AutoCAD 2016 Introduction.
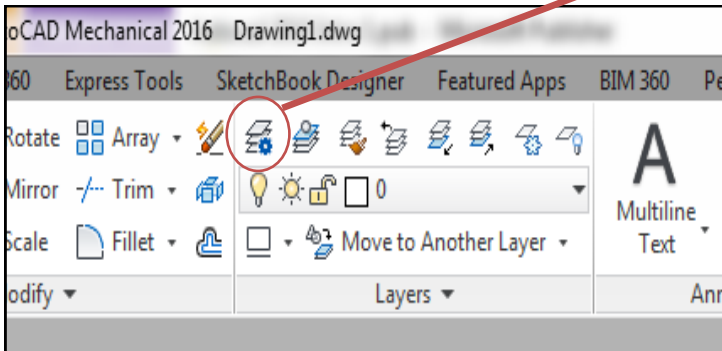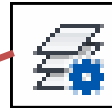
Create 2D drawings in AutoCAD 2016.
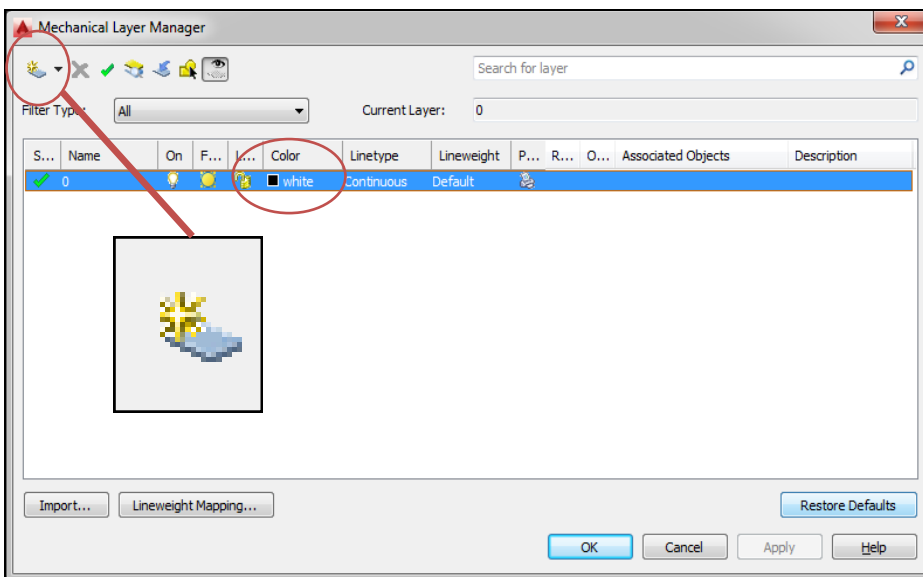
Step # 1. Launch AutoCAD 2016.

Set up some default layers.

On the "**Home tab**" -- "**Layers panel**" - "**Layer Manager**"

(See help "**Layers**")

See Fig 1.0

FIG 1.0

Step # 2. Create some new layers.

Click on the "New Layer" button shown in figure 2.0. A new layer entry will be displayed. Name the layer "**Laser Cut**"

Next click on the "**Colour**" , a standard Windows Colour palette as shown in Figure 3.0 will be displayed.

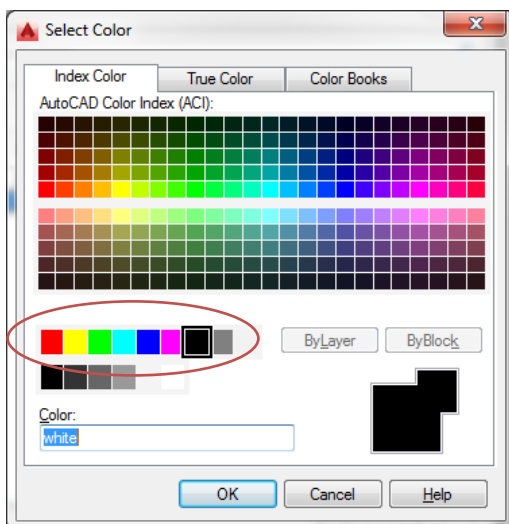Select "**Red**" and OK the dialog box.

See Fig 2.0

FIG 2.0

Step # 3.

Repeat the process to create a second layer called "**Laser Engrave**".

Leave the colour setting as "**Black**"

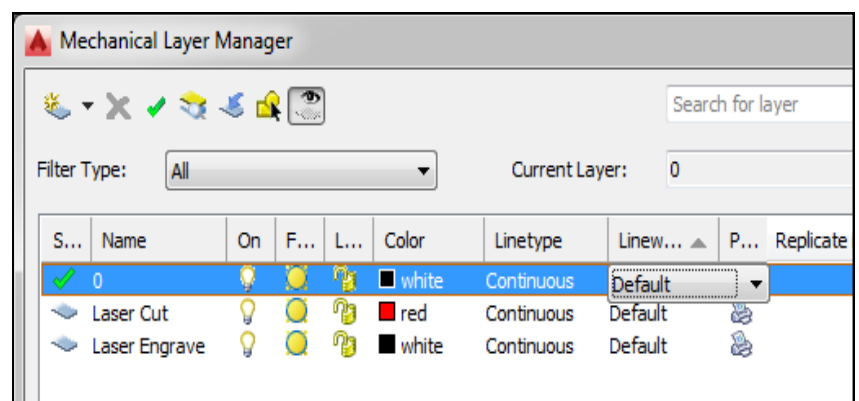OK the Colour dialog box and then the Layer Manager Dialog box.

See Fig 3.0

FIG 3.0

After the new layers are created, it should look like the dialog box in Figure 4.0   **FIG 4.0**

Step # 4. Setting the Object Snaps.

From the "**Status Bar**" select the "**Down Arrow**" , a list of available object snaps will be displayed.   See Fig 5.0
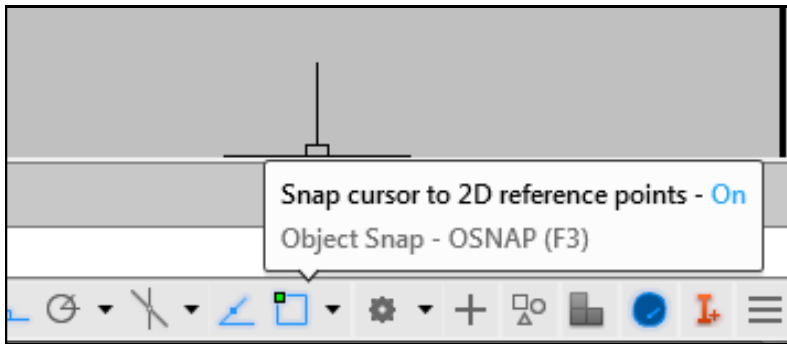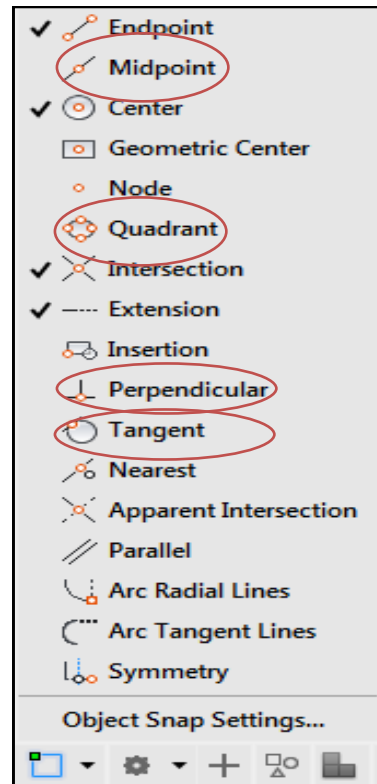

FIG 5.0

Adding extra Object Snaps.

Select "**Midpoint**" , "**Quadrant**" and "**Perpendicular**" and "**Tangent**".

Type "**Enter**" to terminate the selection.

See Fig 6.0

FIG 6.0

Step # 5. Saving a new Drawing Template.

Click on the "**Application Menu icon**" , Select "**Save As**" , and then select "**Drawing Template**".

Name the new Drawing template "SCCC" and click save. Follow the instructions in any displayed prompts.
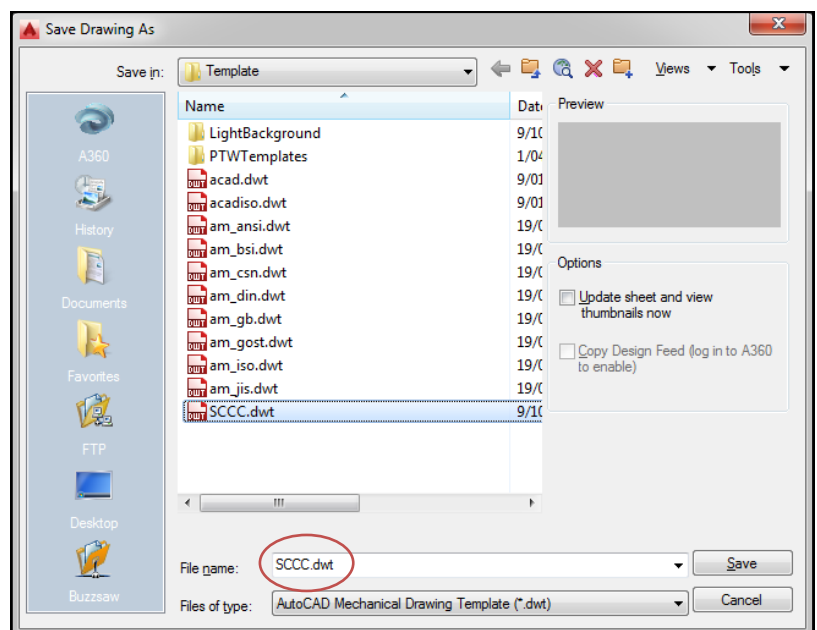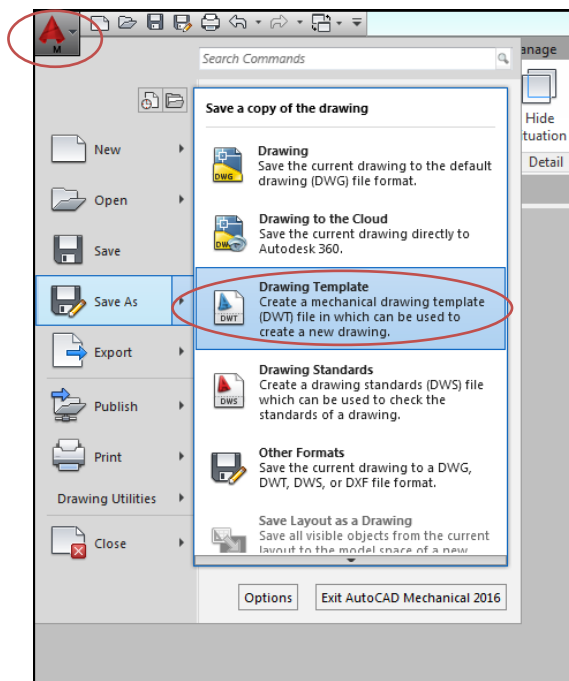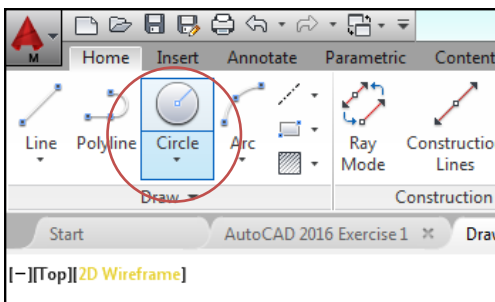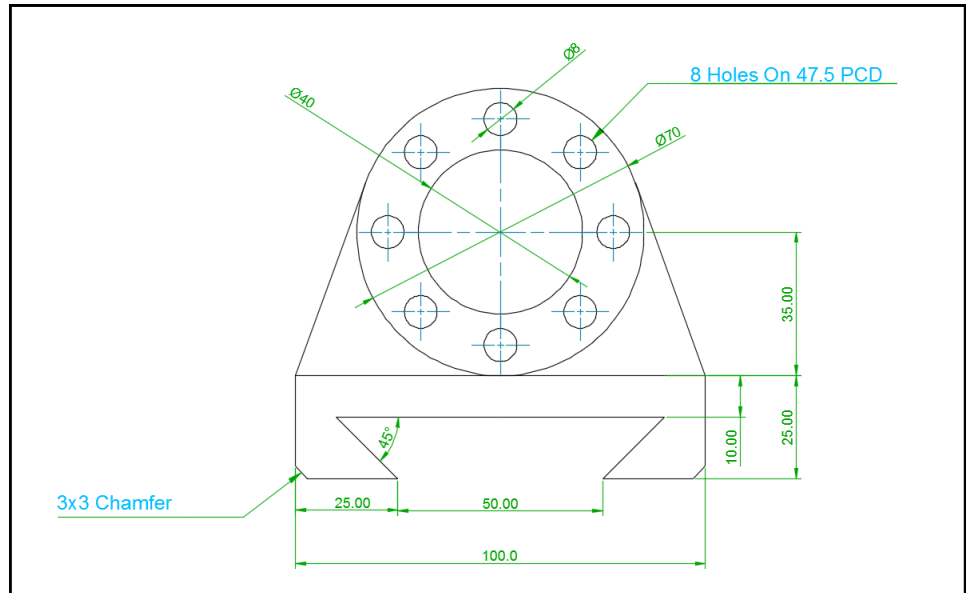
 See Fig 7.0

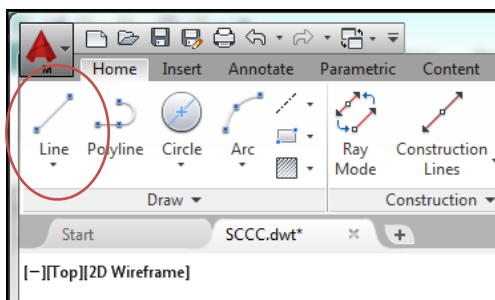**FIG 7.0**

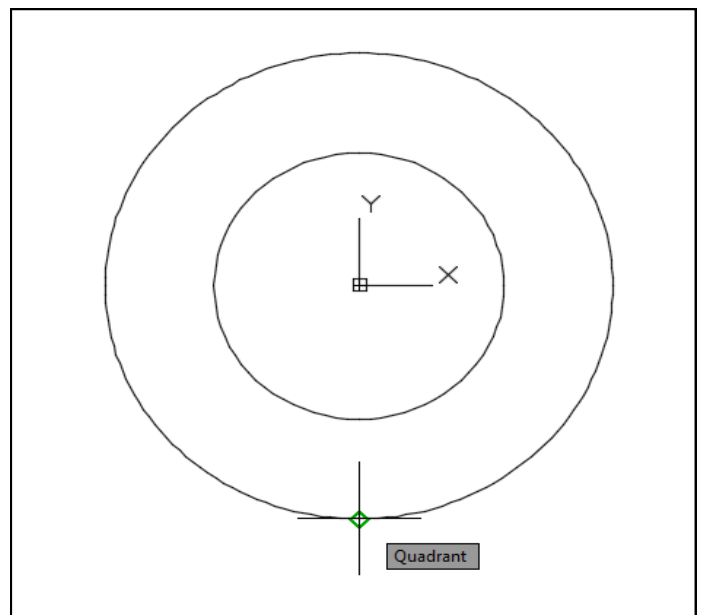**Step # 6. Producing a 2D Drawing.**

See Fig 8.0

FIG 8.0

On the "Home" Tab, select the "**Circle**" (Center, Diameter) command.

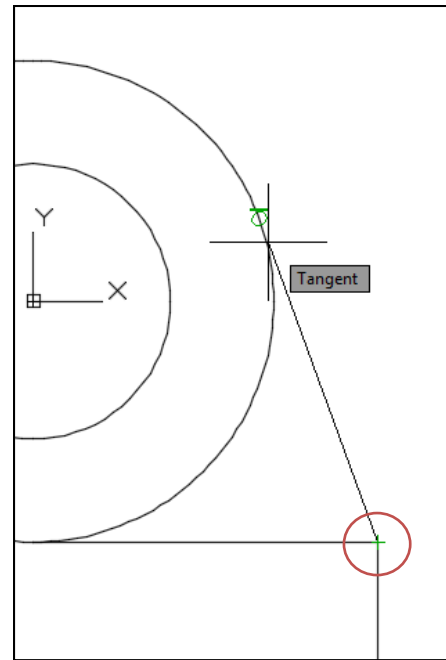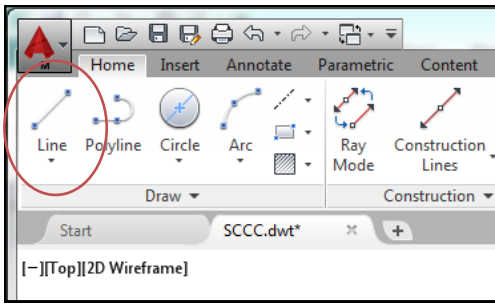1. Specify centre point for circle.   Type 0,0 *enter.*
2. Specify radius of circle or [Diameter]   Type 40 *enter.*
3. Press "Spacebar" or right-click to re-enter "**Circle**" command.
4. Specify centre point for circle.   Hover the cursor over the centre until the "Centre" object snap appears, left click.
5. Type "D" for diameter, *enter.*
6. Type 70 *enter.*

On the "**Home**" Tab, select the "**Line**" command.

1. Specify first point. Place the cursor over the lower quadrant until the snap appears.
2. Turn on "Ortho" mode by pressing "**F8**" or click the "Ortho" mode icon*.*
3. Move the cursor to the right and type "**50**" *enter*.
4. Move the cursor down and type "**25**" *enter*.
5. Move the cursor to the left and type "**25**" *enter*.
6. Type *enter* to terminate the command or right click and select *enter.*

On the "**Home**" Tab, select the "**Line"** command.

1. Specify first point. Place the cursor over the intersection point until the snap appears and click to select.

2. Turn off "Ortho" mode by pressing "**F8**" or click the "Ortho" mode icon**.**

3. Move the cursor the right side of the 70 diameter circle and a "Tangent" snap should appear, click to select.

4. Type *enter* to terminate the command or right click and select *enter.*

Setting "Polar" coordinates.

On the "**Status Bar**" click the "**Polar Tracking**" icon.

Select the **45 Degree** option.

Ensure the "**Polar Tracking**" mode is "**on**"
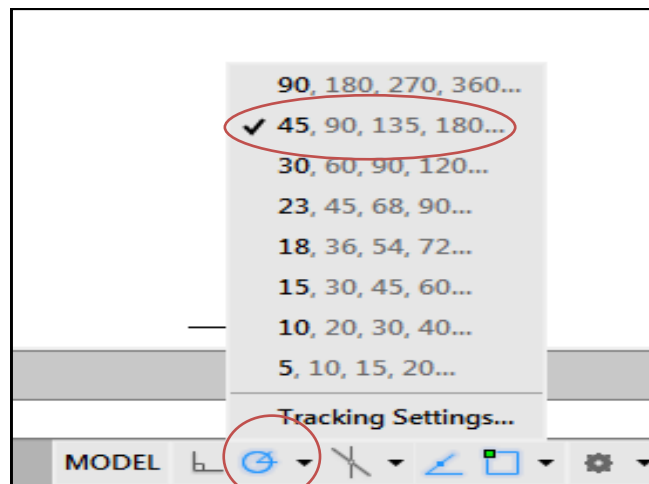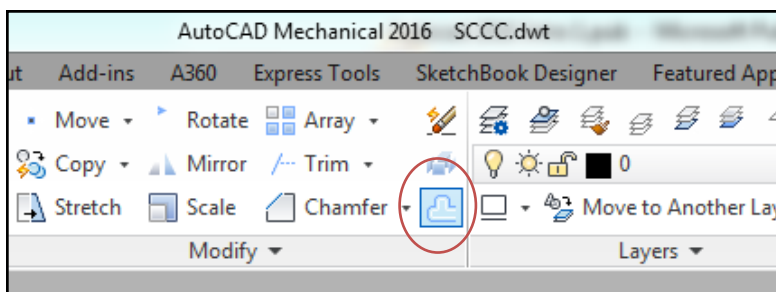
See Fig 9.0

FIG 9.0

On the "**Modify**" Tab, select the "**Offset**" command.

1. Specify offset distance.   Type "**10**" *enter*

2. Select object to offset   Select the horizontal line shown in Figure 10.0

3. Specify point on side to offset.   Move the cursor to below the horizontal line and click.

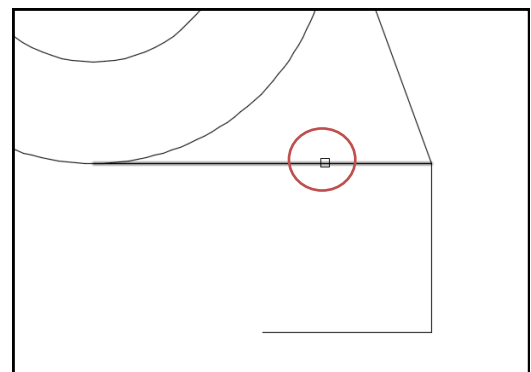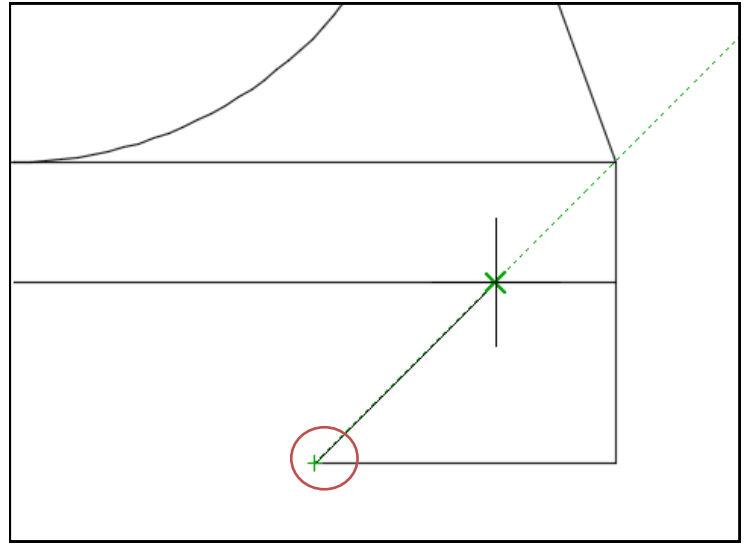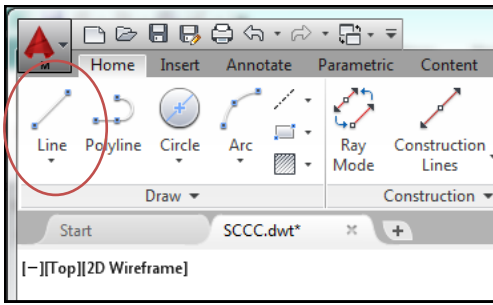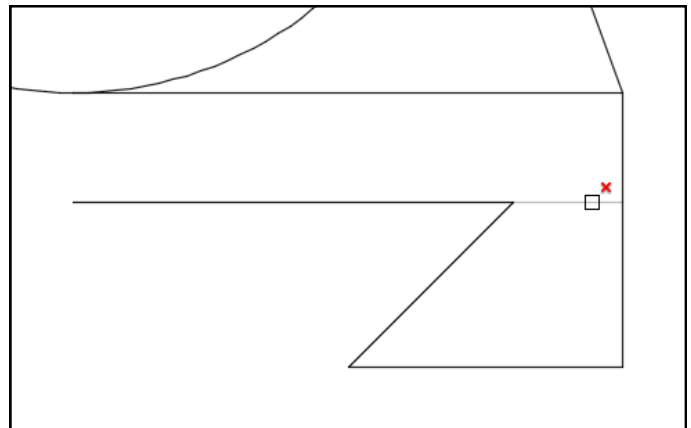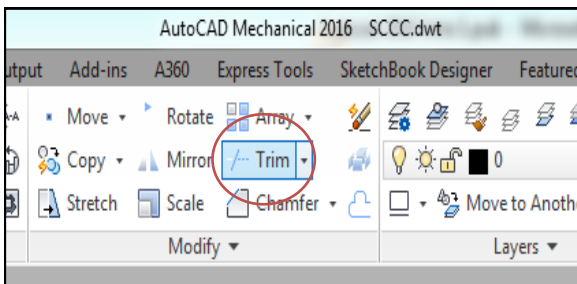4. Type *enter* to terminate the command or right click and select *enter.*

FIG 10.0

On the "**Home**" Tab, select the "**Line**" command.

1. Specify first point. Place the cursor over the endpoint until the snap appears and click to select.

2. Turn off "Ortho" mode by pressing "**F8**" or click the "Ortho" mode icon**.**

3. Move the cursor to the line offset in the previous command. A "Perpendicular" snap should appear, click to select.

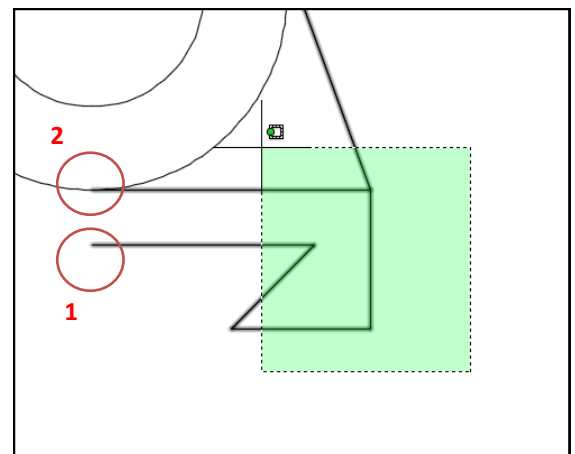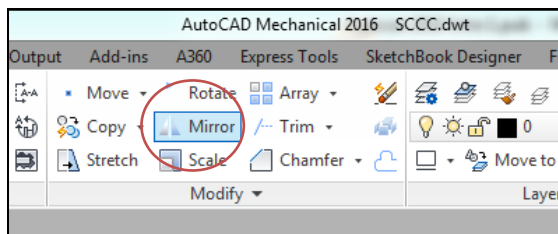4. Type *enter* to terminate the command or right click and select *enter.*

Note : Because Polar Tracking was on, the angle of the line will track to 45 degrees.

On the "**Modify**" Tab, select the "**Trim**" command.

1. Select objects or <select all>: Press *enter, or right click.*

2. Select object to trim.   Select the horizontal line shown in Figure 11.0

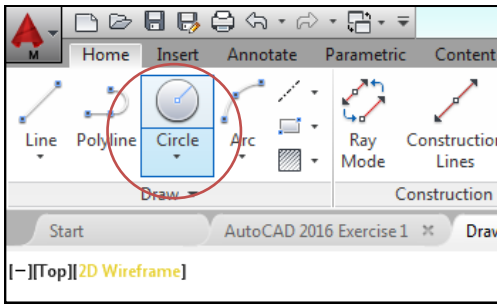3. Type *enter* to terminate the command or right click and select *enter.*

FIG 11.0

On the "**Modify**" Tab, select the "**Mirror**" command.

1. Select objects by crossing window, *right click to finish selection.*

2. Specify first point of mirror line. Select point "**1**"

3. Specify second point of mirror line. Select point "**2**"

4. Erase source objects? [Yes/No] <No>: *enter.*

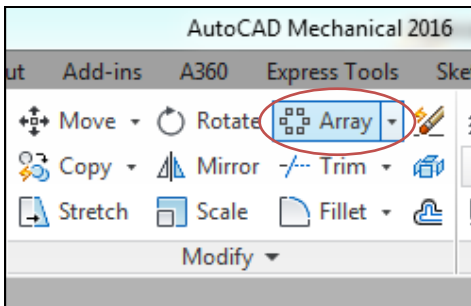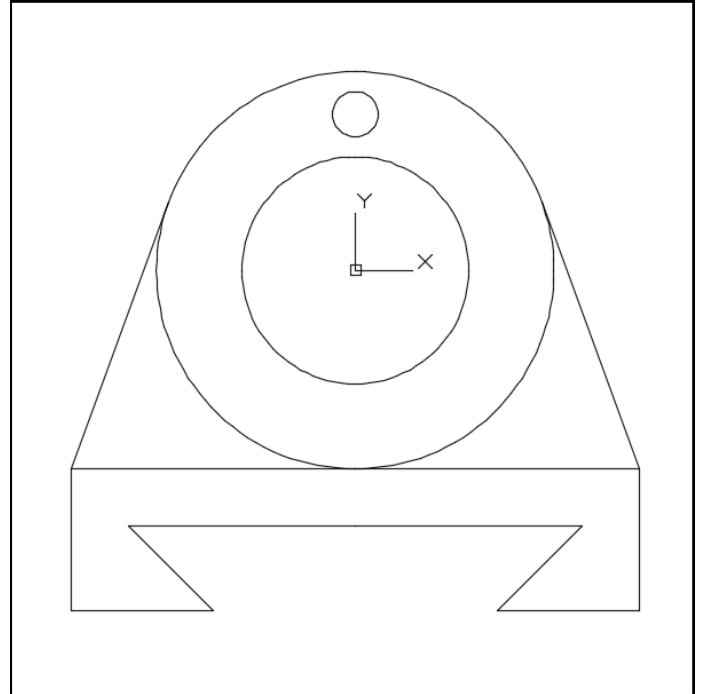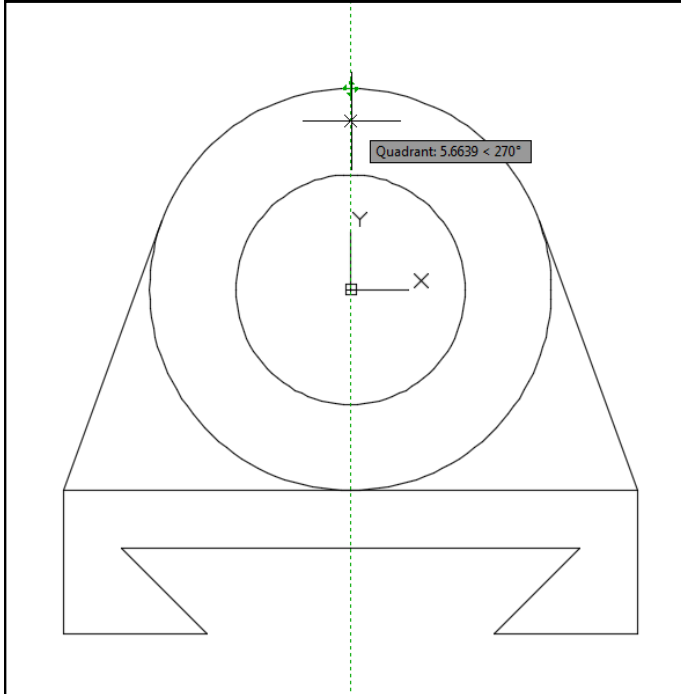3. Type *enter* to terminate the command or right click and select *enter.*

FIG 12.0

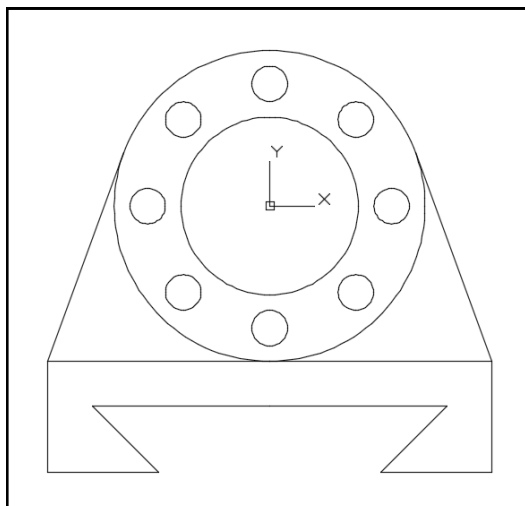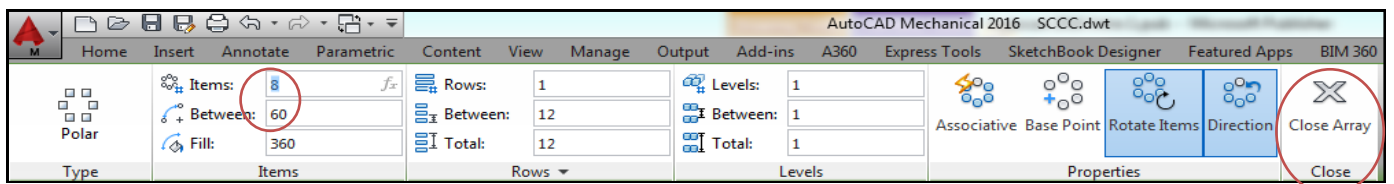On the "Home" Tab, select the "**Circle**" (Centre, Radius) command.

1. To specify centre point for circle.
2. Hover the cursor over the top quadrant, then move the cursor downwards.
3. Wait for a vertical "**Tracking**" line to appear*.
3. Type 7.5 *enter.*
4. Specify radius of circle or [Diameter]   Type 4.0 *enter.*   See figure 13.0
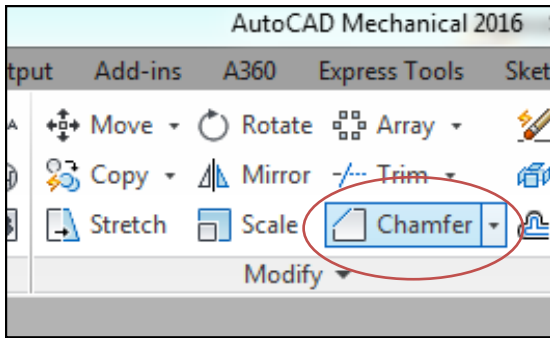
FIG 12.0

On the "**Modify**" Tab, select the "**Array**" command.

1. Select objects : Click on the 8mm circle, then *right click to finish selection.*
2. Specify center point of array .   Click on the centre of the large circle.
3. After clicking the centre of the large circle, the "**Array**" command ribbon is displayed.
4. Set the "**Number of Items**" to 8.
5. Type *enter*, or click on **"Close Array"**

The "Arrayed" circles.   See figure 13.0

FIG 13.0

On the "**Modify**" Tab, select the "**Chamfer**" command.

1. The "**Chamfer**" command ribbon will be displayed.
2. Type 3 in the first distance textbox and *enter.*
3. Type 3 in the second distance textbox and *enter.*
4. Select the 2 lines as shown in Figure 14.0
5. Repeat the process on the other corner.
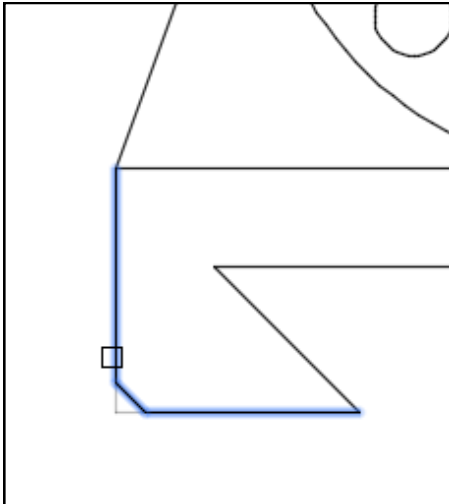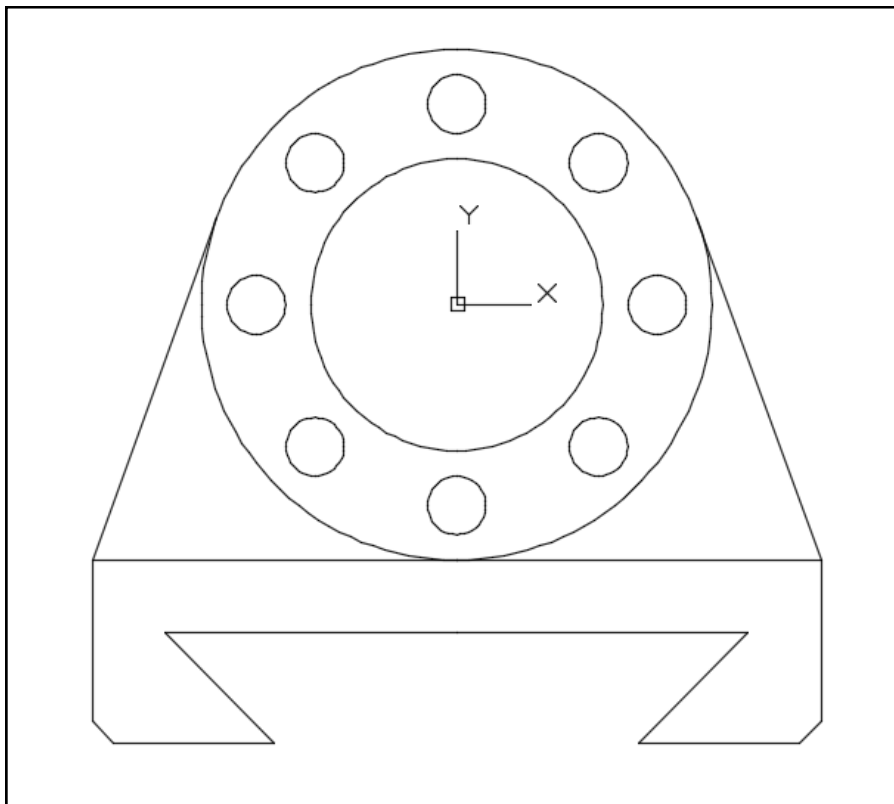6. Type *Escape* or right click and select *Cancel.*

FIG 14.0

Completed drawing.