

INVENTOR FUNDAMENTAL PARAMETRIC MODELLING

4 Days Training

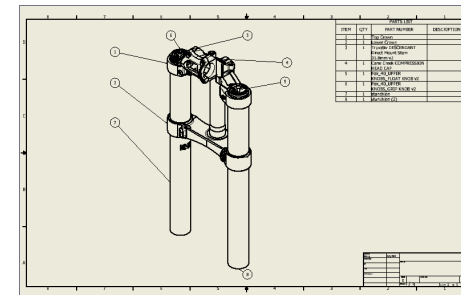
Aiman

aiman@bimageconsulting.com

FUNDAMENTAL

Environment / Extension	Description
Part (.ipt)	All parts modelling, sketching, and complex design takes place on individual parts.
Assembly (.iam)	Parts joined together to create a complete or sub complete product with a working mechanism.
Drawing (.idw / .dwg)	2D drawing that contains multiple views & annotations.

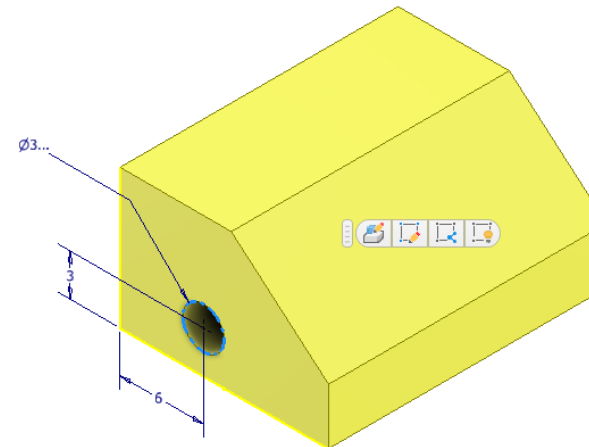
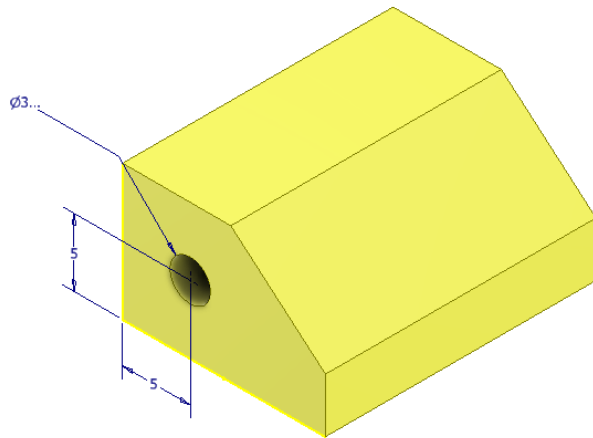
Inventor is fully associative. Any changes made into the model on any environment, will be reflected to the rest of environment.



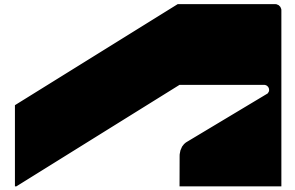
FUNDAMENTAL

Parametric Design

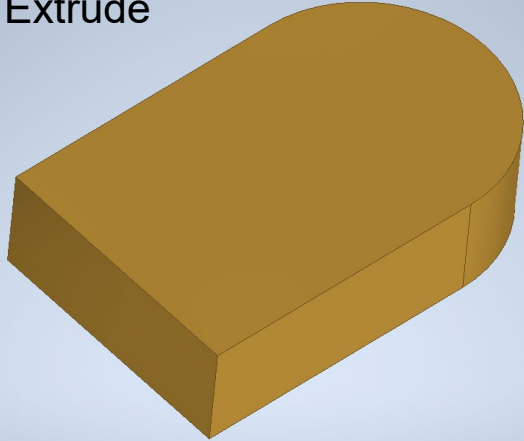
- ☼ All features that are created are parametric.
- ☼ Dimensions & Constraints considered parameters.
- ☼ Any changes of parameters will be reflected to the features thus, changing the shape and size of the model.



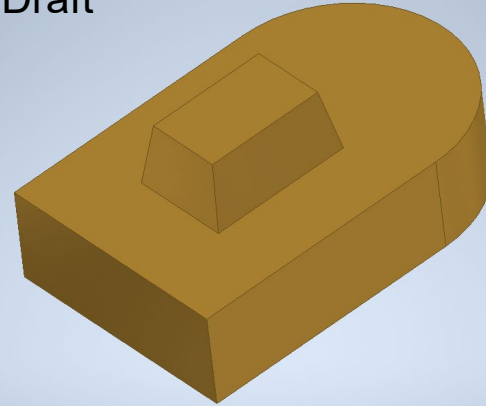
GETTING STARTED



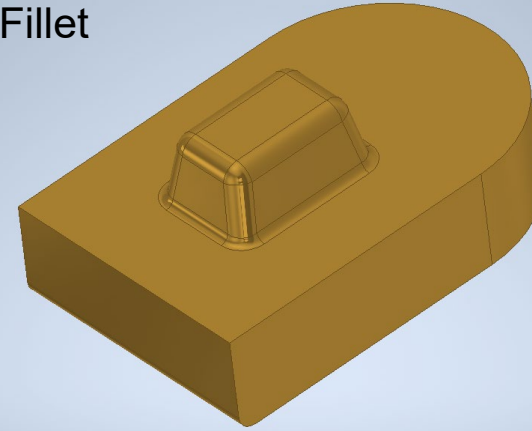
Extrude



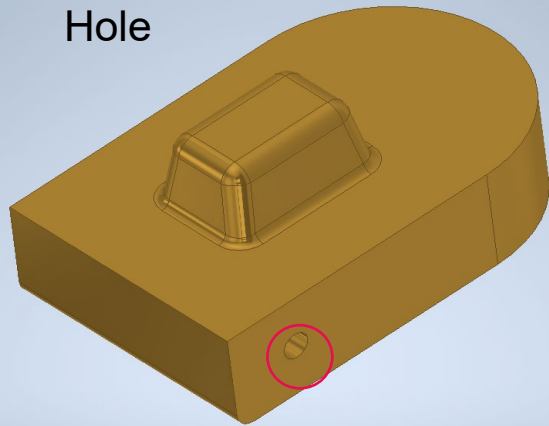
Draft



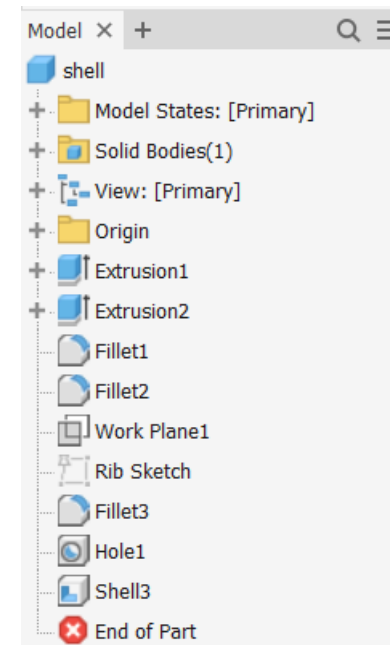
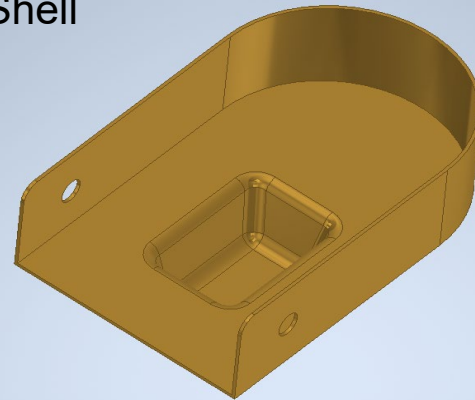
Fillet



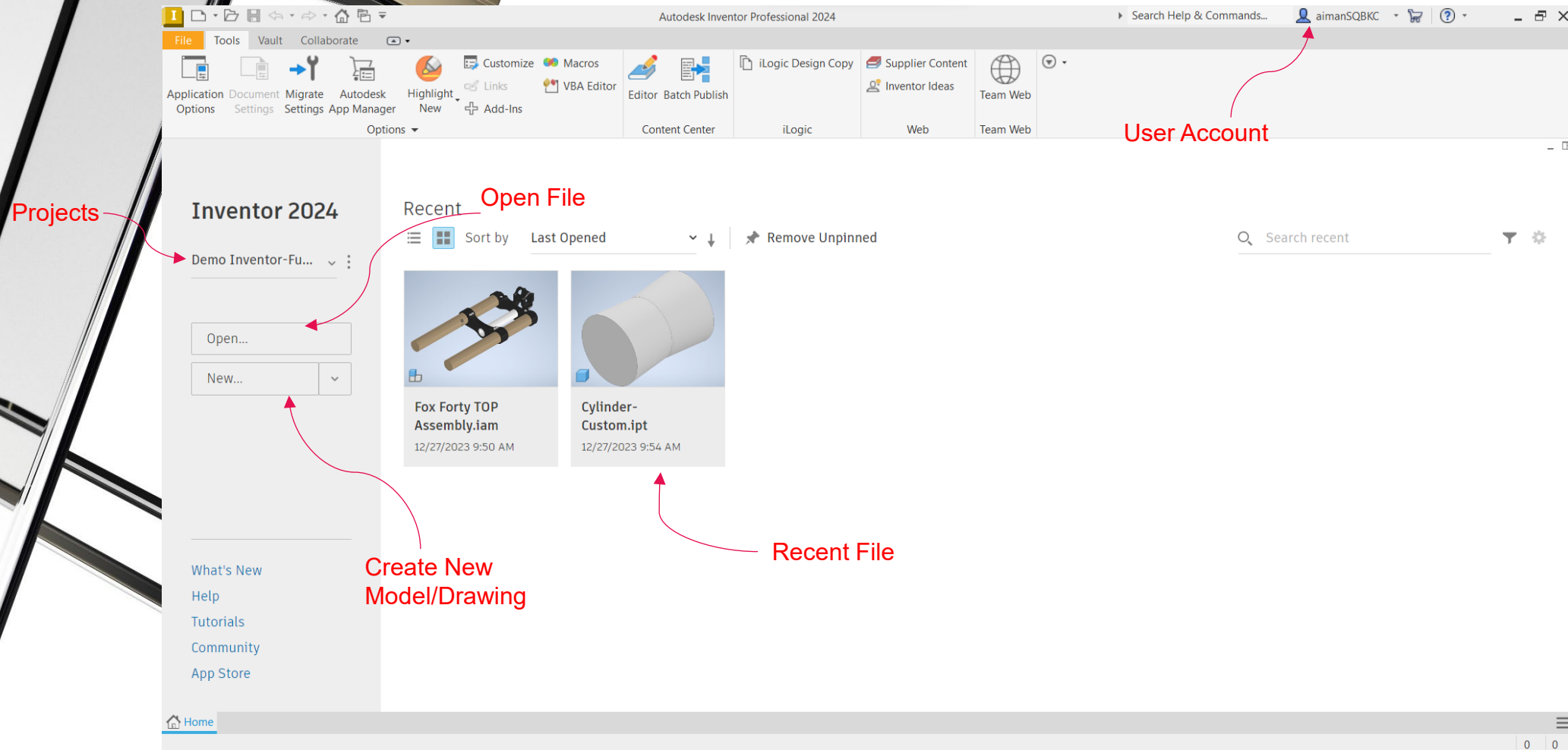
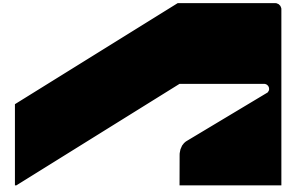
Hole



Shell



GETTING STARTED (Interface)



GETTING STARTED (Interface)



Model
Browser

Quick Access
Toolbar

Feature Tab

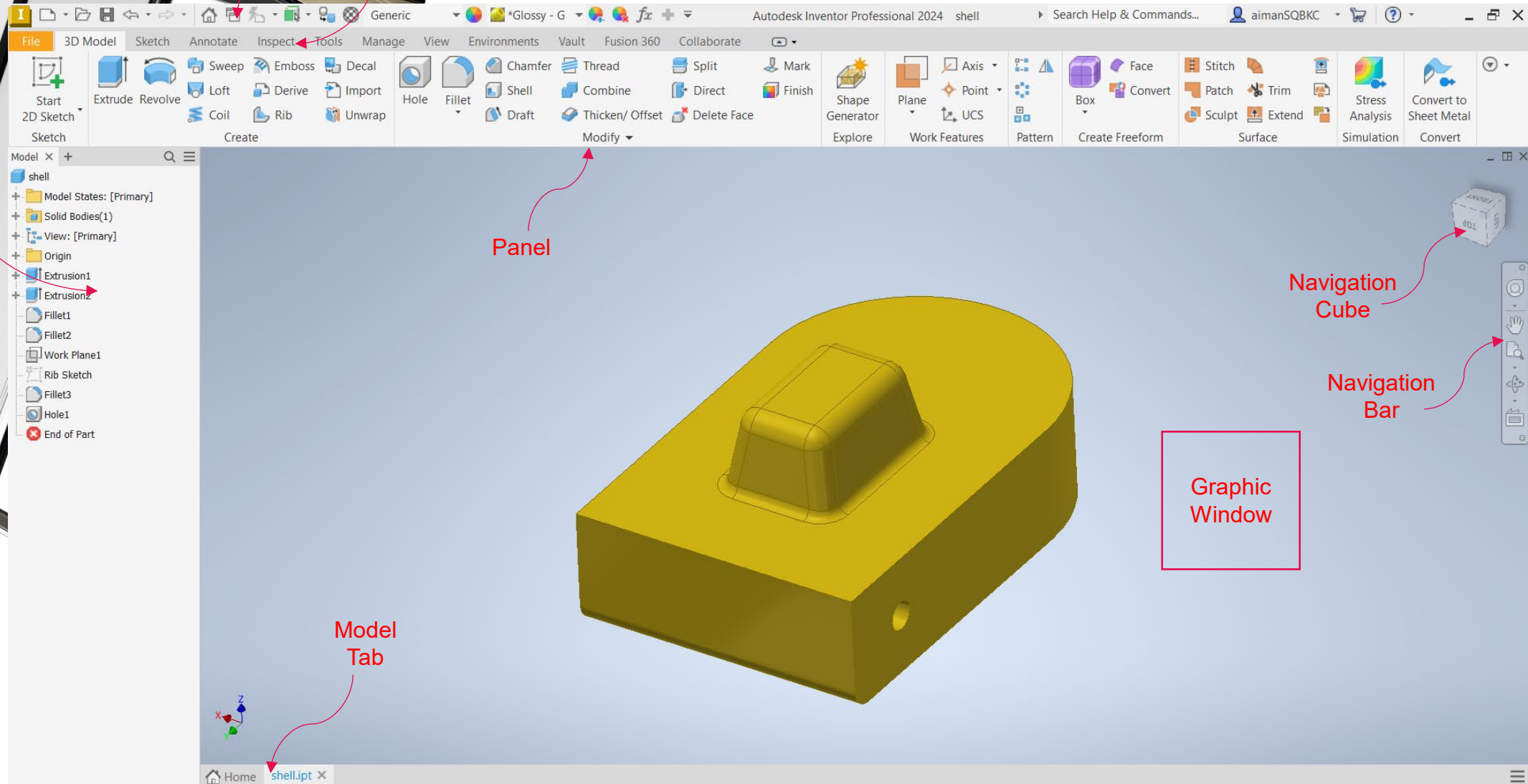
Panel

Navigation
Cube

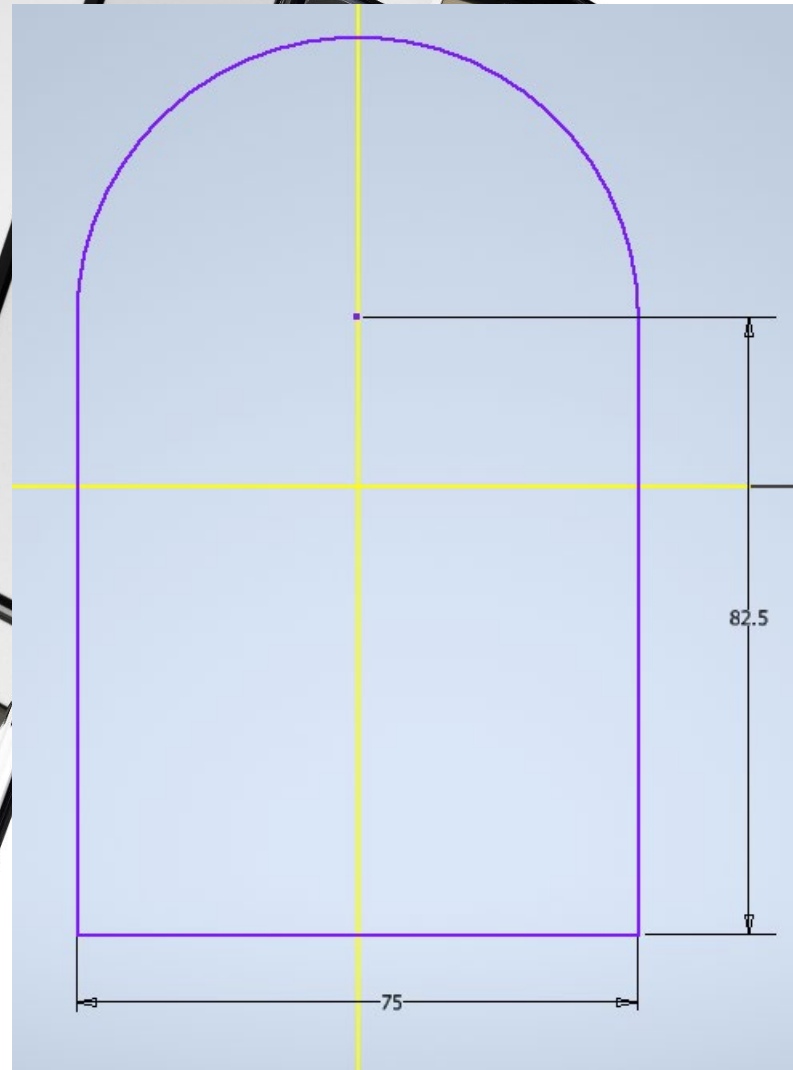
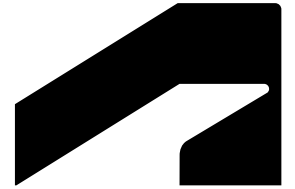
Navigation
Bar

Graphic
Window

Model
Tab



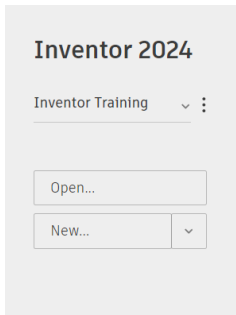
SKETCH



1. Plane
2. Geometrical Tools
 1. Line
 2. Circle
 3. Rectangle
 4. Arc
3. Constrains
4. Dimensions

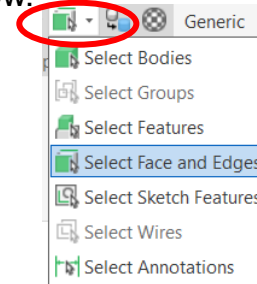
Navigation & Project (Hands On)

1. Click on 3 dots besides 'Open Project' button. Select 'setting'.

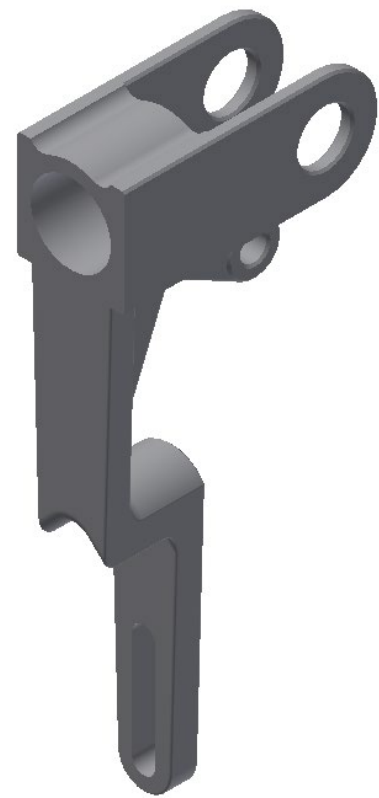


2. Click browse. Find the location of the project file.
(Intro To Modeling.ipj)
3. Click 'Open' on the Project Browser.
4. Open a file 'Joint.ipt'.
 - Rotate by using 'middle mouse' + 'shift'
 - Zoom in & zoom out using 'middle mouse scroll'

5. Click 'Select faces & Edges' feature at the top of the window.



6. Choose each option given and see what can we select on the model.

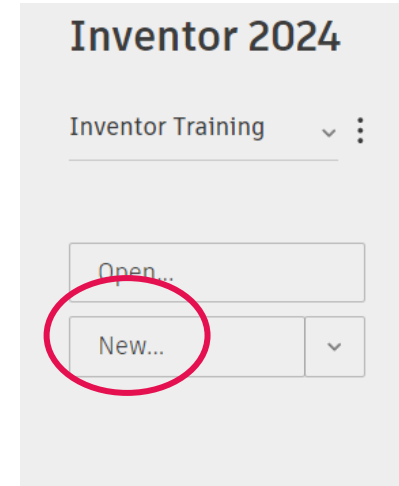


Create New Model (Hands On #1 - Sketch)

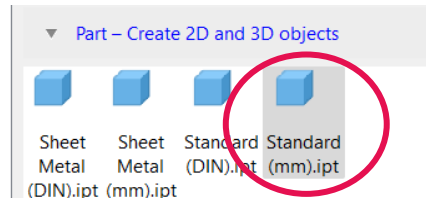
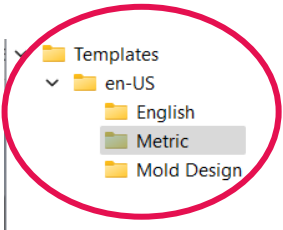
1. Click on 'New' button located on Quick Access Toolbar / Project Browser



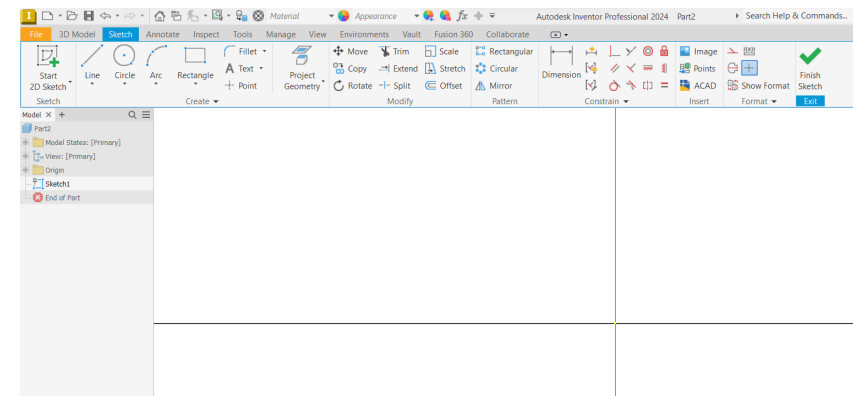
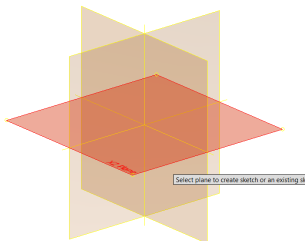
or



2. Select Standard.ipt (mm) template.

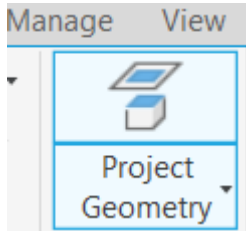


3. Click 'Start 2D Sketch'.
4. In the Graphic Window, select XY Plane. You'll enter 'Sketch' environment.

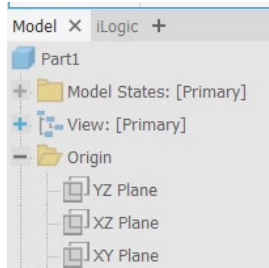


Create New Model (Hands On #1 - Sketch)

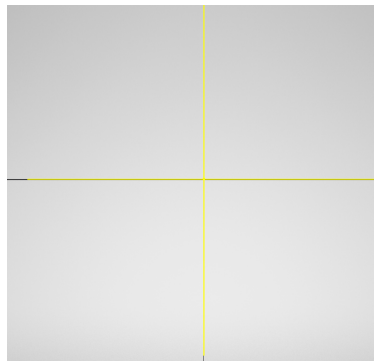
5. Hover cursor towards '**Modify**' ribbon and click it. We want to select different planes as references for our reference lines.



6. Hover towards model browser at the side. Expand 'origin' folder. Select XZ & YZ planes.



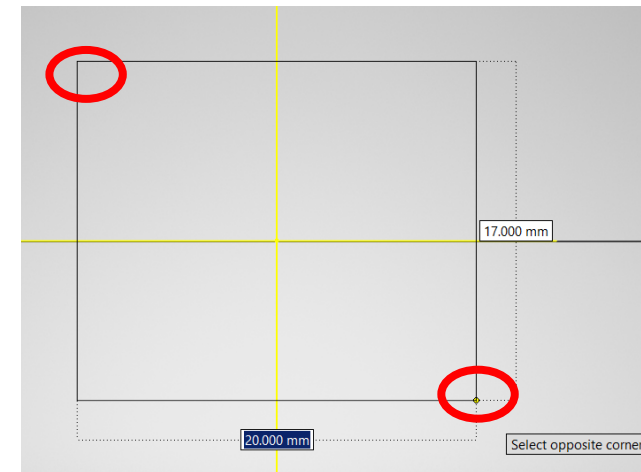
7. You will notice that there will be two lines for reference will appear according to the location of those planes that we have selected.



8. Hover towards 'Create' Panel in the sketch environment. Click on 'Rectangle' drop down ribbon, and select '**Two Point Rectangle**'.



9. Create a rectangle around origin point. Start the rectangle creation by **first corner point** and **second corner point**.

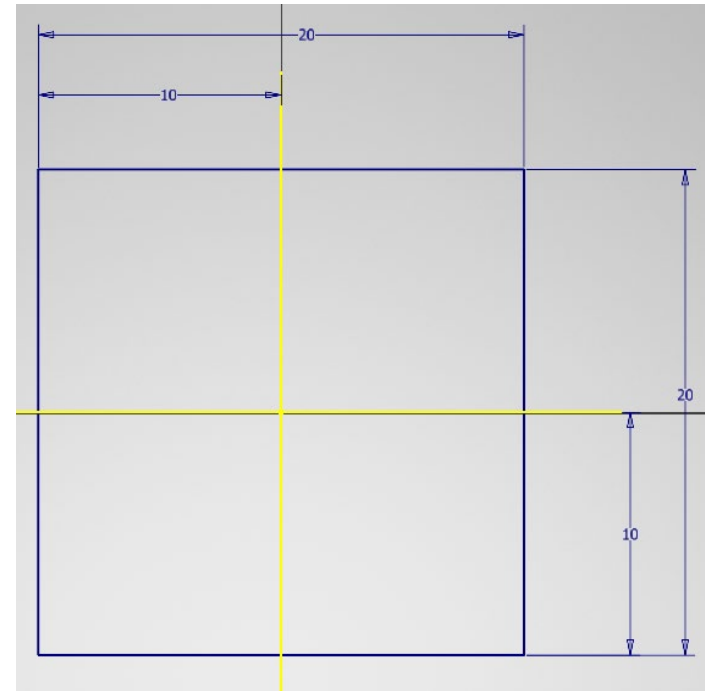
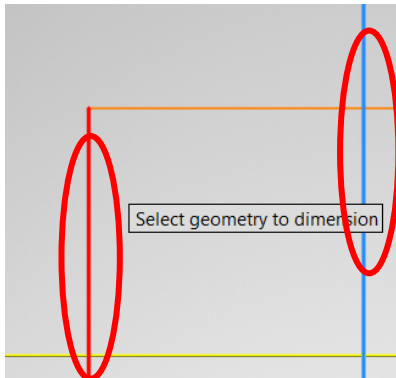


Create New Model (Hands On #1 - Sketch)

10. We want to apply dimensions. Find the ribbon called '**Dimension**' inside of 'Constrain' panel. Click the ribbon.



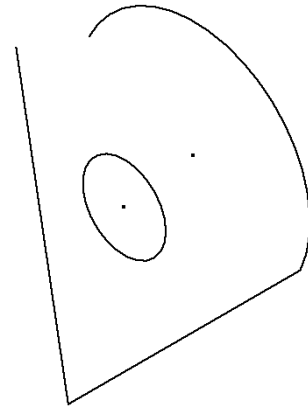
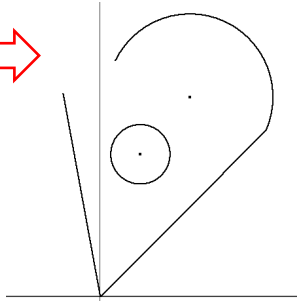
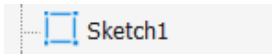
10. To apply the dimension, select the vertical reference line. Secondly, select the vertical line of the rectangle. Continue the dimensioning according to the picture given below.



Sketching Tools

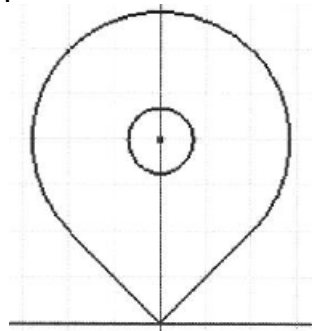
(Hands On #2 – Sketch Constraints)

1. Open 'constraints_1.ipt'. A model with an incomplete sketch will appear.
2. Double click on Sketch1 in feature browser to enter into the sketch environment.



3. Double click on Sketch1 in feature browser to enter into the sketch environment.

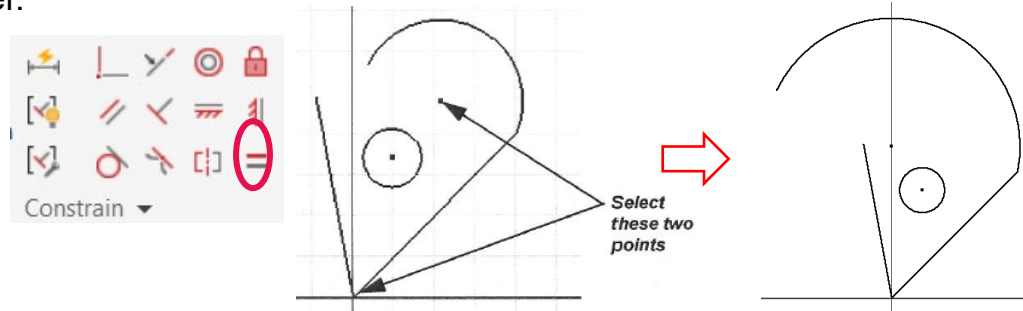
You will have to apply constraints in order to make the sketch looks proper and having design intent ad shown in picture below.



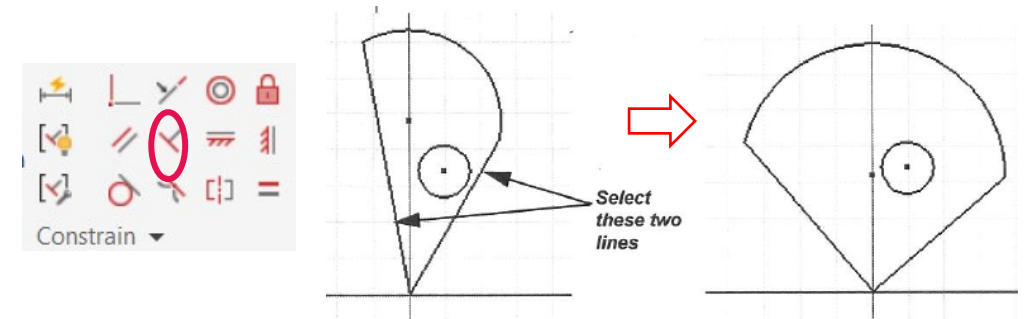
Sketching Tools

(Hands On #2 – Sketch Constraints)

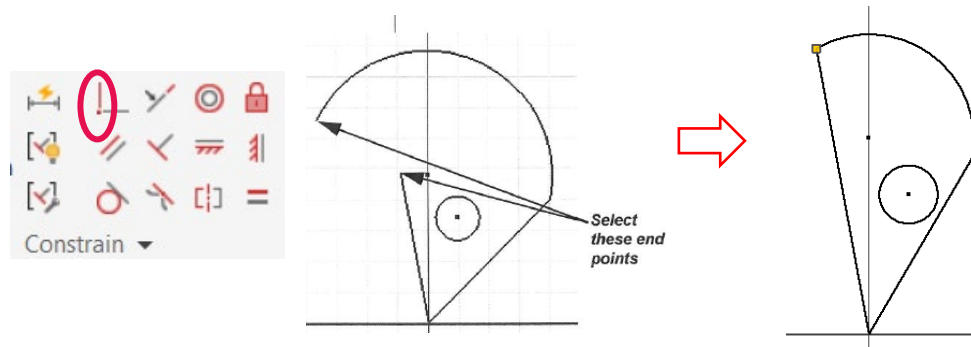
4. In the constraint panel, click 'Vertical Constraint' and select the two (2) projected points. The two points will be moved until they are arranged vertically to each other.



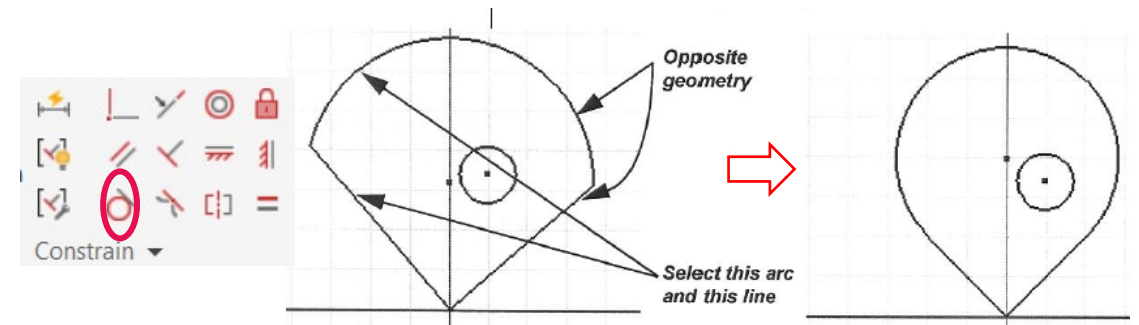
6. In the constraint panel, click 'Perpendicular Constraint' and select the two (2) lines. The two line will be moved until they are perpendicular to each other.



5. In the constraint panel, click 'Coincident Constraint' and select the end point of the arc and line. The two points will be moved until they are stick to each other.

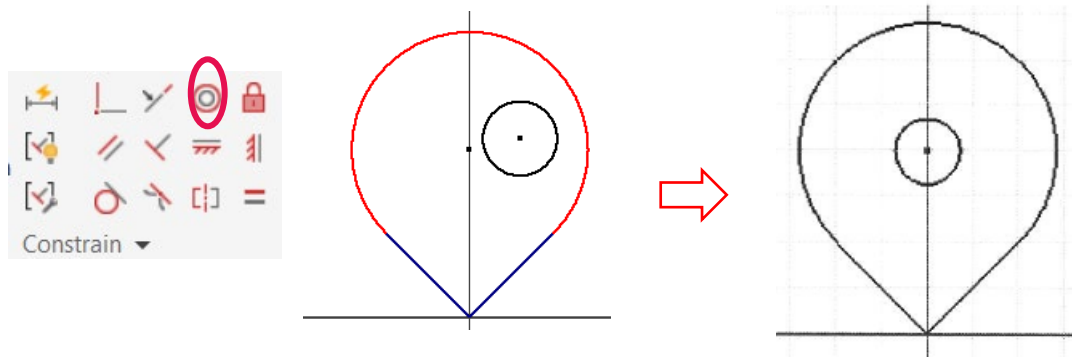


7. In the constraint panel, click 'Tangent Constraint' and select the arc and two (2) lines consecutively. The two line and the arc will be moved until they are tangent to each other.



Sketching Tools (Hands On #2 – Sketch Constraints)

4. In the constraint panel, click 'Concentric Constraint' and select the arc and the circle.
The entities will be moved until they are sharing the same center point.



Sketching Tools

(Hands On #3 – Mirror, Trim & Fillet)

- In this Hands-On, you will create a sketch geometry using 'Mirror', 'Trim' & 'Fillet' commands to complete the sketch. The completed sketch will be as shown in Figure 1.6
- Create a new part using Standard.ipt (mm) template.
 - Create a sketch on the XY plane.
 - Project YZ and XZ planes in the sketch.
 - Start the sketch by sketching two (2) circles and two tangent lines attach to both circle Right and Left side.

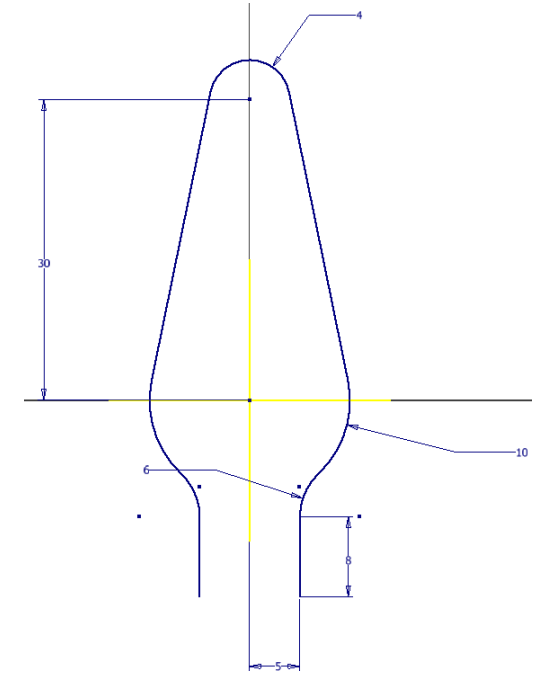
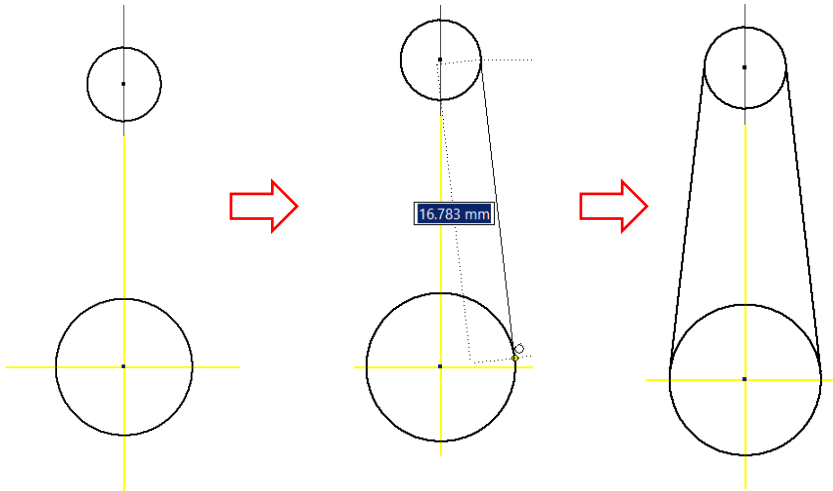
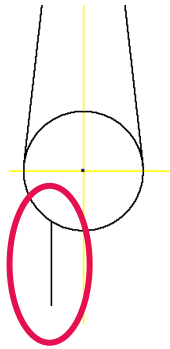


Figure 1.6

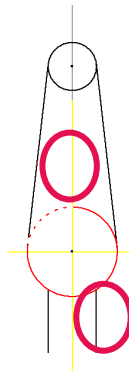
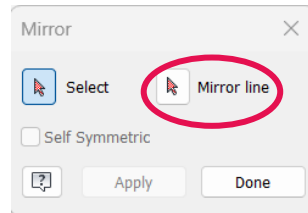
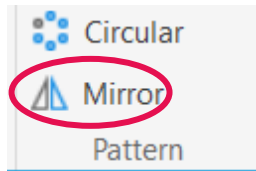
Sketching Tools

(Hands On #3 – Mirror, Trim & Fillet)

5. Create a line below the bottom circle as shown in picture below.



6. Hover cursor to 'pattern' panel, click 'Mirror' command. Select the newly create line, click on 'Mirror Line' in the dialog box and then select the vertical yellow line as a mirror line. You will see The newly created line duplicated at the opposite side.



7. In the 'Modify' panel, click 'Trim' command. Select the unwanted segments of both circles until it becomes as in figure 1.6.

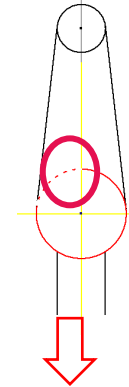
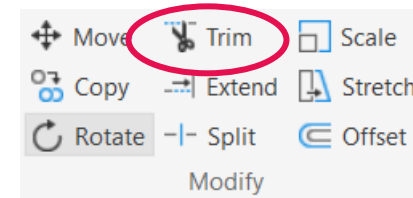
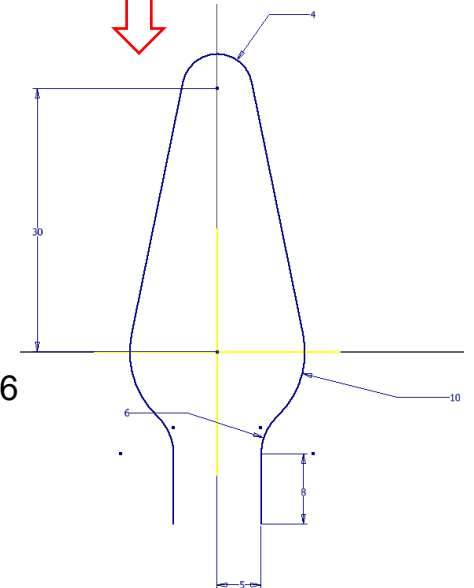


Figure 1.6

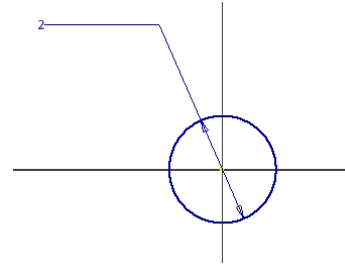


Sketching Tools

(Hands On #4 – Copy & Scale)

- In this Hands-On, you will create a sketch using 'Copy' & 'Scale' commands to complete the sketch. The completed sketch will be as shown in Figure 1.7

1. Create a new part using Standard.ipt (mm) template.
2. Create a sketch on the XY plane.
3. Sketch a circle with its center on the projected origin point.
4. Add 2mm of Diameter to the circle.



5. In the 'Modify' panel, click 'Copy' command. You will see 'Copy' dialog box menu appear.

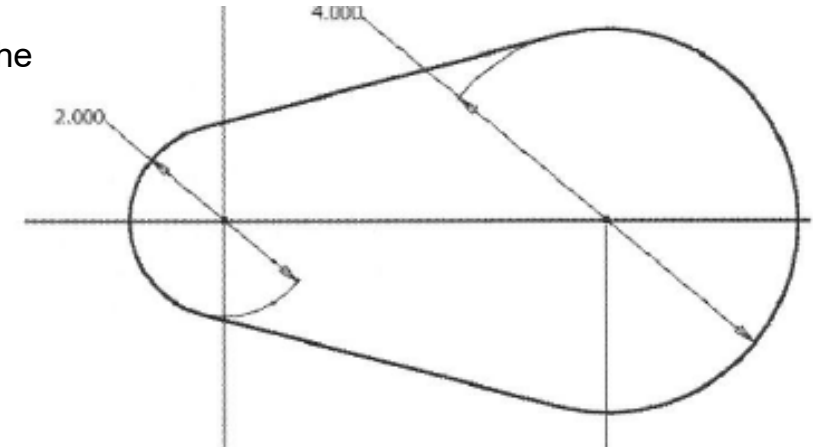
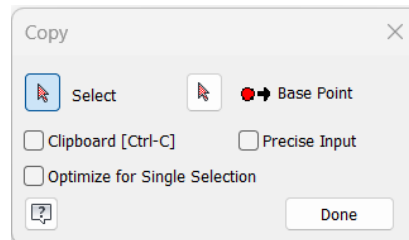
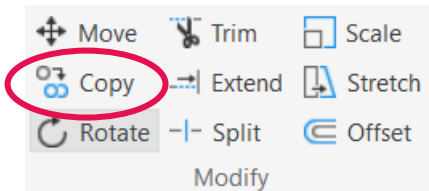
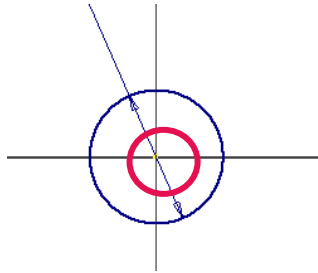
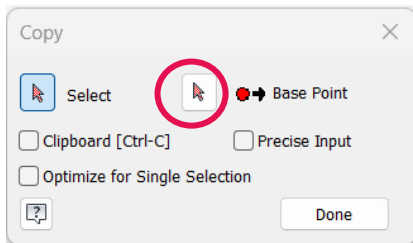


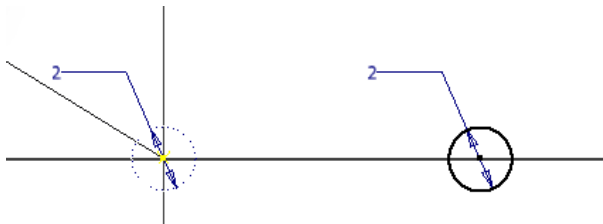
Figure 1.7

Sketching Tools (Hands On #4 – Copy & Scale)

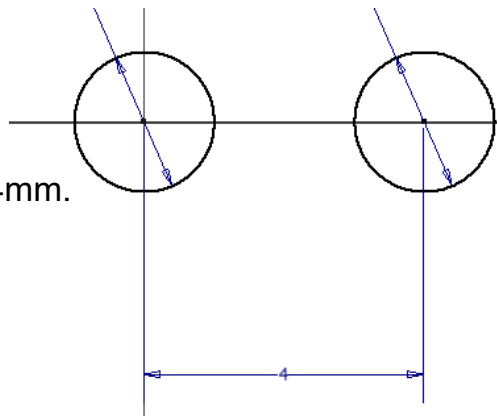
6. Select the 2mm diameter circle. Click on 'Base Point' in the dialog box menu, select the center of the circle as base point.



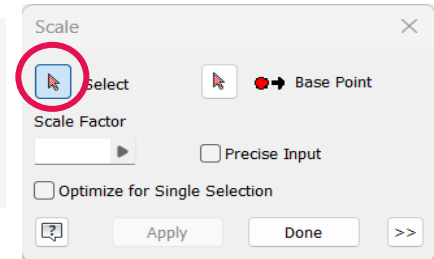
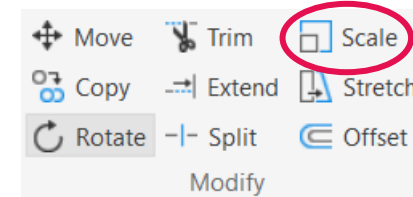
7. A copy of the circle will appear. Drag it to the right side horizontally and click to place it.



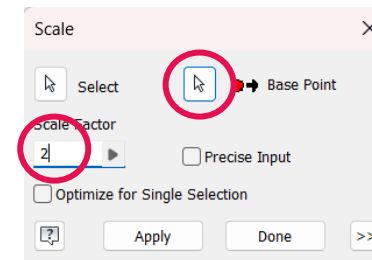
8. Put a dimension between the center of the circles by 4mm.



9. In the 'Modify' panel, click 'Scale' command. Select the newly copied circle.

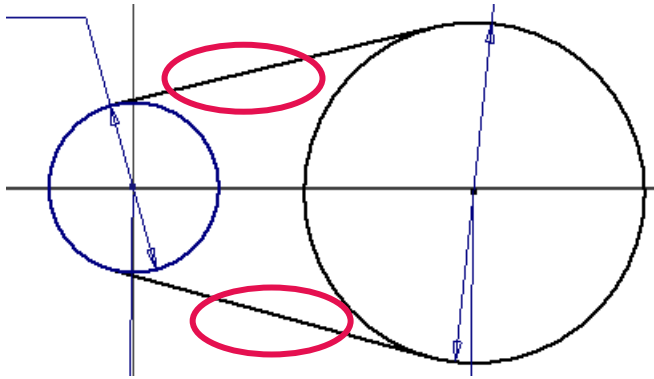


10. Click 'Base Point' in the Scale dialog box menu, and select the center of the selected circle. Enter 2 as the value of scale factor. The circle will become bigger by 2 times.



Sketching Tools (Hands On #4 – Copy & Scale)

11. Draw two (2) lines tangent to the both circles at the top and bottom.



12. Trim the sketch as shown in Figure 1.7.

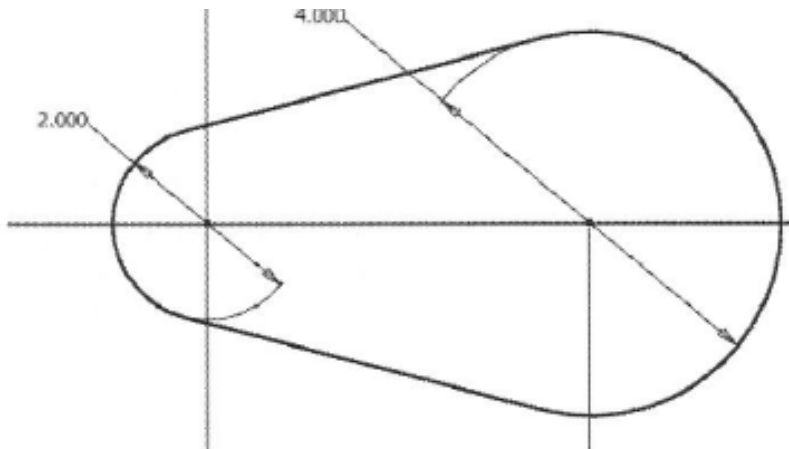


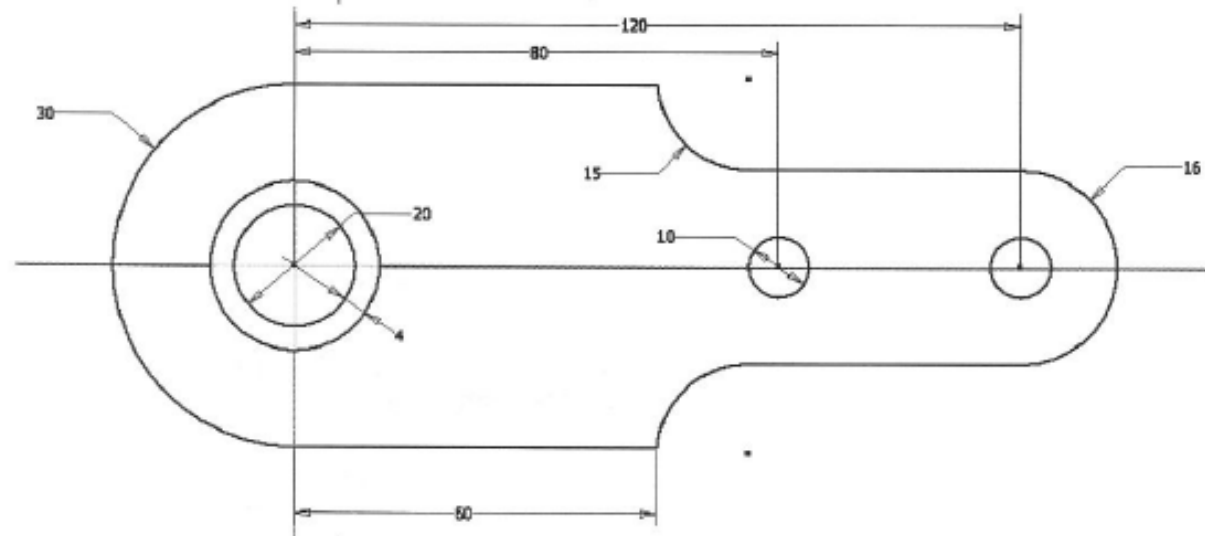
Figure 1.7

Sketching Tools (Practice)

Practice Objectives

- Sketch, dimension, and constrain entities to fully constrain a sketch.
- Use Relax mode to temporarily suspend constraints and dimensions to move constrained sketched entities.

In this practice, you will create the sketch shown in Figure 3–65. To successfully create this sketch you will be required to sketch linear, arc, and circular entities and ensure that they are fully dimensioned and constrained.



Sketching Tools (Hands On #5 – Circular Pattern)

- In this Hands-On, you will pattern the circular sketches as shown in Figure 1.8

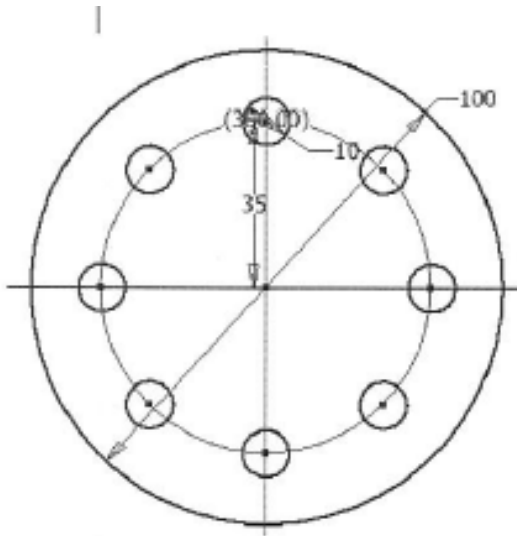


Figure 1.8

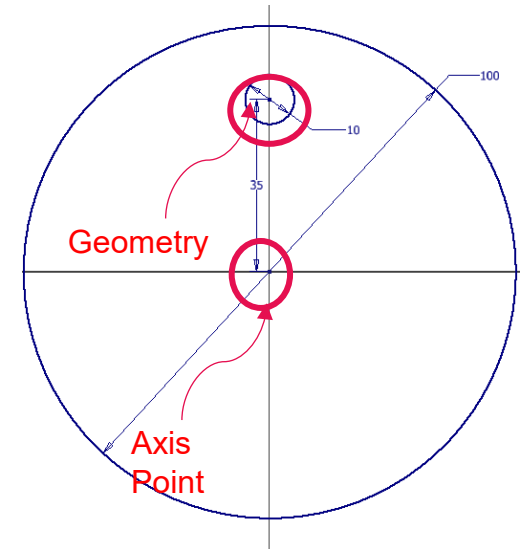
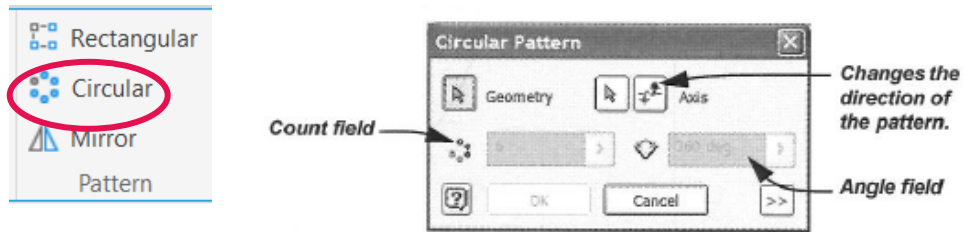
1. Open a file 'pattern_circular.ipt'.
2. Edit the sketch that was provided for you in the model.

Sketching Tools

(Hands On #5 – Circular Pattern)

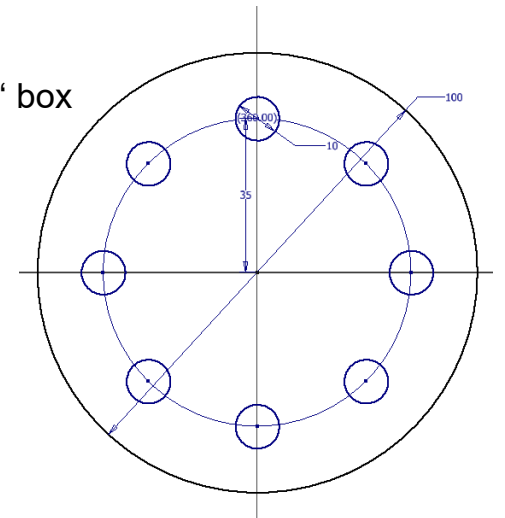
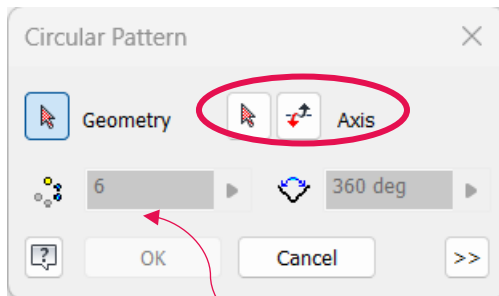
3. You will then see in the sketch there are two circles. We want to duplicate the smaller circle by 7 times in a circular pattern.

In the 'Pattern' panel, click on 'Circular Pattern'. The 'circular Pattern' dialog box menu will appear.



4. With 'Geometry' box selected, select the small circle.

5. In the dialog box menu, select 'Axis' box and select the vertex at the center of the larger circle. Enter 8 in the 'Count' box



Sketching Tools (Hands On #6 – Rectangular Pattern)

- In this Hands-On, you will pattern rectangular sketches as shown in Figure 1.81

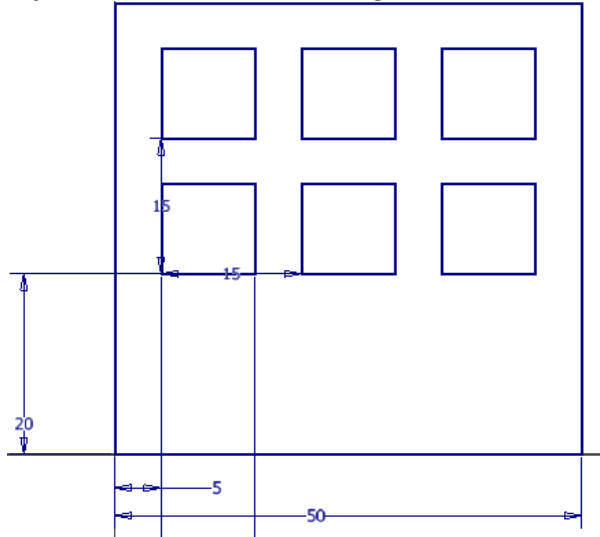


Figure 1.81

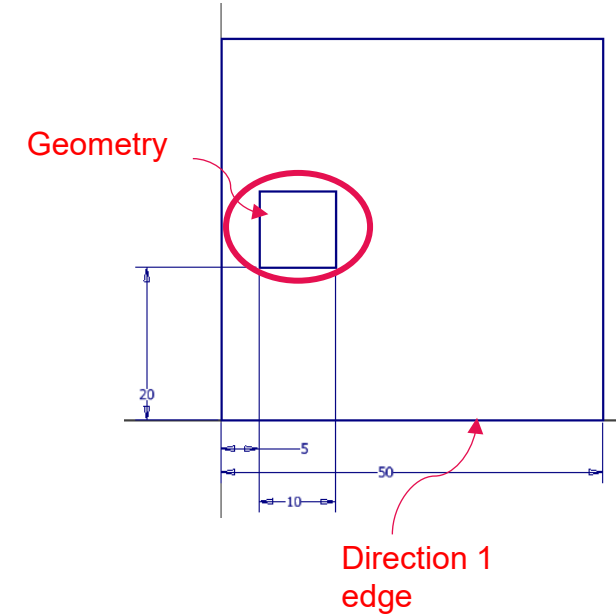
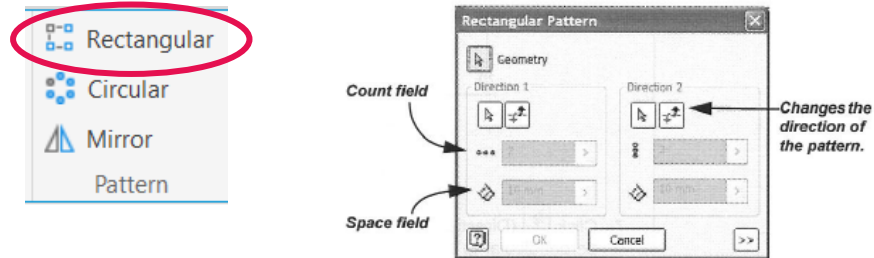
1. Open a file 'pattern_rectangular.ipt'.
2. Edit the sketch that was provided for you in the model.

Sketching Tools

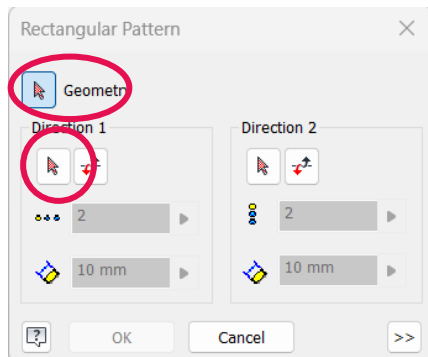
(Hands On #6 – Rectangular Pattern)

- You will then see in the sketch there are two rectangles. We want to duplicate the smaller rectangle vertically and horizontally.

In the 'Pattern' panel, click on 'Rectangular Pattern'. The 'Rectangular Pattern' dialog box menu will appear.



- With 'Geometry' box selected, select the small rectangle.
- In the dialog box menu, select 'Direction 1' box and select the bottom line as a reference for horizontal direction.

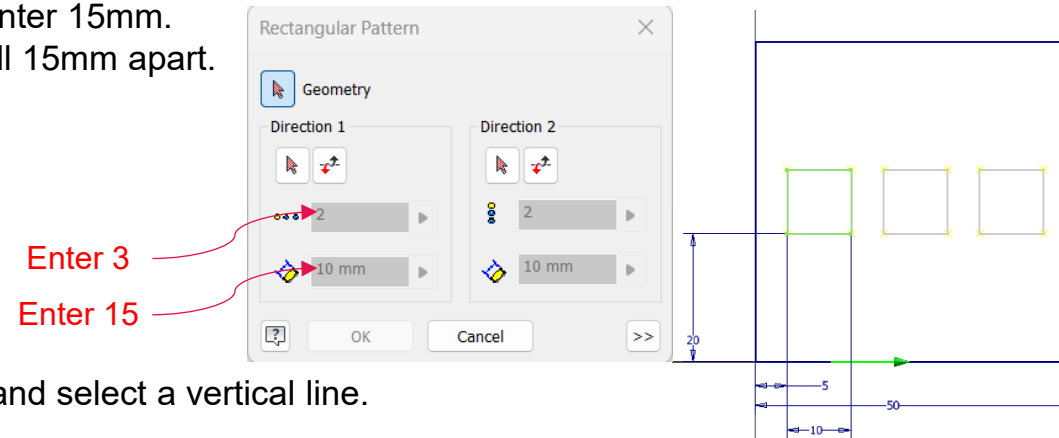


Sketching Tools

(Hands On #6 – Rectangular Pattern)

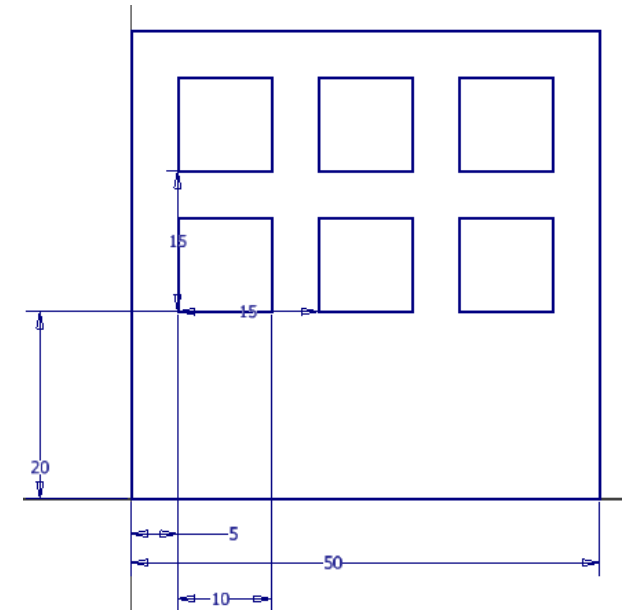
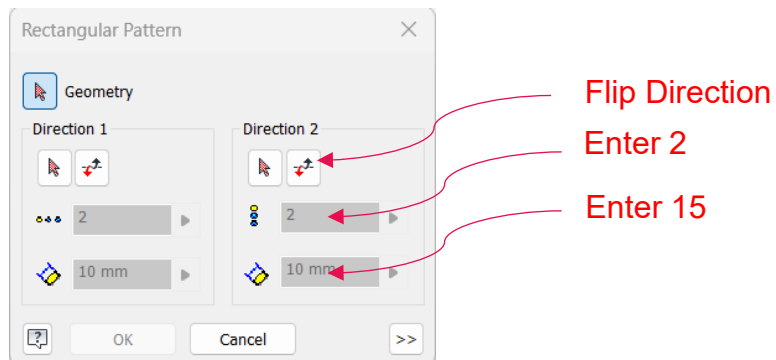
6. By default, the direction will be to the right. You can flip it if you want to change it to left. In this case, we want to maintain to the right.

In the 'Pattern' panel, click on 'Count' field, enter 3 as we want it duplicate 2 additional rectangles to the right. Click on the 'Space' field, enter 15mm. The space between the instances will 15mm apart.



7. In the dialog box, click 'Direction 2', and select a vertical line.
8. On the 'Count' field, enter 2, and on the 'Spacing' field, enter 15. Click 'Flip' direction to change direction into

going upwards.



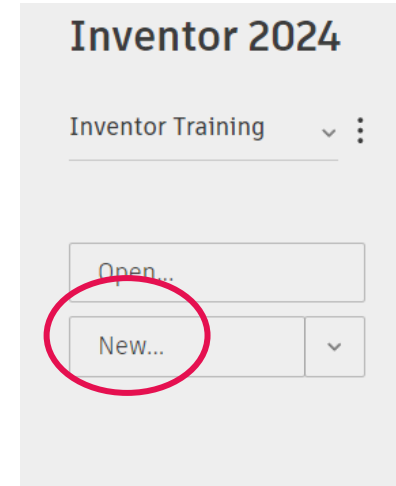
3D Features

(Hands On #7 – Sketch & Extrude)

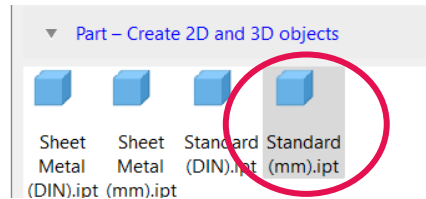
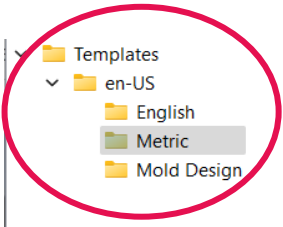
1. Click on 'New' button located on Quick Access Toolbar / Project Browser



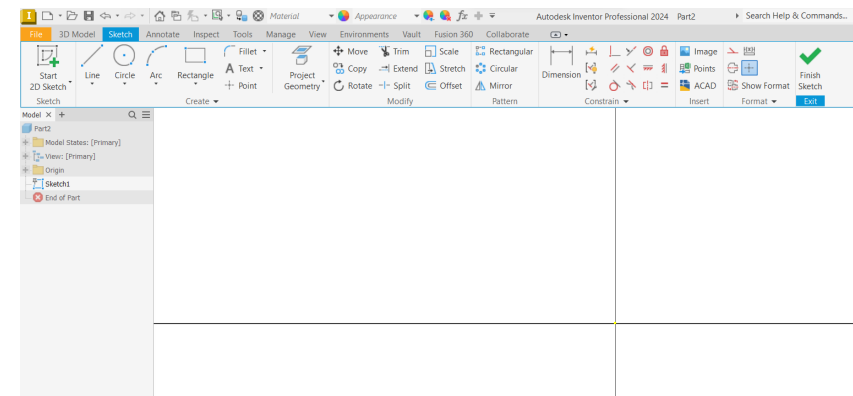
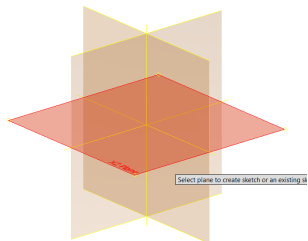
or



2. Select Standard.ipt (mm) template.



3. Click 'Start 2D Sketch'.
4. In the Graphic Window, select XZ Plane. You'll enter 'Sketch' environment.

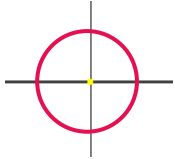


3D Features (Hands On #7 – Sketch & Extrude)

5. Click 'Line' command



6. Create the lines starting from the center of the sketch. Create 6 lines to form a shape as shown in the Figure 1.



7. Find and click on 'Dimension' command in the constraint panel. Place all the dimension needed as shown in Figure 1. After that, click 'Finish Sketch'.
- To place a dimension you may click between end points of a line as per highlighted in Figure 1.
 - To place a dimension you may also click on the line itself as per highlighted in Figure 1.
 - Change the value of dimension accordingly after placing a dimension.

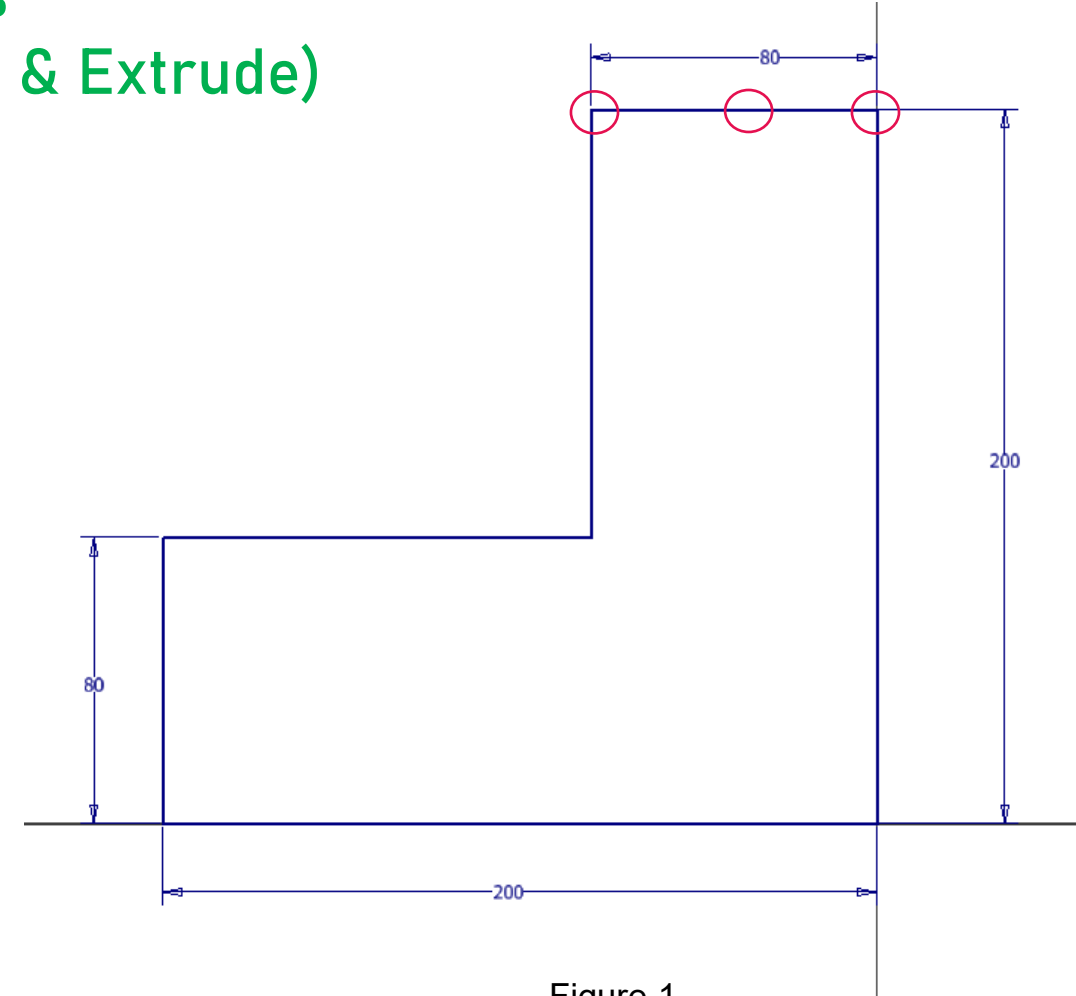
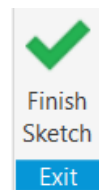
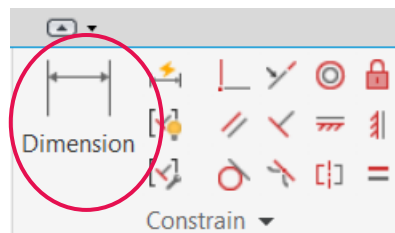
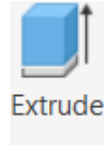


Figure 1

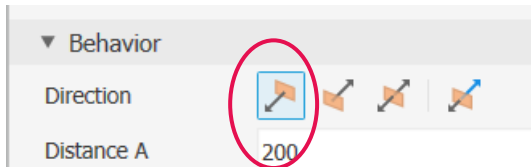
3D Features

(Hands On #7 – Sketch & Extrude)

1. Click 'Extrude' command



2. An extrude properties menu will appear. In behavior section, select direction 'Default'. Enter the value for 'Distance A', 200mm. Optionally you may also enter the value in the flyout box near model as shown in Figure 1.1



- You may also click and hold the handle (arrow) in the graphic window (near model) to adjust the 'Value' for the distance.
3. Click 'Ok' Button in the menu.

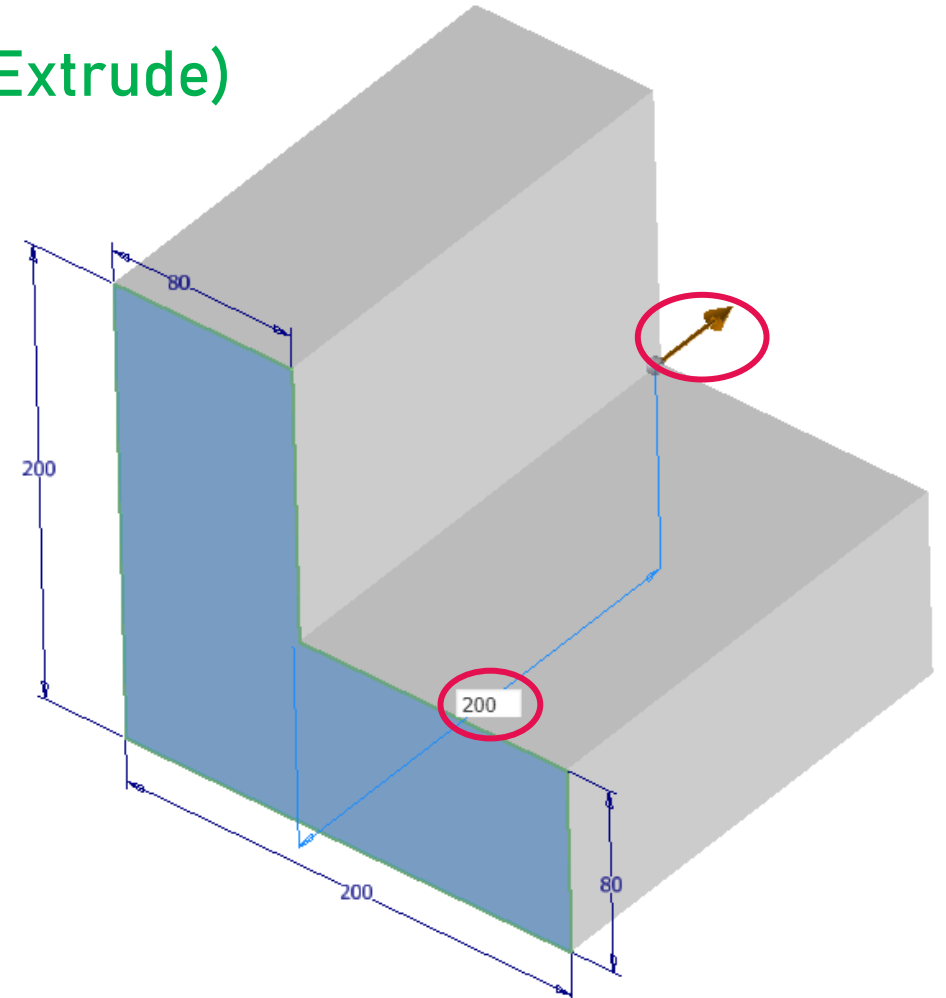


Figure 1.1

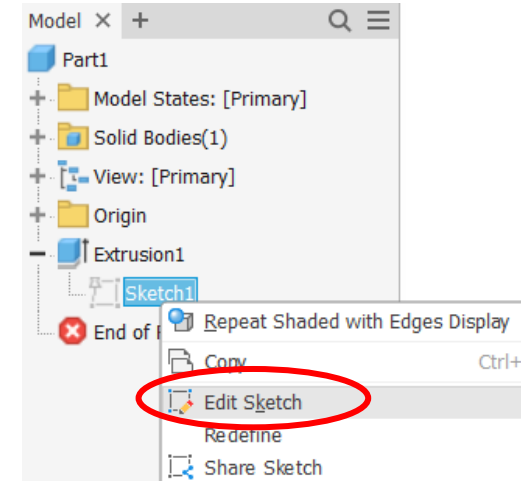
3D Feature

(Hands On #8 – Edit Sketch)

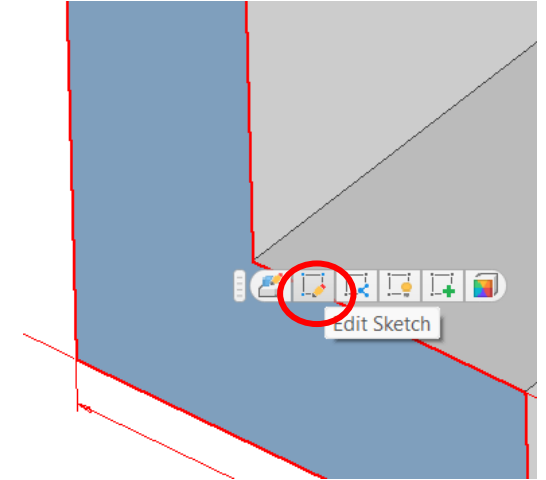
- There are two (2) ways to modify a model.
 - First is through 'sketch edit' command.
 - Second is by using 'show dimension' command.

1. Hover cursor into 'Feature Browser'. Expand 'Extrusion 1' feature. Right click on 'Sketch 1' and click 'Edit Sketch'. You will enter back into sketch environment.

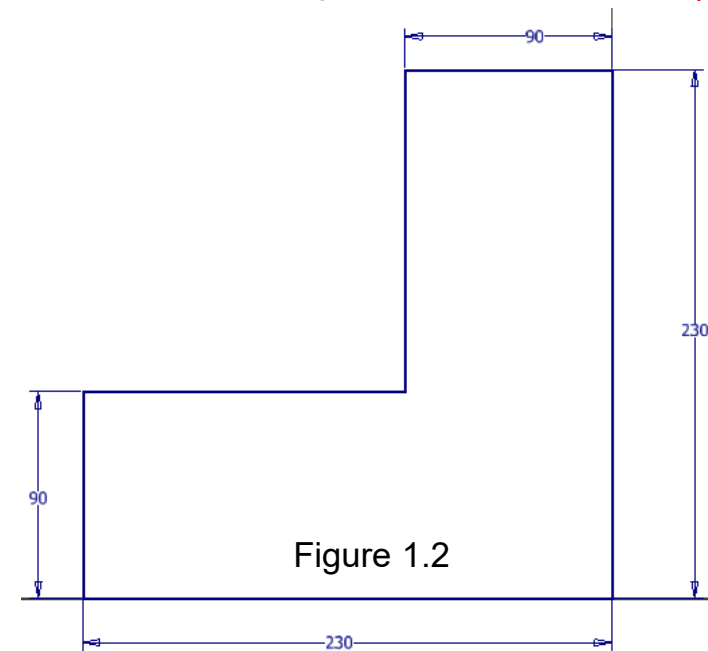
Alternatively, you may click on any surface of the model. A flyout will appear. Click on the 'Edit Sketch' in the flyout menu.



Or



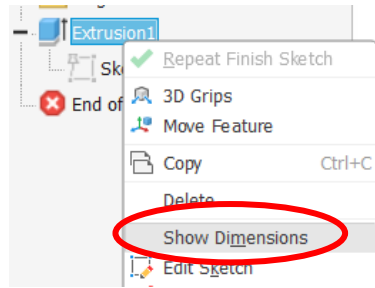
2. Change the dimension value accordingly; (Figure 1.2)
 - 200mm to 230mm
 - 80mm to 90mm
3. Click 'Finish Sketch'. See the update.



3D Feature (Hands On #9 – Show Dimension)

1. Hover cursor into 'Feature Browser'. Right click on 'Extrusion 1'. Click on 'Show Dimension'.

Dimensions will appear in the model view as shown in Figure 1.3



2. Double click on each dimensions and change the dimension value accordingly;
 - 230mm to 200mm
 - 90mm to 80mm

The change will be reflected to the sketch instantly.

3. Click 'Local Update' on the 'Quick Access Toolbar' at the top of window to see the change on the model.

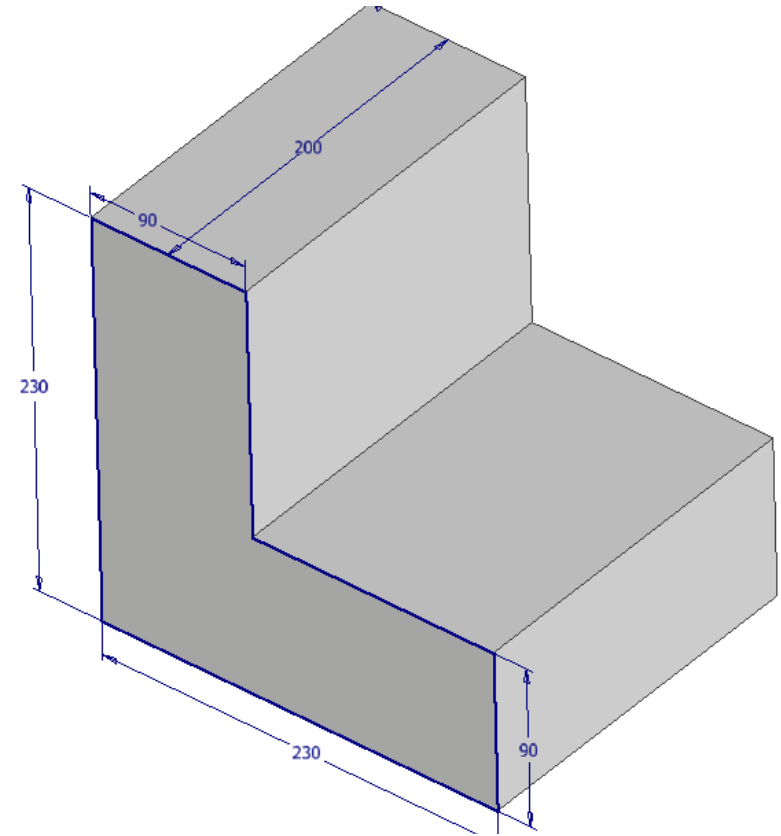
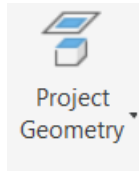


Figure 1.3

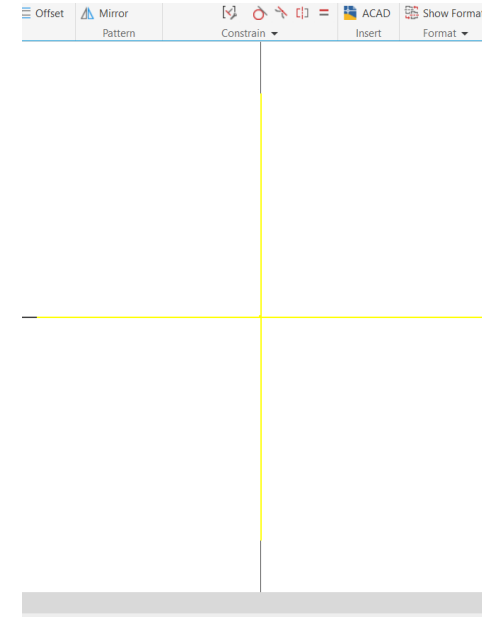
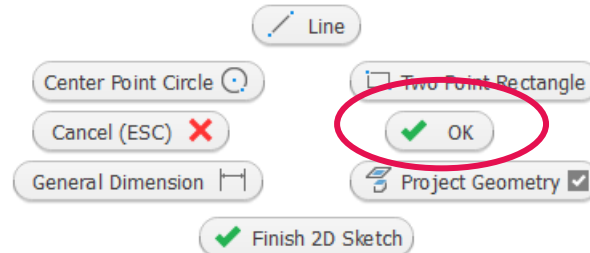
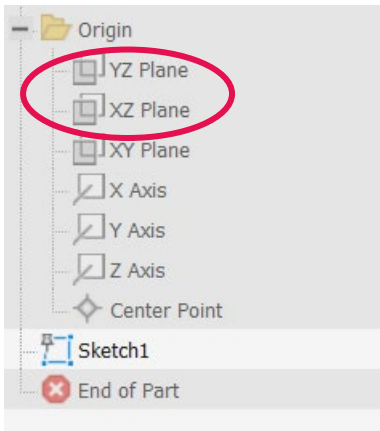
3D Feature (Hands On #10 – Sketch & Revolve)

1. Click on 'New' button and select Standard.ipt (mm) template.
2. Start a sketch using Plane XY.
3. In the sketch environment, hover your cursor to 'Create' panel, click on 'Project Geometry' command.



4. Move on to feature browser, expand a 'Origin' Folder. Select YZ & XZ planes that listed in the folder. Right click, click 'OK' on the flyout.

Those planes will be turned into two (2) reference lines colored by yellow.



3D Feature (Hands On #10 – Sketch & Revolve)

5. By using 'Line' command, create the shape as per shown in the Figure 1.4.

Follow the numbering given for clicking reference. 1 to 5 back to 1.

6. Click on 'Dimension' command, and place all of the dimension needed as shown in Figure 1.5.

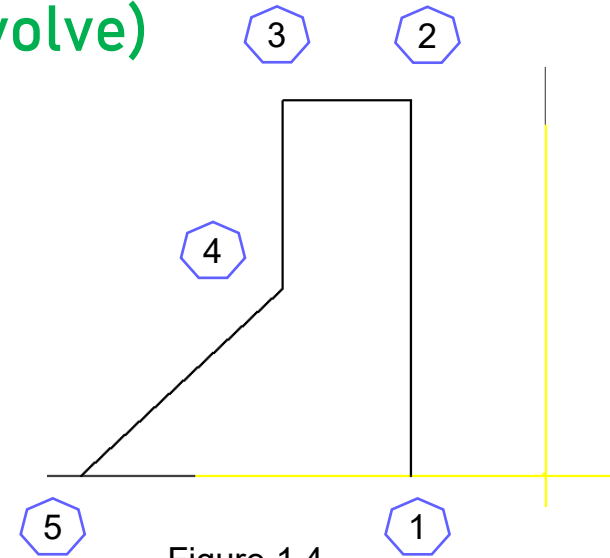


Figure 1.4

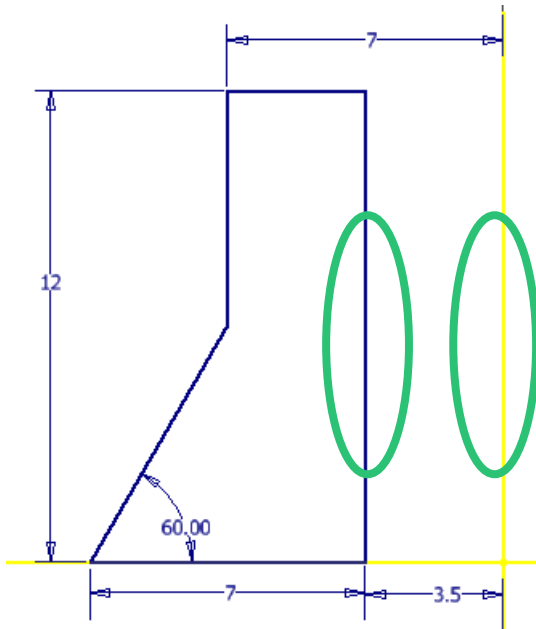
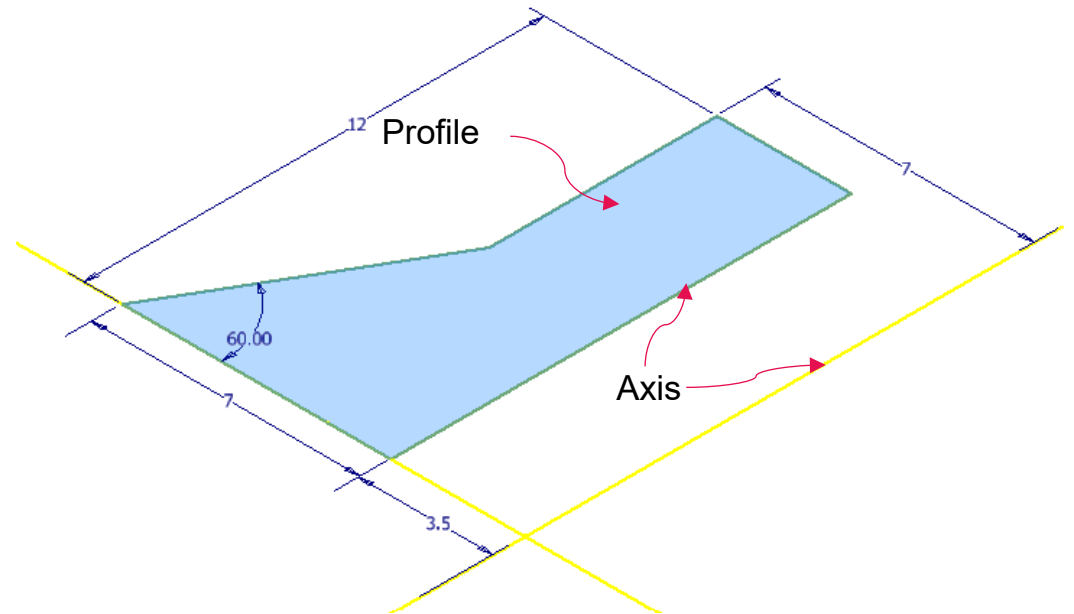
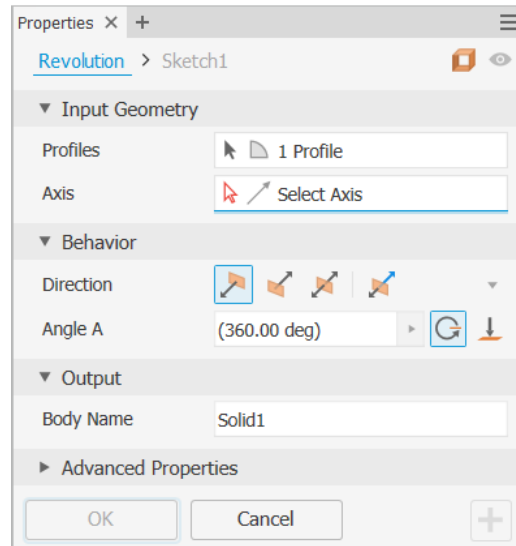
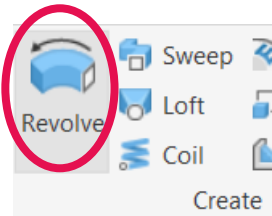


Figure 1.5

- To create 'Angle' dimension, use the same 'Dimension' command, click on the two lines of the angle.
 - Start with the biggest dimension is a good practice.
 - The two (2) highlighted lines will be used as Revolve 'axis'. The shape will become 'Profile'.
5. Click 'Finish Sketch'

3D Feature (Hands On #10 – Sketch & Revolve)

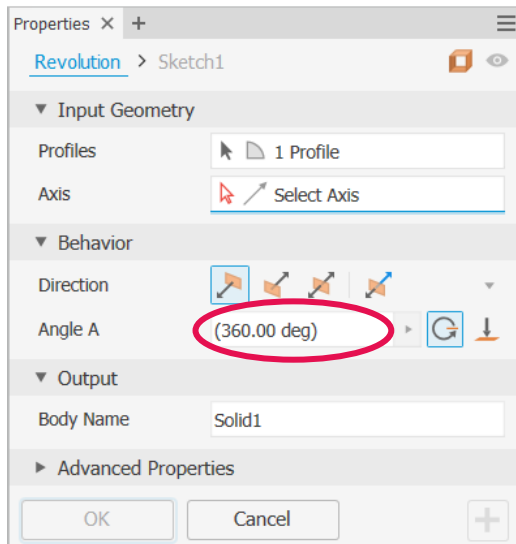
1. Hover cursor to 'Create' Panel, Click on 'Revolve' command. Revolve properties menu will appear.



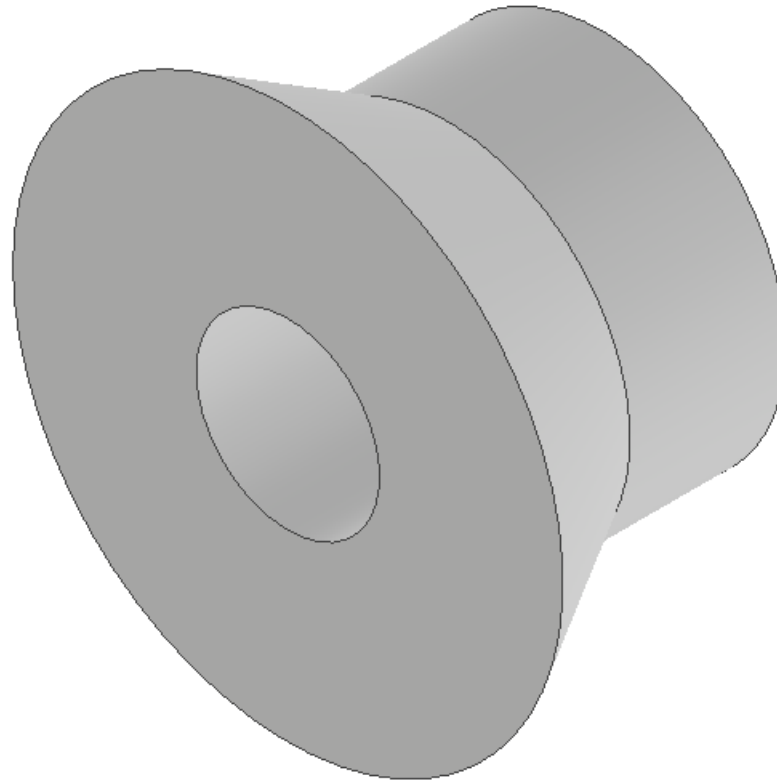
2. Select 'Profile' box inside of the Revolve properties menu, select internal area of the shape.
*Profile normally will be automatically selected.
3. Select 'Axis' box in the Revolve properties menu, select the Projected line as the axis of rotation.

3D Feature (Hands On #10 – Sketch & Revolve)

4. Click on 'Angle A' box. Set the angle of rotation by 360 deg.

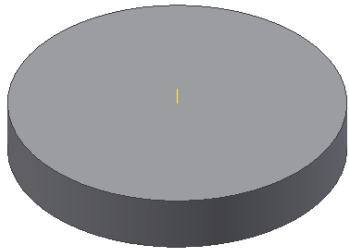


5. Click 'OK' to accept the design.



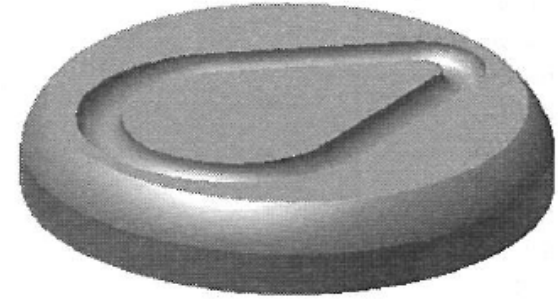
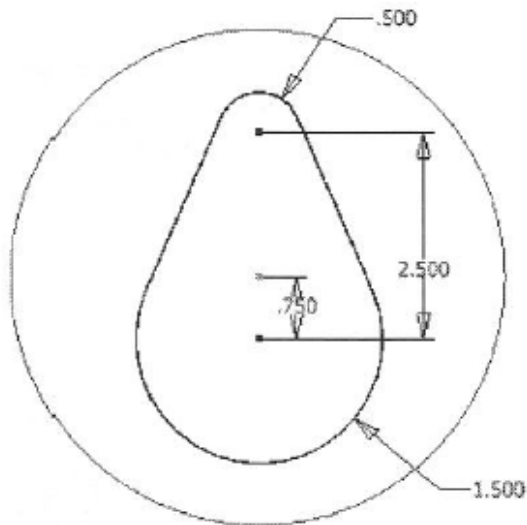
3D Feature (Hands On #11 – Sketch & Sweep)

1. Open 'sweep.ipt' file. The model contains a single base feature.



2. Create a sketch on **top surface** of the base part as shown in the picture below. (For Path)

Rename the Feature as "Path".



3D Feature (Hands On #11 – Sketch & Sweep)

3. Create a New Sketch on YZ Plane. Draw a 0.5in diameter circle. Use 'Coincident' constraint and apply it between center of the new circle to a point at the left most of the 'path' sketch. Finish sketch. (You may need to use 'Project Geometry' to project 'Path' sketch as reference.)

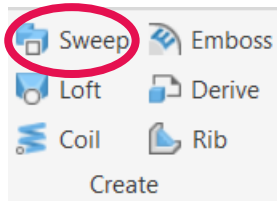


4. Rename the sketch as "Profile".



3D Feature (Hands On #11 – Sketch & Sweep)

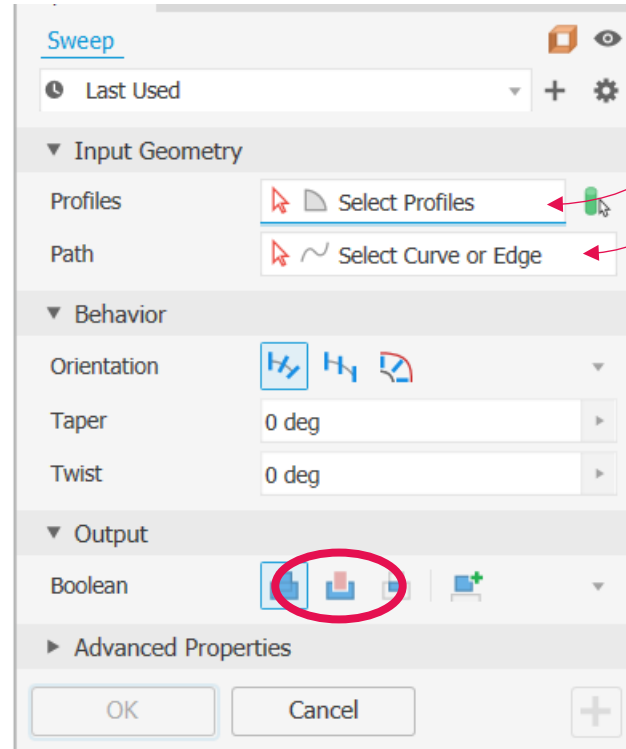
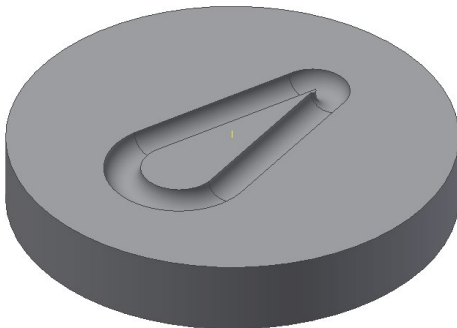
1. In the 'Create' panel, Click 'Sweep' command. The 'Sweep' dialog box menu will appear.



2. Click 'Profile' box in the dialog box, then select the circle profile in the graphic window. Do the same thing for the 'Path' box and the 'Path' sketch.

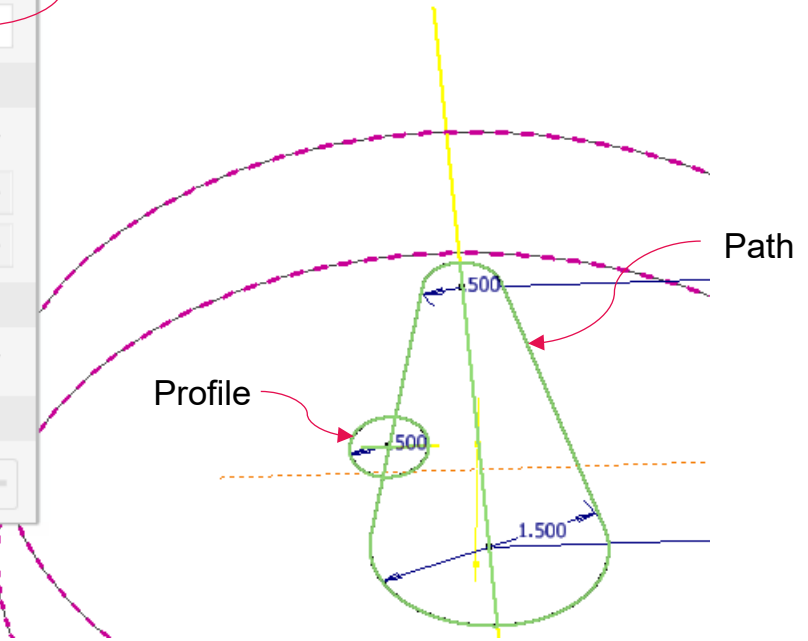
Select 'Cut' option in the dialog box menu under 'Boolean' segment.

3. Click OK.



Profile
Path

Path
Profile

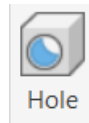


3D Feature (Hands On #12 – Hole Feature)

- In this Hands-On, you will add a coaxial Hole to a cylinder as per shown below.



- Open a file 'cylinder_1'.
- In the 'Modify' panel, Click 'Hole' command.

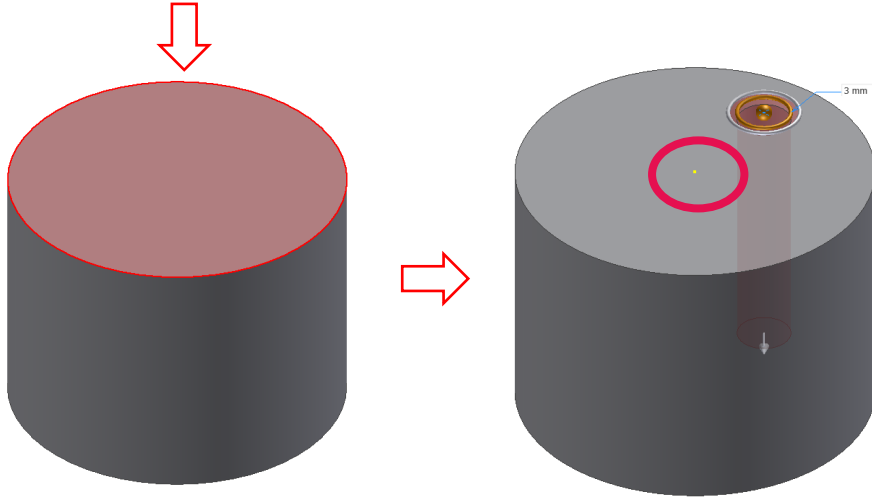


- In the 'Hole' dialog box, select the 'Simple Hole' and 'Counter Sink' options.



3D Feature (Hands On #12 – Hole Feature)

4. Select the Top surface of the cylinder model, then click on the **center point** of the circular surface.



5. Right click at the first hole point, right click, click 'Delete' on the fly-out menu as shown in Figure 1.9.

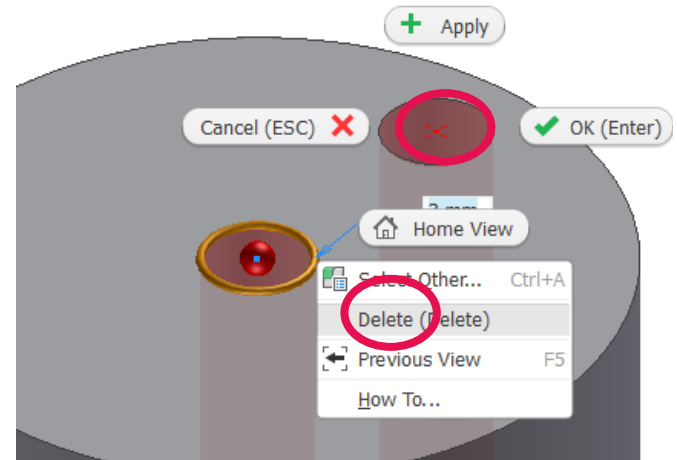
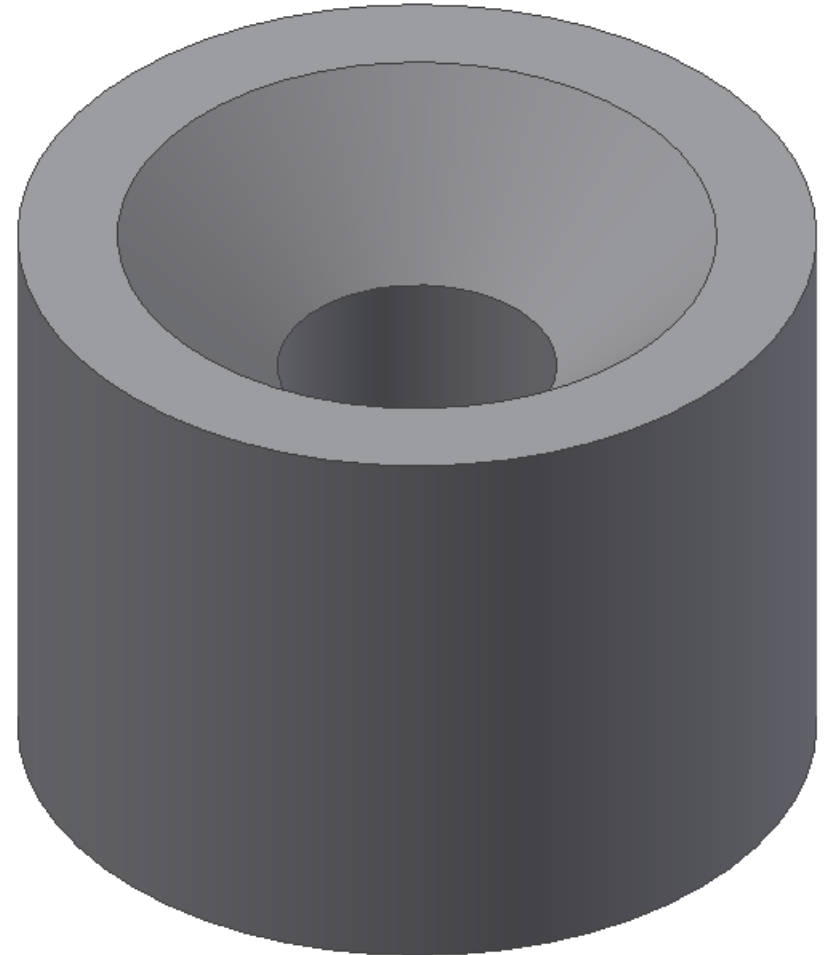
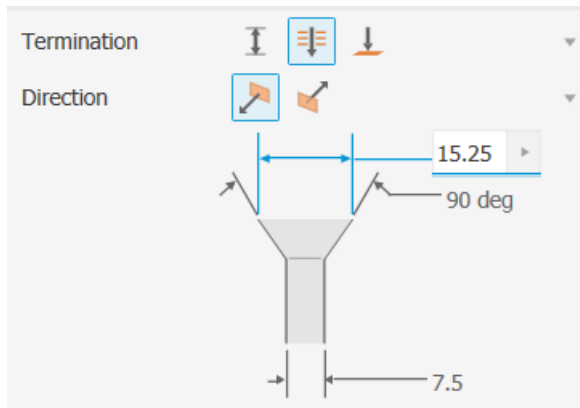


Figure 1.9

3D Feature (Hands On #12 – Hole Feature)

6. Enter the values of 7mm for 'Diameter' of hole, and 15mm for the value of Counter Sink Diameter. The angle is 90deg.



3D Feature (Hands On #13 – Fillet Tool)

1. Open 'fillet.ipt' file.
2. In the 'Modify' panel, click 'Fillet' command. The 'Fillet' dialog box menu will appear.
3. Select all four outside edges of 'extrusion 7', as shown in the Figure 1.91. Fillets will be automatically preview on the model.
4. In the dialog box, enter the Radius value by 6.35. Click OK. Optionally you may click '+' sign.

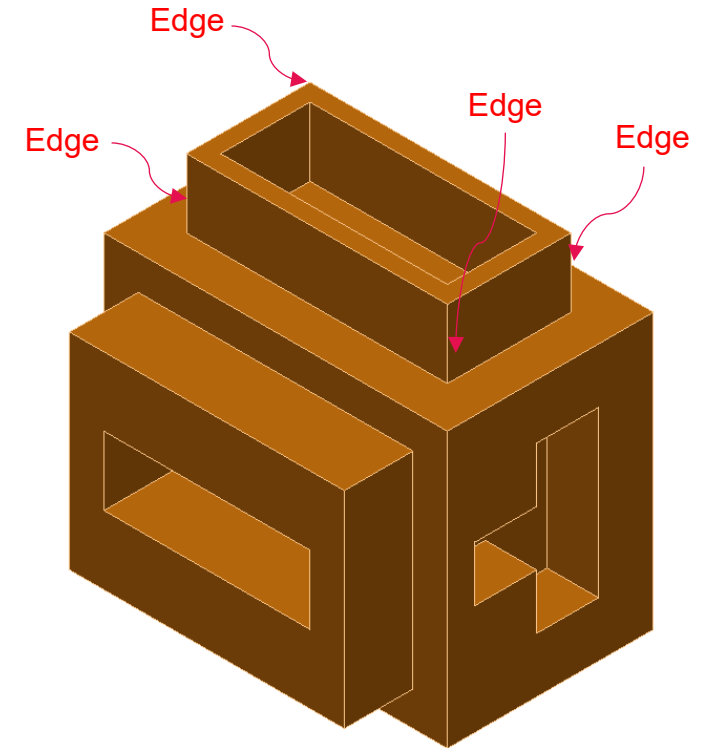
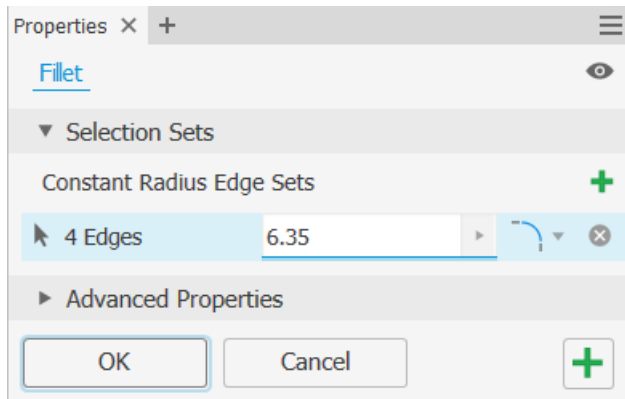


Figure 1.91

3D Feature (Hands On #13 – Fillet Tool)

5. Select the edges shown in the Figure 1.92 Change the value to 9. The changes will be automatically reflected in the model..

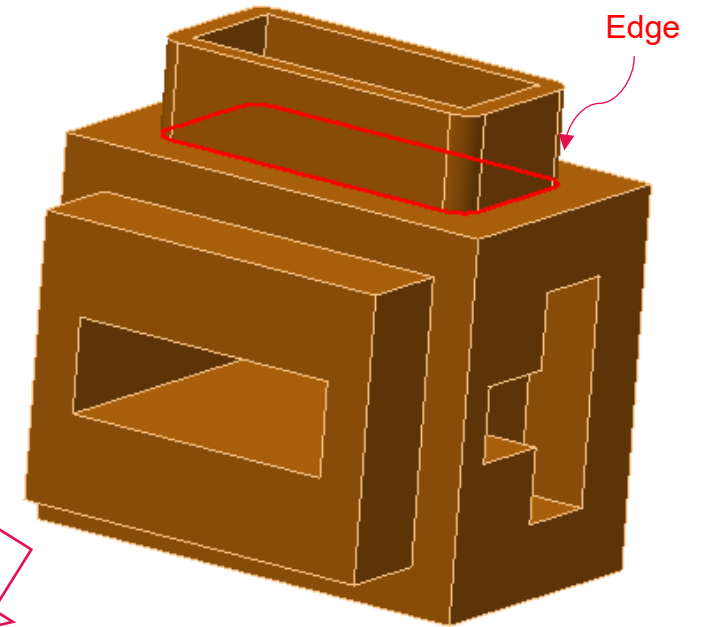
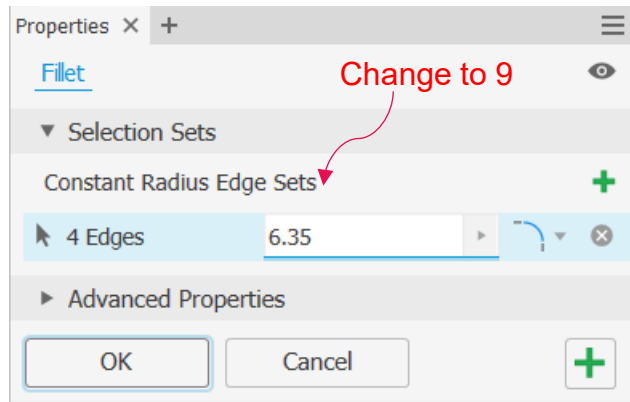
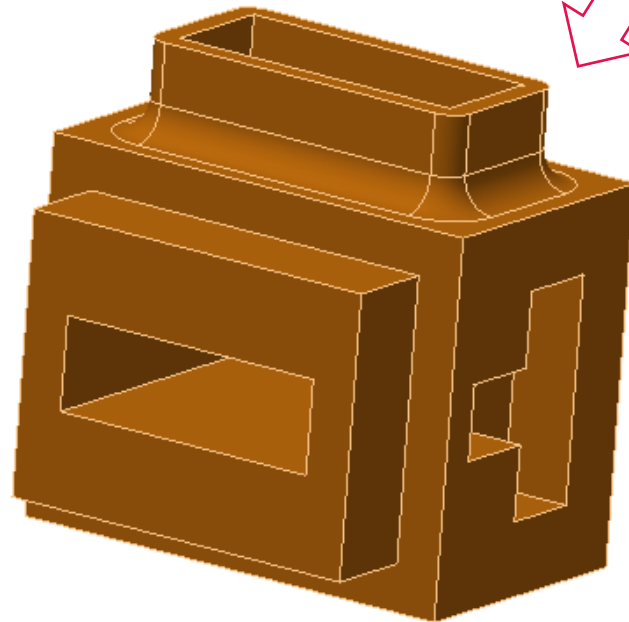
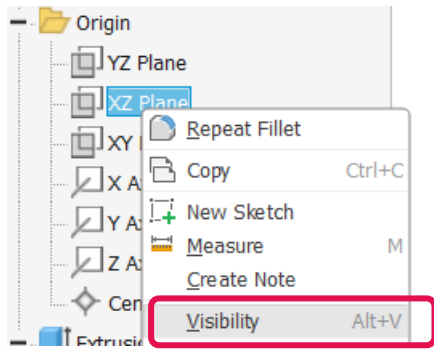


Figure 1.92

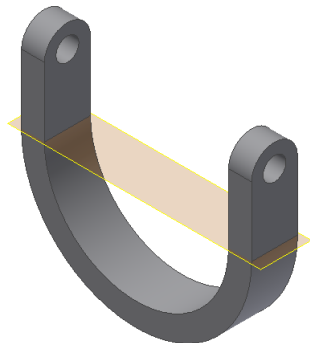


3D Feature (Hands On #14 – Plane Features)

1. Open 'fork.ipt'.
2. Expand 'Origin' folder, Right Click on Plane XZ. Turn On option 'Visibility'. You will see the plane appear.

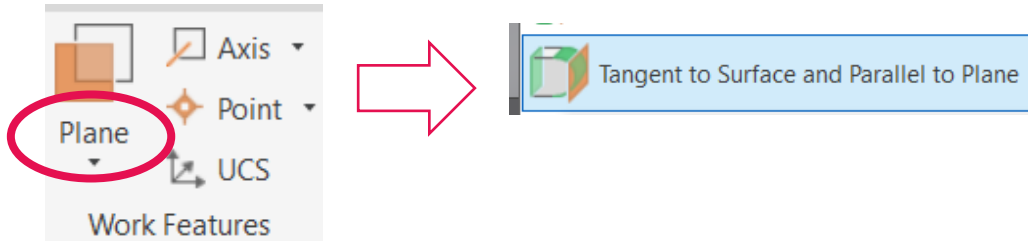


3. Select all four outside edges of 'extrusion 7', as shown in the Figure 1.91. Fillets will be automatically preview on the model.

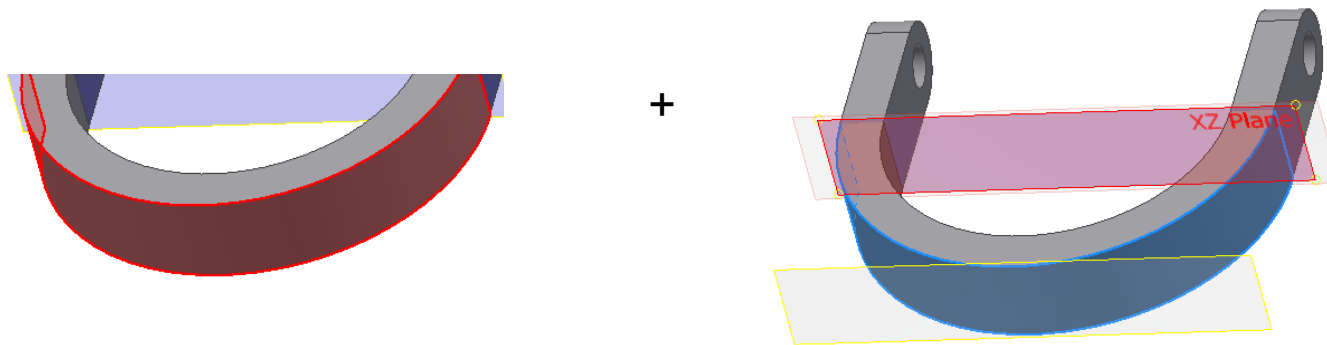


3D Feature (Hands On #14 – Plane Features)

1. In the 'Work' Panel, expand 'Plane' and select 'Tangent to Surface and Parallel to Plane' option.

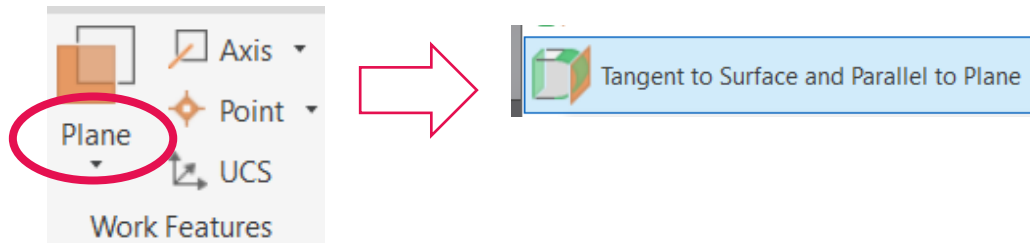


2. select bottom curve surface of the fork (Tangent Reference) and Select XZ plane (parallel reference).

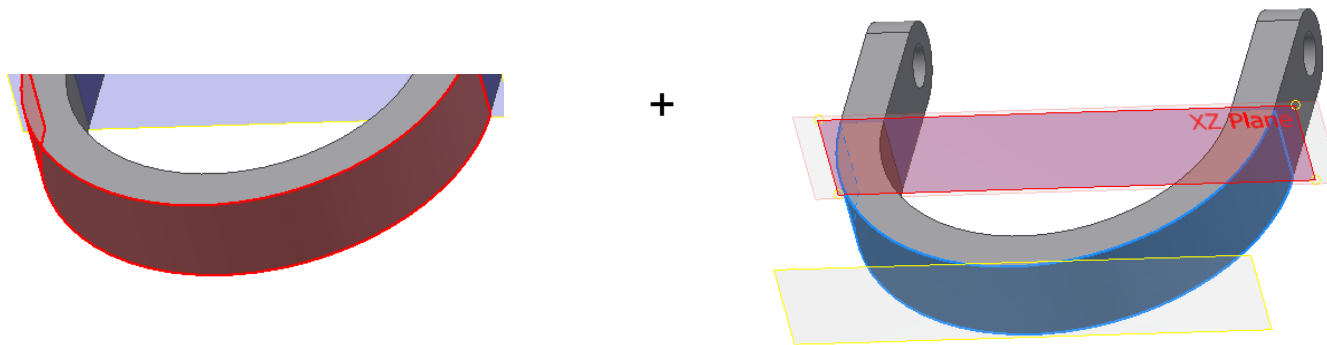


3D Feature (Hands On #14 – Plane Features)

3. In the 'Work' Panel, expand 'Plane' and select 'Tangent to Surface and Parallel to Plane' option.



4. select bottom curve surface of the fork (Tangent Reference) and Select XZ plane (parallel reference).

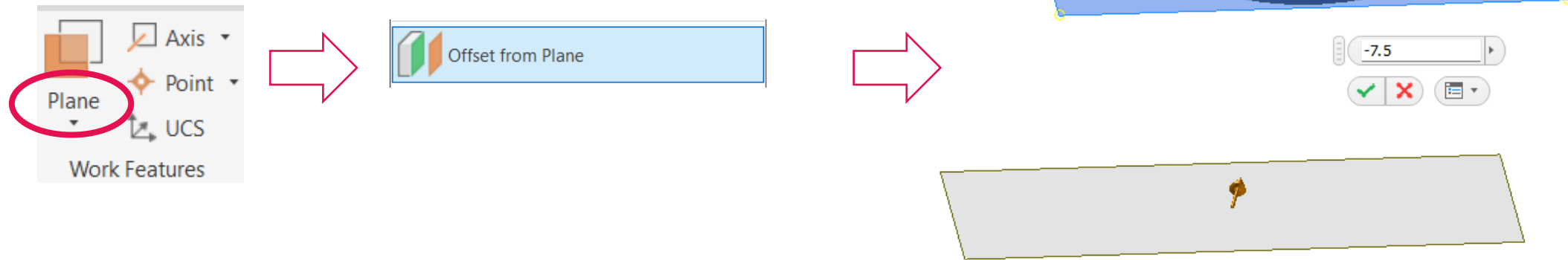


3D Feature (Hands On #14 – Plane Features)

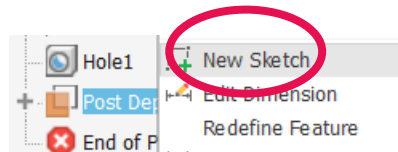
5. In the 'Work' panel, expand 'Plane' drop-down ribbon, select 'Offset from Plane'.

Select plane "Post Tangent Ref", and enter -7.5in for the offset value. Click OK.

Rename the newly created plane as "Post Depth".



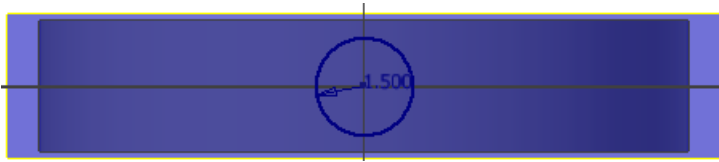
6. Select plane "Post Depth", right click on it, select 'New Sketch' in the fly out menu.



3D Feature (Hands On #14 – Plane Features)

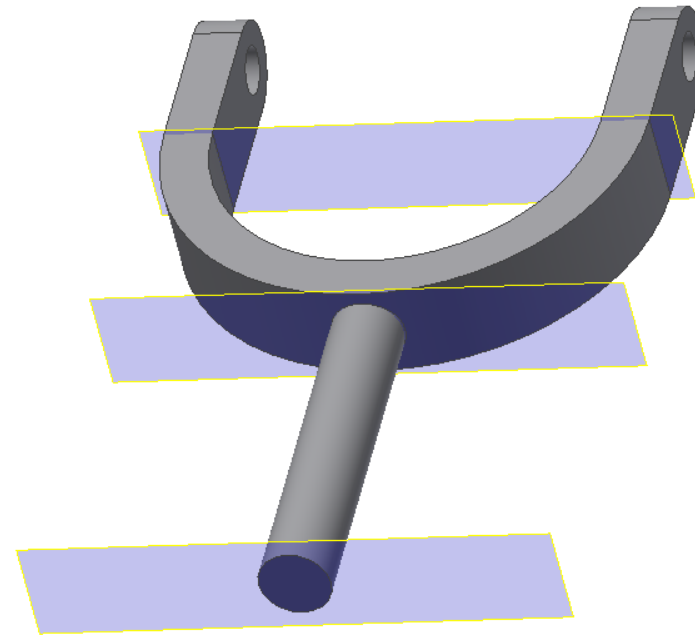
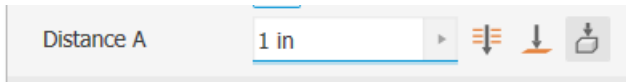
7. In the sketch, draw a Circle at the center point. Enter 1.5in for the diameter.

Click 'Finish Sketch'.



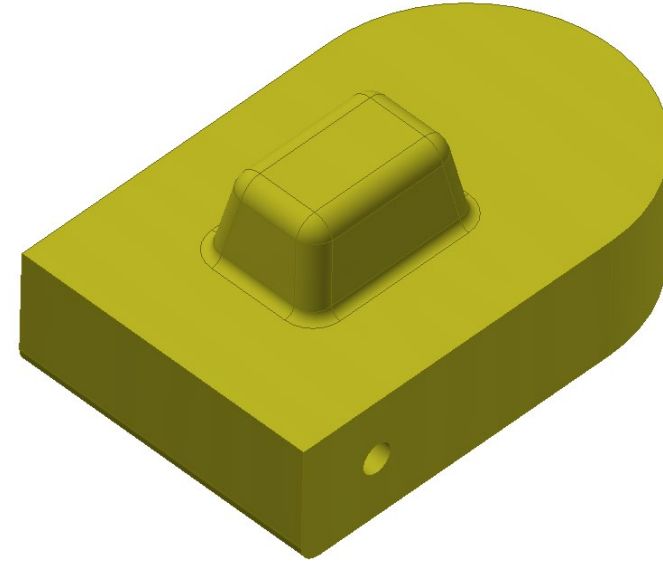
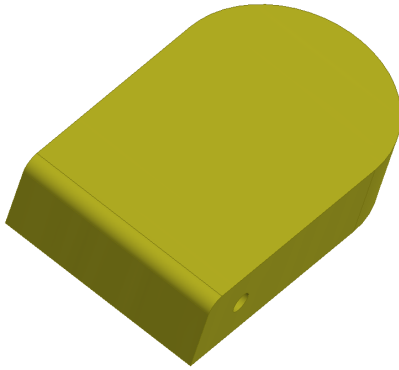
8. In the 'Create' panel, click 'Extrude' to extrude the circle.

Select 'To Next' extent option. Click OK.



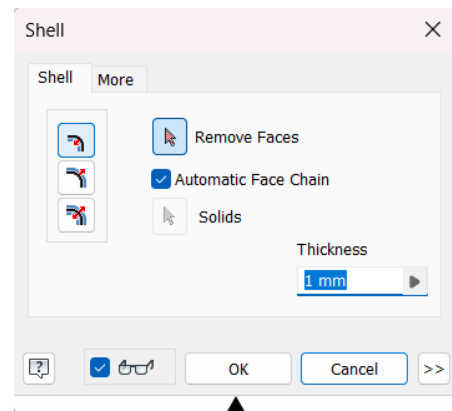
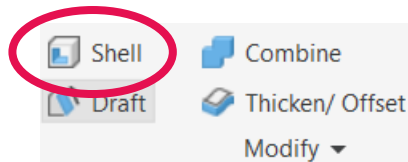
3D Feature (Hands On #15 – Shell Feature)

1. Open 'Shell.ipt' file. A base model will appear.
2. Rotate the view so that we can see the bottom of the model.



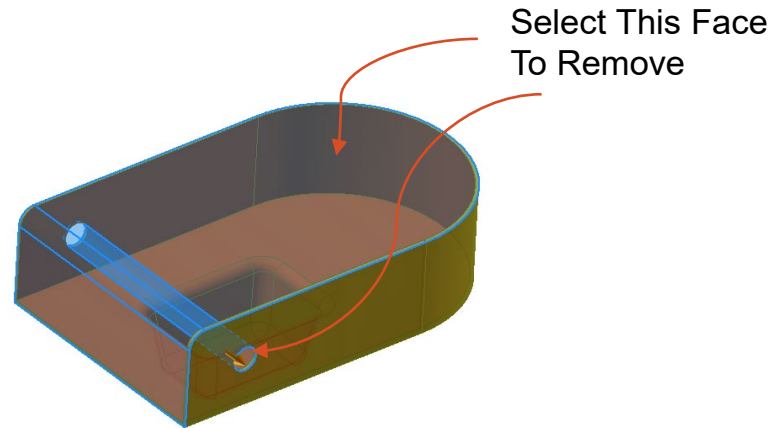
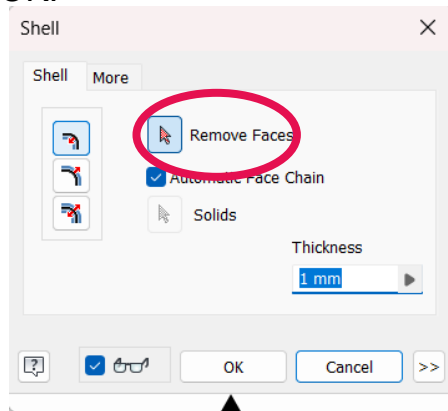
Base Model

3. In the 'Modify' panel, click 'Shell' command. The dialog box menu of Shell will appear. Maintain 1mm thickness.



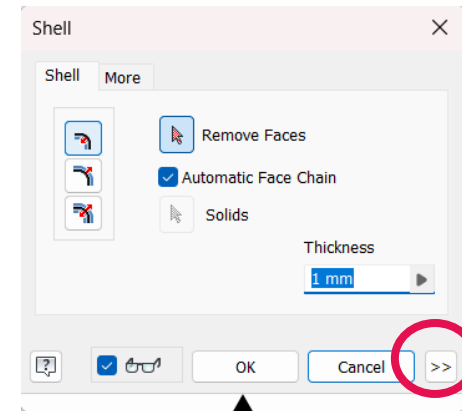
3D Feature (Hands On #15 – Shell Feature)

4. In the dialog box menu, select 'Remove Faces' box. Then in the graphical window, select the bottom face and the hole's inside surface on the model to remove. Click OK.

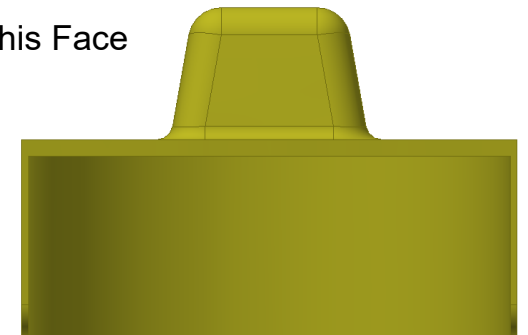
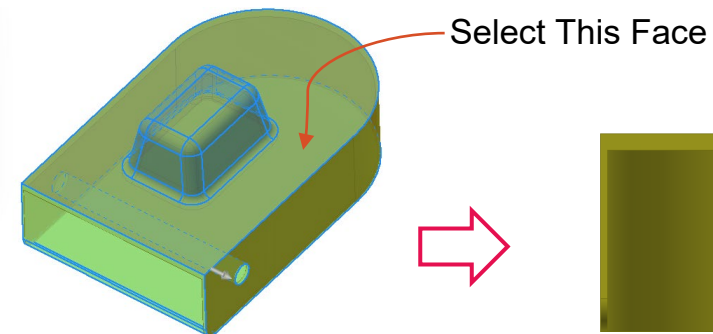
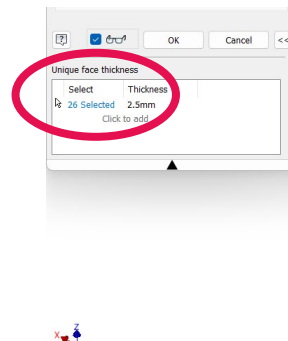
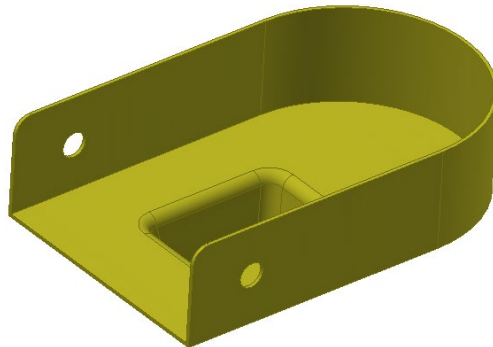


6. In the Feature Browser, Re-Select 'Shell' Feature, Right Click and select edit feature.

7. Click on 'Double Arrow' button in the dialog box menu.

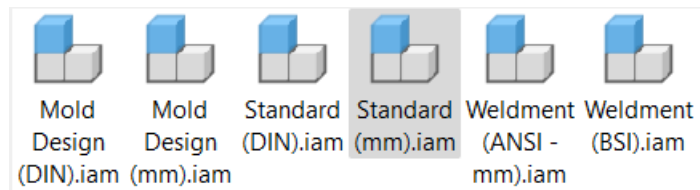


5. You will see, the changes is reflected to the model. All Surfaces are 1mm thick.

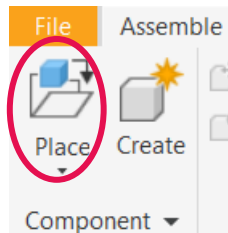


Assembly Basic (Hands On #16)

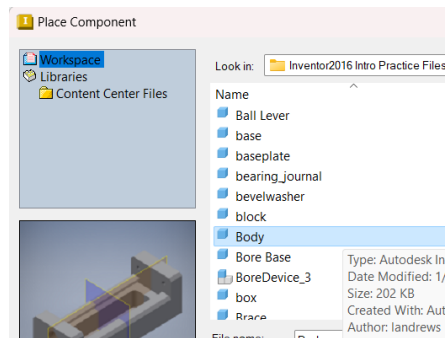
1. Create 'New' Assembly file using 'Standard.iam' template. Inventor will bring you into Assembly Environment.



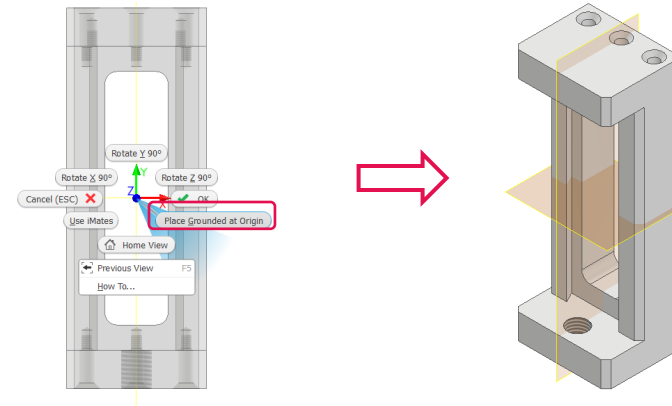
2. In the 'Component' Panel, click 'Place' to insert all required files/parts.



3. Find a file called 'Body.ipt' in the 'Place' dialog box menu, Double Click on the file.

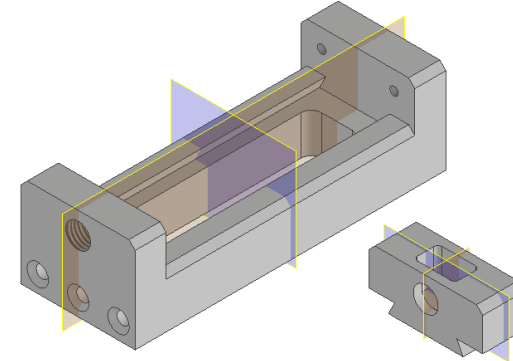


4. In the graphic window, Right Click, select 'Place Grounded at Origin' option. The part will be FIX to the (0,0,0) coordinate and can't be moved.



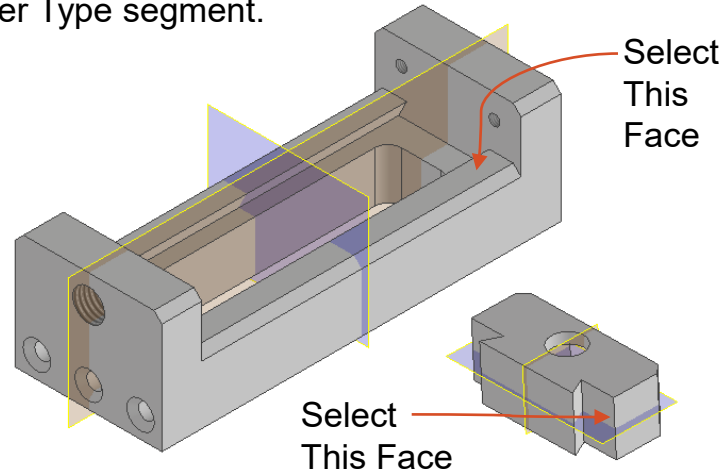
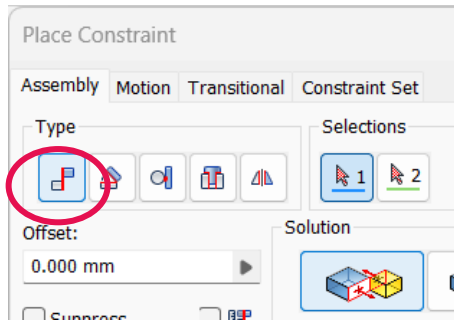
5. Using the same method, insert another part called 'MovingJaw.ipt'. **Don't** use 'Place Grounded At Origin'. Just click anywhere near the Body.ipt.

Rotate the View until the view is the same as in the picture below.

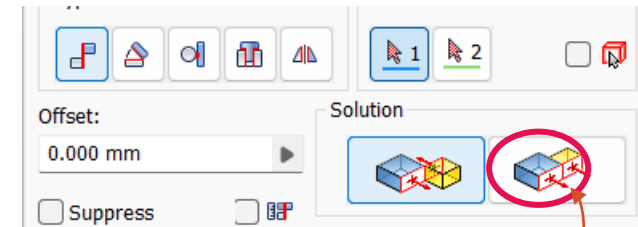


Assembly Basic (Hands On #16)

6. In the 'Relationship' Panel, click 'Constraint'. The 'Constraint' dialog box will appear. Select 'Mate' Option under Type segment.

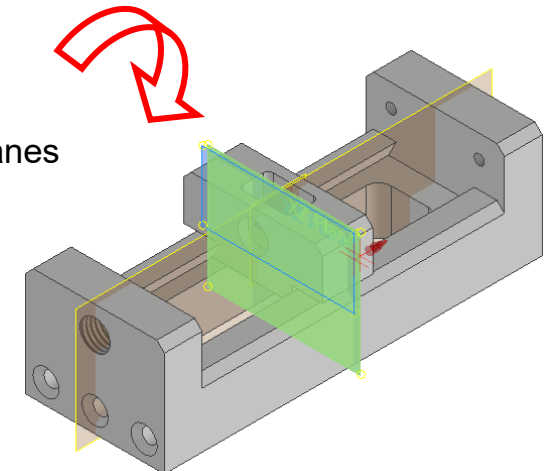
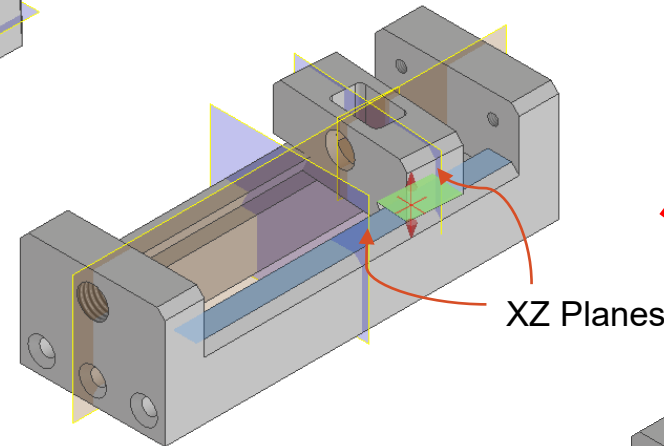
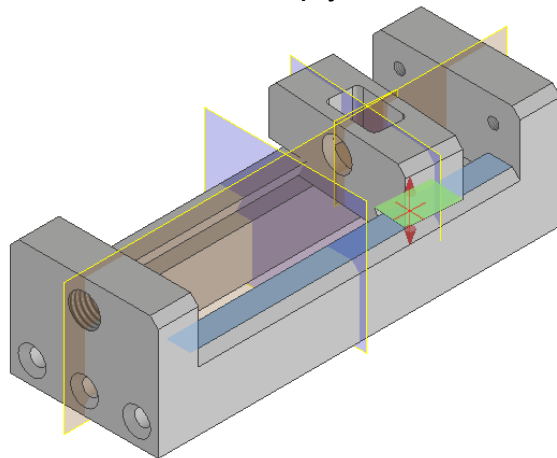


7. Note that the 'MovingJaw.ipt' part is still movable. Click 'Contraint' command, Select 'Flush' in to 'Solution segment. Maintain 'Mate' type. Select the XZ Plane of each Parts (Body & MovingJaw).



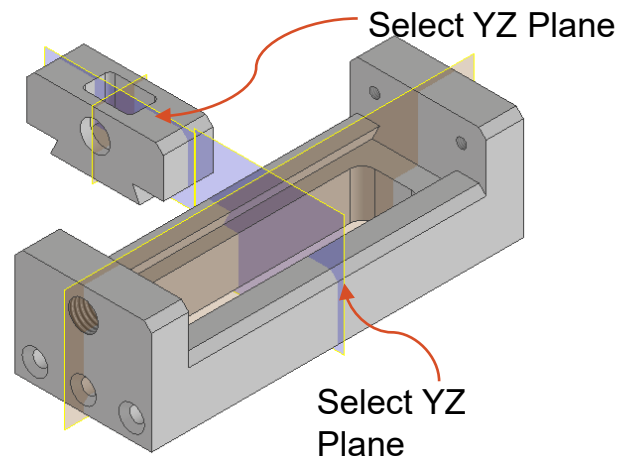
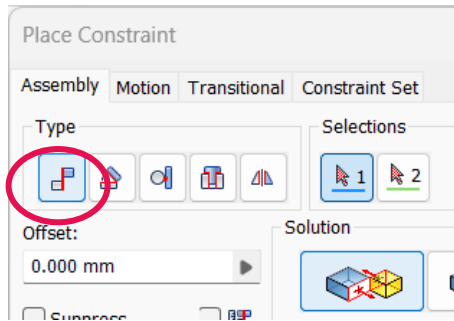
Flush

The part will moved until it complies with the constraint that we have applied.
Click OK

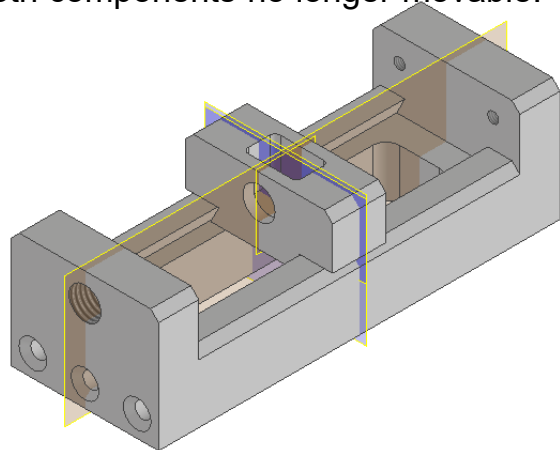


Assembly Basic (Hands On #16)

8. The 'MovingJaw.ipt' is still movable side to side. Using the same constraint method, select YZ planes of both components. Use 'Flush' selection.

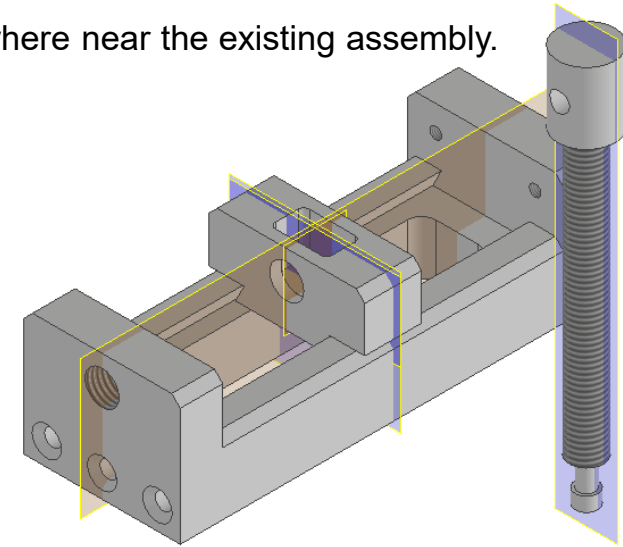


The part will moved until it complies with the constraint that we have applied. Click OK. Both components no longer movable.



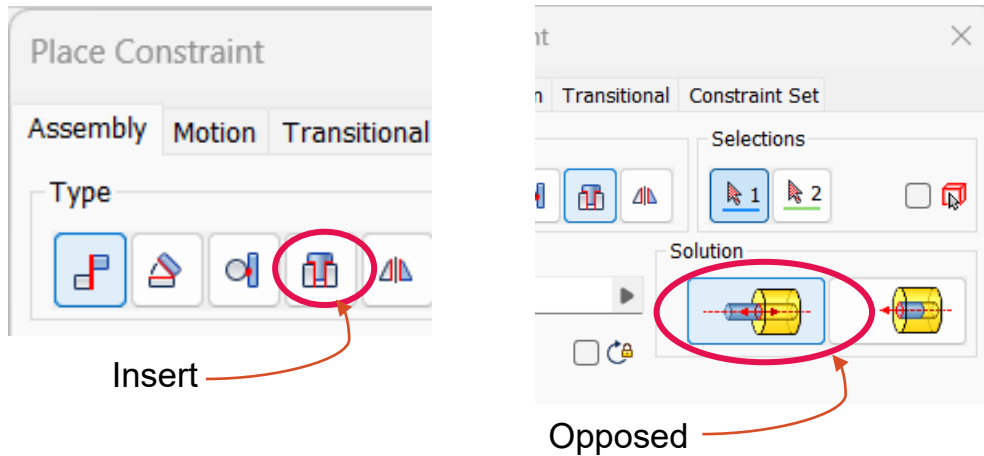
9. Insert another part called 'Spindle.ipt' into the assembly environment. (Use same method as 'Body.ipt' & 'MovingJaw.ipt')

Place anywhere near the existing assembly.

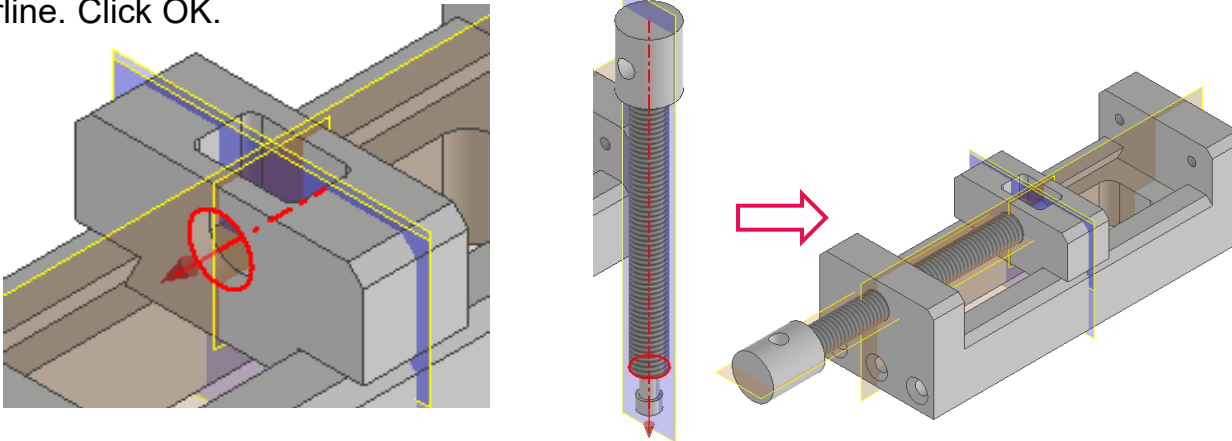


Assembly Basic (Hands On #16)

10. Click 'Constraint' command. In the dialog box menu, select 'Insert' option in the 'Type' segment. Select 'Opposed' in the Solution segment.

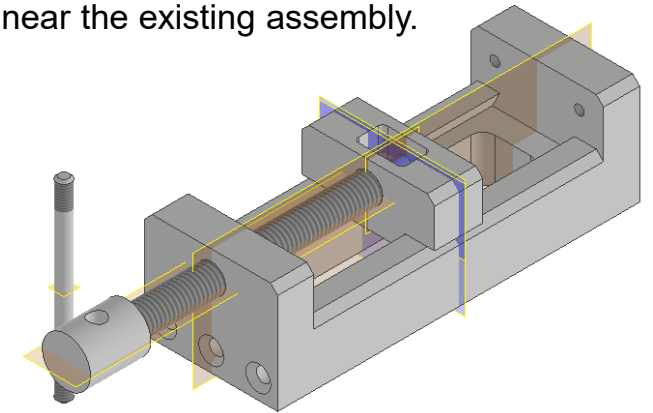


Select the cylindrical wall of 'MovingJaw' hole. A centerline will appear, select the centerline. Click OK.

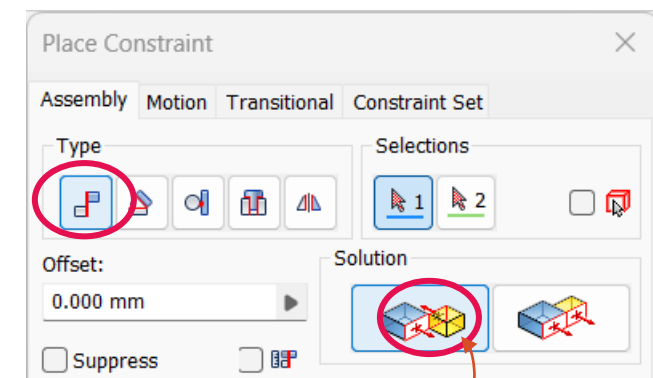


11. Insert another part called 'Pin.ipt' into the assembly environment. (Use same method as 'Body.ipt' & 'MovingJaw.ipt')

Place anywhere near the existing assembly.



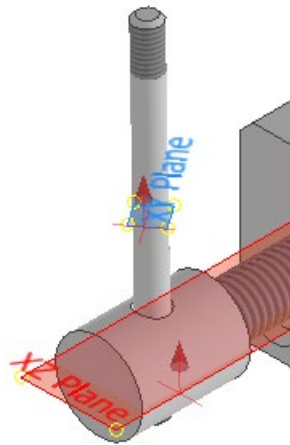
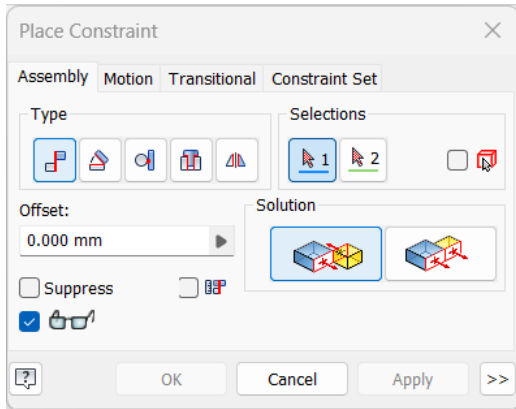
12. Click 'Constraint' command. In the dialog box menu, select 'Mate' option in the 'Type' segment. Use 'Flush'.



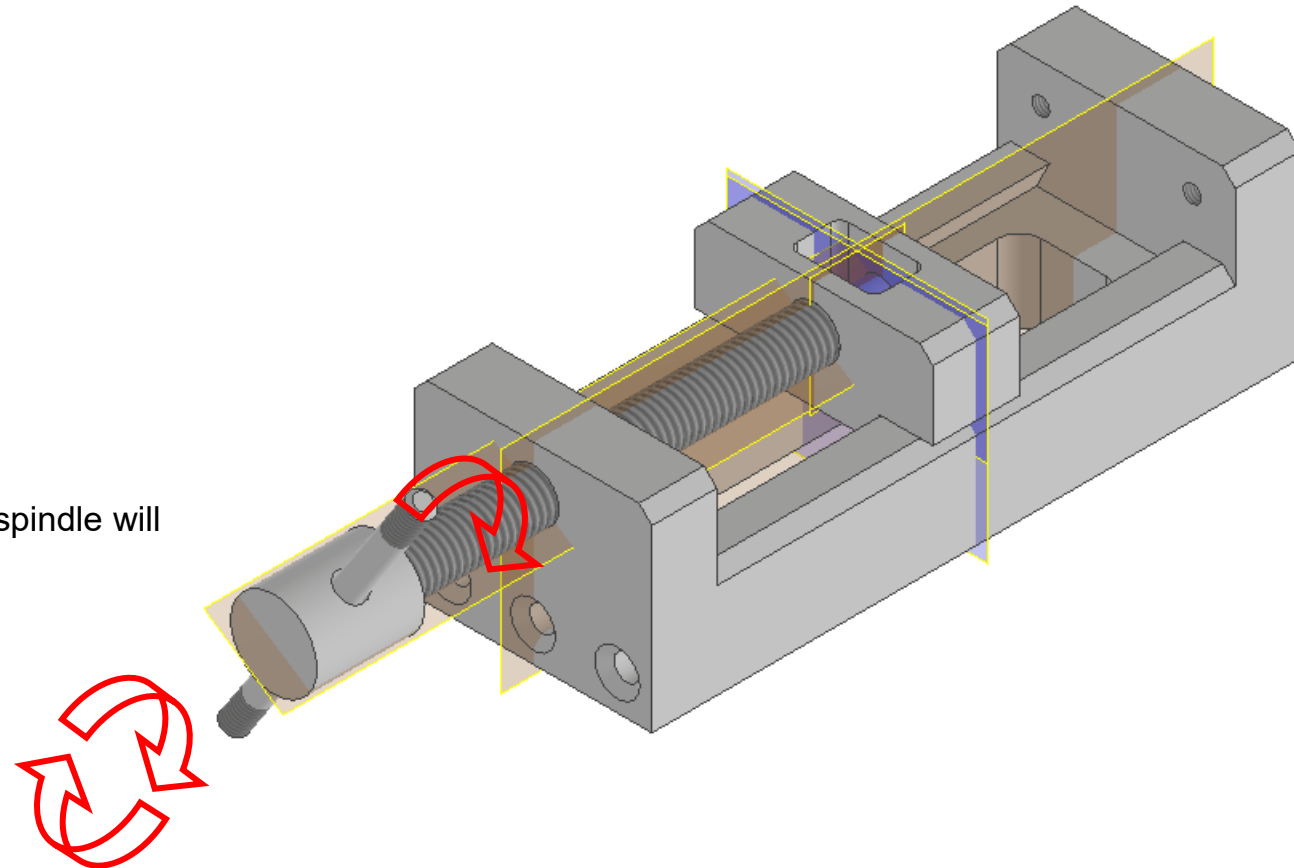
Flush

Assembly Basic (Hands On #16)

13. Repeat 'Constraint' command, 'Use 'Mate' type and 'Flush' selection, Select XZ plane for 'Spindle.ipt', and XY plane for 'Pin.ipt'.



The Assembly is now completed. Notice that if you move the pin, the spindle will also moved in rotational movement.

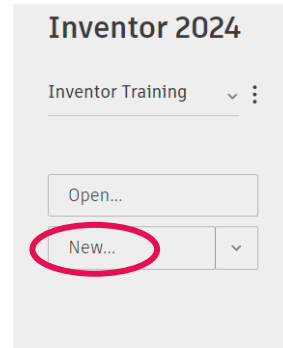


Assembly (Hands On #17 - Joint)

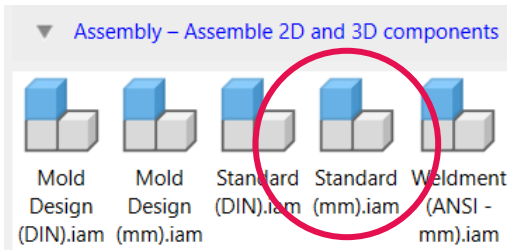
1. Click on 'New' button located on Quick Access Toolbar / Project Browser



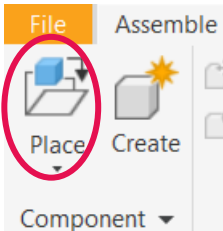
or



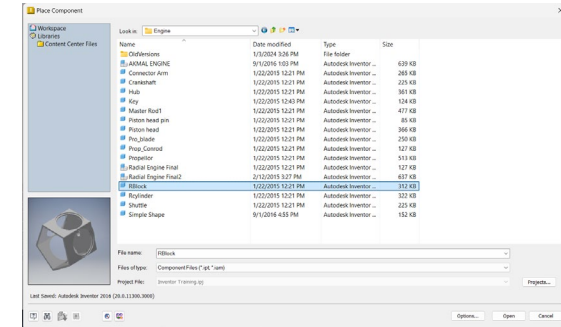
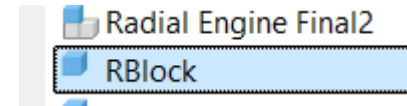
2. Select Standard.iam (mm) template.



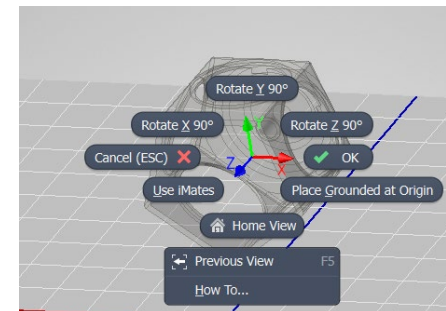
3. In the 'Component' Panel, click 'Place' to insert all required files/parts.



4. Find a file called 'RBlock.ipt' in the 'Place' dialog box menu, Double Click on the file.



5. While having the part hovering around in the assembly environment, right click on an empty space, select 'Place Grounded at Origin'.



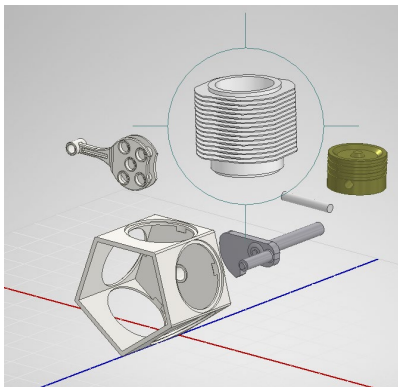
This way, will make our part **FIXED** at the origin by 0,0,0 coordinate and at the same time, unmovable.

Assembly (Hands On #17 - Joint)

6. Repeat the step (4) to insert the rest of the parts (**Crankshaft.ipt**, **MasterRod.ipt**, **Piston Head.ipt**, **Piston Head Pin.ipt**, **Rcylinder.ipt**). Do not use 'Place Grounded At Origin'.

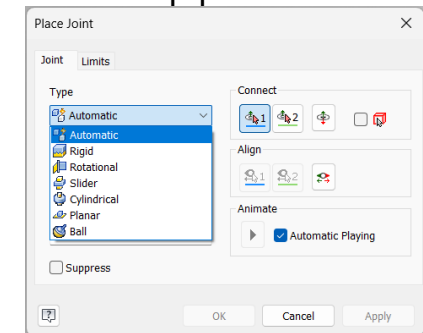
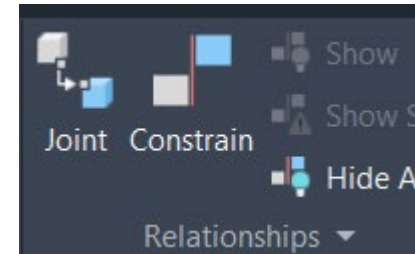
Component Path	Modified	Author	Size
Crankshaft	1/22/2015 12:21 PM	Autodesk Inventor ...	225 KB
Hub	1/22/2015 12:21 PM	Autodesk Inventor ...	361 KB
Key	1/22/2015 12:43 PM	Autodesk Inventor ...	124 KB
Master Rod1	1/22/2015 12:21 PM	Autodesk Inventor ...	477 KB
Piston head pin	1/22/2015 12:21 PM	Autodesk Inventor ...	85 KB
Piston head	1/22/2015 12:21 PM	Autodesk Inventor ...	366 KB
Pro_blade	1/22/2015 12:21 PM	Autodesk Inventor ...	250 KB
Prop_Conrod	1/22/2015 12:21 PM	Autodesk Inventor ...	127 KB
Propellor	1/22/2015 12:21 PM	Autodesk Inventor ...	513 KB
Radial Engine Final	1/22/2015 12:21 PM	Autodesk Inventor ...	127 KB
Radial Engine Final2	2/12/2015 3:27 PM	Autodesk Inventor ...	637 KB
RBlock	1/22/2015 12:21 PM	Autodesk Inventor ...	312 KB
Rcylinder	1/22/2015 12:21 PM	Autodesk Inventor ...	322 KB
Shuttle	1/22/2015 12:21 PM	Autodesk Inventor ...	225 KB

7. Arrange each part away from each other so that we can pick them easily. Use 'Rotate Part' command (or Press G button on Keyboard) to rotate the parts if necessary.



8. We want to assemble the Crankshaft into the RBlock. The Crankshaft has to be able to rotate.

Find a command 'Joint' inside of Relationship panel.



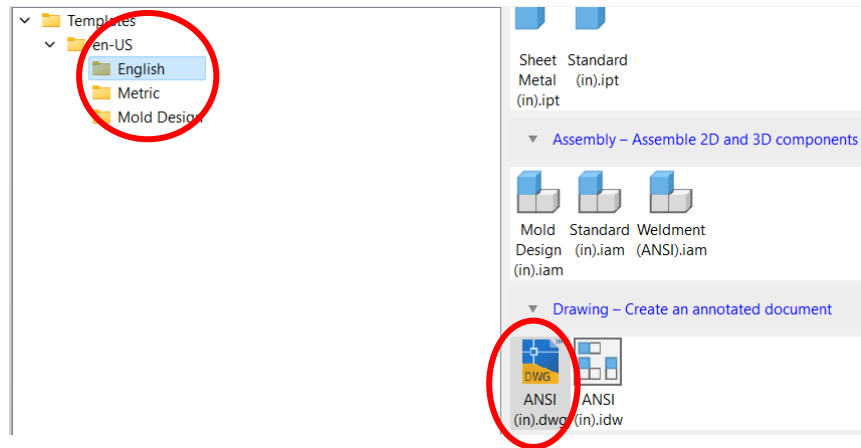
This command/function allows you to create specific mechanism while putting constraint/relationship between the part.

There will be several different type of Joint you may choose depending on the mechanism. There is also Automatic type of Joint.

9. Please follow the instruction from the trainer to complete the assembly.

Drawing Basic (Hands On #18)

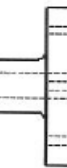
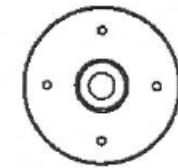
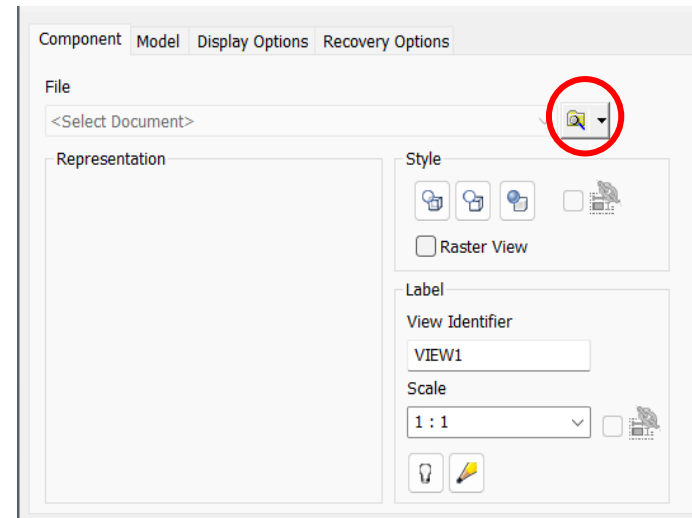
1. Start 'New' drawing using the 'ANSI (in).dwg' template.



2. Click 'Base' command in the 'Place View' Tab. A dialog box will appear.

Click 'Open' command within the dialog box, select and open 'relation.ipt' part/file

Hover cursor into the drawing area to switch between different views. Click the spot that you want to place the views.

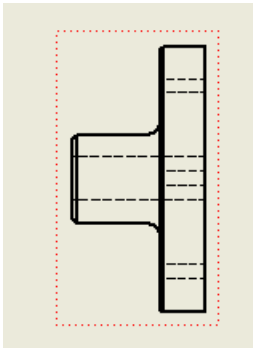
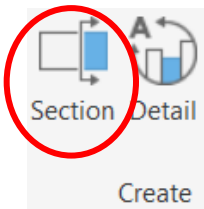


Create all the view accordingly as shown in the picture above. Set the scale to 2:1.

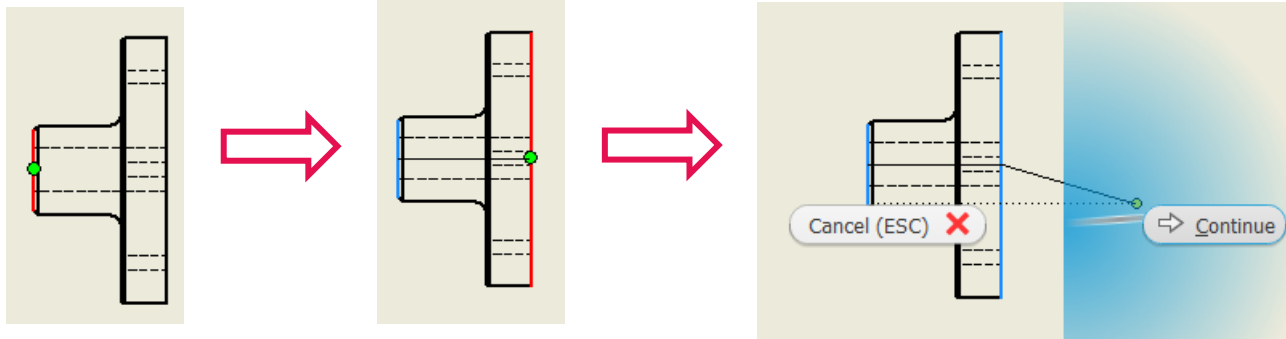
Drawing Basic (Hands On #18)

3. Click 'Section' command in the 'Create' Panel.

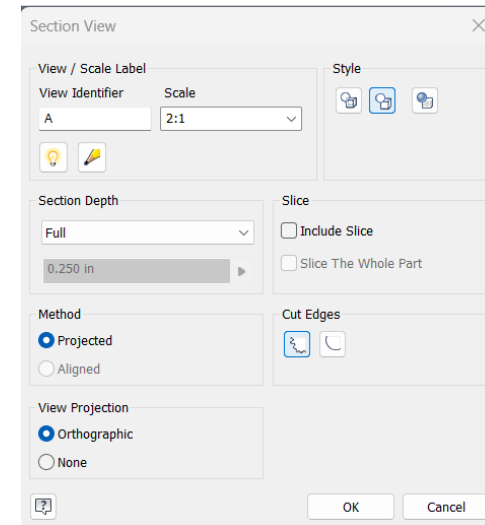
Select 'Side' view in the drawing area as shown in picture.



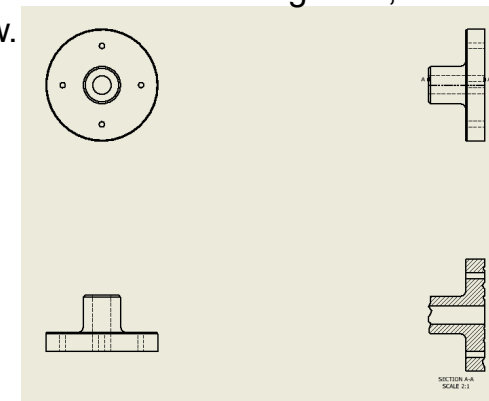
4. Select First Point and Second Point as shown in picture below. Right click, click on Continue.



5. A dialog box menu of 'Section' command will appear.



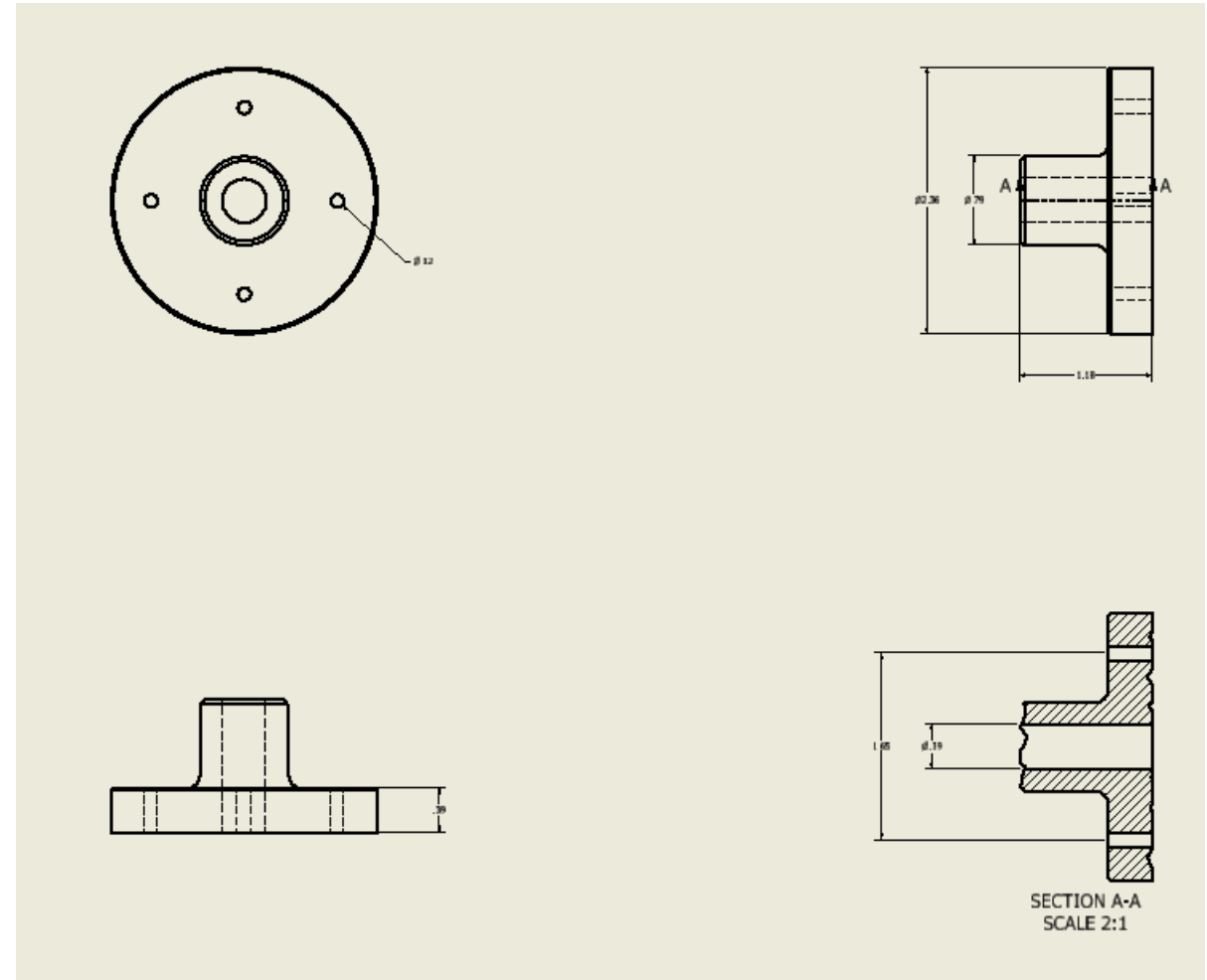
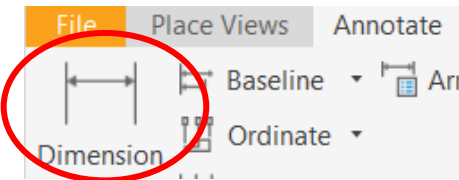
6. Hover cursor within the drawing area, and click to place the section view.



Drawing Basic (Hands On #18)

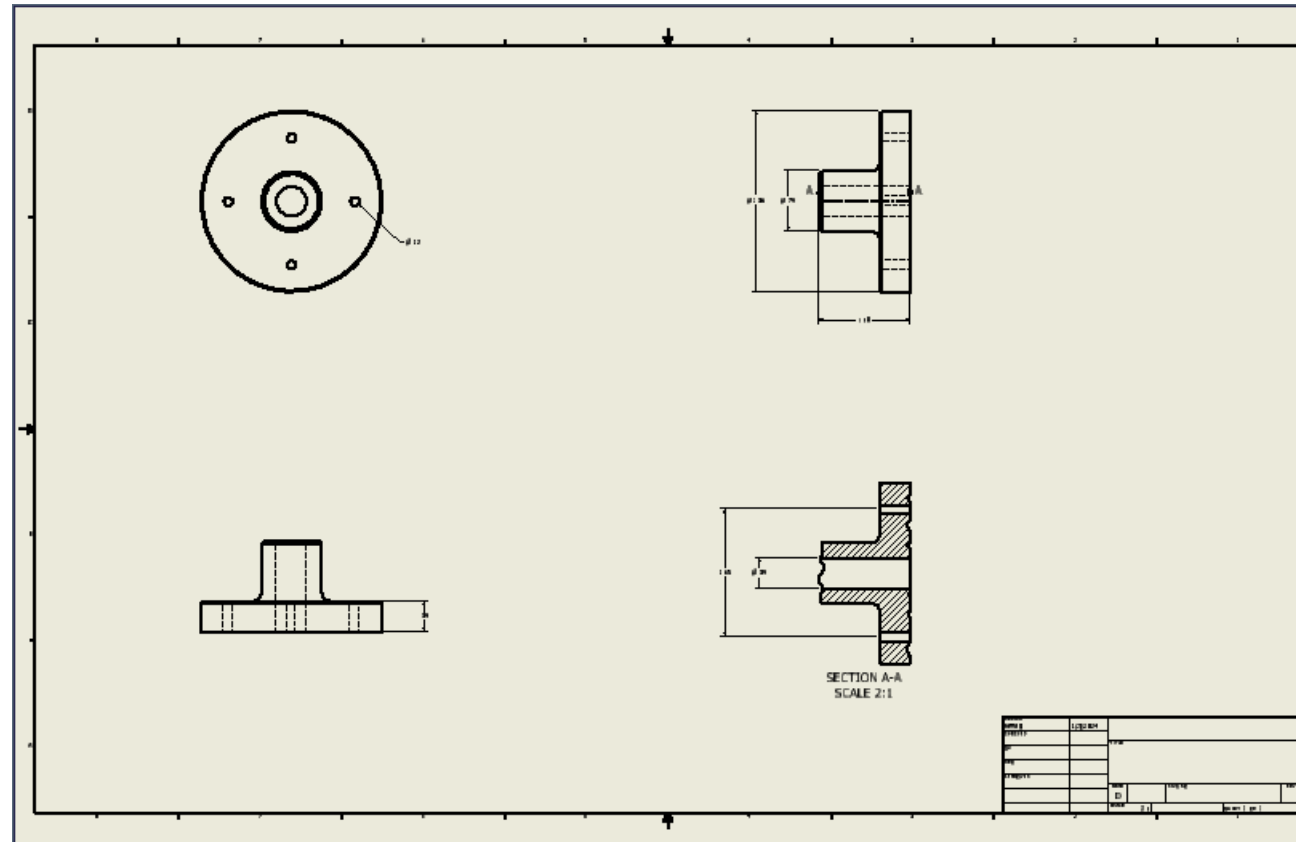
7. Click on 'Dimension' in 'Annotate' Tab. Apply all the dimension according to the picture below.

The method for applying Dimension in drawing, is the same as in Sketch environment.



Drawing Basic (Hands On #18)

8. The drawing is now Completed.



End Of Training

Thank You

Aiman

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