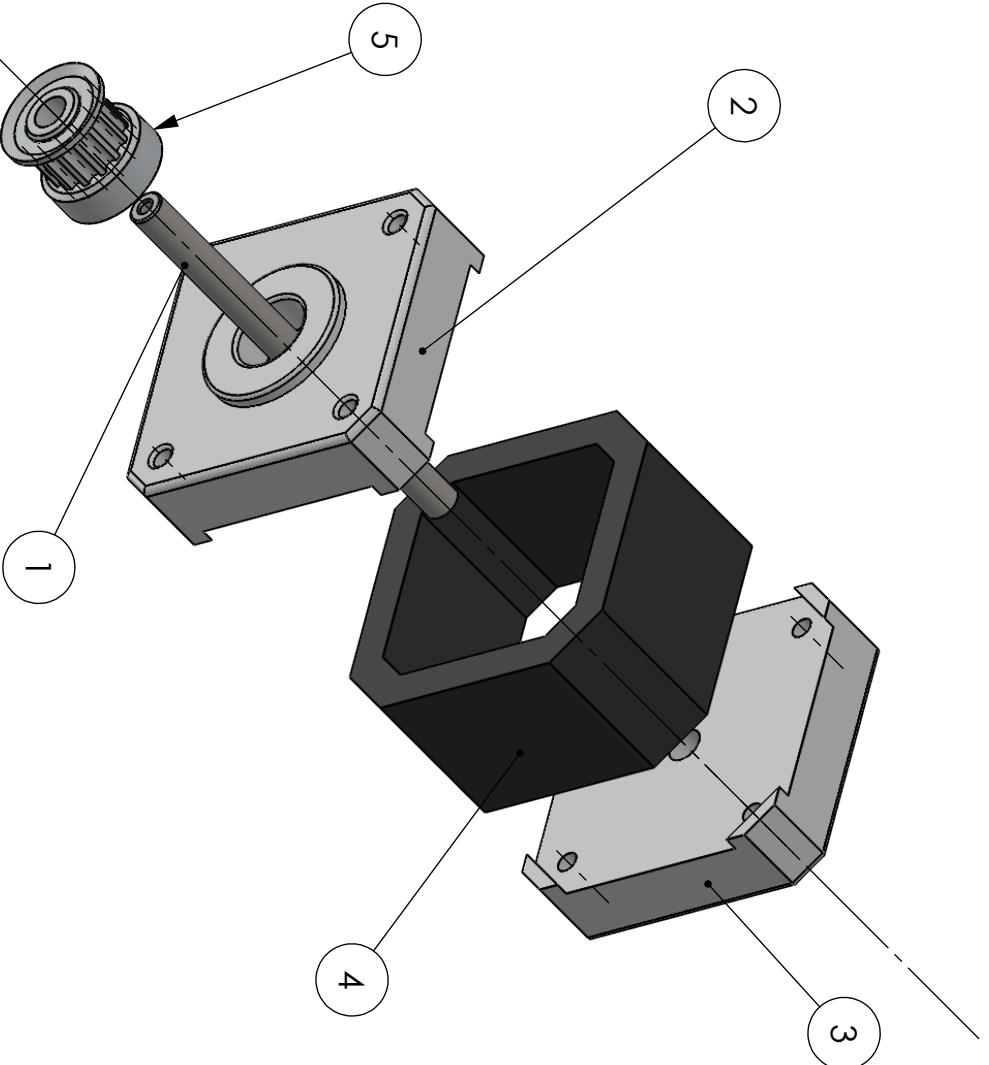
THIRD
ANGLE
PROJECTION

SCALE: 2:1

WEIGHT:

SHEET 1 OF 1





Assembly

PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
IS PROHIBITED.

UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MM

TOLERANCES:

FRACTIONAL M.I.A

ANGULAR: MACH $\pm 0.1^\circ$

BEND $\pm 1^\circ$

ONE PLACE DECIMAL

± 0.1

TWO PLACE DECIMAL

± 0.01

INTERPRET GEOMETRIC
TOLERANCING PER:

MATERIAL

FINISH

COMMENTS:

DRAWN

CHECKED

ENG APPR.

MFG APPR.

Q.A.

TITLE:

Axial rod

Back plate

Base

Body

Pulley

Part number

Part name

NAME

DATE

5/02/2019

TITLE:

Part number	Part name
1	Axial rod
2	Back plate
3	Base
4	Body
5	Pulley

SIZE

DWG. NO.

A

SCALE: 1:2

WEIGHT:

A

SHEET 1 OF 1

REV

A



Sri Lanka Institute of Information and Technology

Department of Mechanical Engineering

Engineering Drawing – ME2031

SolidWorks Laboratory - 4

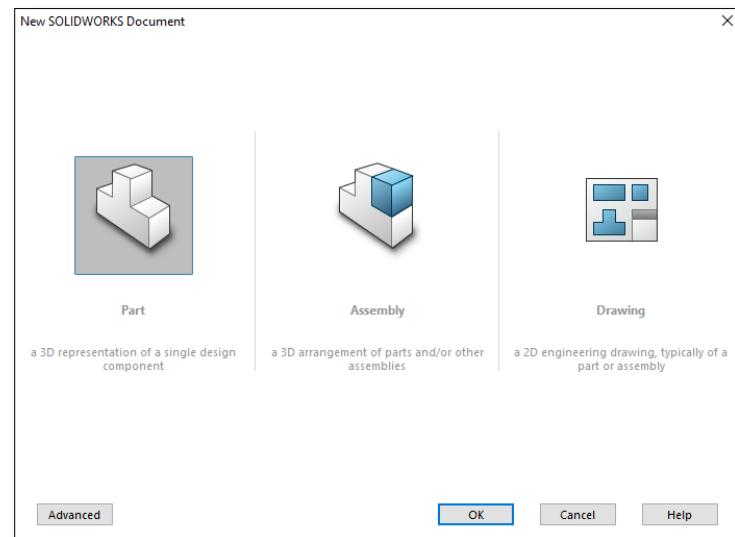
Mr. Thilina Weerakkody
Mr. Kulunu Samarakkrama

Time: 2 hours
Date : 12/03/2019

Targeted out comes of this lab - Creating a sheet metal box with custom holes

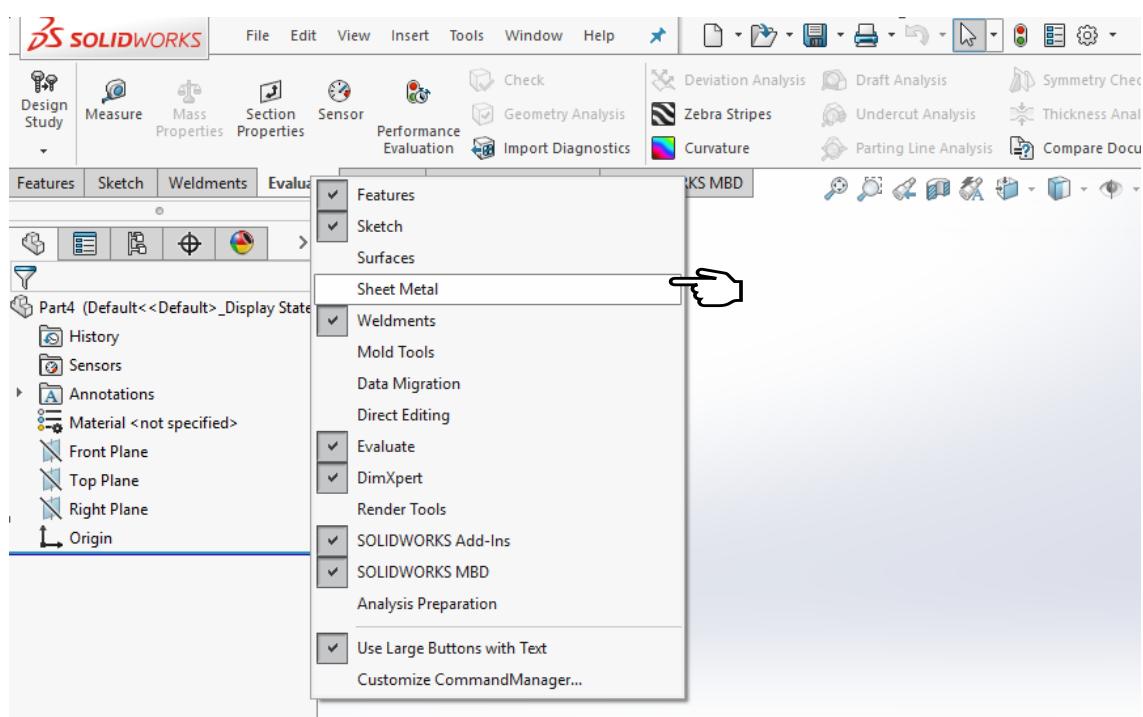
- Sheet Metal feature
- Hole Wizard
- Detailed Drawing

1. Open a new part file in SolidWorks

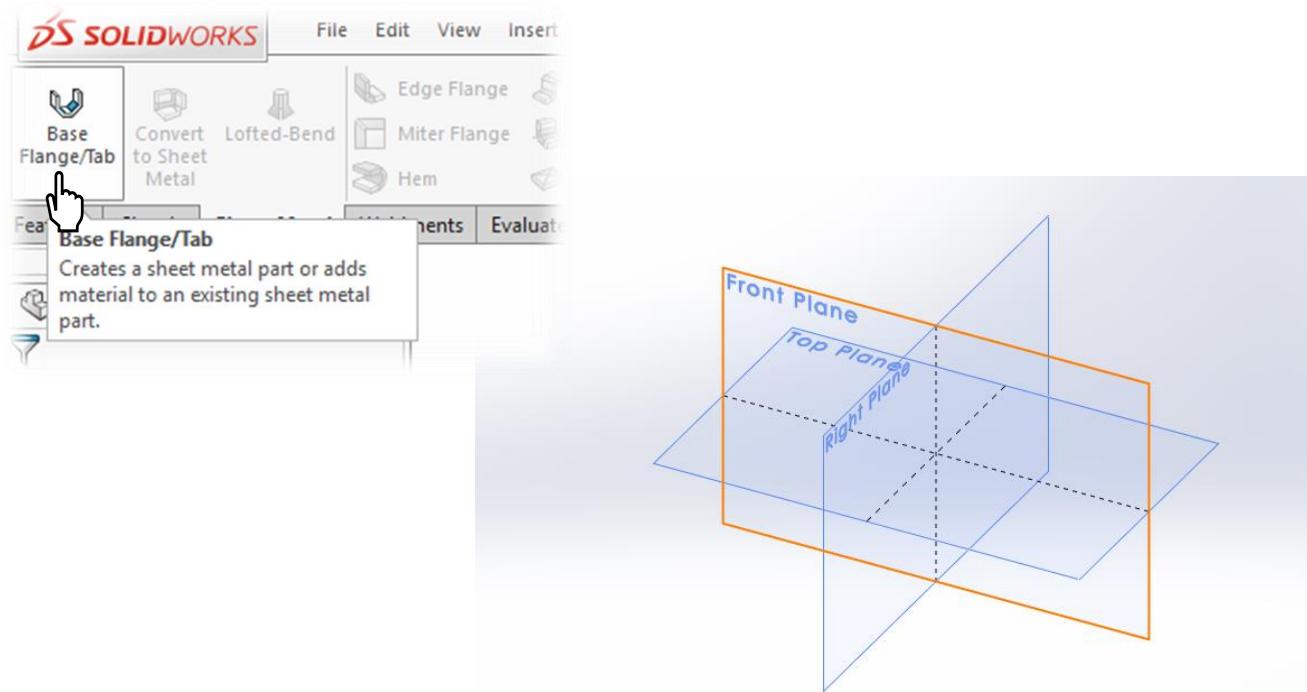


2. Select Sheet Metal on Tool Bar (Sheet Metal tool does not show in SOLIDWORKS default):

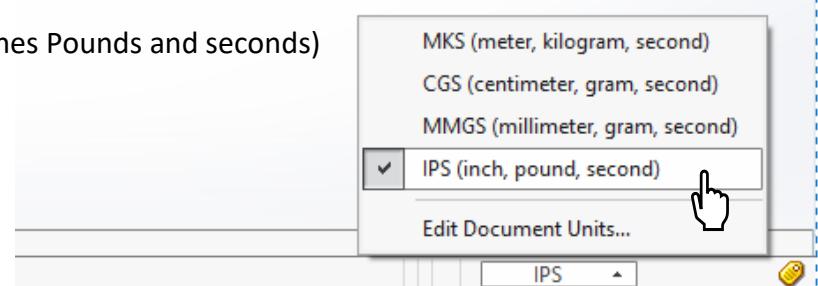
Right click on Tool Bar: >> Select **Sheet Metal** feature



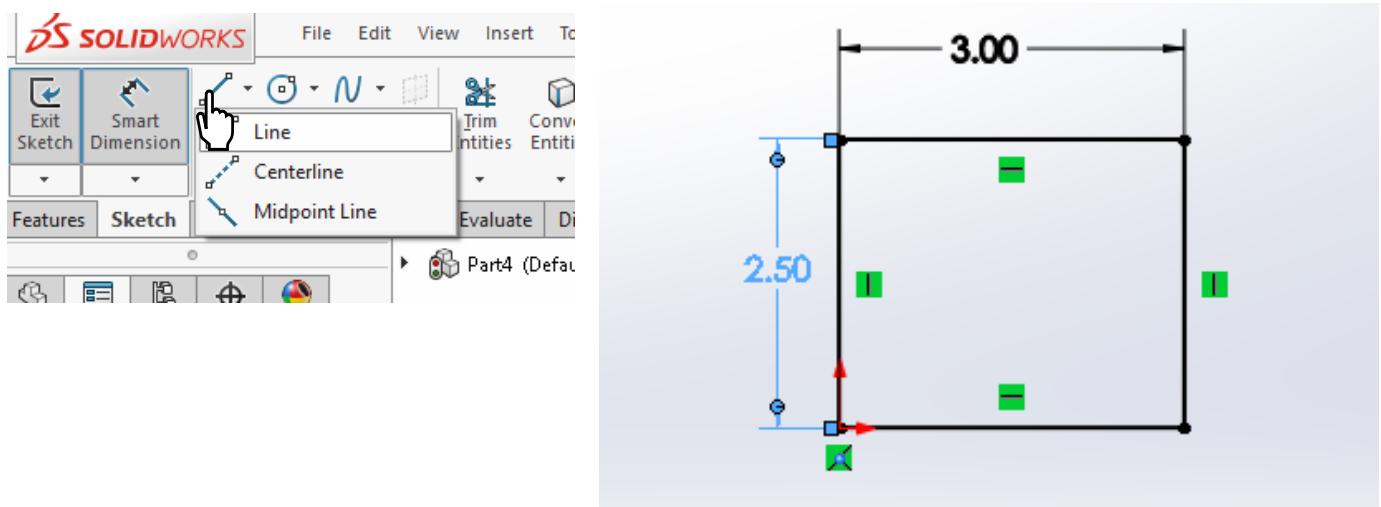
3. Select **Base Flange/tab** option and select the **Front-plane** to start sketching



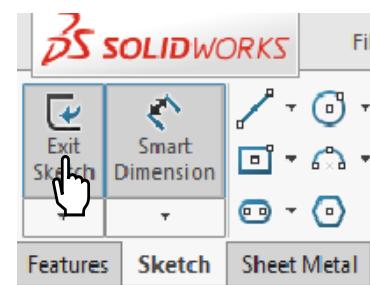
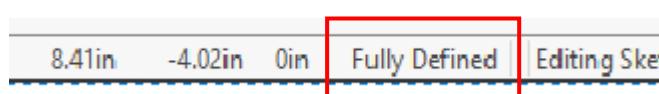
4. Change the unit system to IPS (Inches Pounds and seconds)



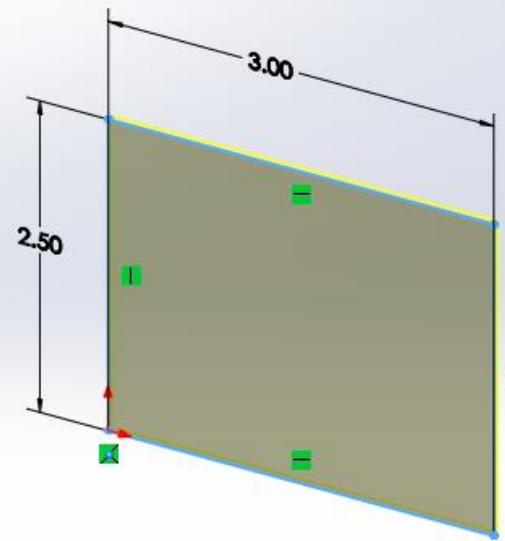
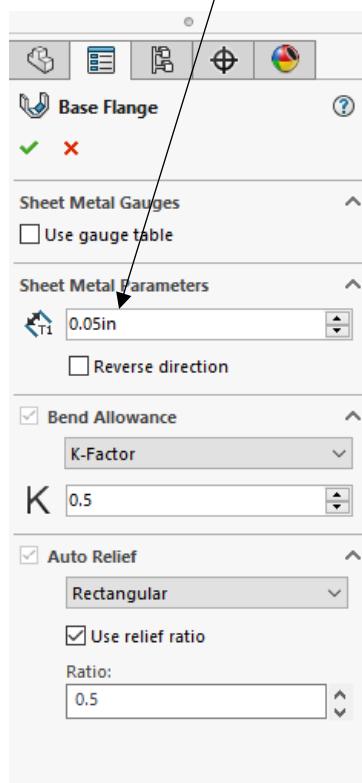
5. Sketch a square on Front plane using Lines. (3 inches * 2.5 inches)



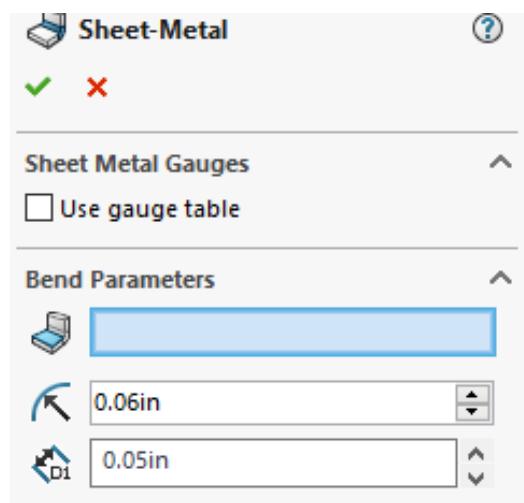
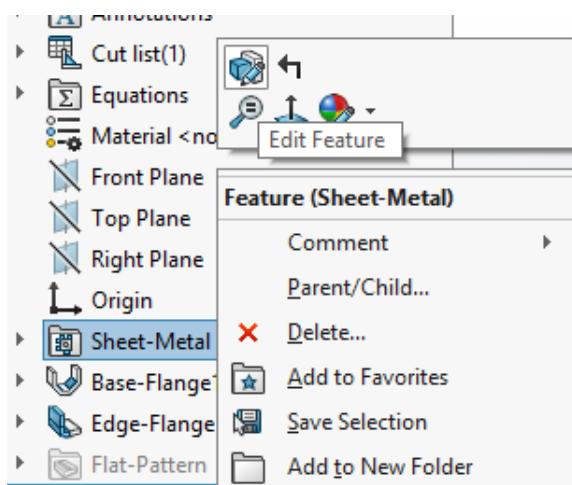
6. After sketching the square and **fully-defining** the sketch – **exit the sketch**



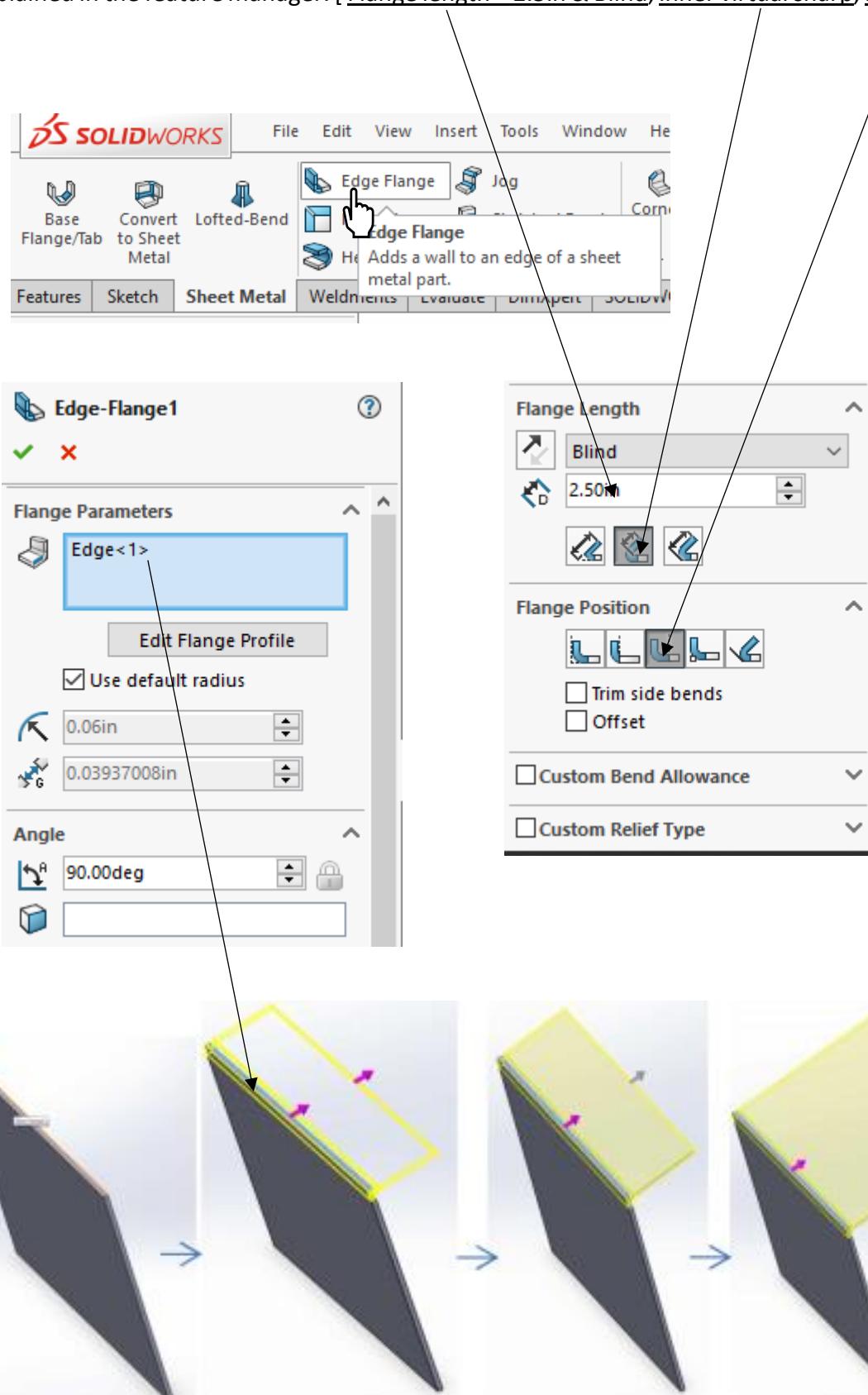
7. Enter the **Sheet Metal Parameter** in the property manager window and click OK
: Leave other areas as they are for the time being



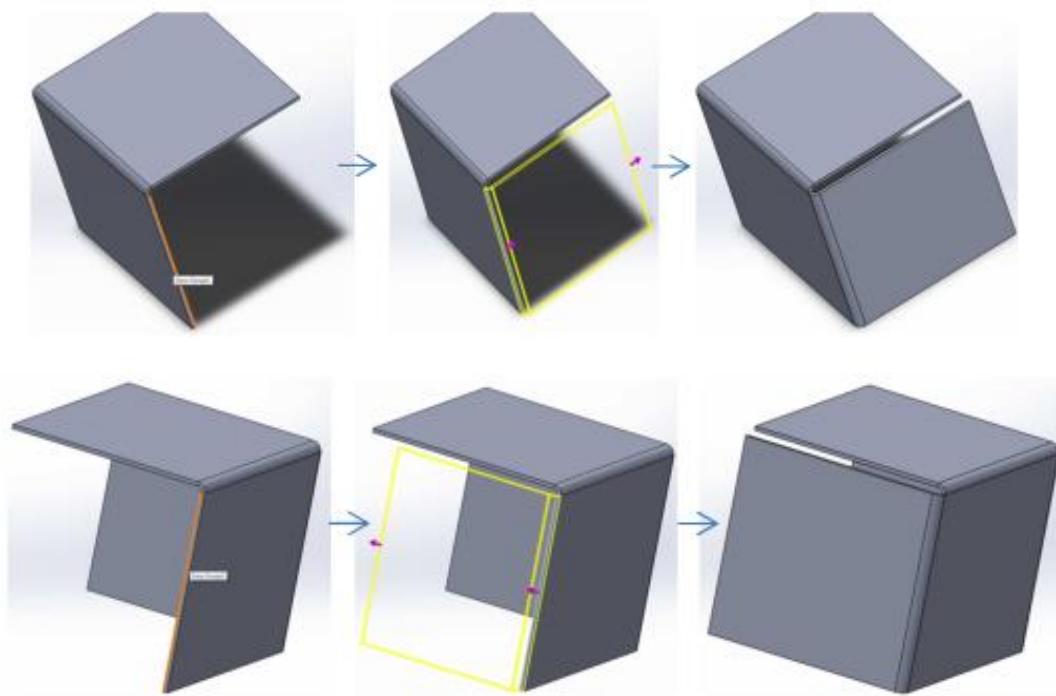
8. Right click on **Sheet Metal** from **Feature manager tab** and select **Edit Feature**. Radius in bend parameter as 0.06 inches



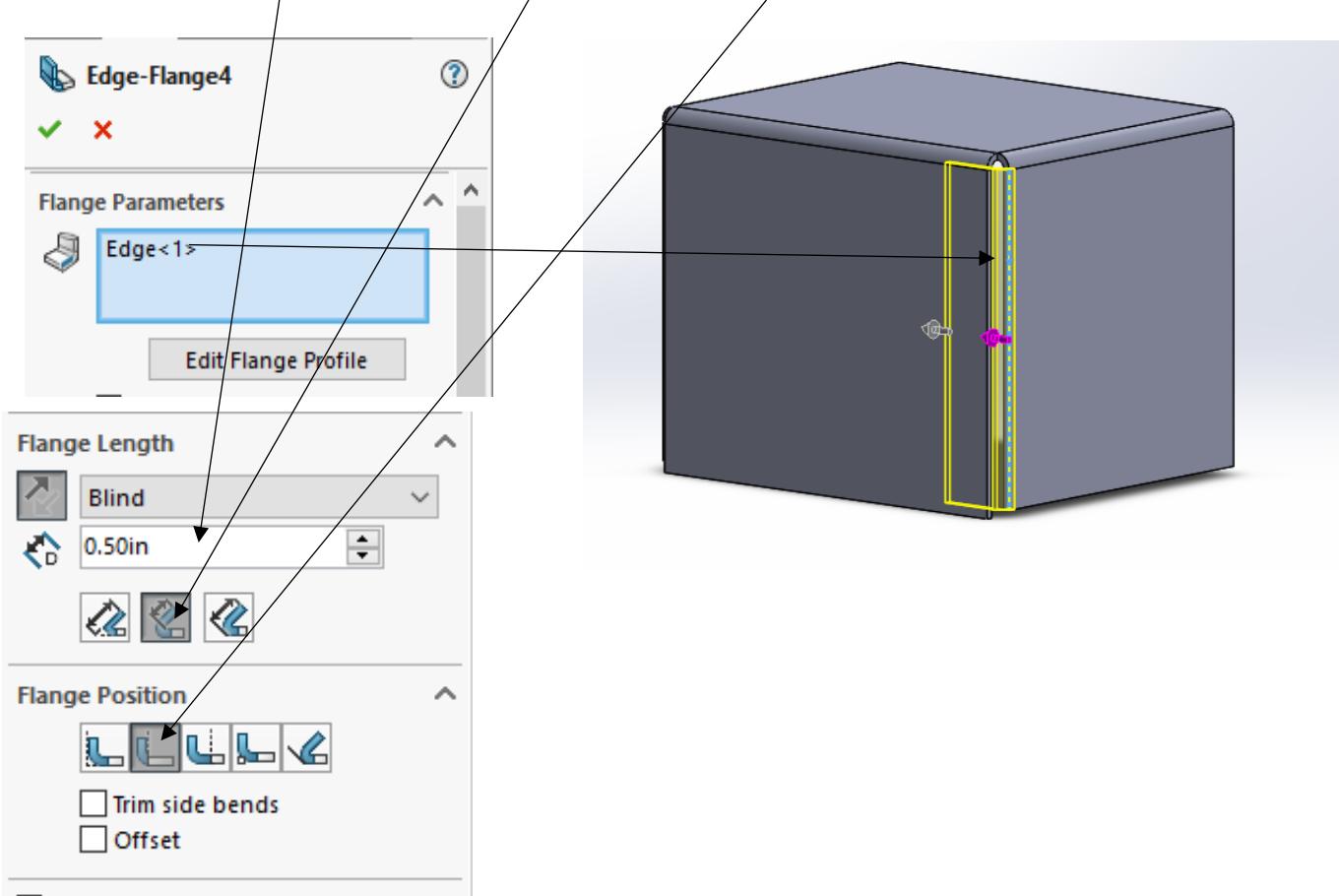
9. Select **Edge flange** to create an edge flange. Enter the given parameters and select the options as explained in the feature manager. [Flange length = 2.5in & Blind, inner virtual sharp, Bend outside]



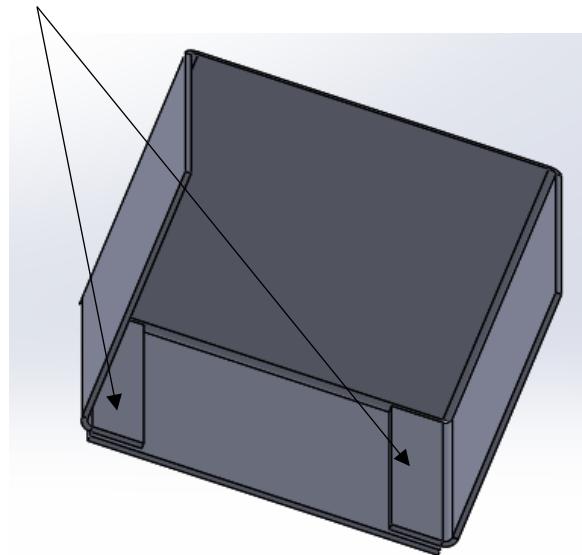
10. Using the same settings and parameters complete the edge flange for next 2 sides as shown



11. Now Create the corner edge flange. Select the parameters as shown
[Flange length = 0.5in & Blind, inner virtual sharp, Material outside]

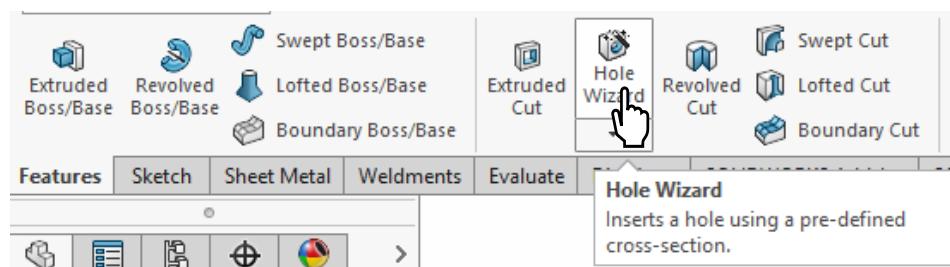


12. Repeat the step for the other open edge

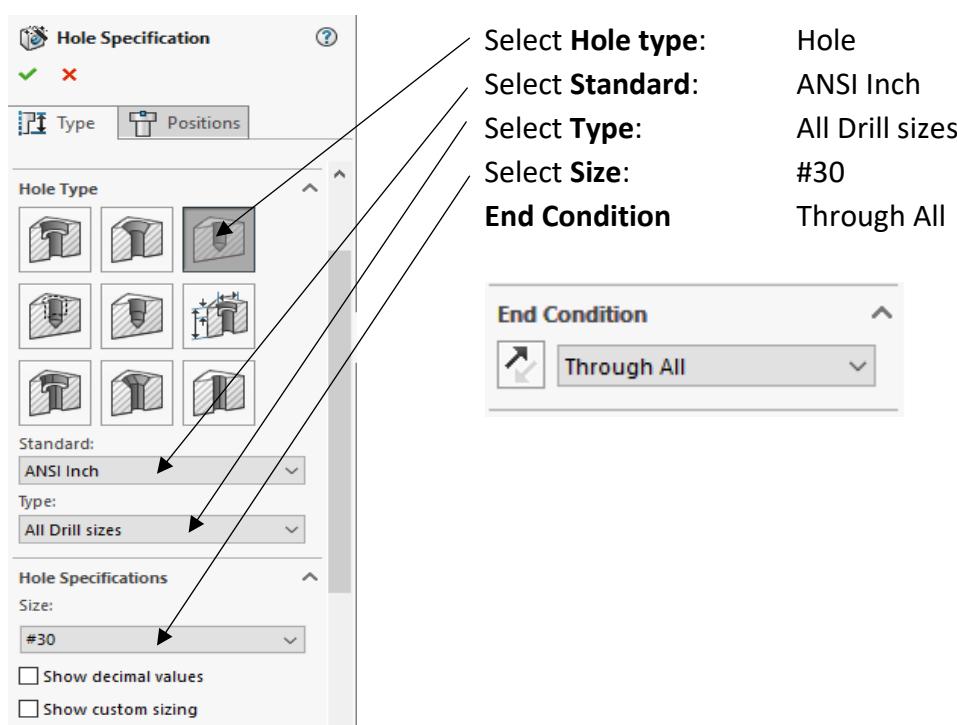


Hole Wizard

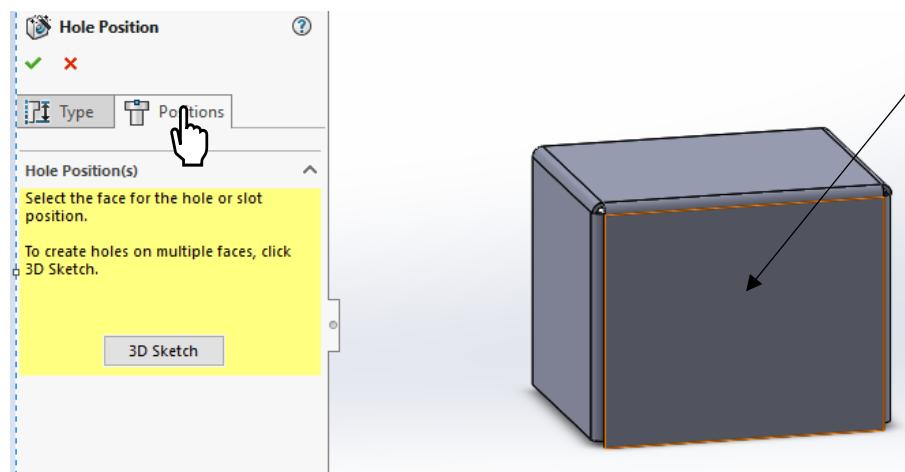
1. Select Hole Wizard from **Features** toolbar



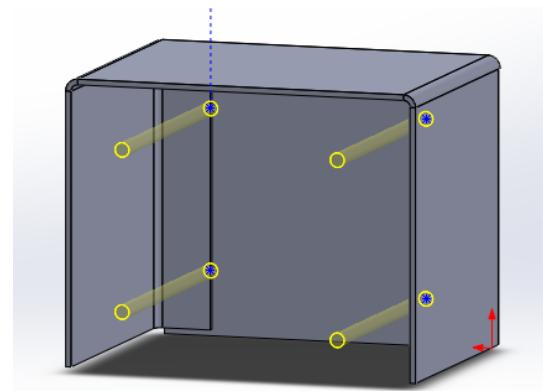
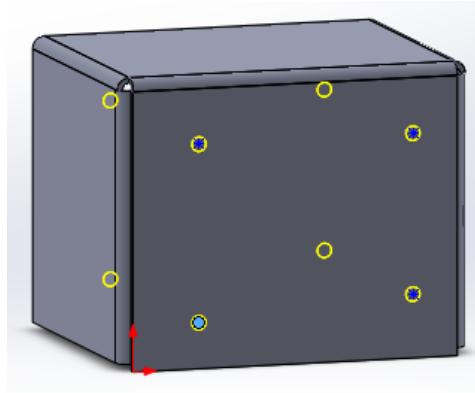
2. Set the parameters as shown below in the Property Manager window and press OK



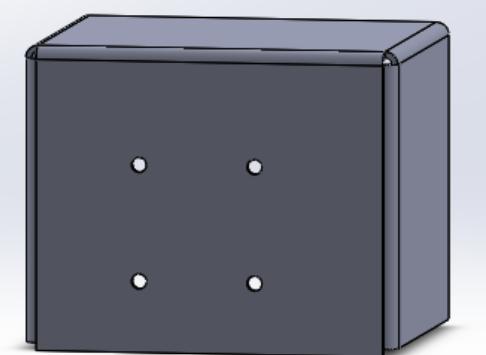
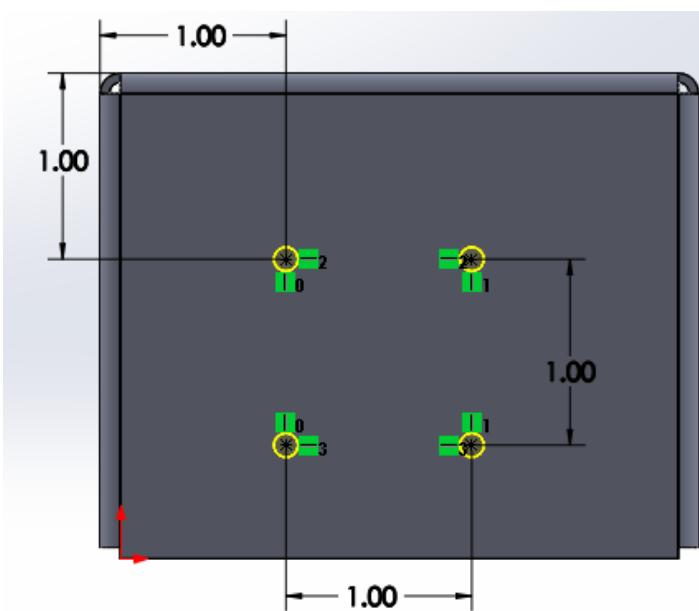
3. You will be requested to define the location of the hole. Click OK and Select the Face as shown



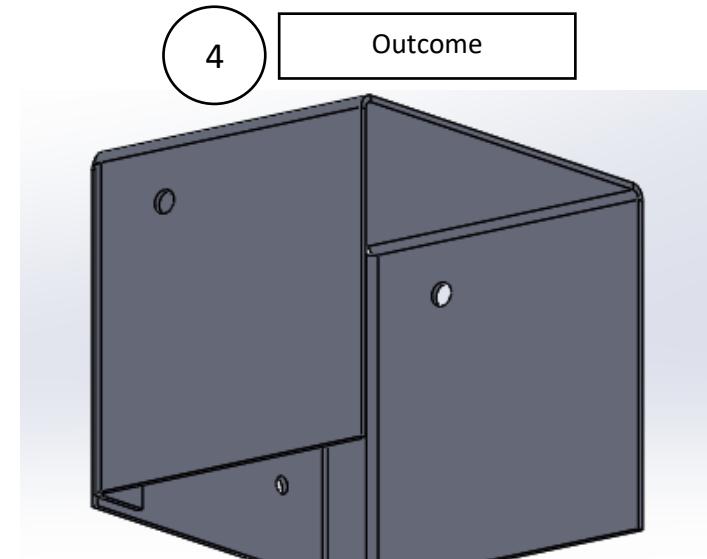
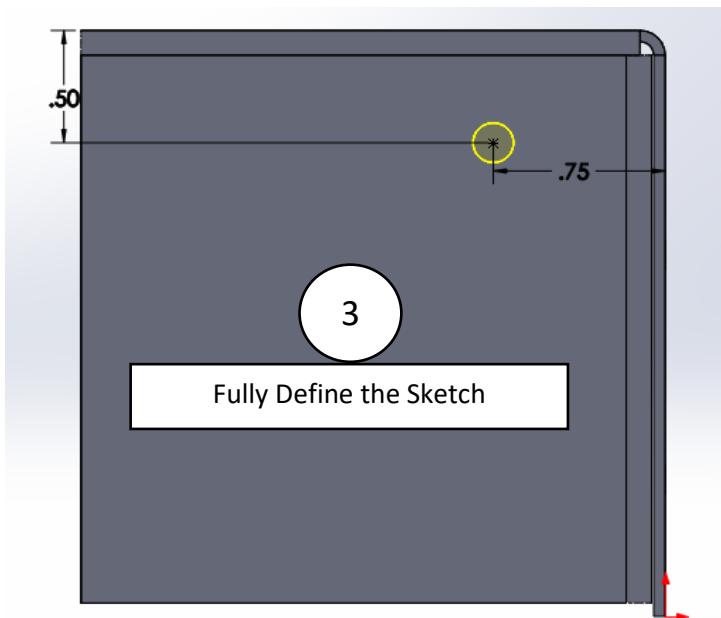
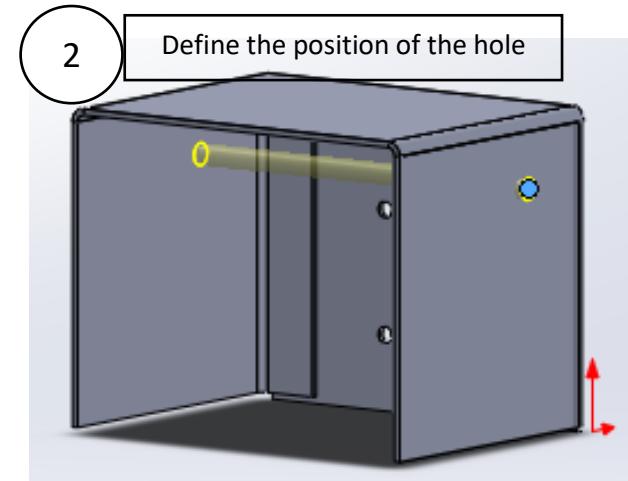
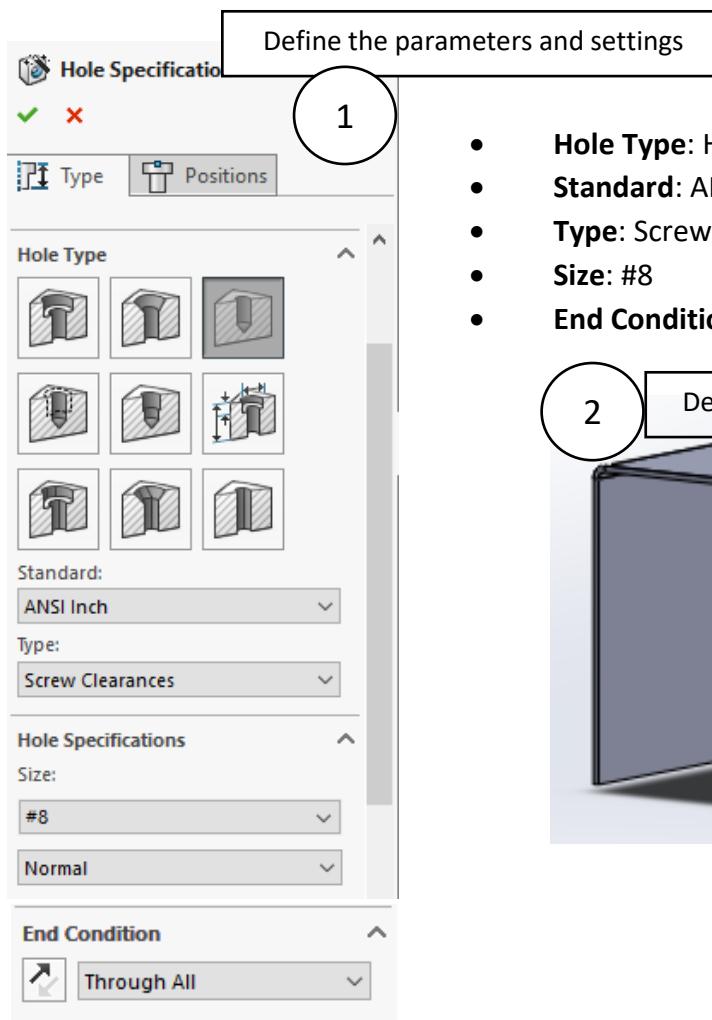
4. Place 4 holes on the face on **arbitrary locations**



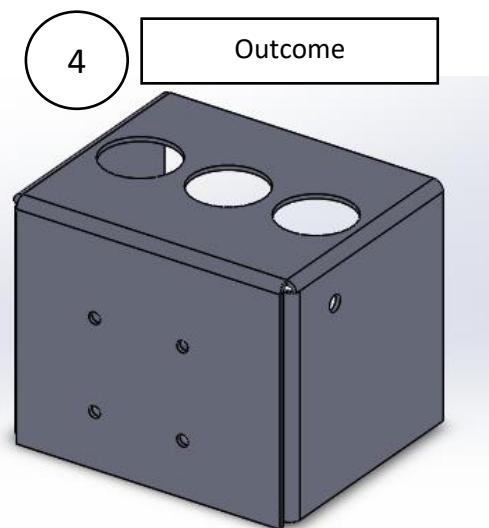
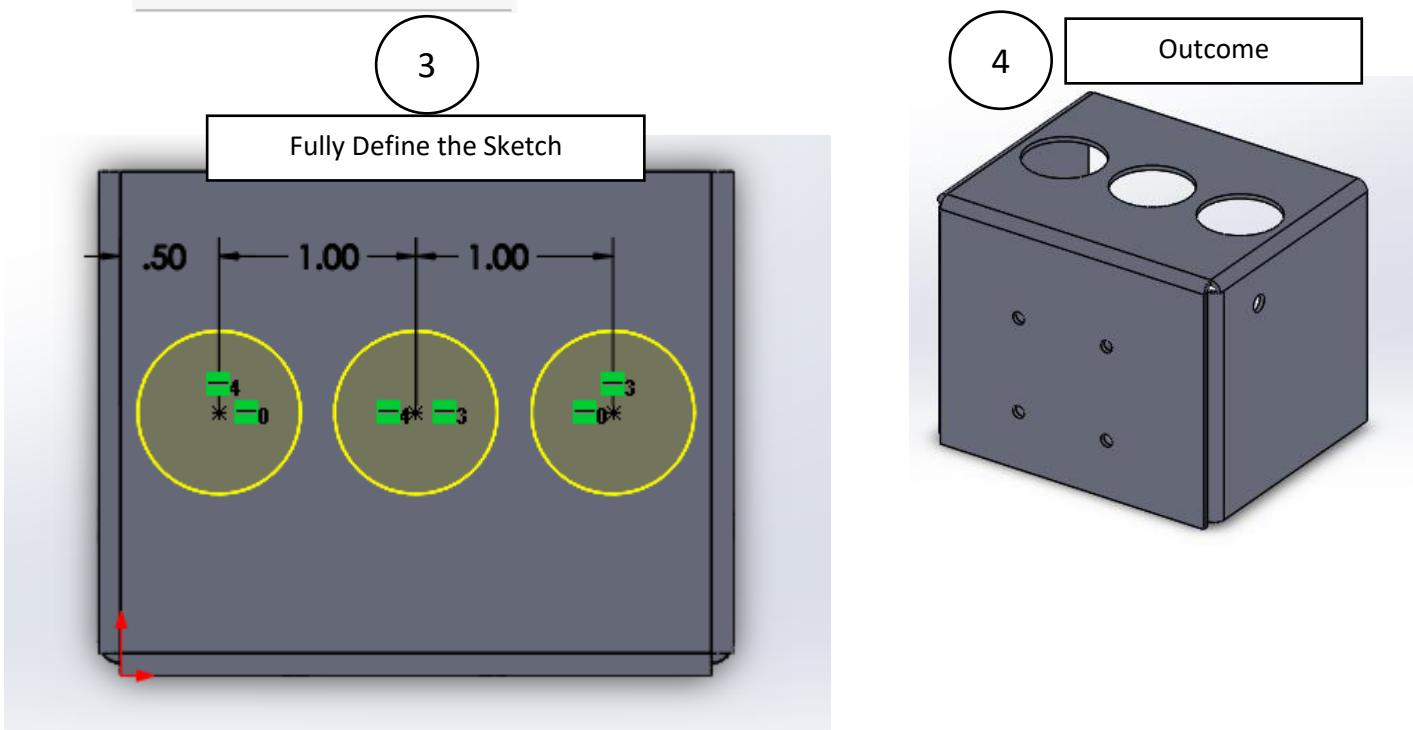
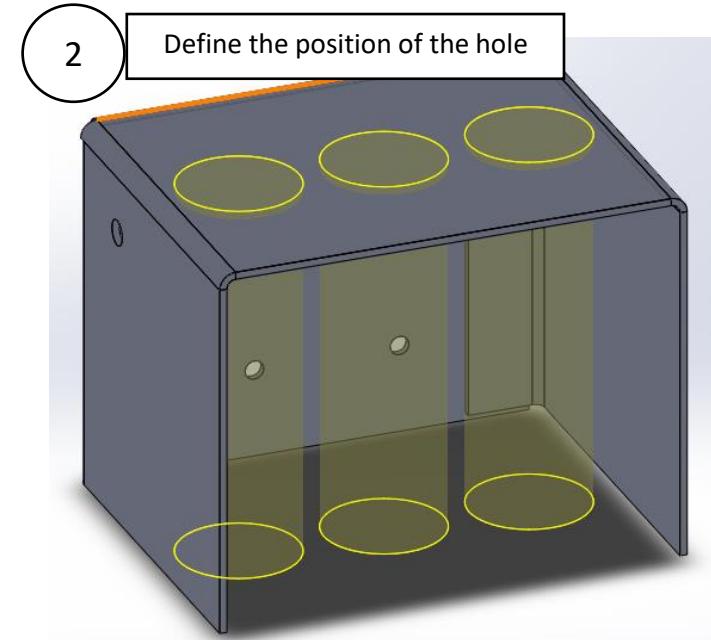
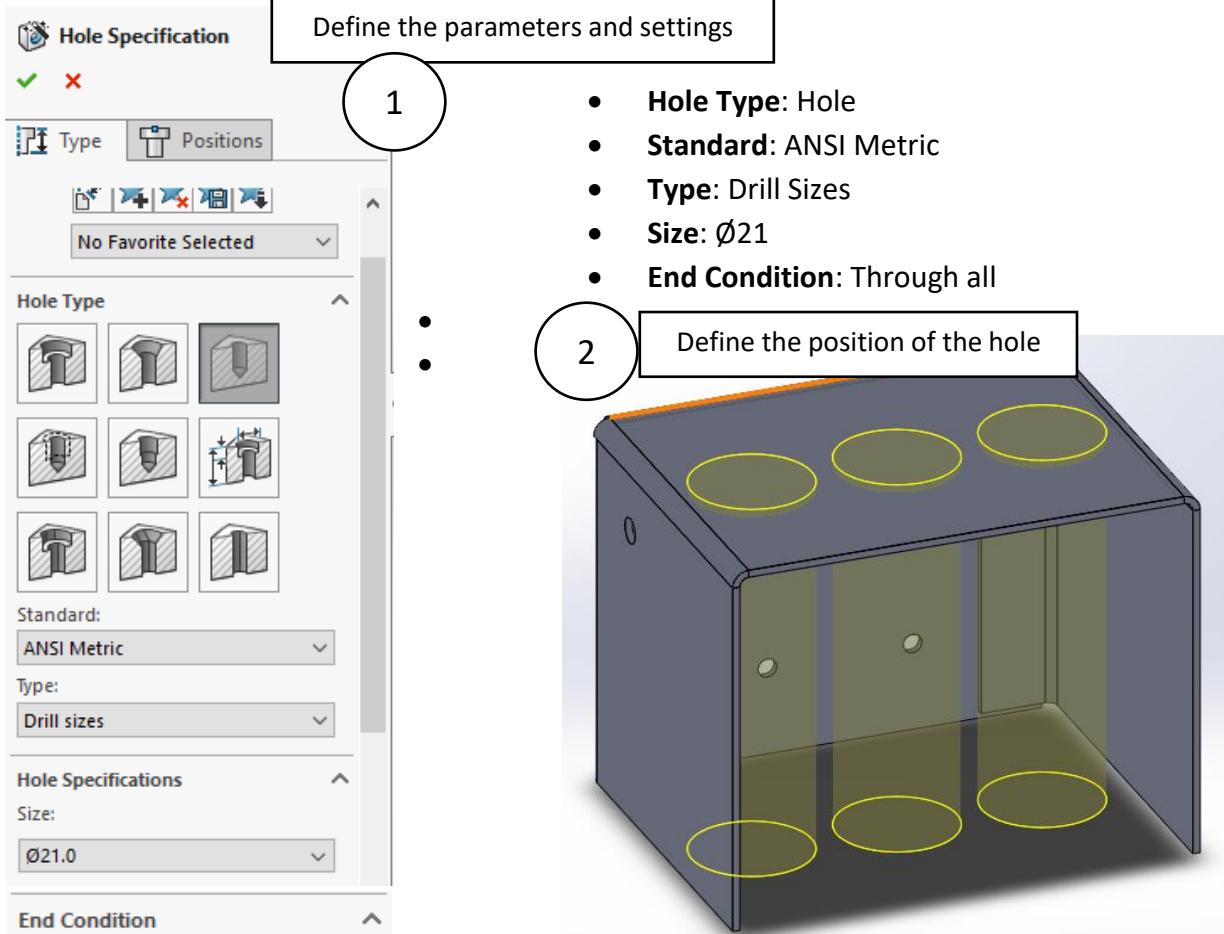
5. Use **Smart dimensions** and **sketch relations** to **Fully define** the Hole sketch (This must be done while inside the hole wizard – Hole position window in on). And press OK



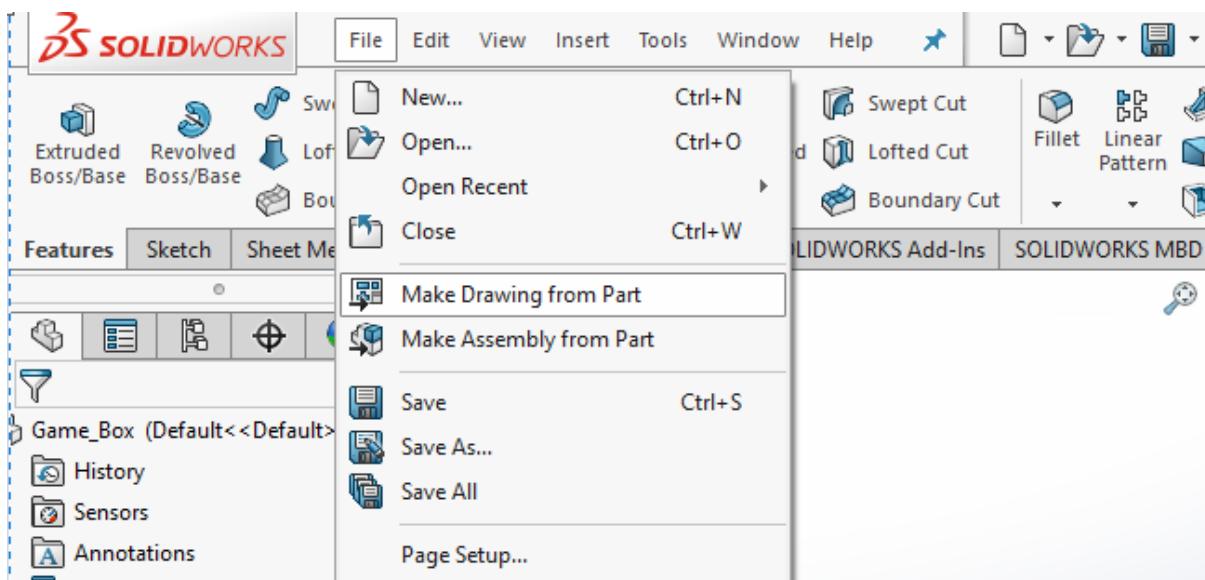
6. Repeat the Hole wizard process to create 2 more holes. Use the parameters given below. Follow the steps 1-4!



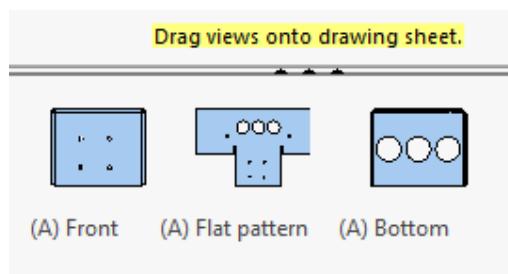
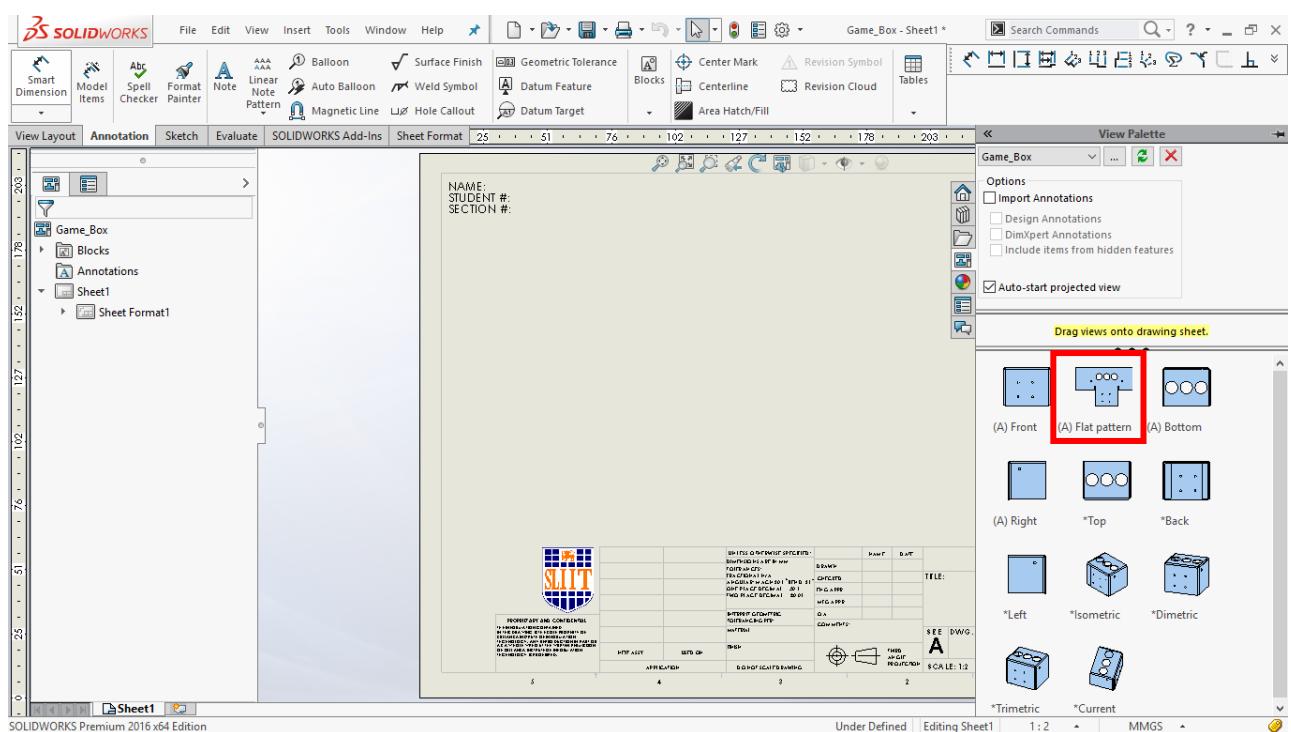
7. Repeat hole Wizard to create another 2 holes



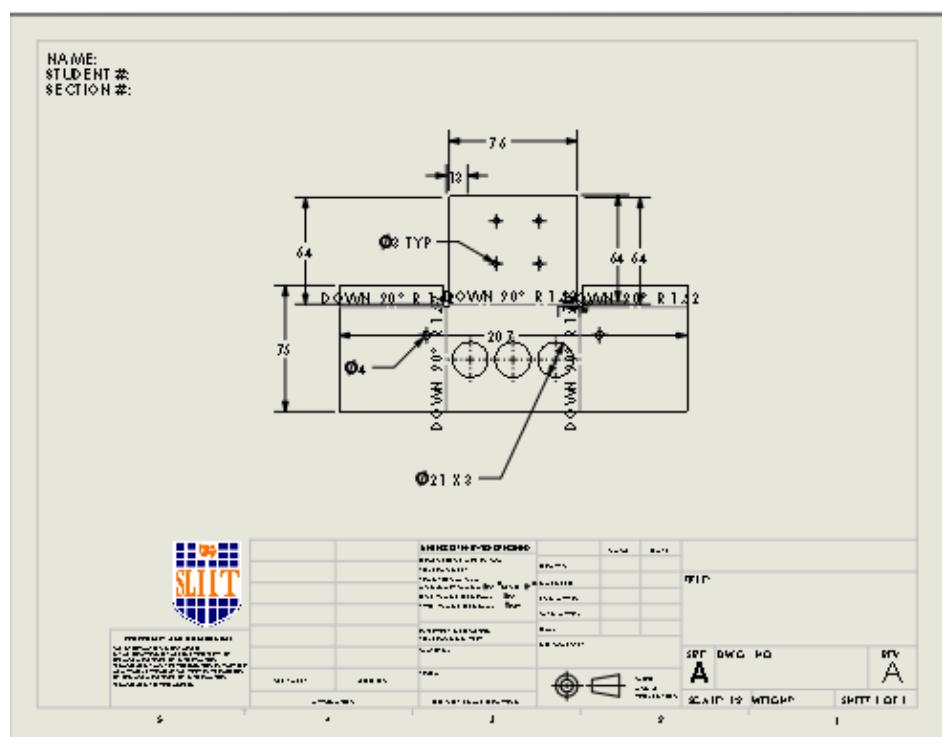
1. Files >> Make Drawing from part



2. Select Flat pattern view from the View palette on right side of the window

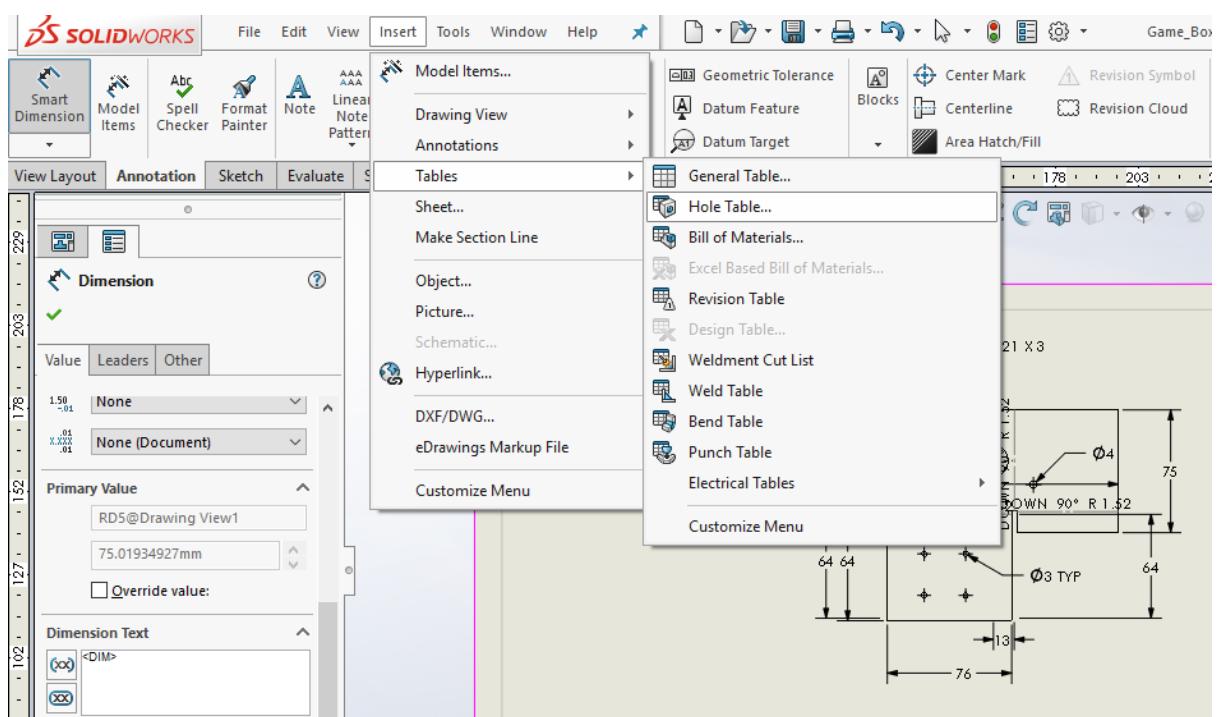


3. Drag and Drop the View on to the drawing sheet. Complete Dimensioning using **Smart dimensions** and **model items**

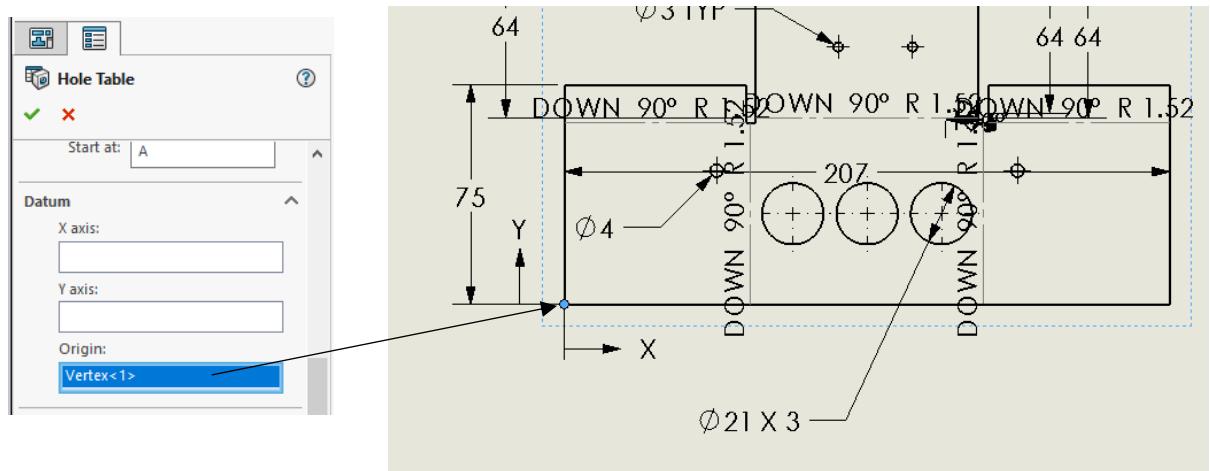


4. Insert a **Hole table**.

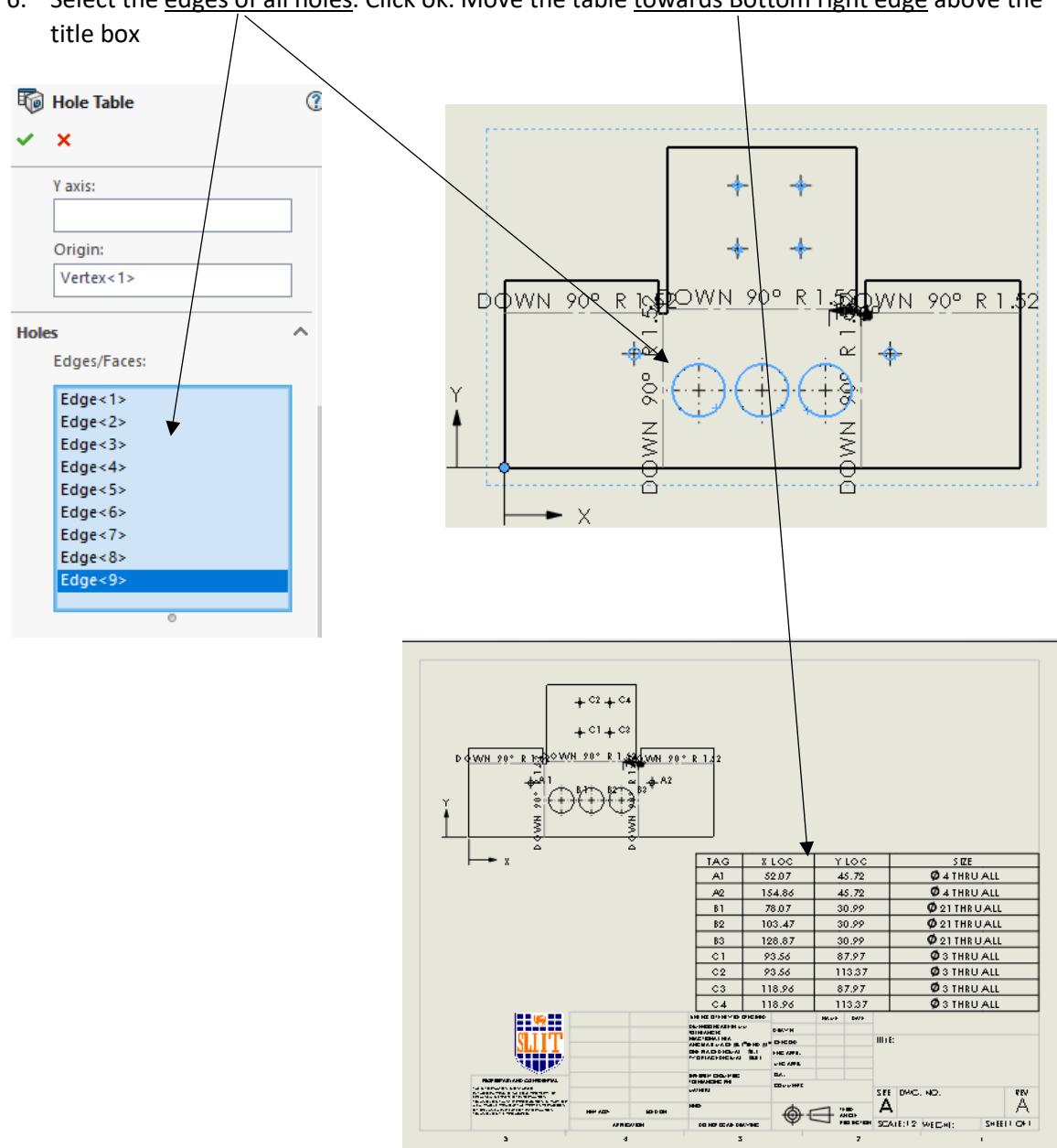
Select **Insert >> Tables >> Hole Table**



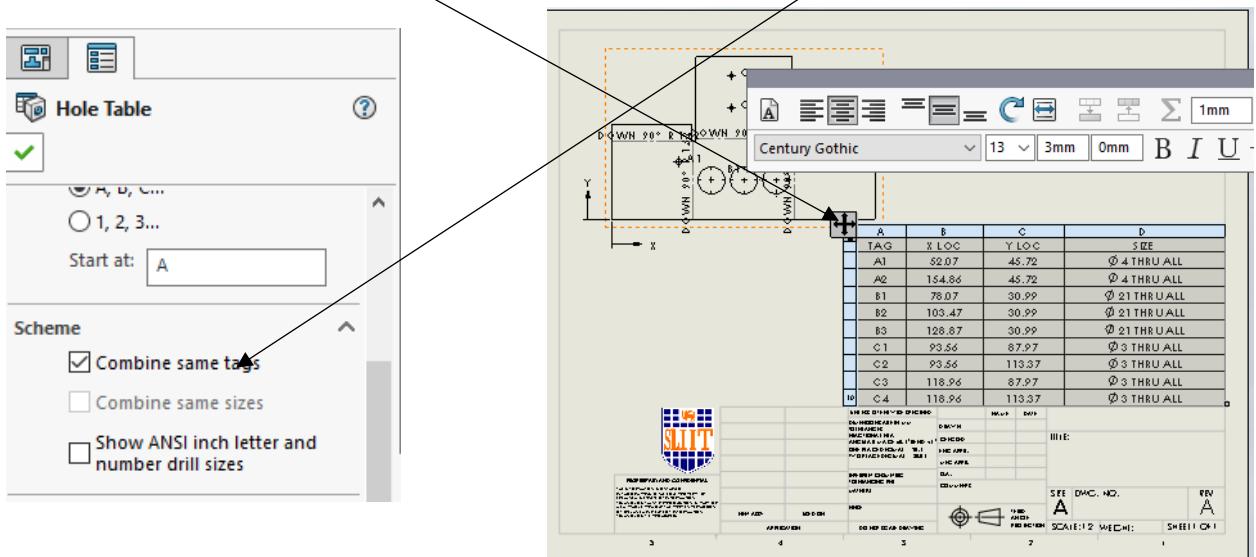
5. In the hole table property manager select/insert following parameters and settings



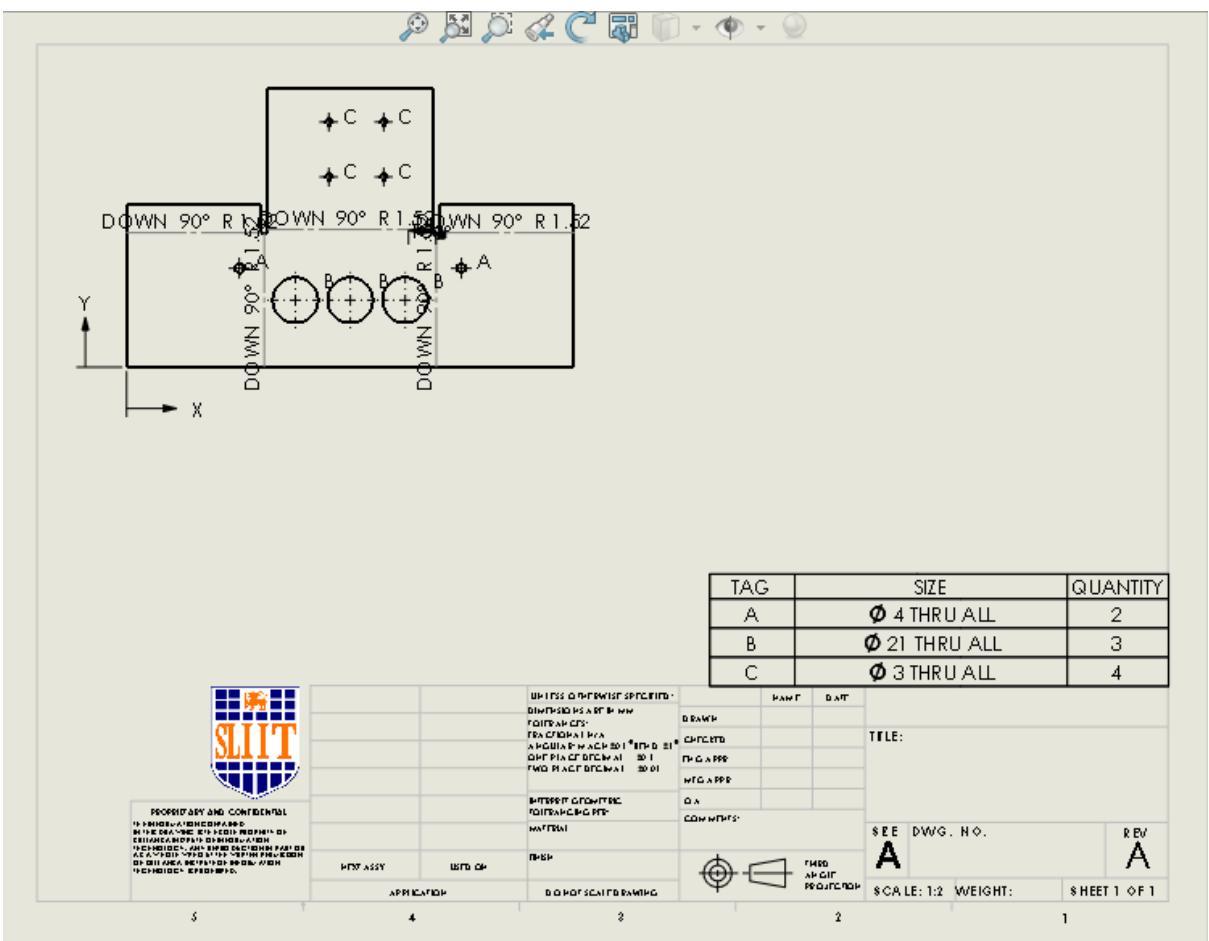
6. Select the edges of all holes. Click ok. Move the table towards Bottom right edge above the title box



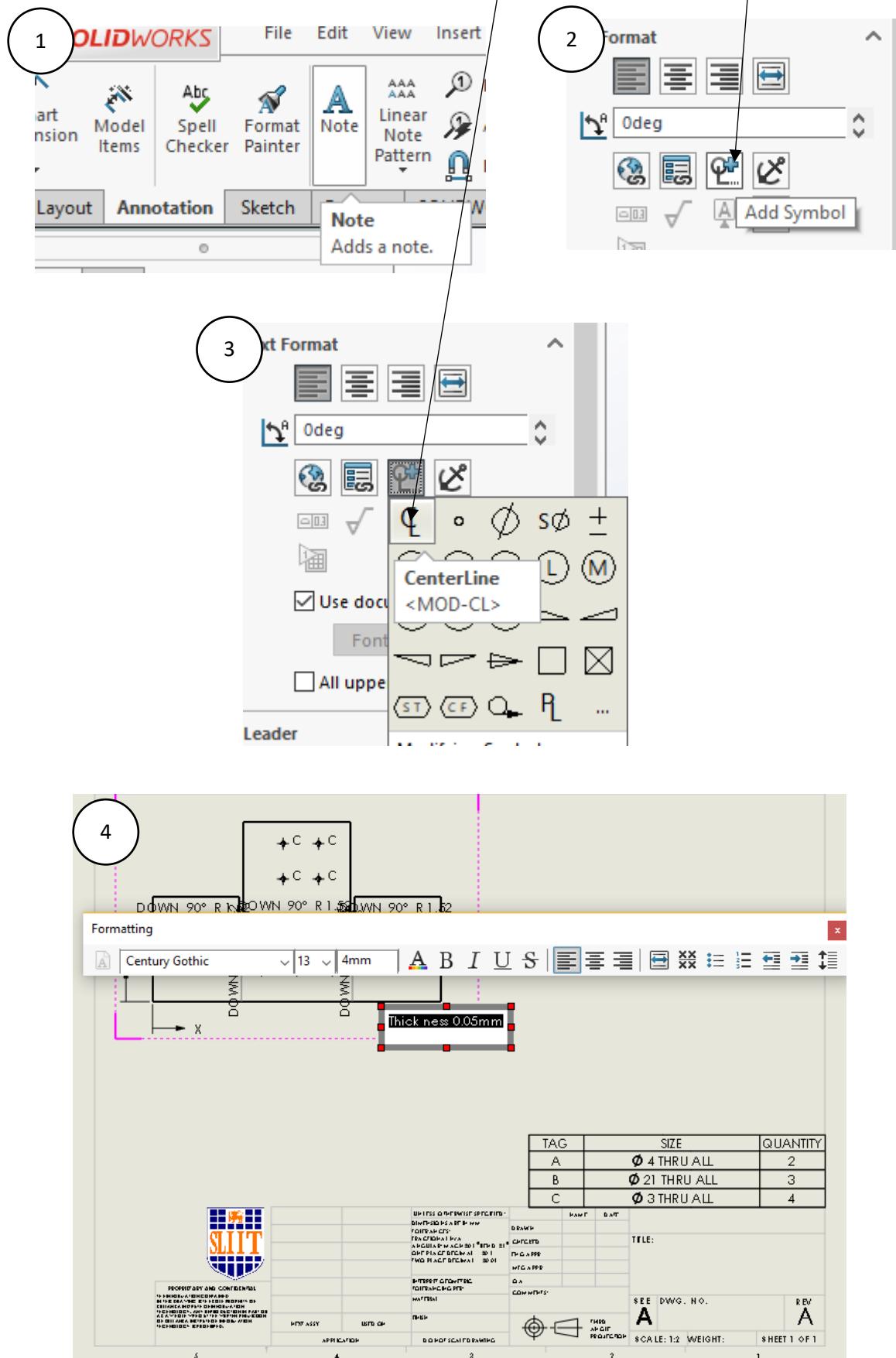
7. To simplify the table of holes click on the table itself and tick **combine same tags** under the **scheme** tab of hole table property manager



8. Simplified table.



9. To add notes click “notes” feature from Annotations toolbar. Select “Add symbol” from Notes property manager. Then select the centreline symbol to insert any notes.





Sri Lanka Institute of Information and Technology

Department of Mechanical Engineering

Engineering Drawing – ME2031

SolidWorks Laboratory - 5

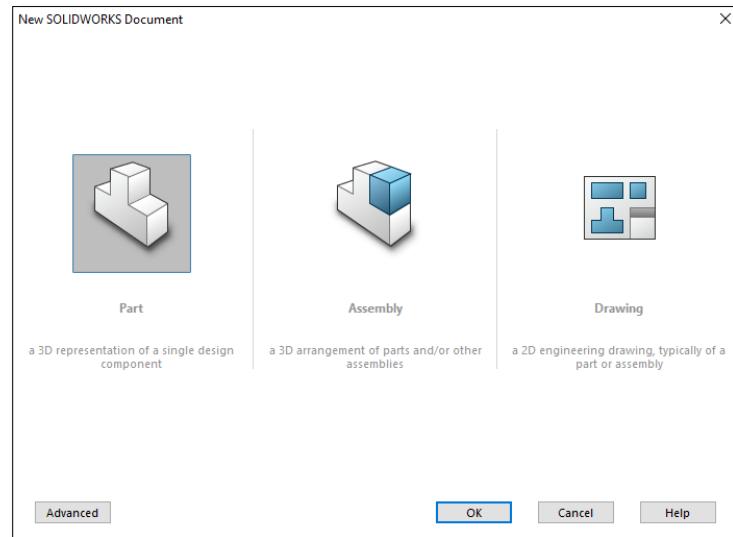
Mr. Thilina Weerakkody
Mr. Kulunu Samarakkrama

Time: 2 hours
Date : 19/03/2019

Targeted out comes of this lab

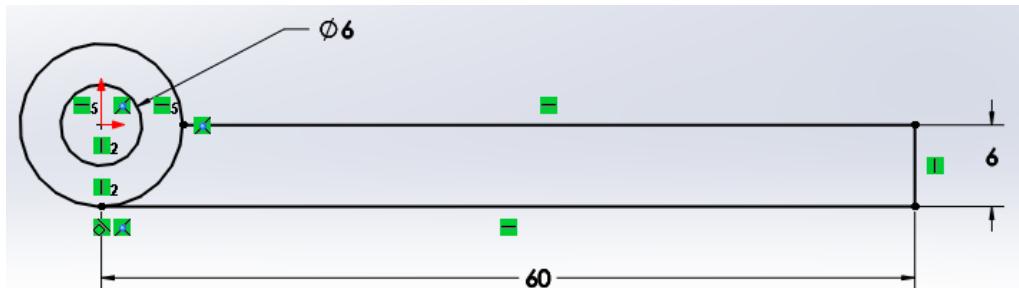
- Fundamental tools recap
- Wrap Feature
- Advanced Mates
- Exploded Views in Drawings

1. Open a new part file in SolidWorks

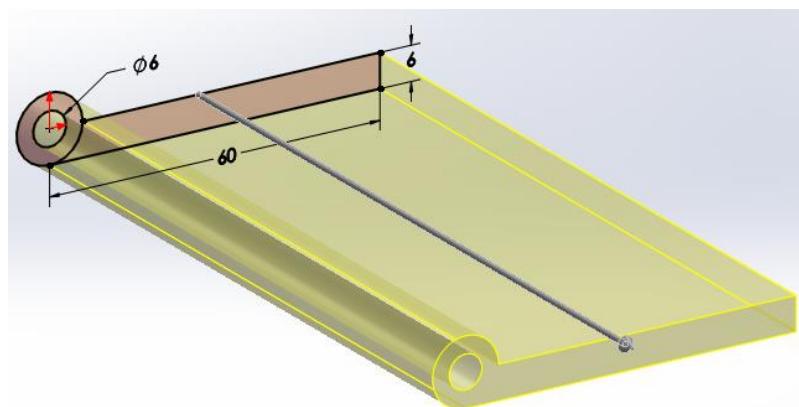


2. Create a new sketch in Right-Plane and

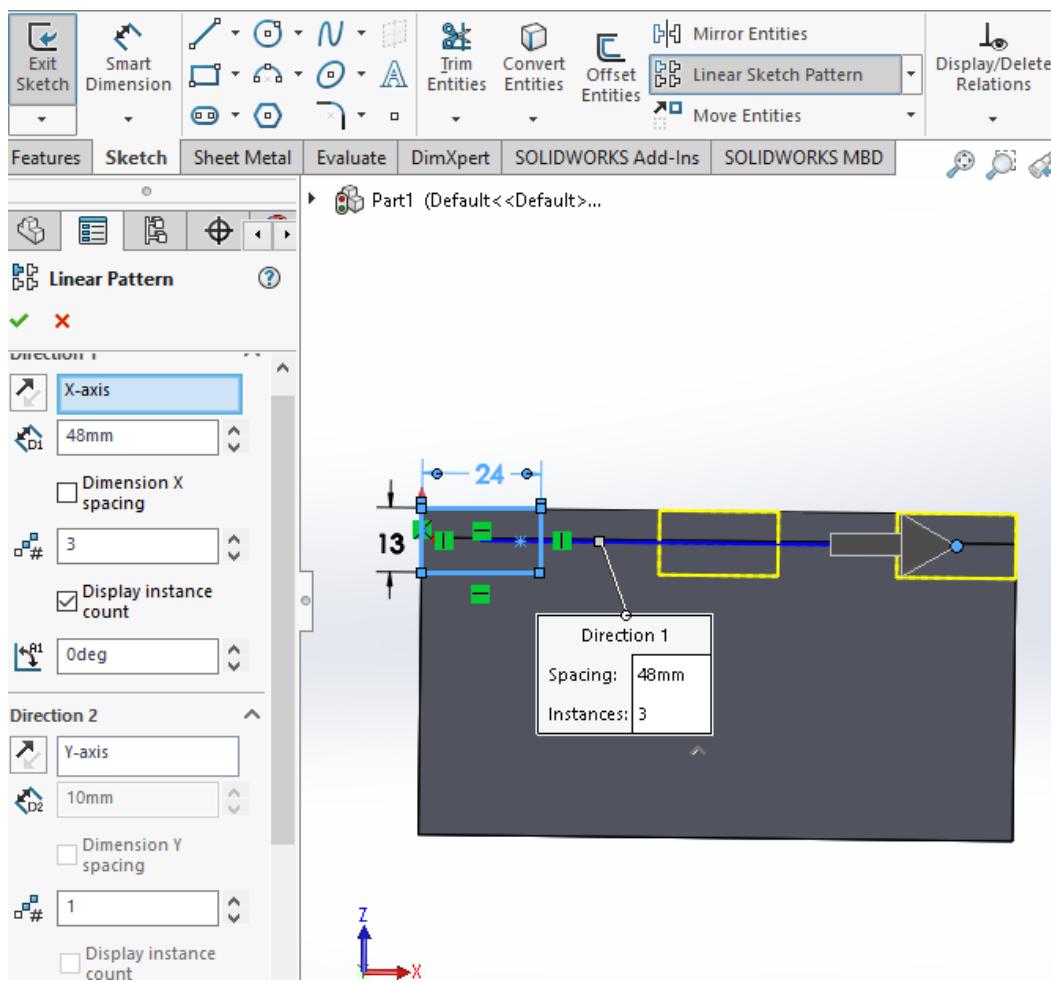
Sketch the following figure. Keep track on all relations and dimensions as described in the drawing given



3. Boss-Extrude the surface for 120mm

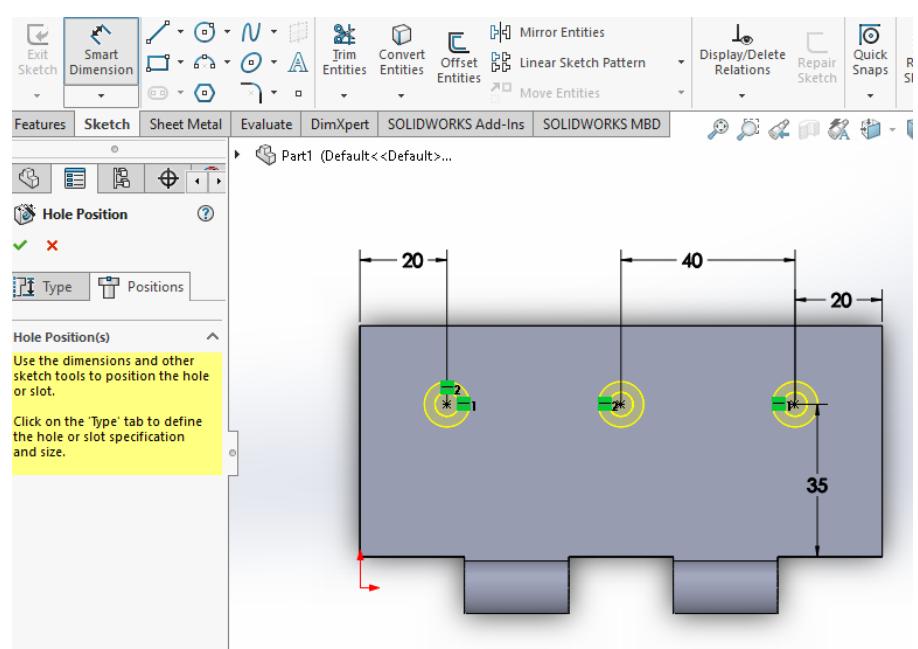


4. Press Cntrl + 6 to obtain following view Orientation. Select the following sketch surface. Draw a rectangle to given dimensions and use Linear pattern feature to copy the pattern. Cut-extrude the patterns **through all**.

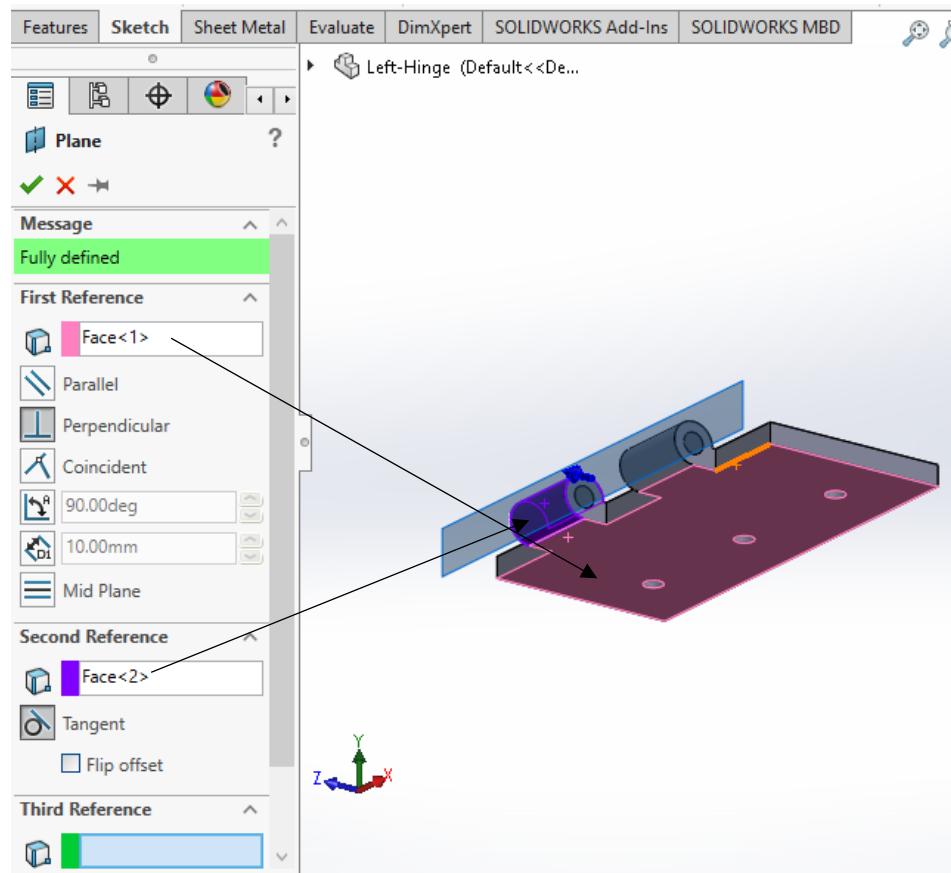


5. Use hole wizard to drill holes. Position the holes as shown

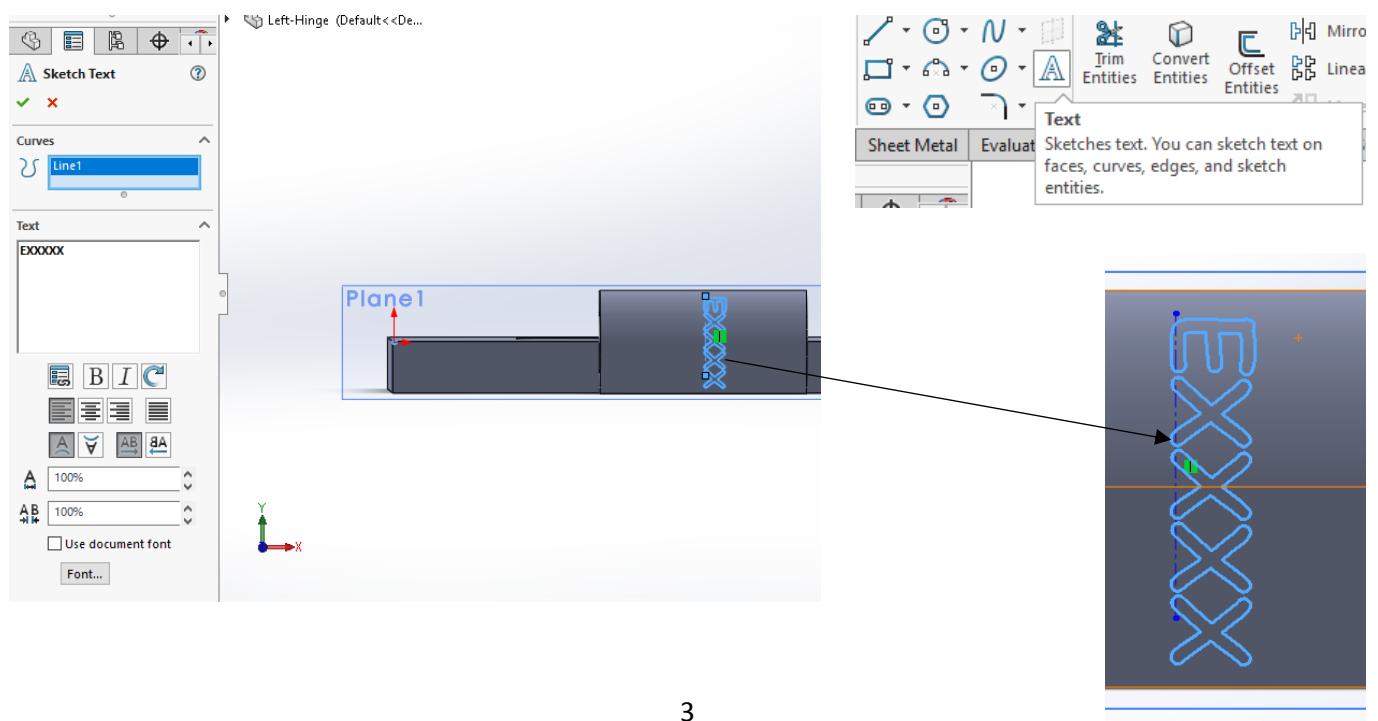
- Type: Countersink
- Standard: ISO
- Type: CTSK Flat ISO 7046-1
- Size: M6
- Fit: Normal
- End Condition: Through All



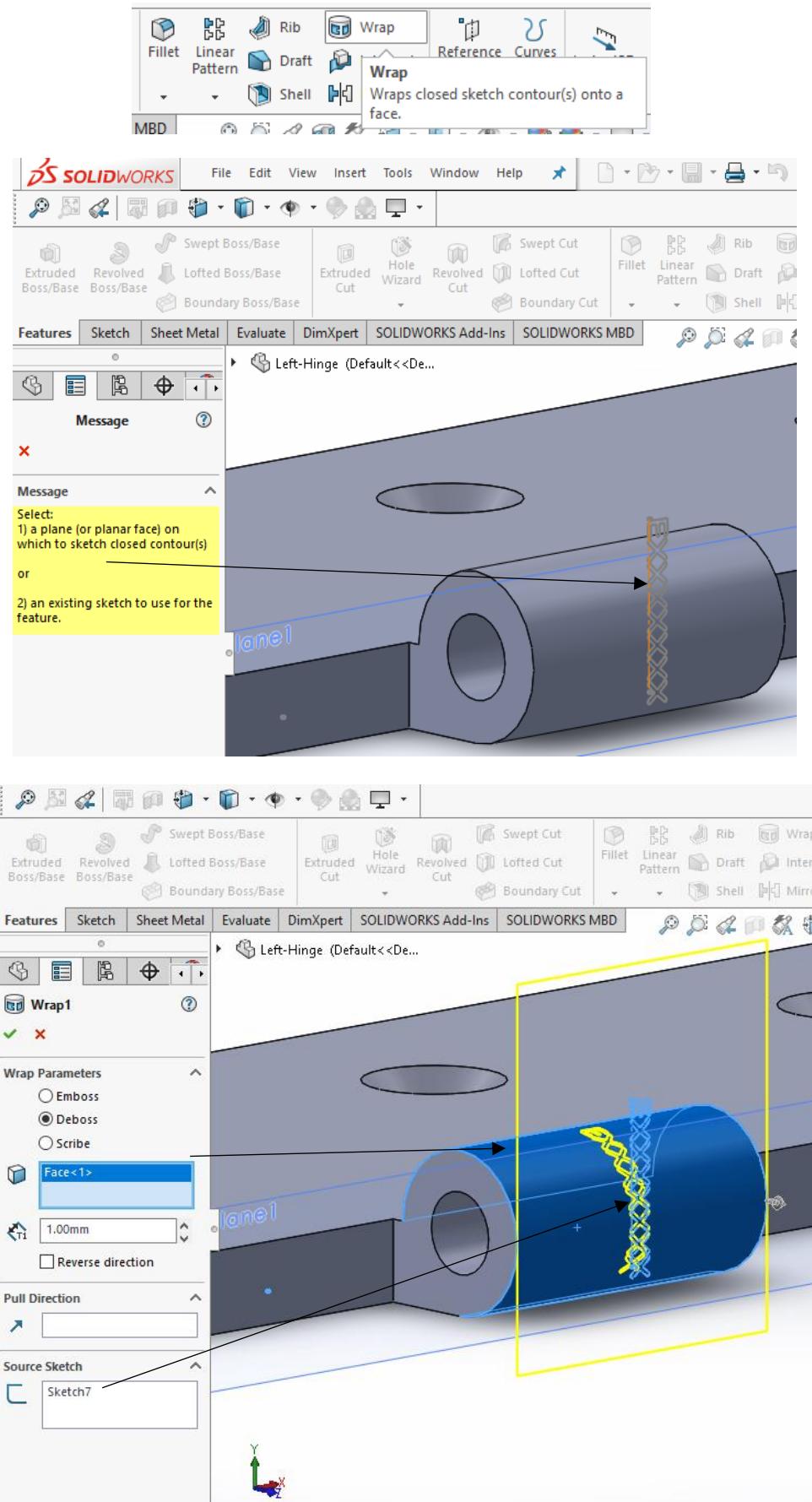
6. Select Material from **Feature Manager Design Tree** and select (Right Click) Edit Material. Select “**AISI 1020 Steel Cold Rolled**” from the available list and click apply.
7. Sketch a plane using **Insert > Reference Geometry > Plane**. Apply the settings and check OK



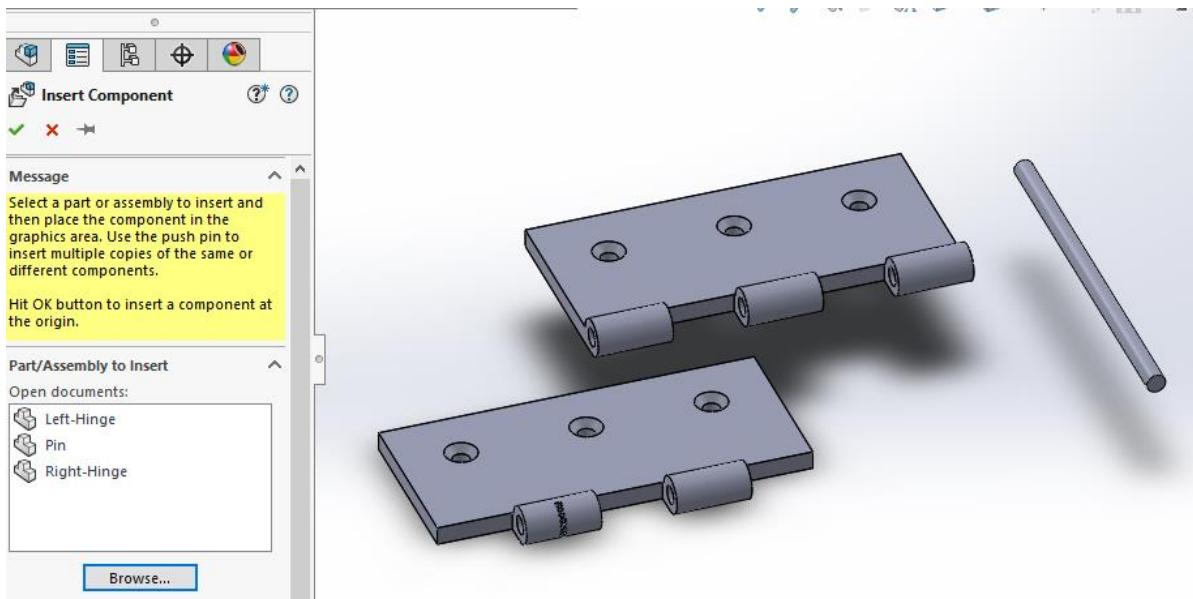
8. First draw a vertical centerline and then sketch a text on the plane created and enter your student number on it



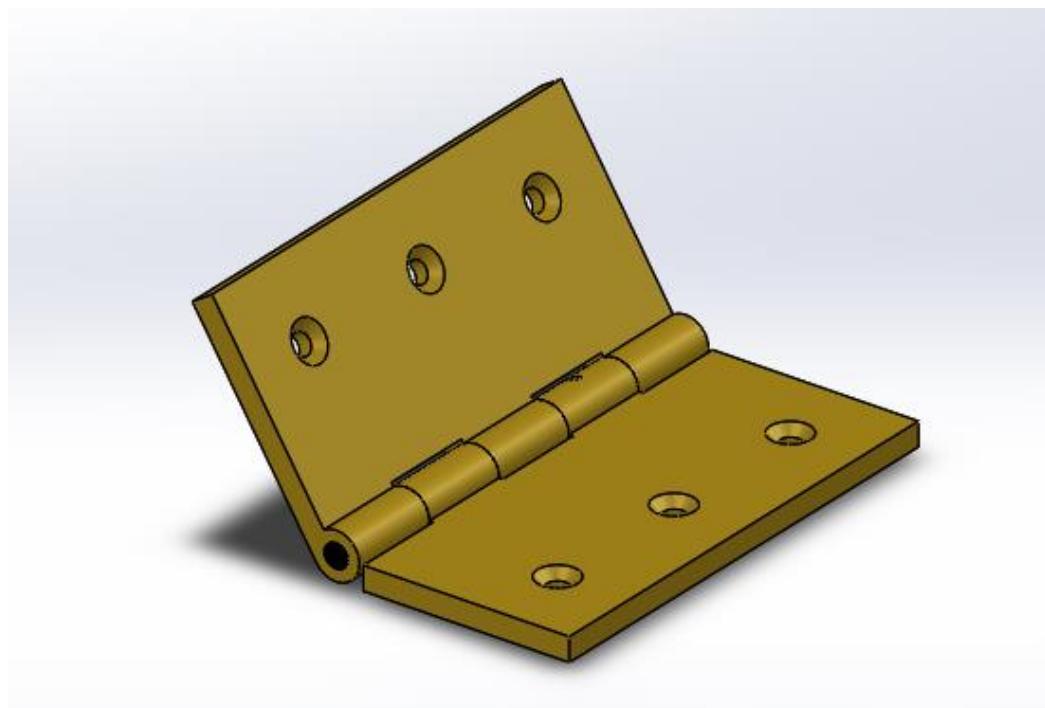
9. Exit the Sketch and use the Wrap feature in the Feature tab of Command Manger to deboss the typed text on to the hinge. When prompted to select a sketch plane, select the centerline which text is aligned. After debossing check ok



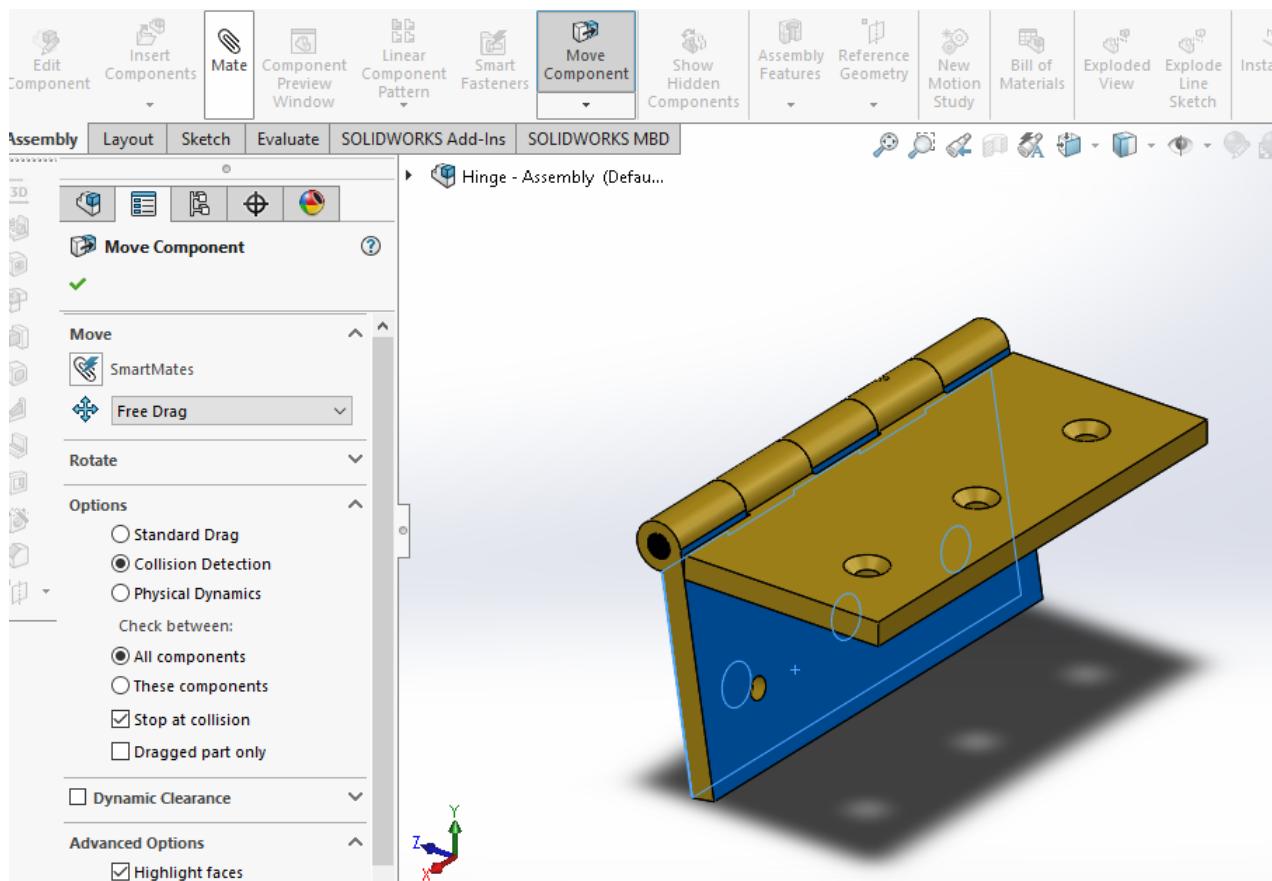
10. Save the part as **Left-Hinge** in a folder named with your **Index number_Lab05**
11. Repeat steps 1-6 to complete the other half of the hinge and save as **Right-Hinge** inside your folder
12. Create a new part file and design the **Pin** with a diameter of 6mm and length of 120mm
13. Create a new Assembly file named **Hinge-Assembly** and insert the two components to assembling environment as shown



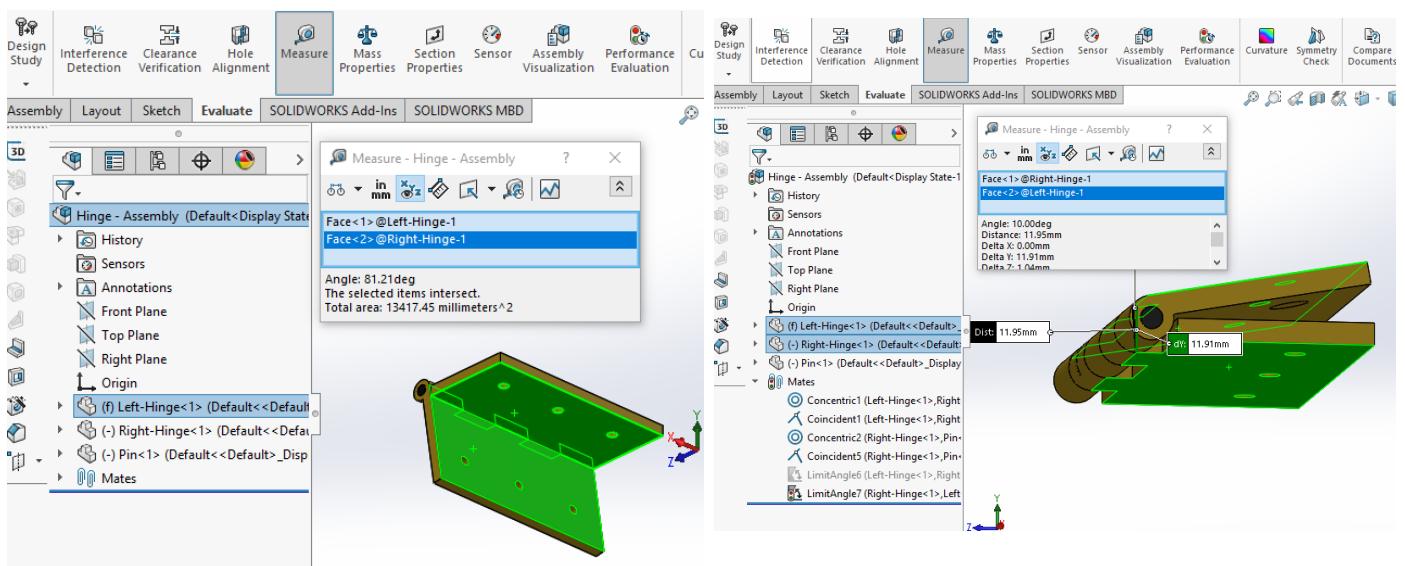
14. Use appropriate coincident and concentric mate to assemble the hinge. Edit appearances/colors and rebuild



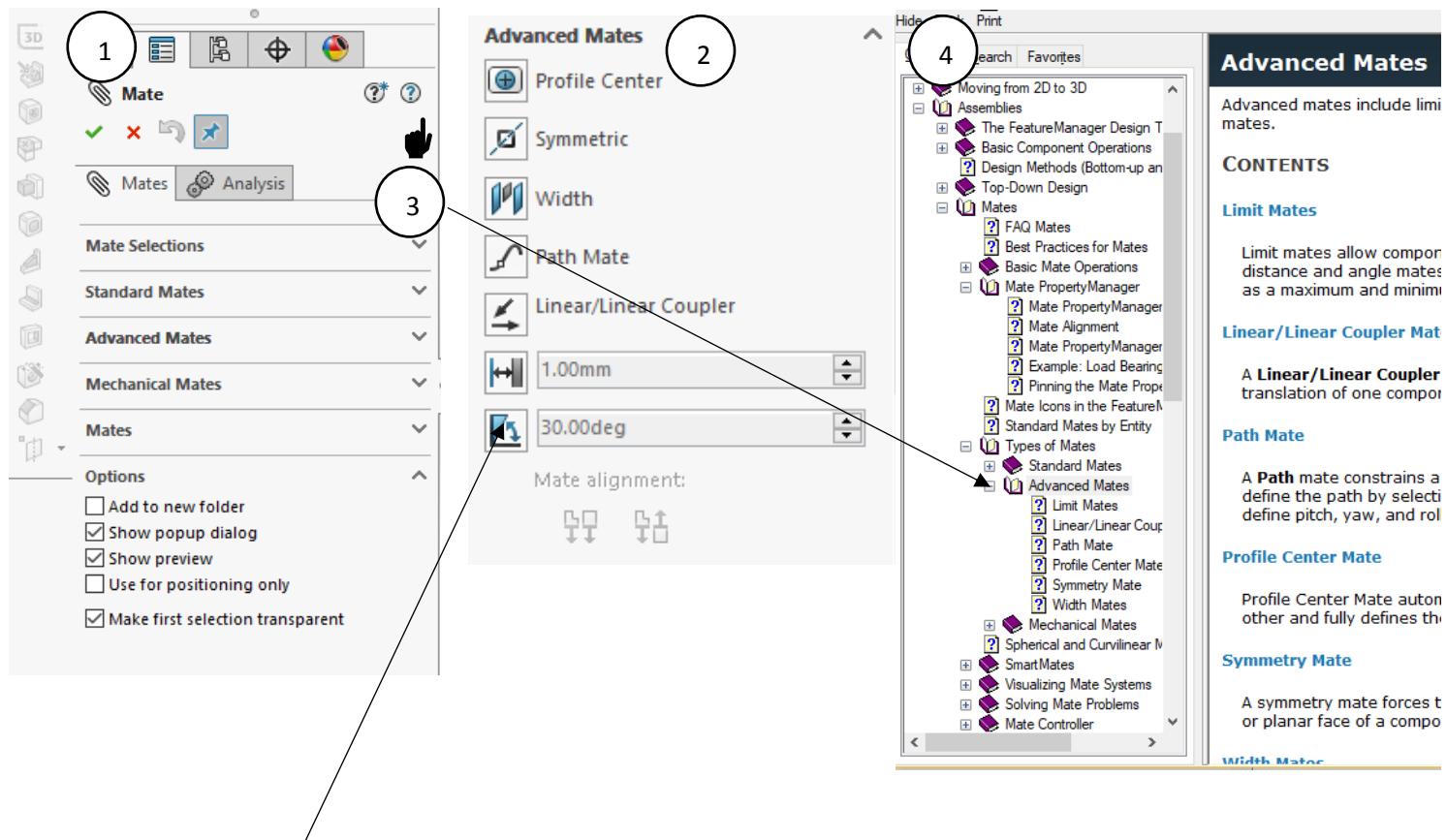
15. Select **Move Component** under Assembly tab in the **Command Manger**. In the Move Component **Property Manager** select **collision detection** under **Options** and rotate the hinge to its extremes. Check Ok



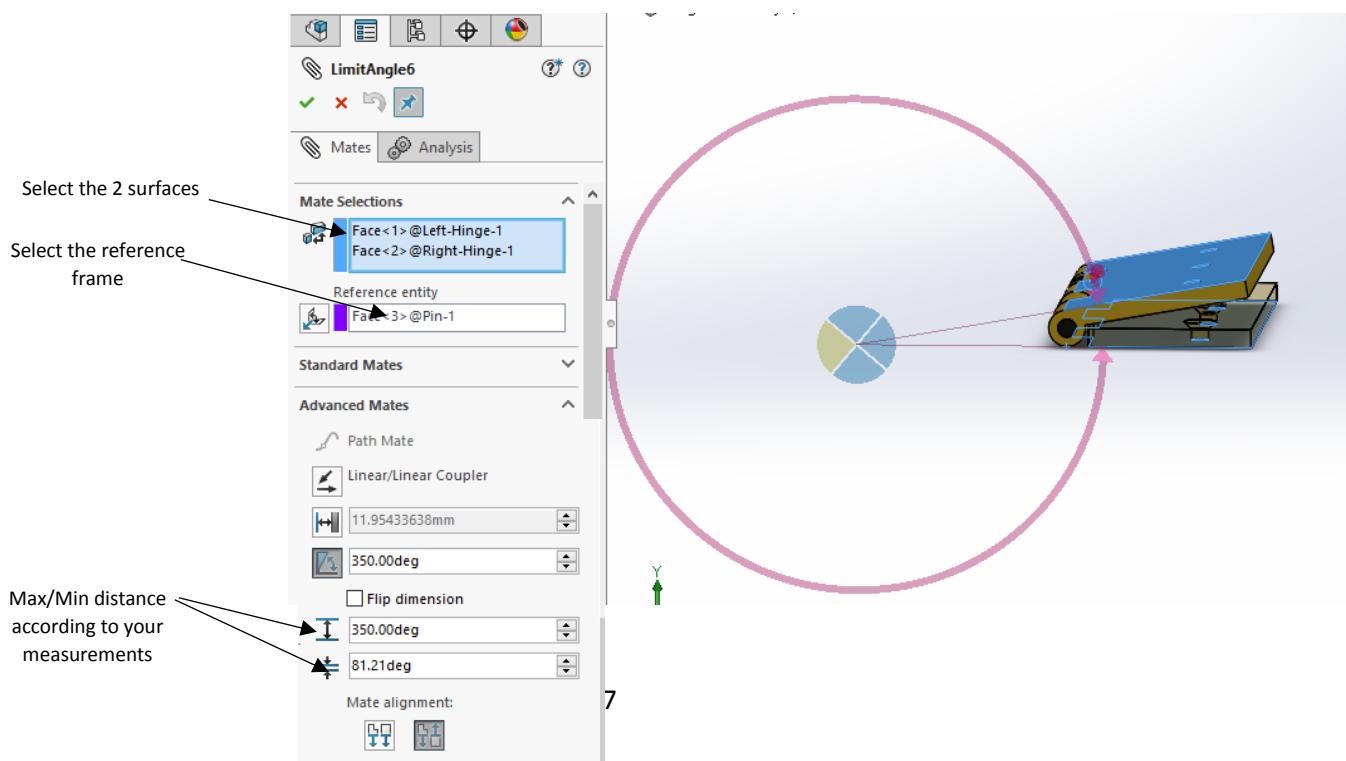
16. While hinge is rotated to one of its extremes. Select **Measure** under **Evaluate** tab in the Command Manager. Select the two faces of the hinge to measure the **angle** of rotation. Record the **maximum** and **minimum** angles. [Max: , Min:]



17. Go to **Assembly** tab and select **mate** feature. Select **Advanced mates** tab. So far in the lab series you have used standard mates. Explore the Advanced mates in this section. For more details about mates you can click on help and navigate to Advanced mate section

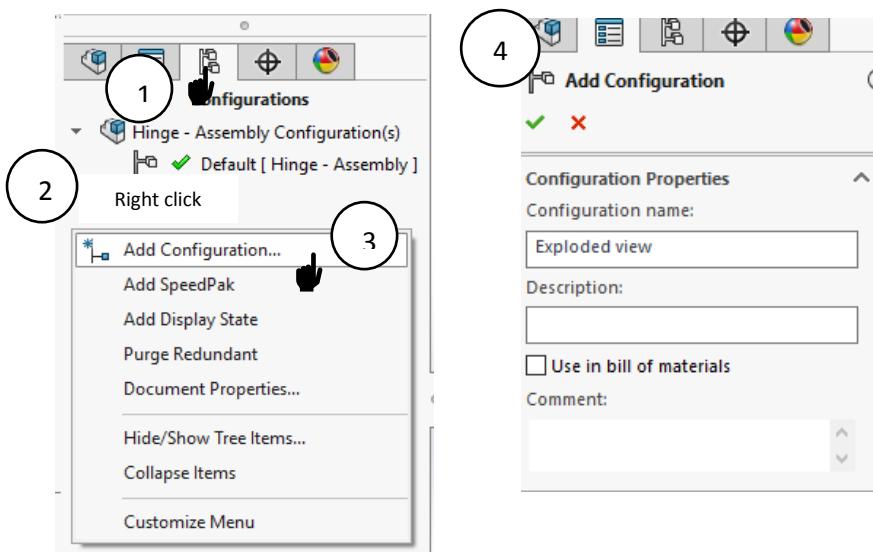


18. Click on the “Angle” option under advanced mates. Constrain the movement of the two hinges between the maximum and minimum angles.

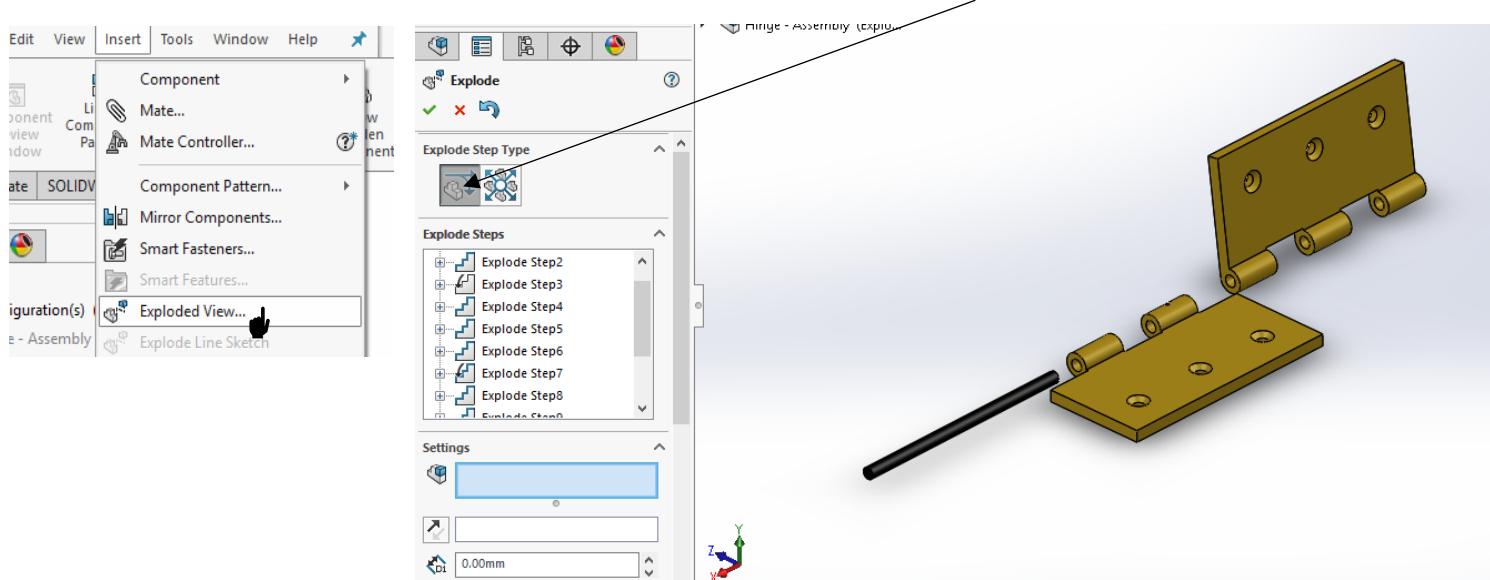


19. Create the Part file – drawings of the 3 components. Dimension them properly. Include The name of the part, Drawing number, Date, And INDEX Number in title. Generate the PDF versions of the drawings.

20. Create an exploded view of the hinge. Go to configuration Manager >> Right Click >> Add Configuration. Name new configuration as “exploded View” and check ok

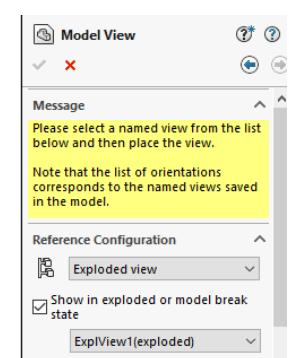


21. Go to Insert >> Exploded view. Create the exploded view using Regular step



22. Create Drawing from Assembly as usual. In the **Model view** select **Assembly file** and then **Exploded view** as configuration

23. Zip the Part files – **Right-Hinge**, **Left-Hinge**, **Pin**, Assembly file - **Hinge Assembly**, And Drawing files – **Left-Hinge.PDF**, **Right-Hinge.PDF**, **Assembly-Hinge.PDF**, **Pin.PDF** & **Left-Hinge.SLDDR**, **Right-Hinge.SLDDRW**, **Pin.SLDDRW**, **Assembly-Hinge.SLDDRW**, Files to Zip folder named with your index number and Submit online on 19/03/2019 before 4.00pm





Sri Lanka Institute of Information and Technology

Department of Mechanical Engineering

Engineering Drawing – ME2031

SolidWorks Laboratory - 6

Mr. Thilina Weerakkody
Mr. Kulunu Samarakrama

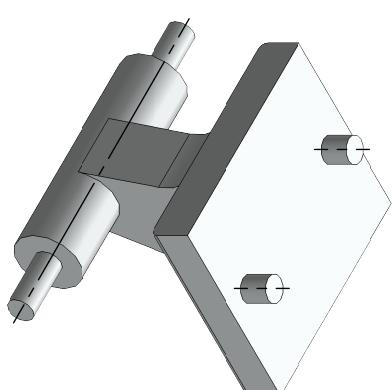
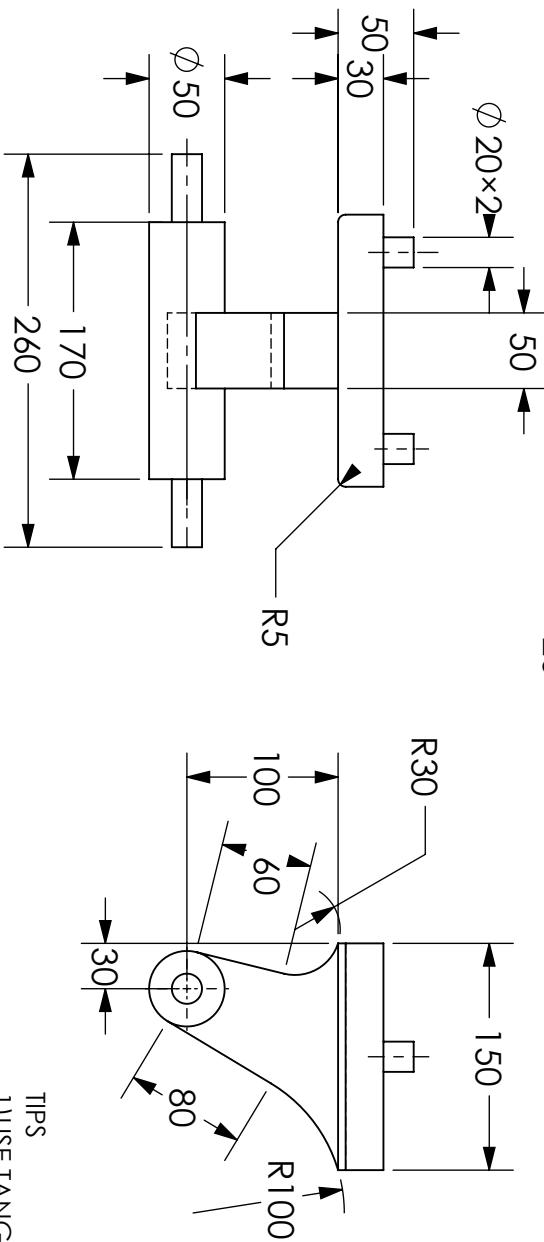
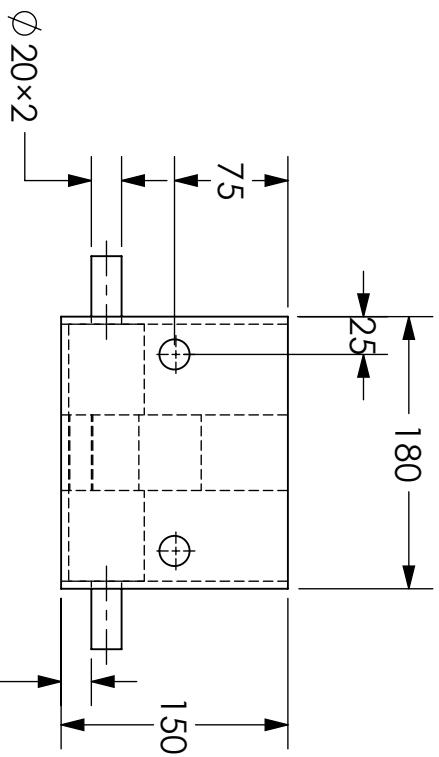
Time: 2 hours
Date : 02/04/2019

This Lab session tests your knowledge and skills on following topics and is a **preparation session for Midterm test.**

- Sketching in SolidWorks
- Implementation of Basic features
 - Extrude Boss/Base, Extrude cut
 - Revolve cut & Mirror
 - Linear Pattern & Fillet
 - Mass properties
- Assembly in SolidWorks
 - Mates feature
- SolidWorks Drawings
 - Analyzing Part file Drawing
 - Analyzing Assembly Drawing

Strictly Follow the Instructions!

- Analyze the drawings of the part files and the assembly.
- Construct the 3D models of the given drawings as separate SolidWorks part files
- Save the part files with their respective names.
- Create the SolidWorks Assembly as described in the Drawing – Assembly.
- Use the most appropriate mate features to assemble.
- Use different colors to parts and improve clarity of the assembly.
- Save the Assembly as Assembly 1
- Zip .SLDPRT files and the .SLDASM file
- Zip the submission files with the name: Surename_Index.No_Lab6 (e.g: Smith_EN14XXXXXX_Assignment6)
- Submission should be done on Courseweb at the end of the Lab session.



TIPS
1) USE TANGENT ARC TO DESIGN ARCS (e.g. A)
2) START BY SKETCHING THE PROFILE VIEW



PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
OF SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY IS PROHIBITED.

TRUCK

ME2031 - Lab 6

**INTERPRET GEOMETRIC
TOLERANCING PER:**
MATERIAL
FINISH
COMMENTS:
ALL DIMENSIONS ARE IN mm

UNLESS OTHERWISE SPECIFIED:

DIMENSIONS ARE IN MM

TOLERANCES:

FRACTIONAL N/A

ANGULAR: MACH $\pm 0.1^\circ$ BEND $\pm 1^\circ$

ONE PLACE DECIMAL
TWO PLACE DECIMAL

± 0.1
 ± 0.01

ENG APPR.
MFG APPR.

Q.A.

APPLICATION
DO NOT SCALE DRAWING

THIRD
ANGLE
PROJECTION

A

SCALE: 1:5

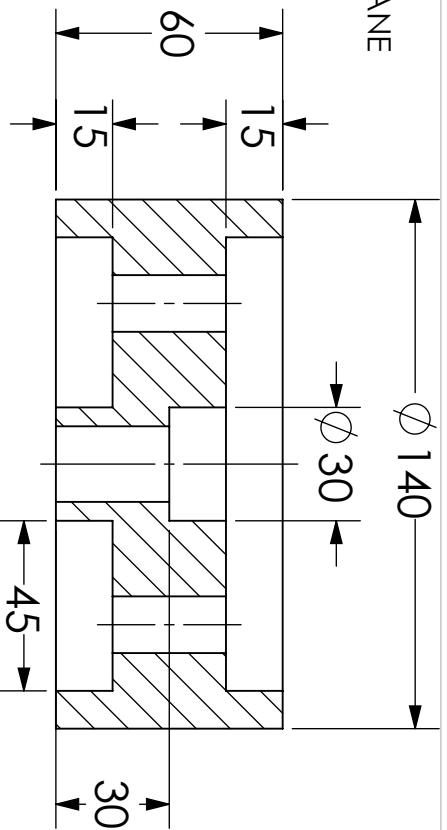
WEIGHT:

A

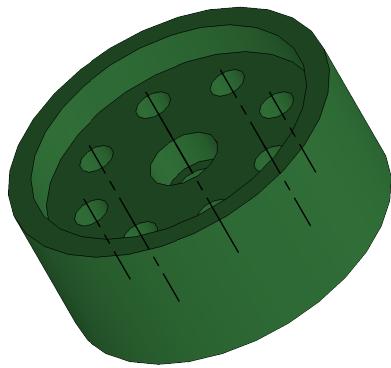
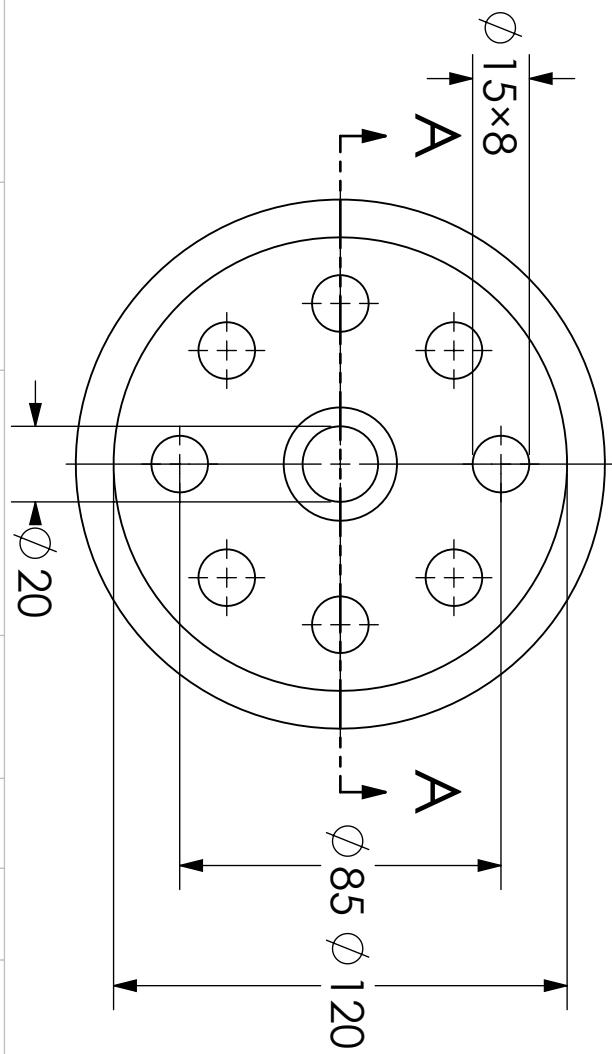
SHEET 1 OF 1

TIPS

- 1) EXTRUDE DIRECTION : MID PLANE
- 2) USE SELECTED CONTOURS



SECTION A-A



UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MM
TOLERANCES:

FRACTIONAL N/A
ANGULAR: MACH $\pm 1^\circ$ BEND $\pm 1^\circ$
ONE PLACE DECIMAL ± 0.1
TWO PLACE DECIMAL ± 0.01

INTERPRET GEOMETRIC
TOLERANCING PER:
MATERIAL

ALL DIMESIONS ARE IN mm

COMMENTS:

REV

SCALE: 1:2 WEIGHT: A

DRAWN MR DATE
CHECKED ENG APPR.
MFG APPR.

NAME DATE
21/03/18

ME2031 - Lab 06

WHEEL



PROPRIETARY AND CONFIDENTIAL

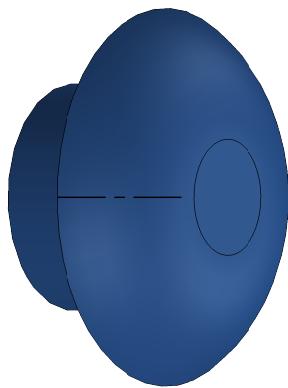
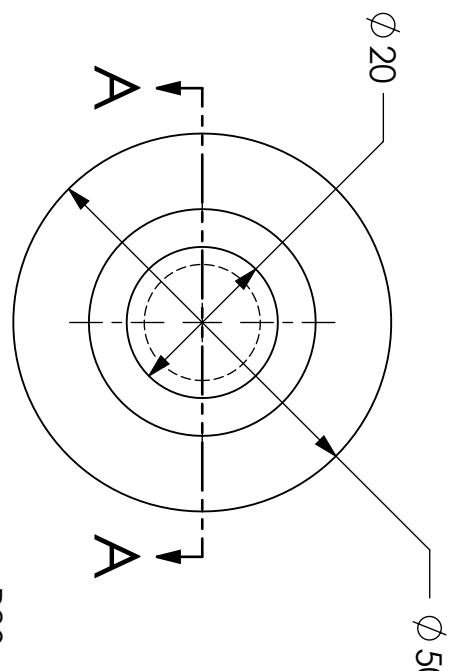
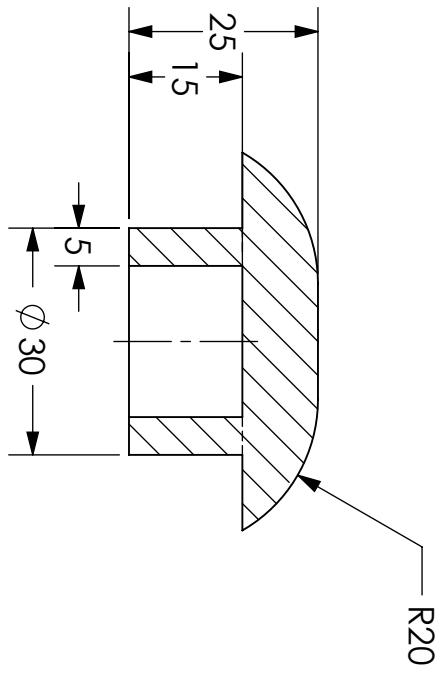
THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
OF SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY IS PROHIBITED.

APPLICATION	USED ON	FINISH	MATERIAL	DO NOT SCALE DRAWING
		GREEN LOW GLOSS PLASTIC		



PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND ANY REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
IS PROHIBITED.

SECTION A-A

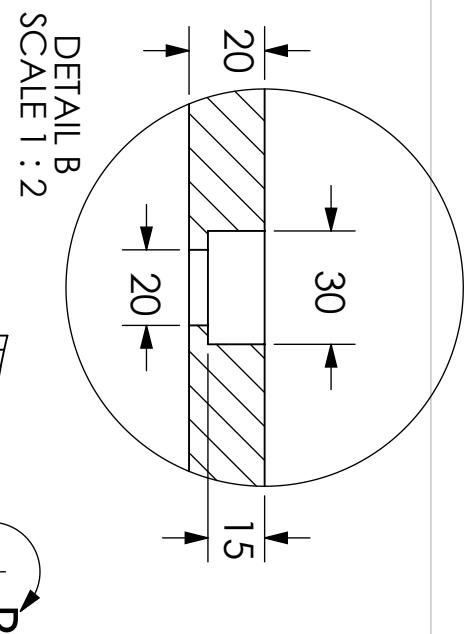


LOCK

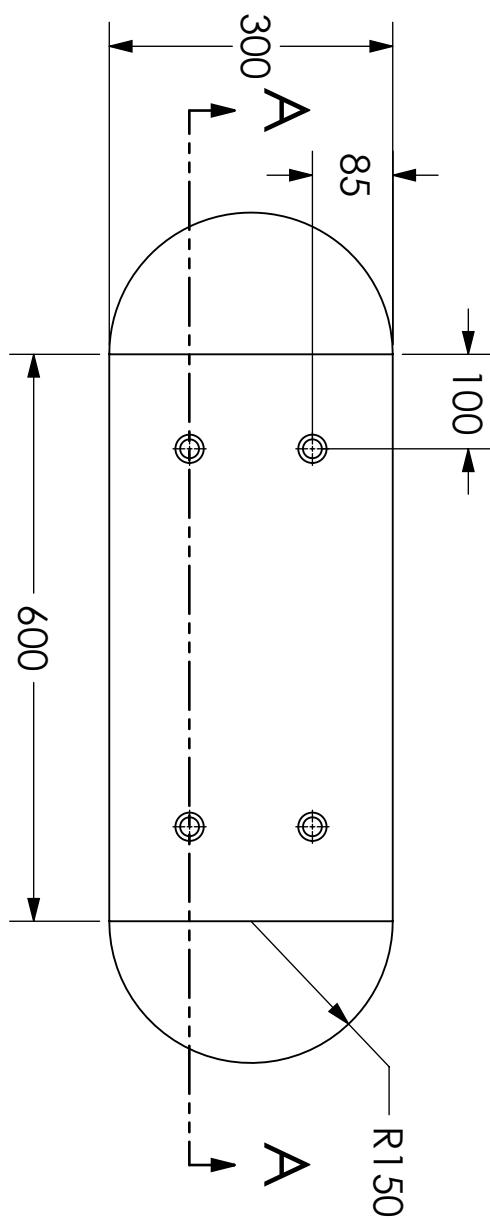
APPLICATION		USED ON	FINISH	MATERIAL	INTERPRET GEOMETRIC TOLERANCING PER:			COMMENTS:		
			BLUE ANODIZED ALUMINIUM		UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MM	DRAWN	NAME MR	DATE 21/03/18	ME2031 - Lab 06	TOLERANCES: FRACTIONAL MIN. ANGULAR: MACH $\pm 0.1^\circ$ BEND $\pm 1^\circ$ ONE PLACE DECIMAL TWO PLACE DECIMAL
					CHECKED ± 0.1	ENG APPR.			LOCK	ENG APPR.
					Q.A. MFG APPR.					MFG APPR.
					ALL DIMENSIONS ARE IN mm	SIZE A	DWG. NO. 3 OF 5	REV A	SCALE: 1:1	WEIGHT:
					THIRD ANGLE PROJECTION					SHEET 1 OF 1

TIPS

- 1) SKETCH THE FRONT FACE FIRST
- 2) EXTRUDE TO GET RECTANGULAR TOP FACE
- 3) USE EXTRUDED CUT TO DESIGN SEMI-CIRCULAR SECTIONS



SECTION A-A



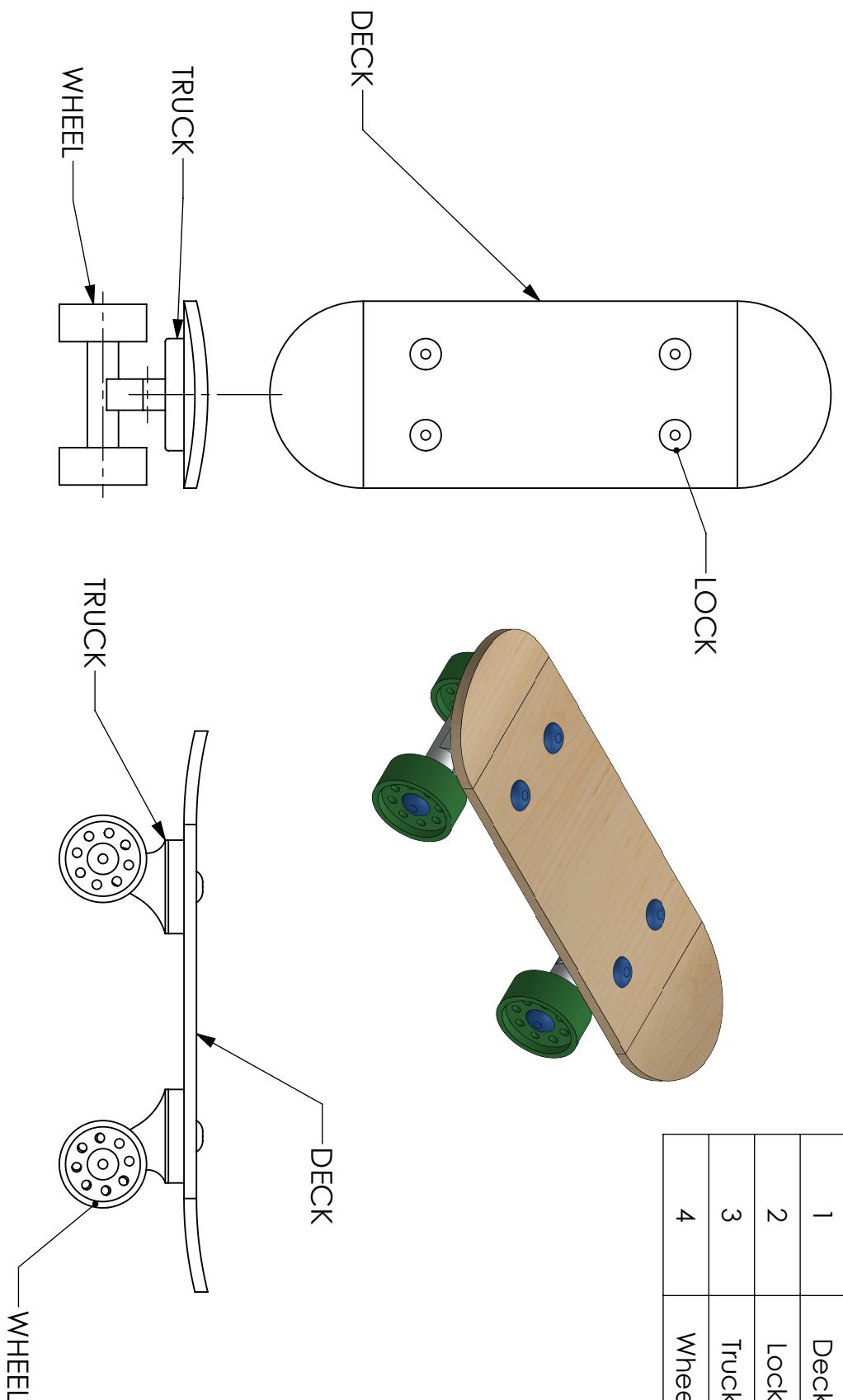
PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
IS PROHIBITED.

APPLICATION	USED ON	FINISH	INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL	DRAWN	NAME	DATE	MR
	POLISHED MAPLE 2D				ME2031 - Lab 06	21/03/18	
				CHECKED			
				ENG APPR.			
				MFG APPR.			
				Q.A.			
			COMMENTS: ALL DIMENSIONS ARE IN mm				

SIZE **A** DWG. NO. **4 OF 5** REV **A**

ITEM NO.	PART	QTY.
1	Deck	1
2	Lock	8
3	Truck	2
4	Wheel	4



PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND ANY REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
IS PROHIBITED.

SKATEBOARD ASSEMBLY

APPLICATION	USED ON	FINISH	DO NOT SCALE DRAWING
INTERPRET GEOMETRIC TOLERANCING PER:	MATERIAL	COMMENTS:	
DRAWN	MR	DATE	ME2031 - Lab 06
TOLERANCES: FRACTIONAL N/A ANGULAR: MACH $\pm 0.1^\circ$ BEND $\pm 1^\circ$ ONE PLACE DECIMAL TWO PLACE DECIMAL ± 0.1 ± 0.01	CHECKED	ENG APPR.	MFG APPR.
Q.A.			
SIZE A	DWG. NO. 5 OF 5	REV A	SCALE: 1:10 WEIGHT: SHEET 1 OF 1



Sri Lanka Institute of Information and Technology

Department of Mechanical Engineering

Engineering Drawing – ME2031

SolidWorks Laboratory – 8

Mr. Thilina Weerakkody
Mr. Kulunu Samarakkrama

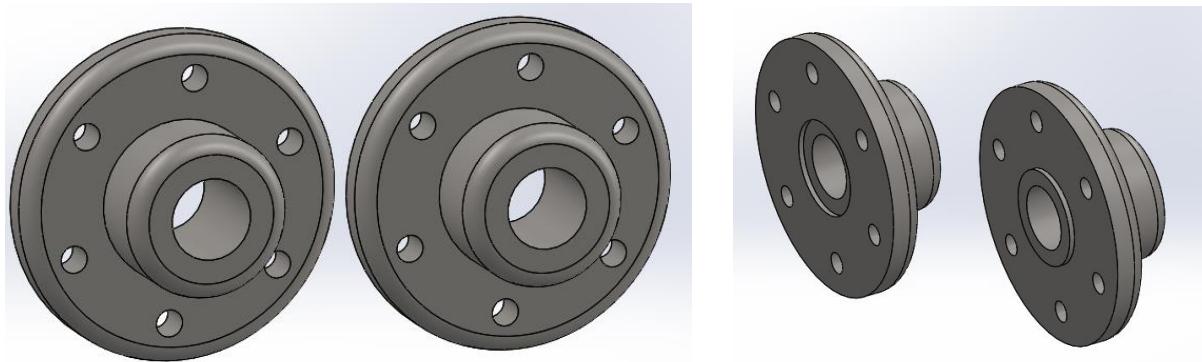
Mechanical Components- Flange, Keys, Nut & Bolt
Date : 06/05/2019

Targeted out comes of this lab

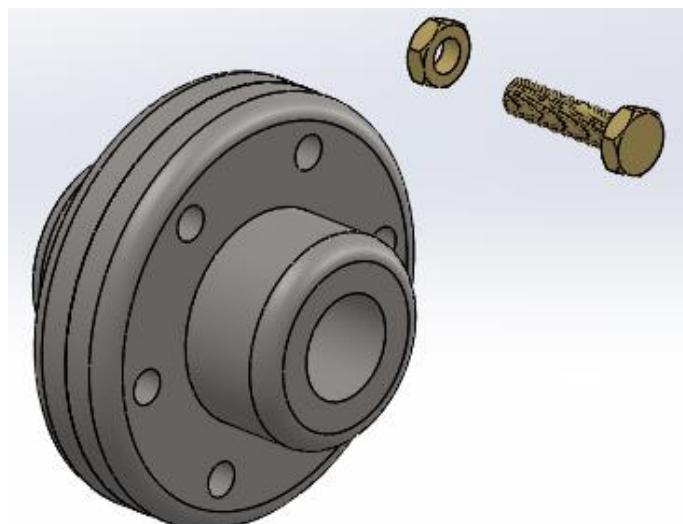
- Flange & Keys
- Nut & Bolt

Instructions

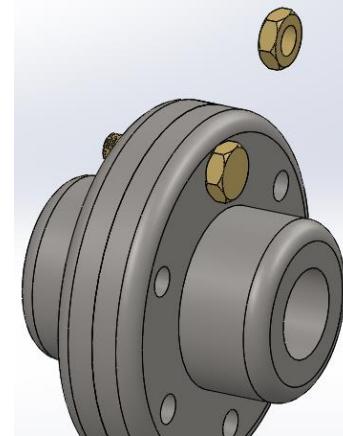
- Analyze the drawings and model the following Mechanical components with the guidelines given in the lab session



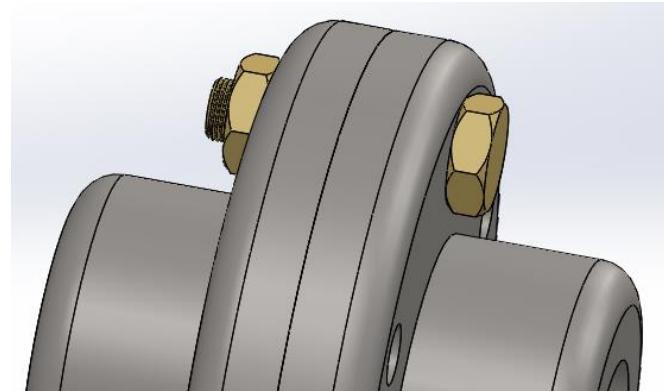
- Mate the Flange 1 with assembly using **Front, Right and Top Planes**.
- Using **Concentric mate, Coincident mate** and **Top Plane/Concentric mate** for a bolt hole, mate the 2nd flange.
- Insert a M8 Nut and Bolt to the assembly.



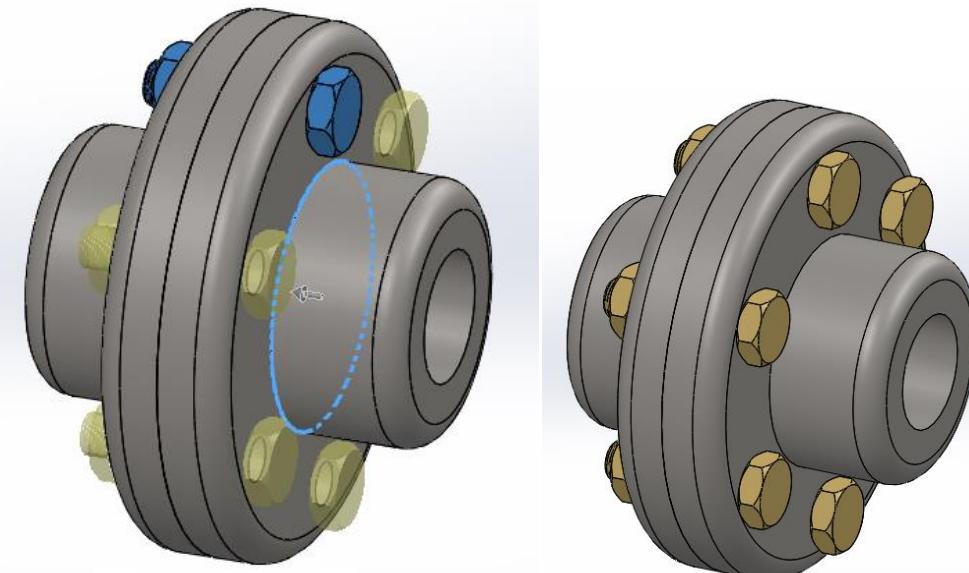
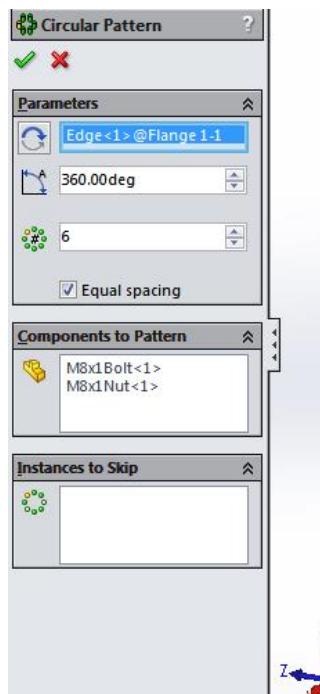
- Mate the Bolt with the flange using **concentric mate**, **Coincide mate** with the flange and top plane **Parallel mate** with the assembly.

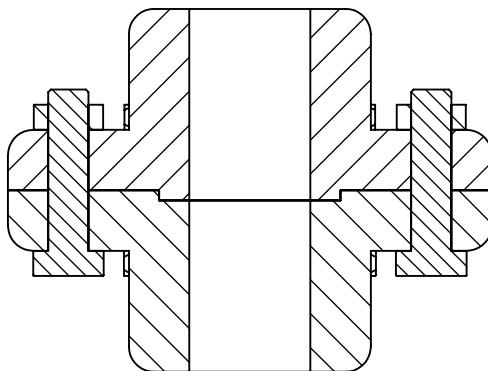
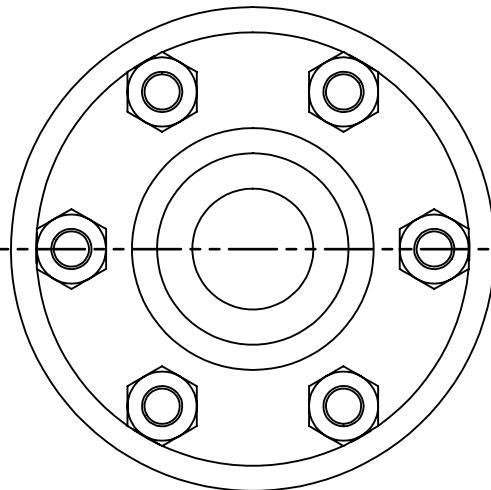
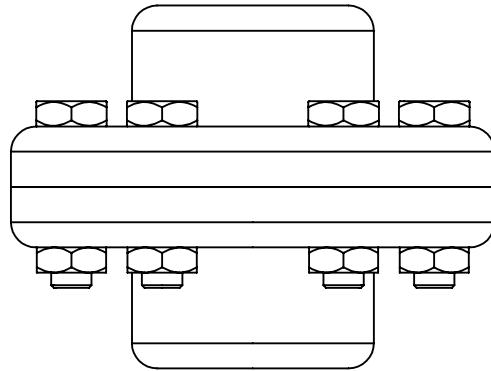


- Assemble the Nut with the Bolt using same set of mates provided in above bolt mate section.

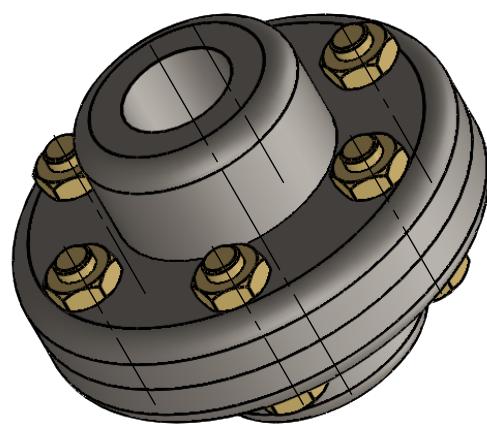


- Using **Assembly-> Circular patters** feature, obtain all 6 nuts and bolts.





SECTION B-B
SCALE 1 : 1.5



PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
OF SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY IS PROHIBITED.

UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MM
TOLERANCES:

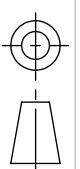
FRACTIONAL N/A
ANGULAR: MACH $\pm 0.1^\circ$ BEND $\pm 1^\circ$
ONE PLACE DECIMAL
TWO PLACE DECIMAL

± 0.1
 ± 0.01

INTERPRET GEOMETRIC
TOLERANCING PER:
MATERIAL

COMMENTS:

FINISH



APPLICATION

DO NOT SCALE DRAWING

PROJECTION

THIRD ANGLE

PROJECTION

SCALE: 1:2

WEIGHT:

SHEET 1 OF 1

ME2031 - Lab8

TITLE:

Assembly

A

DWG. NO.

5

A

REV



PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
OF SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY IS PROHIBITED.

SECTION A-A SCALE 1 : 1

UNLESS OTHERWISE SPECIFIED:

DIMENSIONS ARE IN MM
TOLERANCES:

FRACTIONAL N/A

ANGULAR: MACH $\pm 0.1^\circ$

BEND $\pm 1^\circ$

ONE PLACE DECIMAL

TWO PLACE DECIMAL

± 0.1

± 0.01

Q.A.

ENG APPR.

MFG APPR.

CHECKED

DRAWN

NAME

DATE

TITLE:

ME2031 - Lab 8

Flange 1

INTERPRET GEOMETRIC
TOLERANCING PER:

COMMENTS:

MATERIAL

FINISH

USED ON

APPLICATION

DO NOT SCALE DRAWING

PROJECTION

THIRD ANGLE

PROJECTION

SIZE

A

DWG. NO.

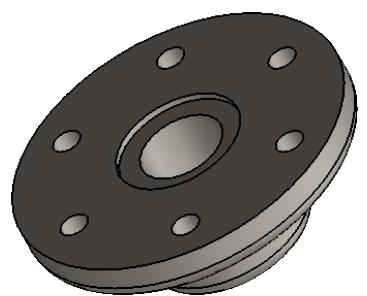
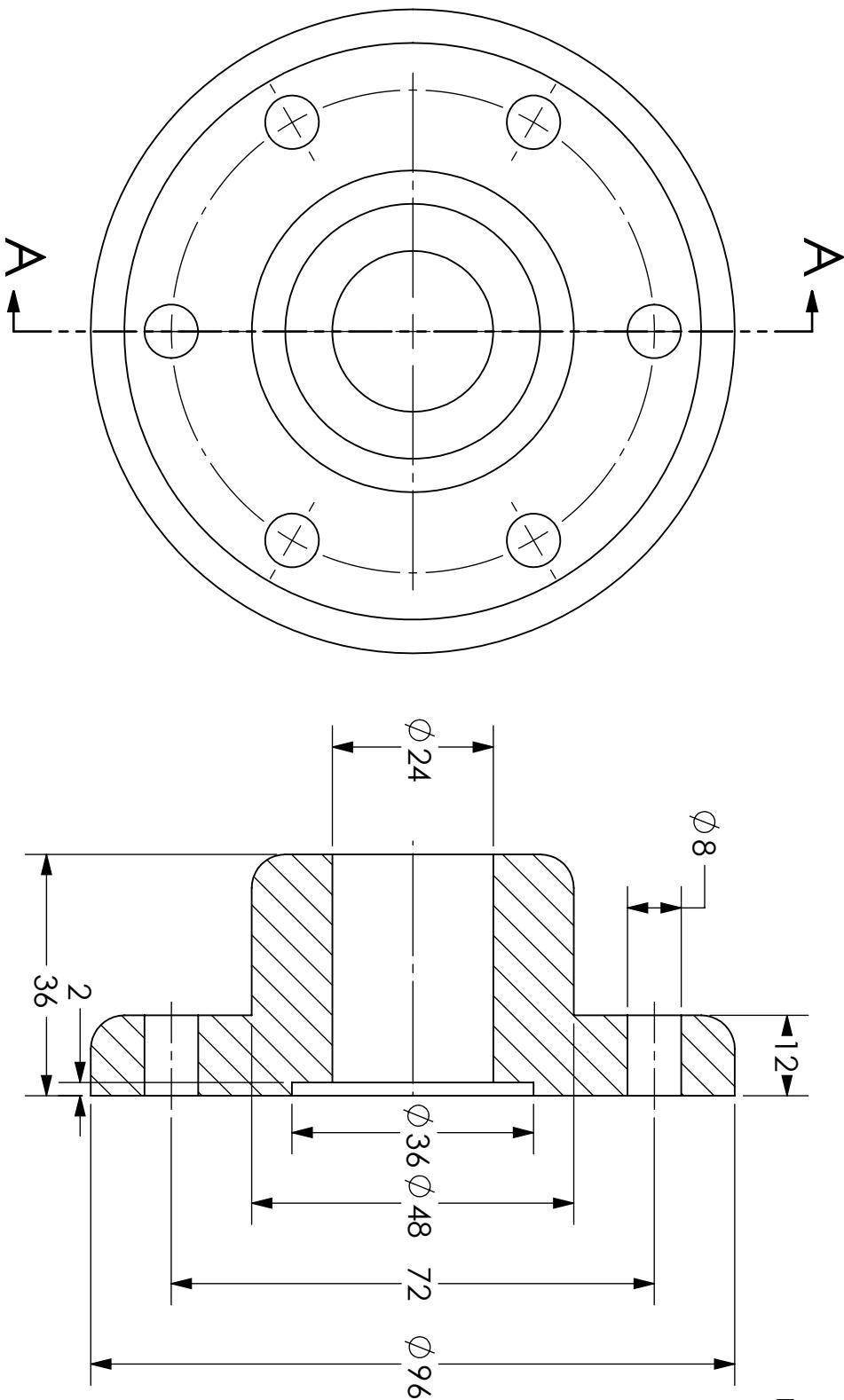
1

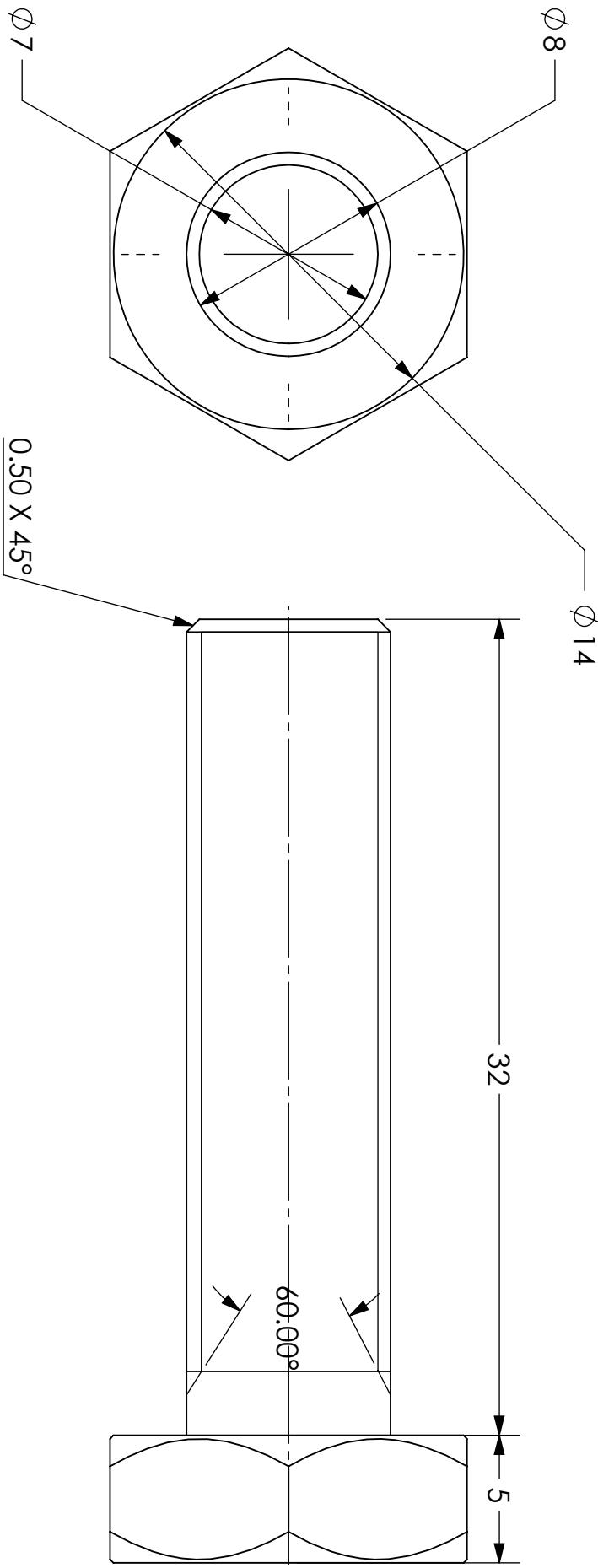
SCALE: 1:2

REV:

WEIGHT:

SHEET 1 OF 1





UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MM
TOLERANCES:
FRACTIONAL N/A
ANGULAR: MACH $\pm 0.1^\circ$ BEND $\pm 1^\circ$
ONE PLACE DECIMAL
TWO PLACE DECIMAL

DRAWN
CHECKED
 ± 0.1
 ± 0.01

ENG APPR.
MFG APPR.
Q.A.

INTERPRET GEOMETRIC
TOLERANCING PER:

MATERIAL

COMMENTS:

THIRD
ANGLE
PROJECTION

SCALE: 2:1

WEIGHT:

SHEET 1 OF 1

NAME DATE

TITLE:

Bolt

REV A

SIZE A

DWG. NO. 3

APPLICATION

USED ON

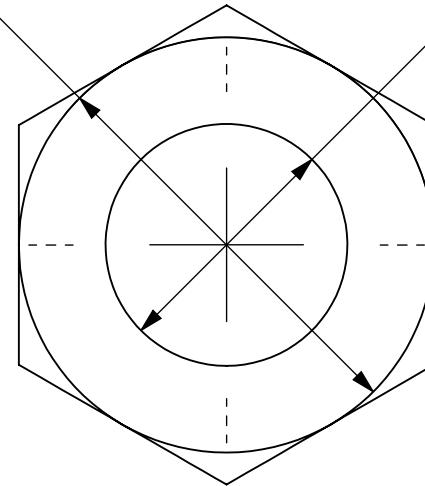
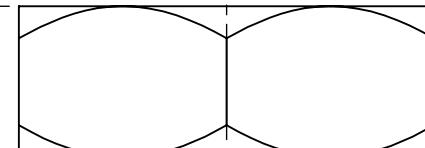
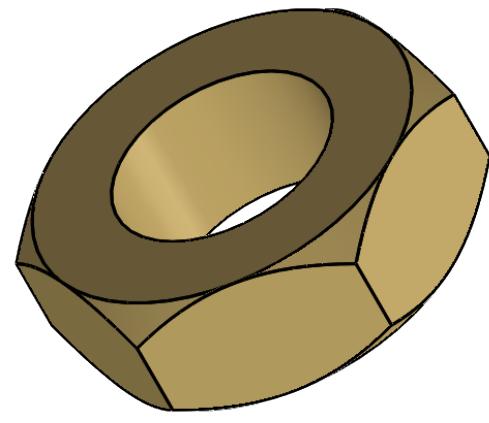
FINISH

DO NOT SCALE DRAWING

PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED
IN THIS DRAWING IS THE SOLE PROPERTY OF
SRILANKA INSTITUTE OF INFORMATION
TECHNOLOGY AND REPRODUCTION IN PART OR
AS A WHOLE WITHOUT THE WRITER'S PERMISSION
IS PROHIBITED.



ME2031 - Lab 8

PROPRIETARY AND CONFIDENTIAL		THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF SRILANKA INSTITUTE OF INFORMATION TECHNOLOGY AND REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITER'S PERMISSION OF SRILANKA INSTITUTE OF INFORMATION TECHNOLOGY IS PROHIBITED.		
 <p>ME2031 - Lab 8</p> <p>Nut</p> <p>A 4 A</p> <p>SCALE: 5:1 WEIGHT: SHEET 1 OF 1</p>				
<p>UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MM TOLERANCES: FRACTIONAL N/A ANGULAR: MACH $\pm 0.1^\circ$ BEND $\pm 1^\circ$ ONE PLACE DECIMAL TWO PLACE DECIMAL</p> <p>DRAWN NAME DATE</p> <p>CHECKED TITLE:</p> <p>ENG APPR. MFG APPR.</p> <p>Q.A.</p> <p>INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL</p> <p>USED ON</p> <p>FINISH</p> <p>DO NOT SCALE DRAWING</p> <p>APPLICATION</p> <p>COMMENTS:</p> <p>THIRD ANGLE PROJECTION</p>				
  				
<p>5</p> <p>4</p> <p>3</p> <p>2</p> <p>1</p>				



Sri Lanka Institute of Information and Technology

Department of Mechanical Engineering

Engineering Drawing – ME2031

SolidWorks Laboratory - 8

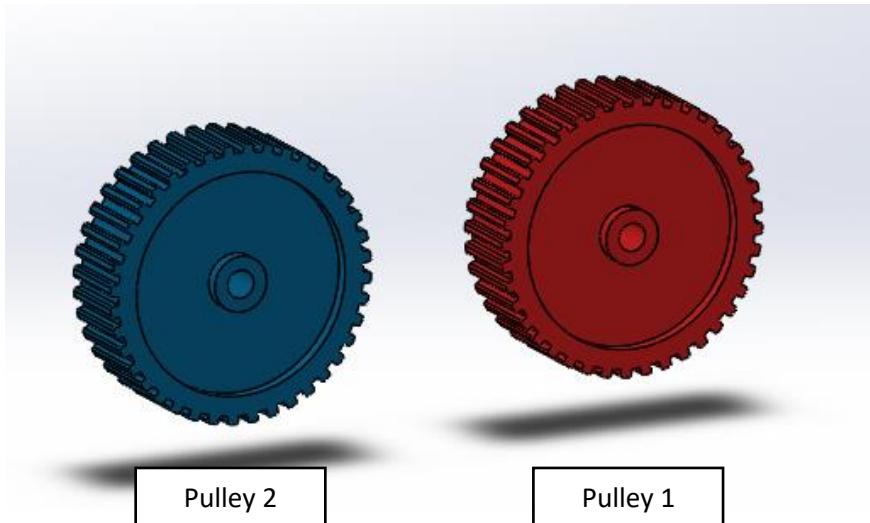
Mr. Thilina Weerakkody
Mr. Kulunu Samarakkrama

Mechanical Components- Pulley & Belt
Date : 06/05/2019

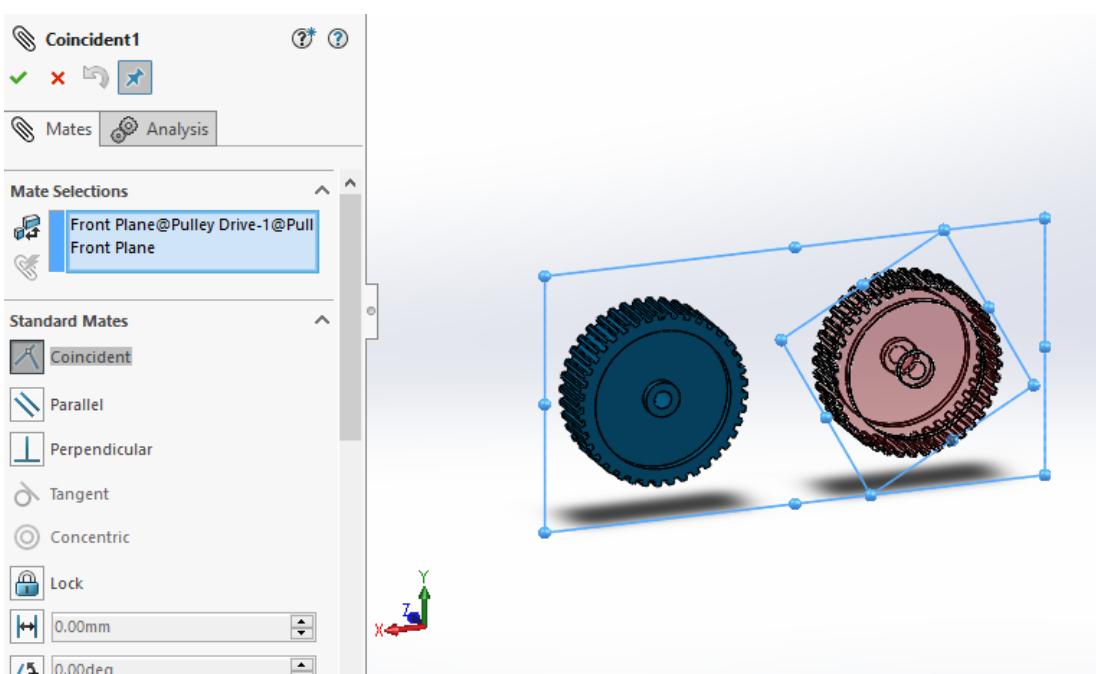
Targeted out comes of this lab

- Pulley belt Assembly

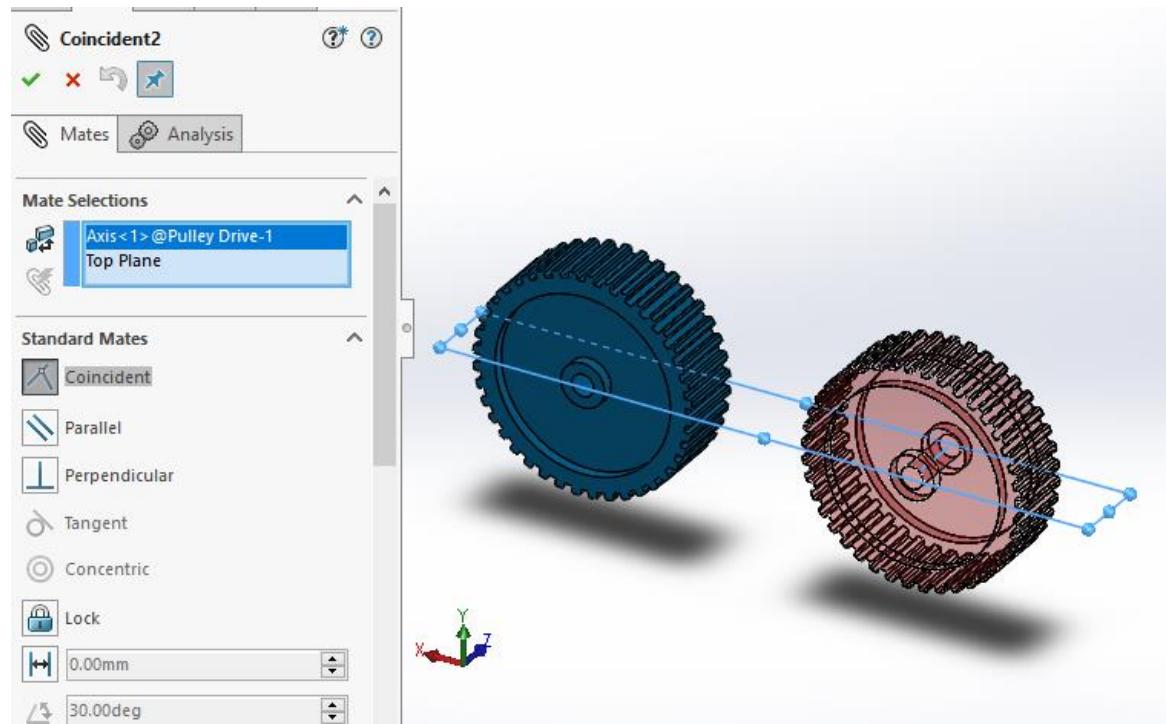
1. Model two gears as described in the drawings given
2. Create a SolidWorks Assembly and Insert the two Components as below



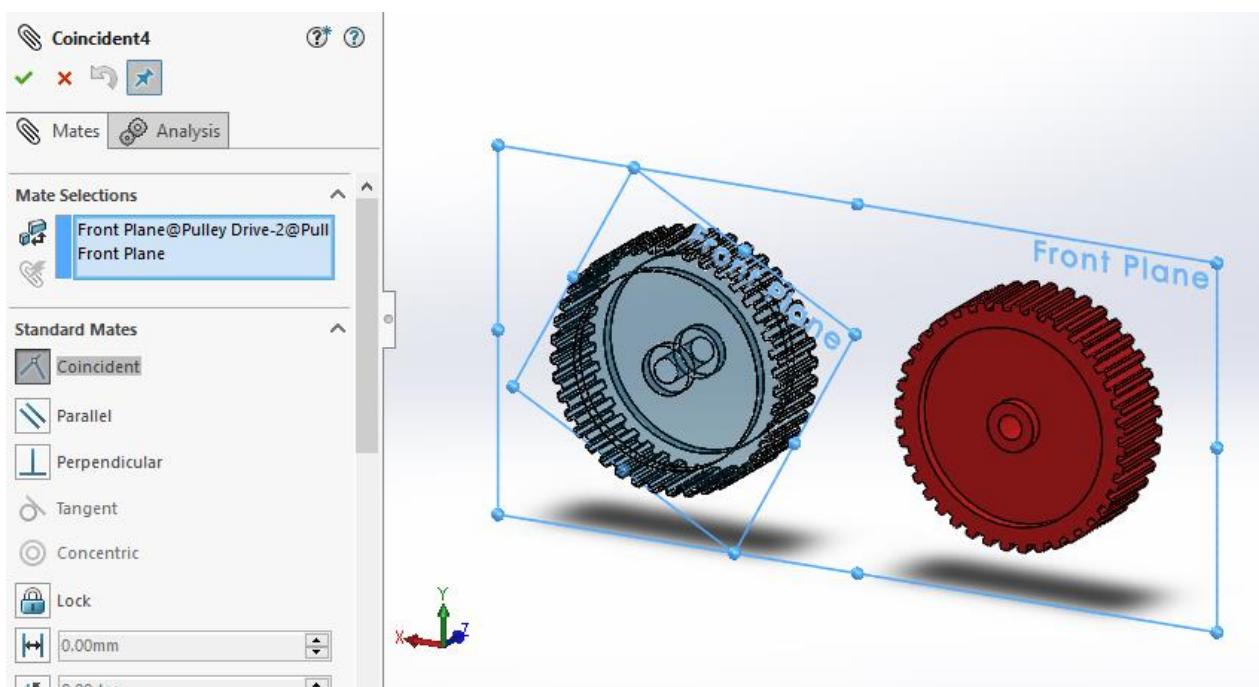
3. Coincide the **Front plane - Pulley 1** with **Front plane - Assembly** using **Coincident mate feature**



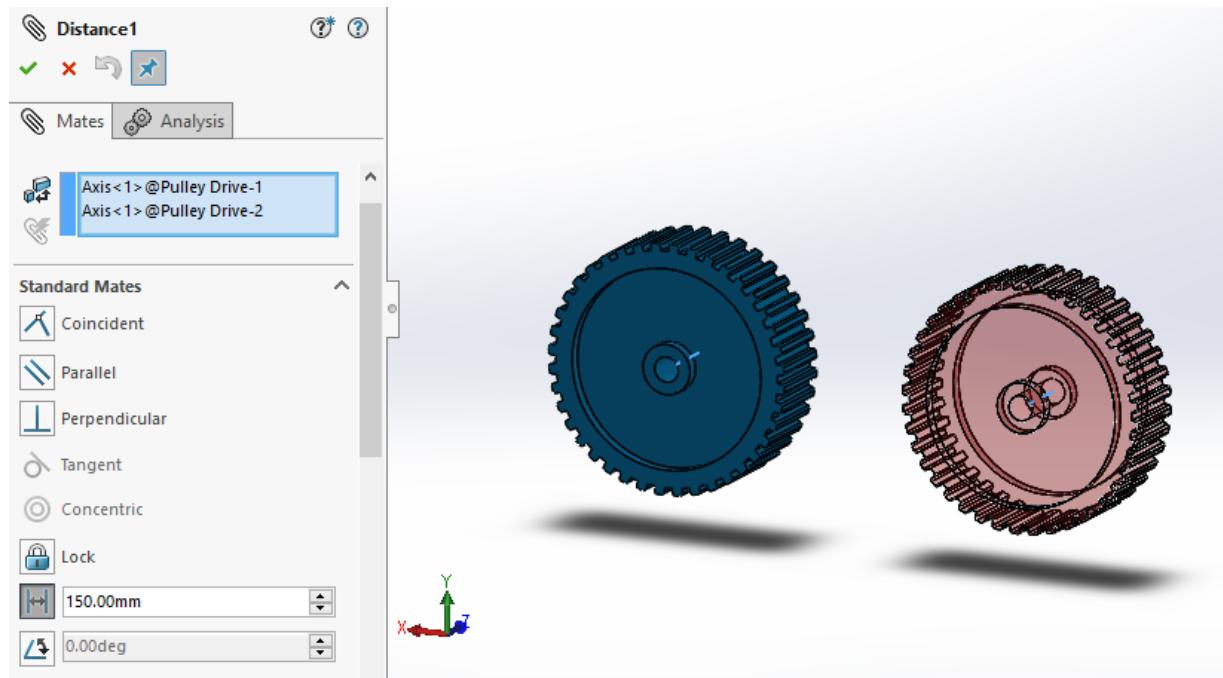
4. Coincide the Axis – Pulley 1 with the Top plane- Assembly using coincident mate feature



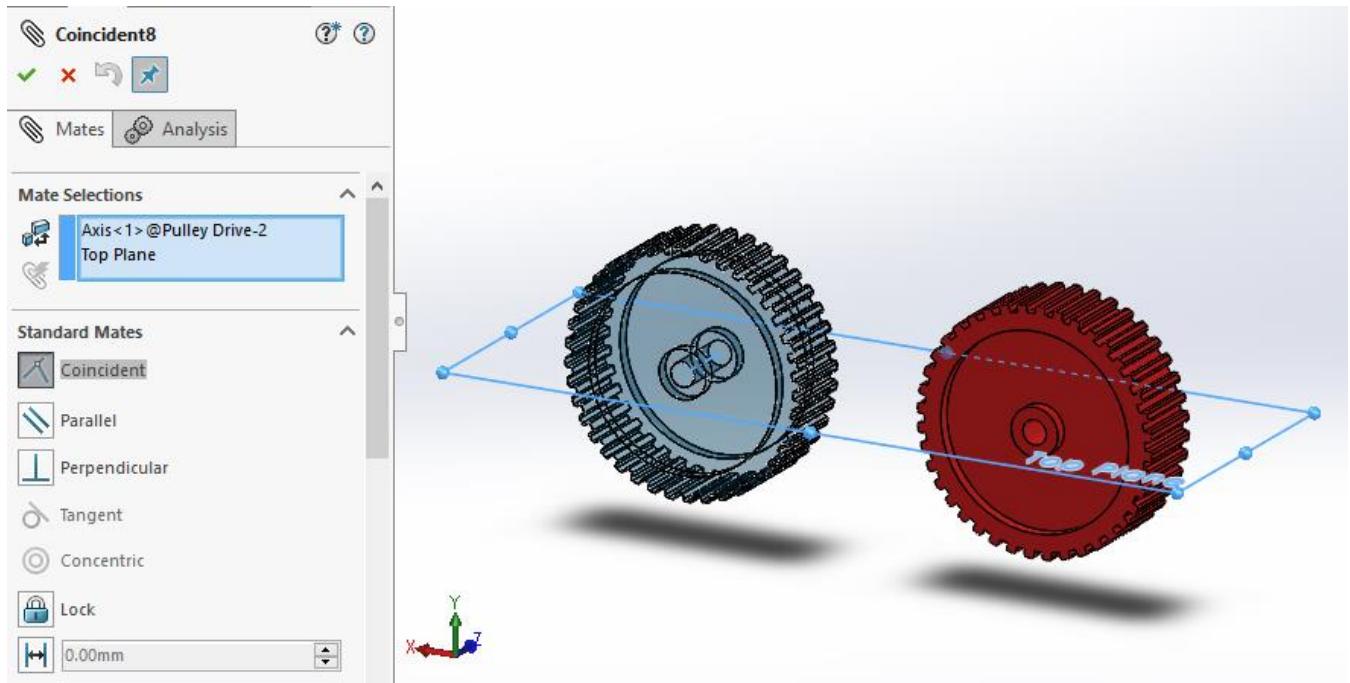
5. Coincide the Front Plane – Pulley 2 with Front plane – Assembly using coincident mate



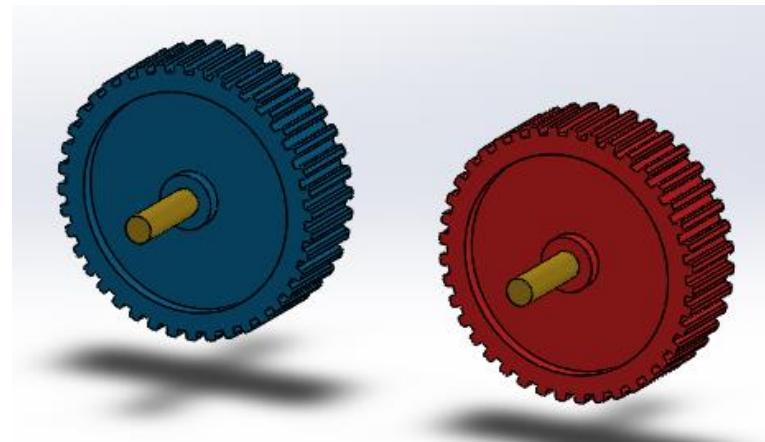
6. Apply distance mate between Axis - Pulley 1 and Axis - Pulley 2 (150 mm)



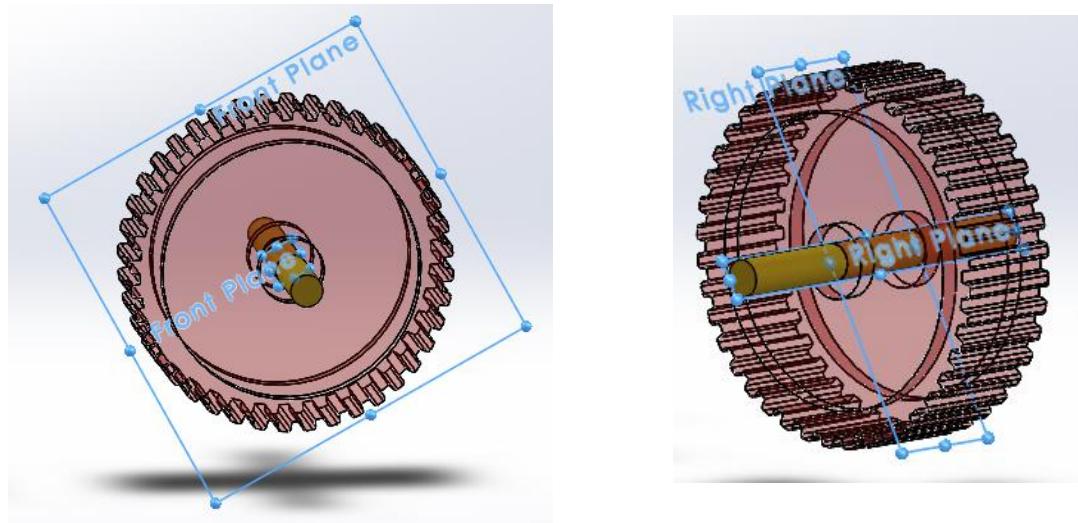
7. Coincide the Axis – Pulley 2 with Top plane – Assembly using coincident mate



8. Use concentric mates to insert the Pulley Shafts as shown



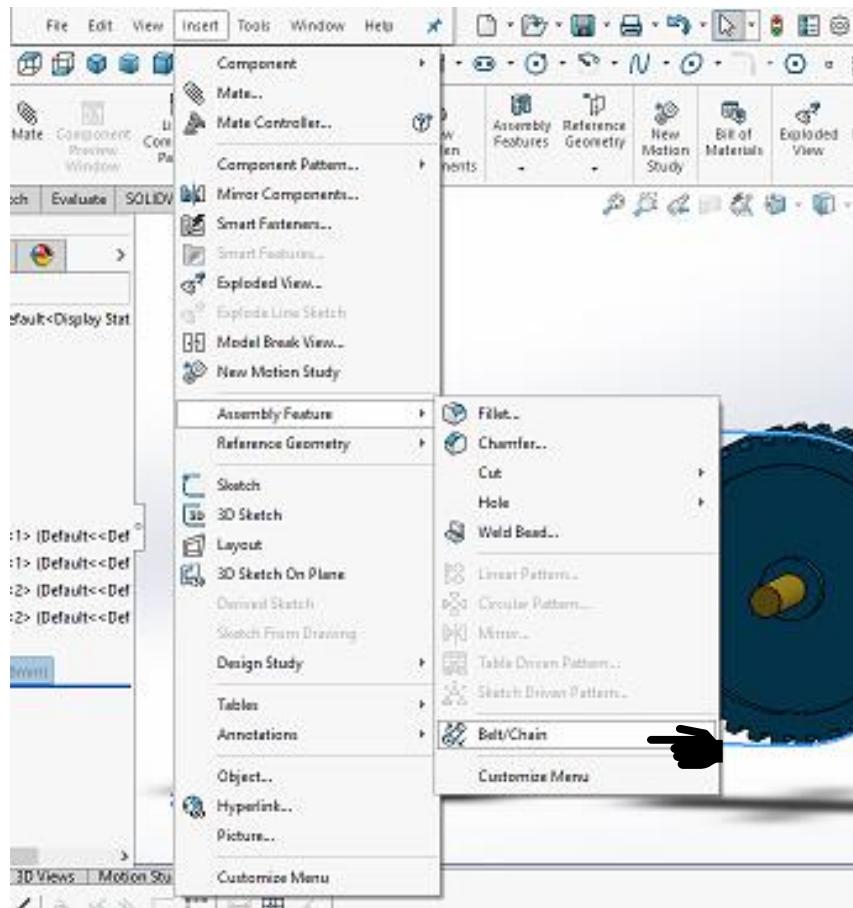
9. Coincide the **Front plane – Pulley 1** with **Front plane - Pulley Shaft 1** and **Right plane – Pulley 1** with **Right plane - Pulley Shaft 1** as below



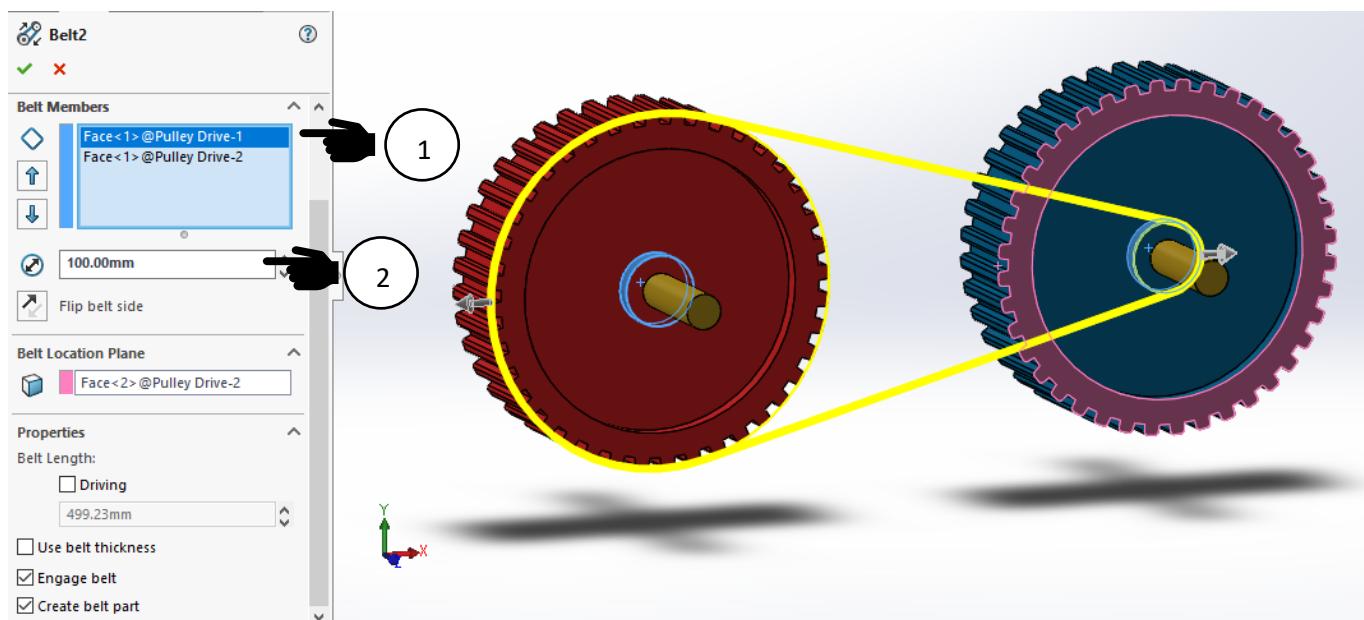
10. Repeat the same mate instance to **Pulley 2** with **Pulley Shaft 2**

Assembling the Belt

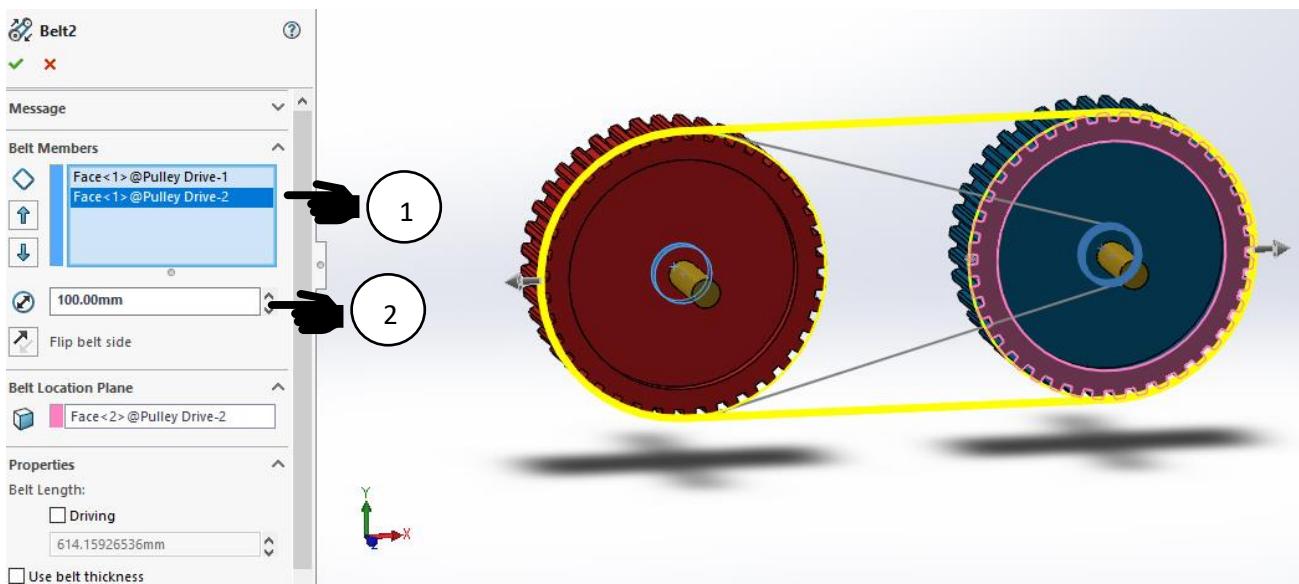
11. Click Belt/Chain (Assembly toolbar) or Insert > Assembly Feature > Belt/Chain



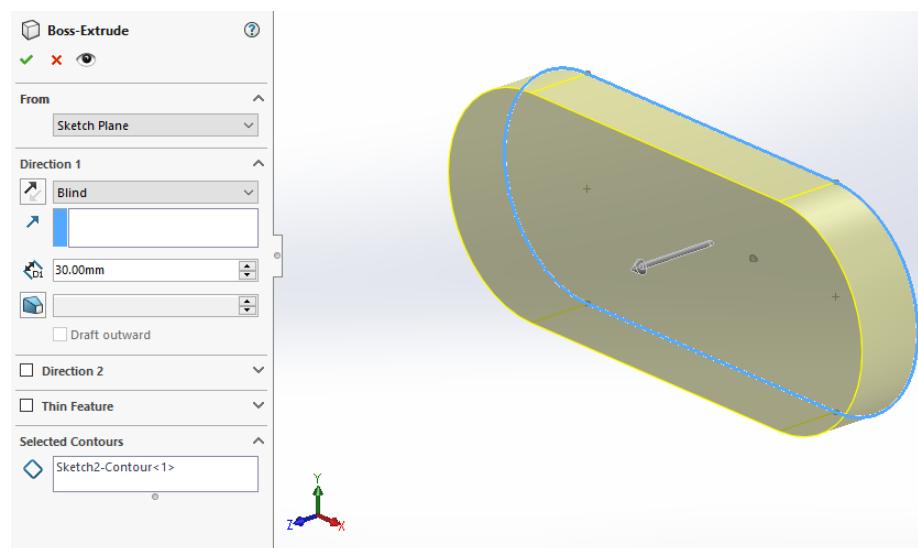
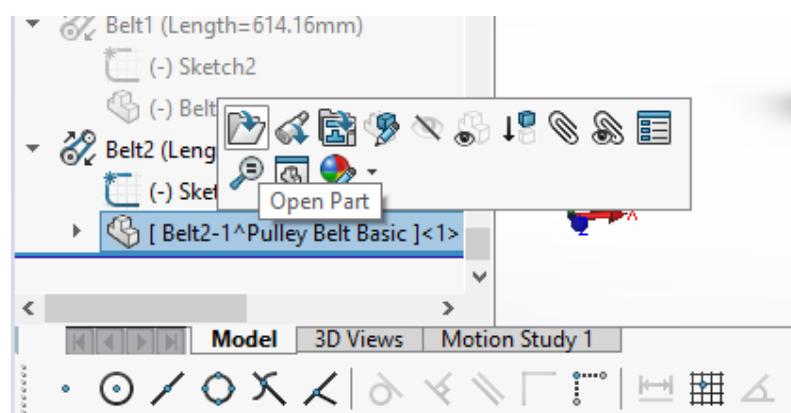
12. Set the PropertyManager options. Select Face of Pulley 1



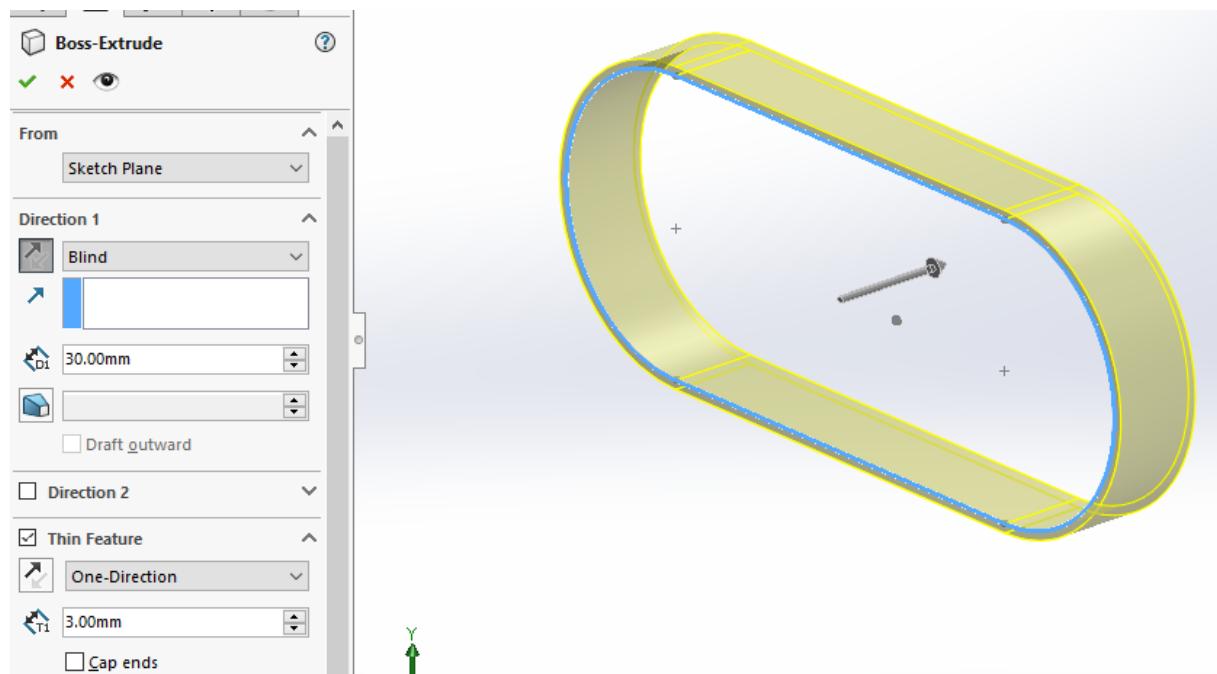
13. Select the Second face of Pulley 2 and set value. Check OK



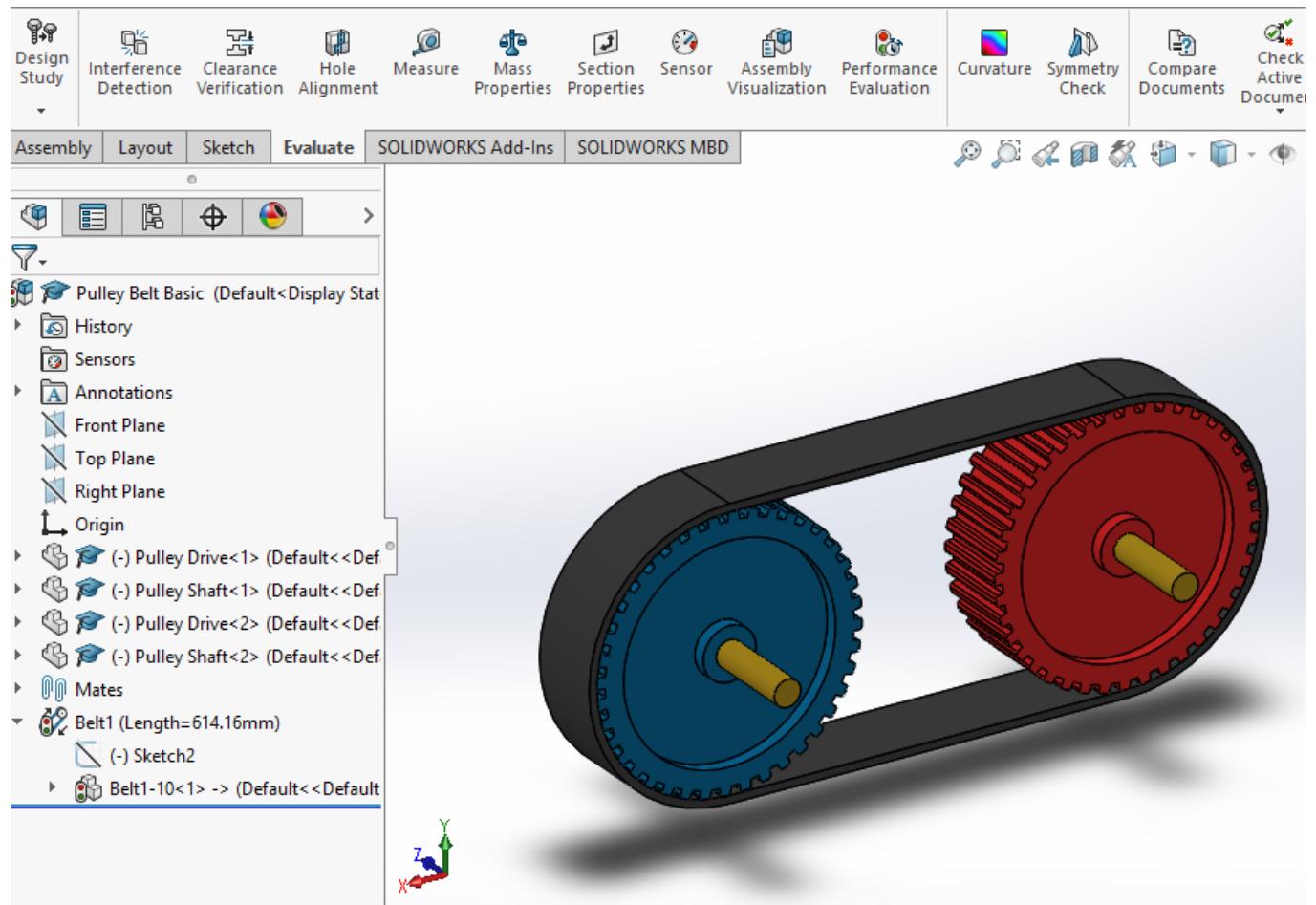
15. Extrude the sketch for 30mm

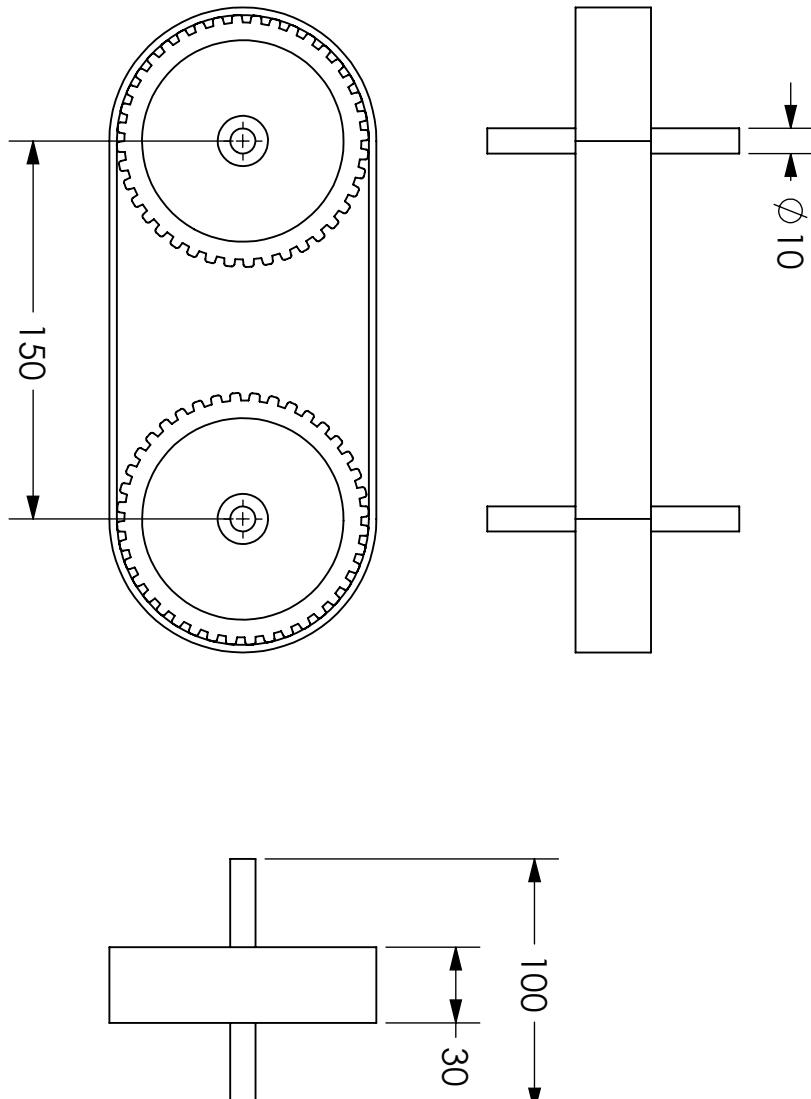
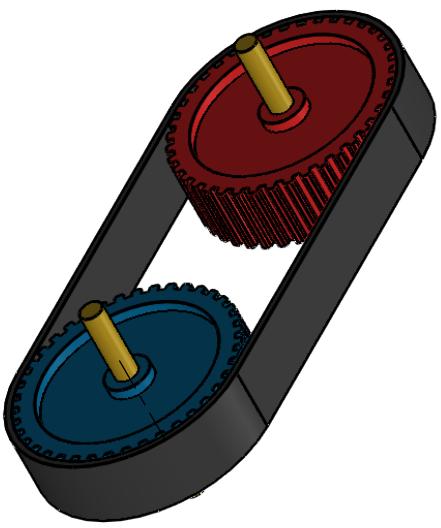


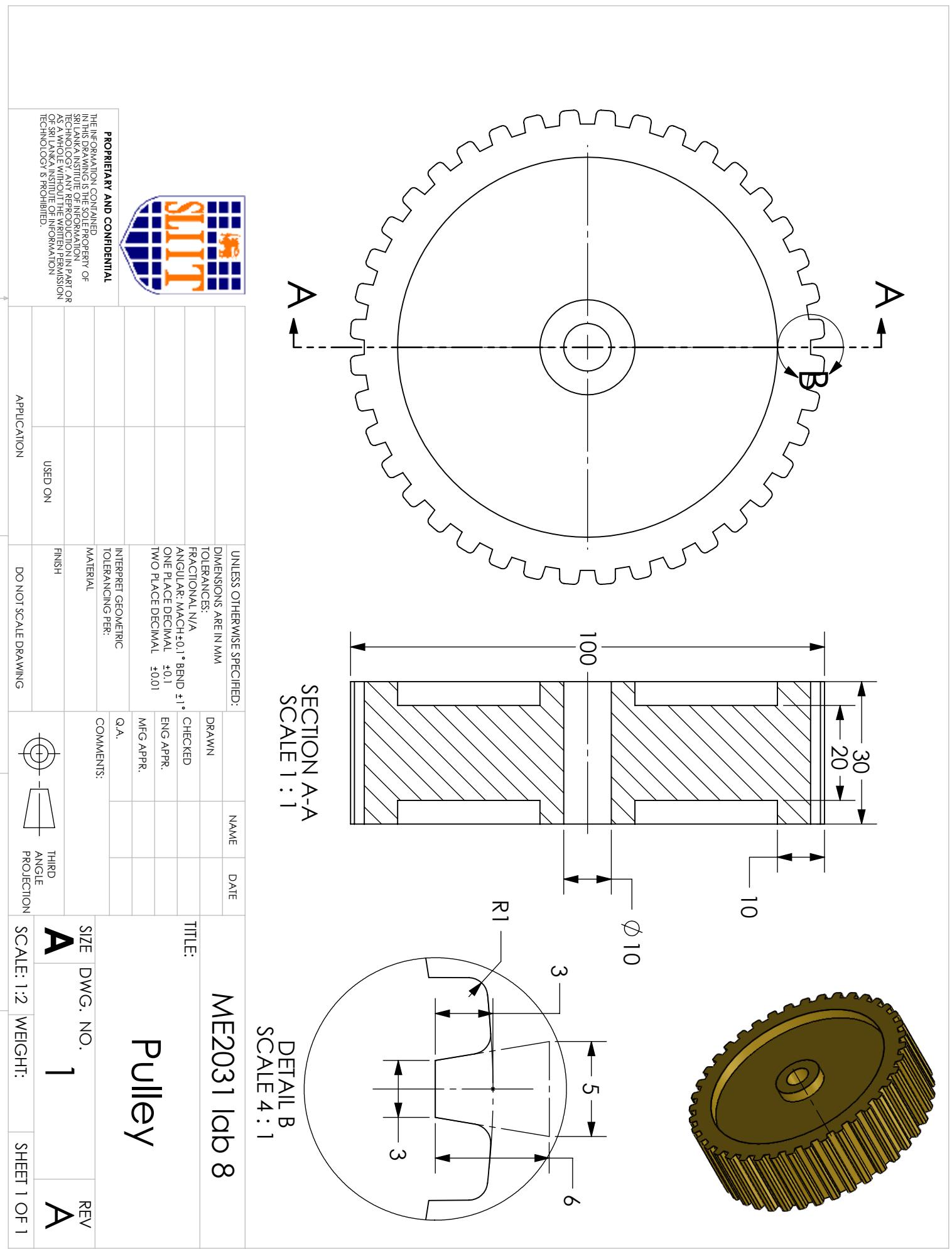
16. Select thin feature when extruding the belt. Check OK



17. Apply black colour to the belt. Rebuild the Assembly and save it



 <p>PROPRIETARY AND CONFIDENTIAL THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF SRILANKA INSTITUTE OF INFORMATION TECHNOLOGY AND REPRODUCTION IN PART OR AS A WHOLE WITHOUT THE WRITER'S PERMISSION OF SRILANKA INSTITUTE OF INFORMATION TECHNOLOGY IS PROHIBITED.</p>		<p>UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MM TOLERANCES: FRACTIONAL N/A ANGULAR: MACH$\pm 0.1^\circ$ BEND $\pm 1^\circ$ ONE PLACE DECIMAL TWO PLACE DECIMAL</p> <p>DRAWN CHECKED ± 0.1 ± 0.01</p> <p>ENG APPR. MFG APPR.</p> <p>Q.A.</p> <p>INTERPRET GEOMETRIC TOLERANCING PER: MATERIAL</p> <p>COMMENTS:</p>		<p>NAME: DATE: TITLE: ME2031 Lab 8</p> <p>USED ON: FINISH: DO NOT SCALE DRAWING</p> <p>APPLICATION: MATERIAL:</p> <p>THIRD ANGLE PROJECTION</p> <p>SIZE: A DWG. NO. 2 REV: A</p> <p>SCALE: 1:5 WEIGHT: SHEET 1 OF 1</p>	
 <p>The technical drawing shows a pulley belt system. It consists of two pulleys and a belt. The top pulley has a diameter of 150 mm and a bore diameter of 10 mm. The bottom pulley has a diameter of 100 mm and a bore diameter of 30 mm. The distance between the centers of the pulleys is 150 mm. The belt is shown in a third-angle projection.</p>					
 <p>The 3D view shows a red pulley on the left and a blue pulley on the right, both mounted on black cylindrical shafts. A black belt connects the two pulleys. The pulleys have a textured surface.</p>					





Sri Lanka Institute of Information and Technology

Department of Mechanical Engineering

Engineering Drawing – ME2031

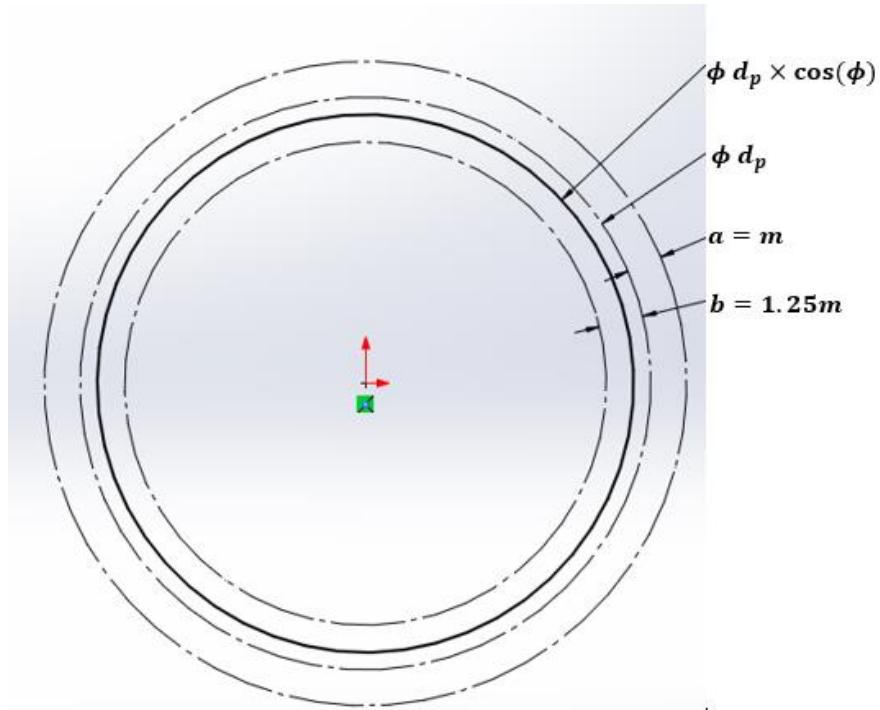
SolidWorks Laboratory - 9

Mr. Thilina Weerakkody
Mr. Kulunu Samarakkrama

Mechanical Components- Involute Spur gears

Date : 07/05/2019

1. Open SolidWorks and sketch the following figure



d_p = Pitch circle diameter

t = Number of teeth

ϕ = Pressure angle of the teeth ($14\frac{1}{2}^\circ$, 20°)

When d_p , t and ϕ are known

$$\text{Module } m = \frac{d_p}{t}$$

Addendum = $a = 1m$

Dedendum = $b = 1.25m$

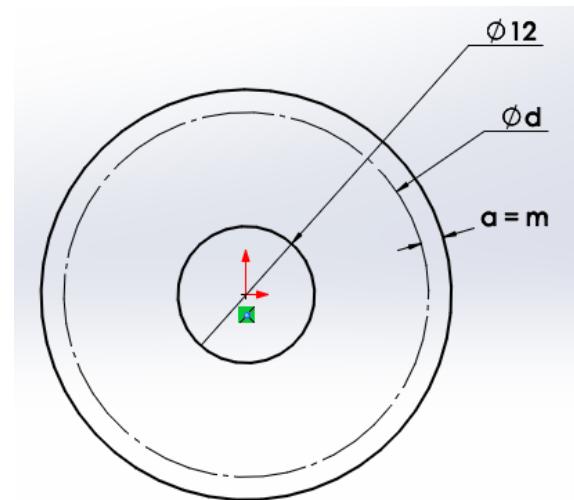
Base circle diameter = $d_p \times \cos(\phi)$

$$x(t) = \frac{d_b}{2} (\cos t + t \sin t)$$

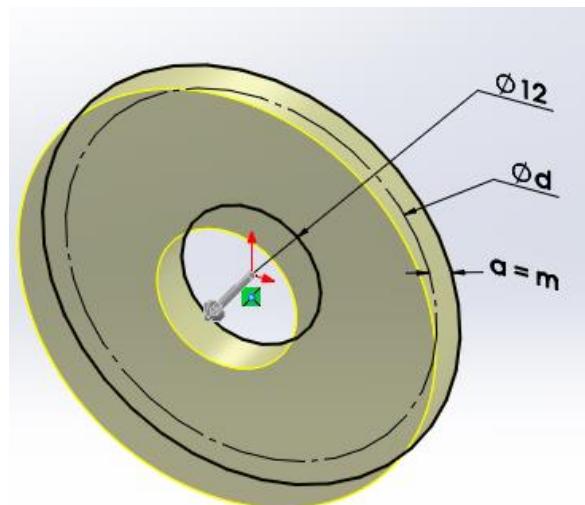
$$y(t) = \frac{d_b}{2} (\sin t - t \cos t)$$

Let $d_p = 32mm$, $m = 2mm$ and $\phi = 20^\circ$

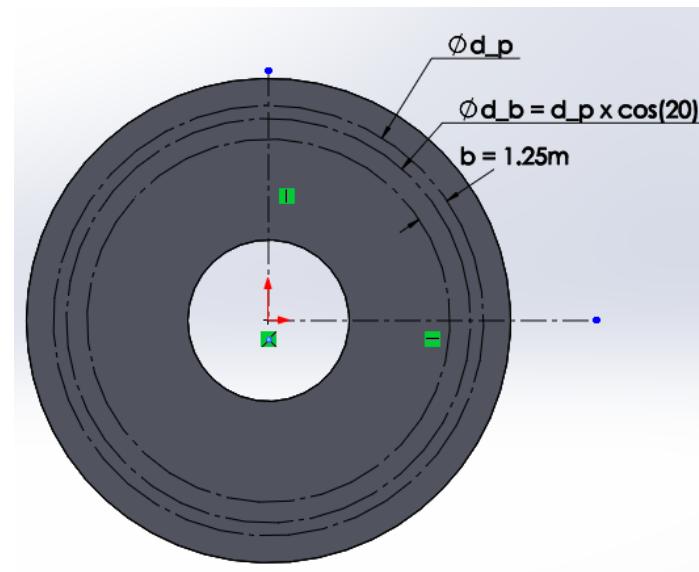
Step 2



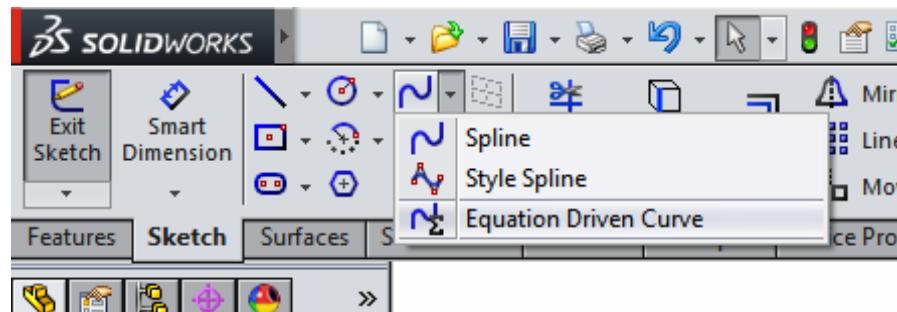
Step 3 – Extract Boss/base 2m



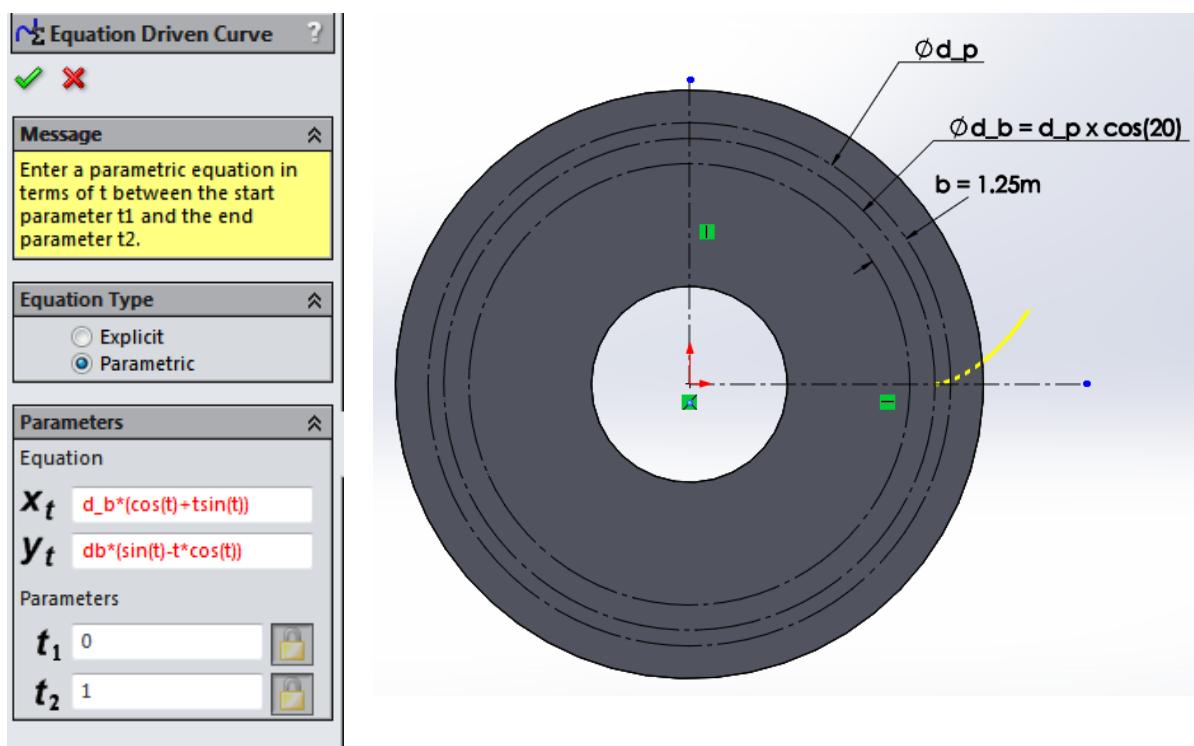
Step 4 – Sketch following construction lines on extruded surface



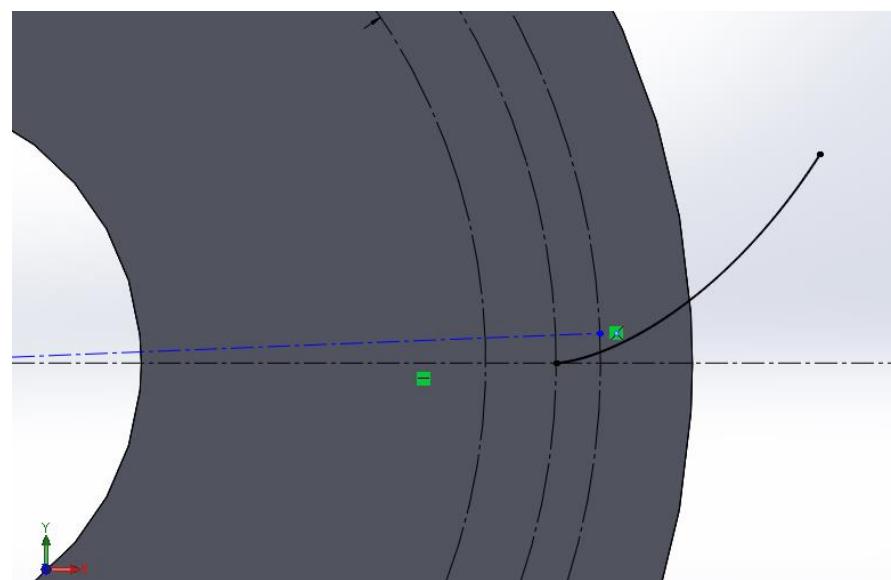
Step 5- Sketch an Equation driven Curve



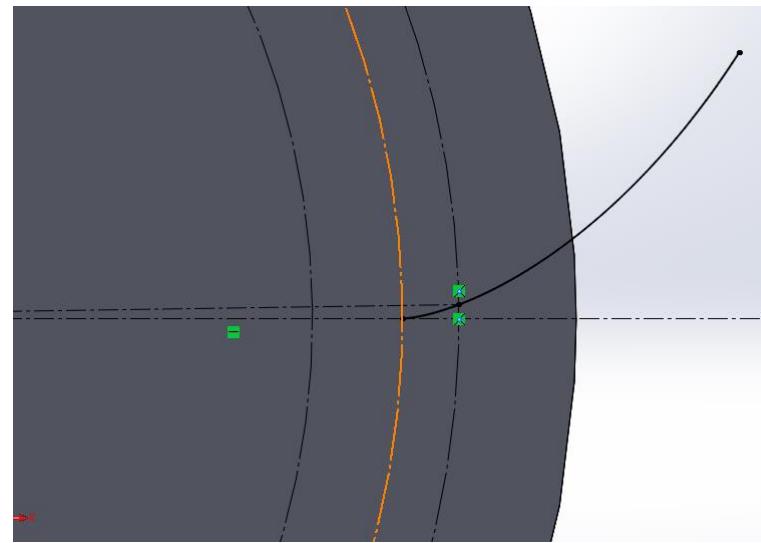
Step 6 – Enter following Equations and parameters



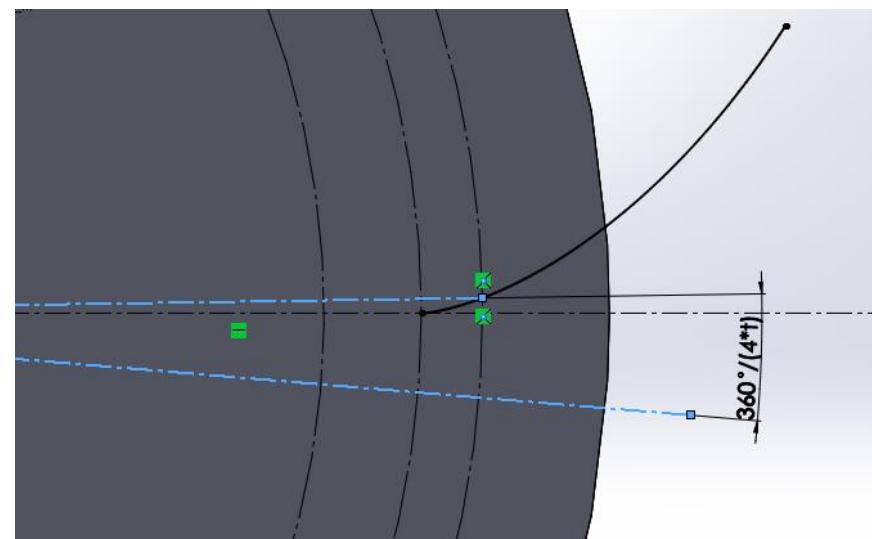
Step 7 - Draw a construct a construction line starting from origin to the circle d_p . Coincide the end point of the line and circle d_p .



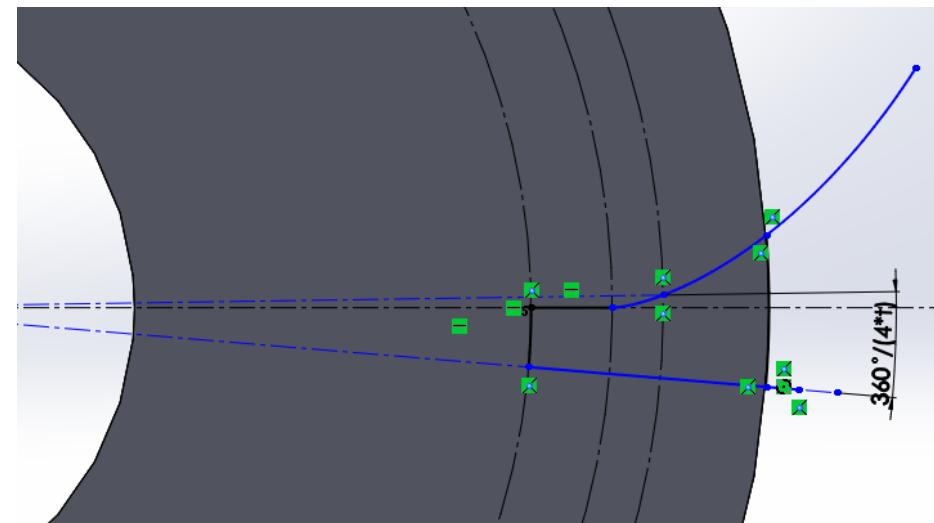
Step 8 – Draw the next construction line with an angle of $\frac{360}{4 \times t}$. $t = \text{No. of teeth}$



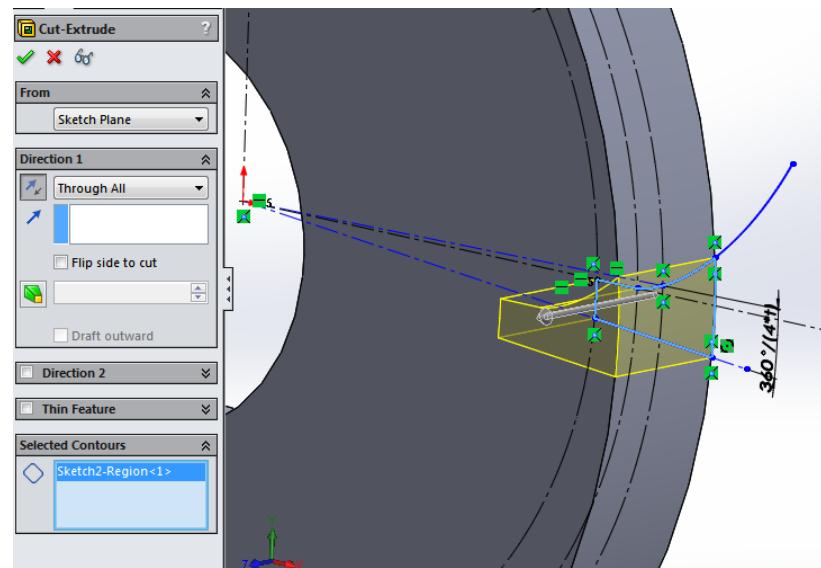
Step 9 - Draw the next construction line with an angle of $\frac{360}{4 \times t}$. $t = \text{No. of teeth}$



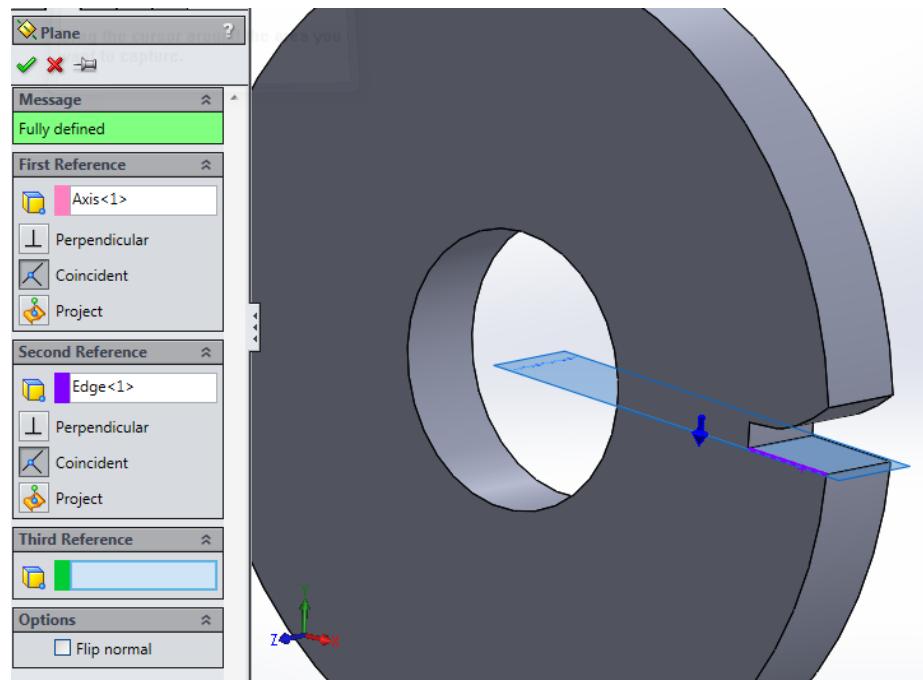
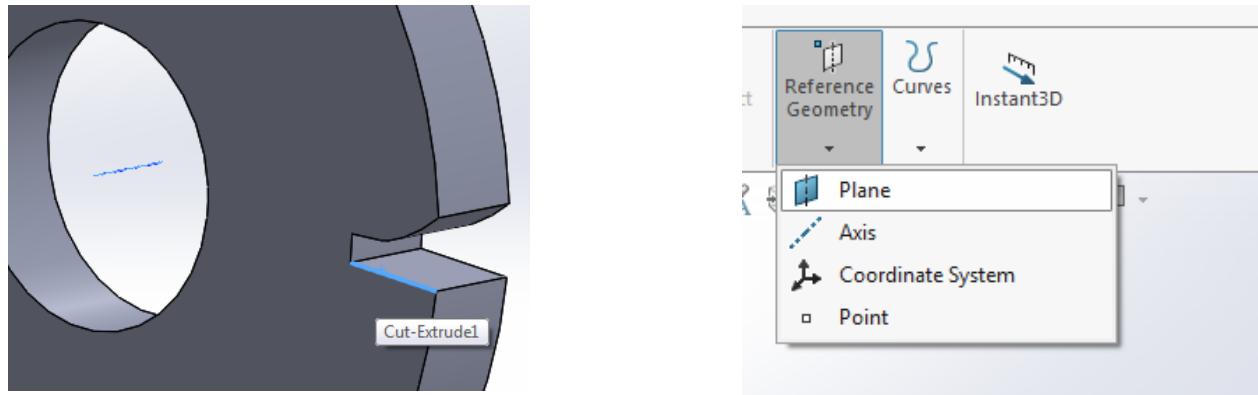
Step 10 - Complete the sketch using construction lines, lines and center points as shown is below.



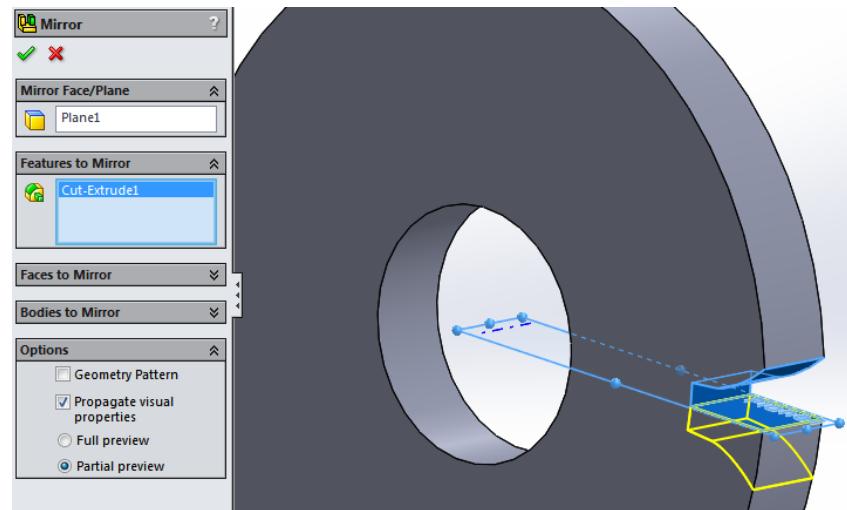
Step 11 – Extrude cut the section as shown



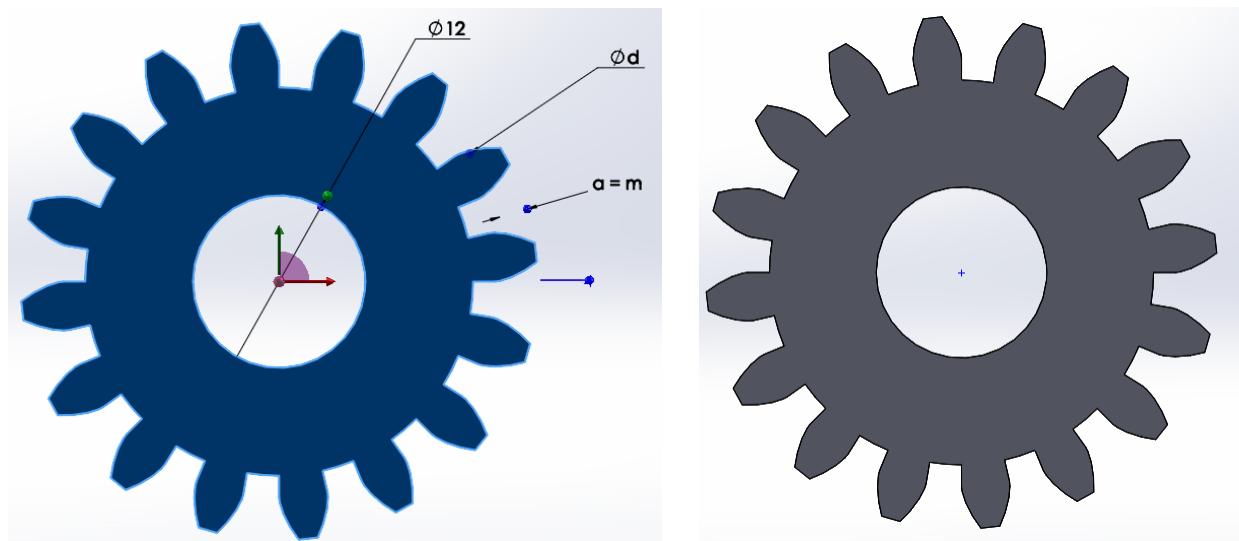
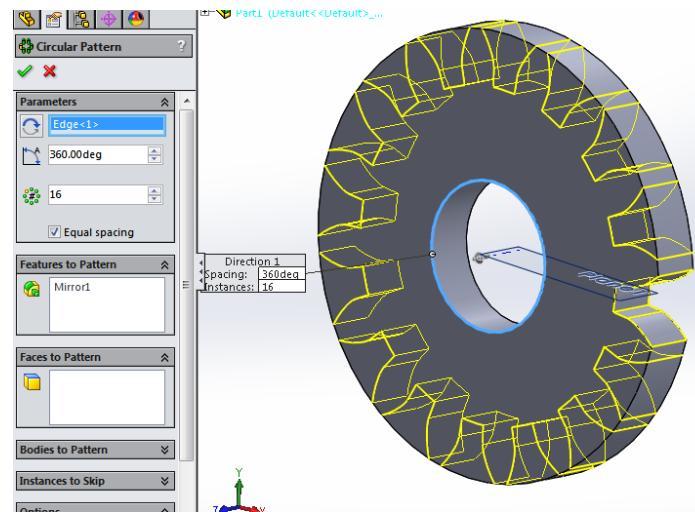
Step 12 – Create a plane using the axis and edge shown below



Step 13 - Mirror the extrude cut part using the created plane



Step 14 - Circular pattern the mirrored teeth.



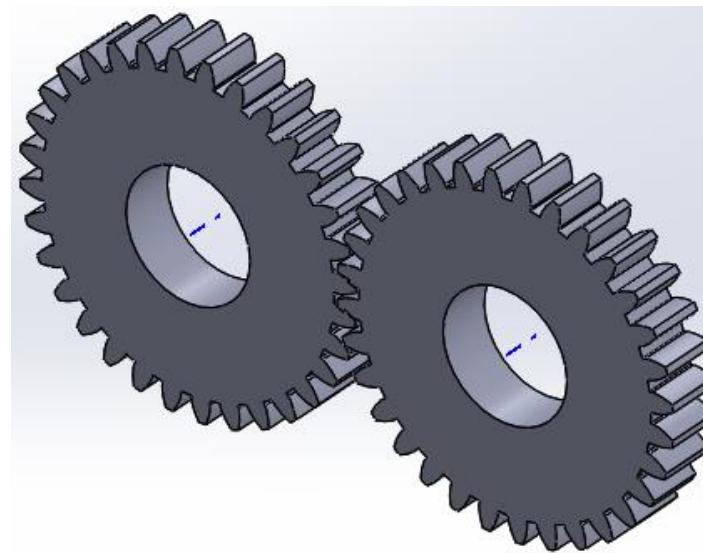
Gear Mate Procedure

Step 1

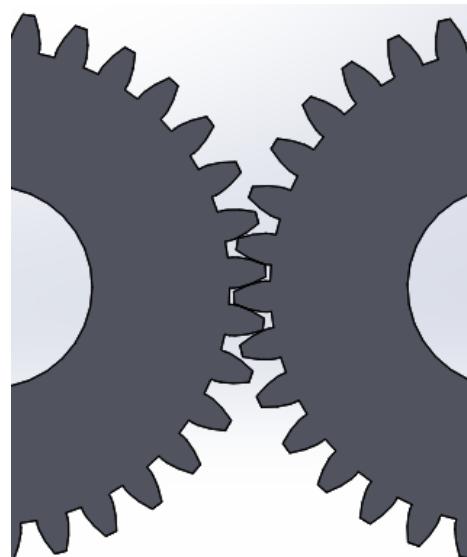
- Import the gear1 to the assembly.
- Mate the Front Plane of the gear1 with Assembly Front Plane.
- Mate the axis of the gear1 with assembly Top plane.
- Mate the axis of the gear1 with assembly Right plane.

Step 2

- Import the gear2 to the assembly or drag another gear using gear 1 by while pressing Ctrl key.
- Mate the Front Plane of the gear2 with Assembly Front Plane.
- Mate the axis of the gear2 with assembly Top plane
- Distance mate the axis of the gear1 and gear2 with a distance of d_p .



Step 3 - Align two sets of gear teeth as shown below.



Step 4 - Select Mates > Mechanical Mates > Gear. Select two circles and set the ratio accordingly. In this example both ratios must be equal.

