



PDHonline Course G206 (6 PDH)

AUTOCAD 2-D BASICS

AutoCAD 2019 TO 2023

John R. Andrew, P.E.

PDH Center

2410 Dakota Lakes Drive
Herndon, VA 20171-2995

Phone: 703-478-6833
www.PDHonline.com
www.PDHcenter.org

An Approved Continuing Education Provider

AUTOCAD 2-D BASICS

Course Content

AutoCAD 2-D Basics is divided into 10 sections. Step-by-step illustrated examples show how to use tools to create all types of two dimensional engineering diagrams and drawings.

The lesson material below applies to all releases of AutoCAD from version 14 to 2014.

It is not necessary to have AutoCAD to study this lesson.

The AutoCAD drawing commands and tools described in this lesson are applicable to the: Mechanical, Structural, Industrial, Chemical, Electrical, and Civil engineering disciplines.

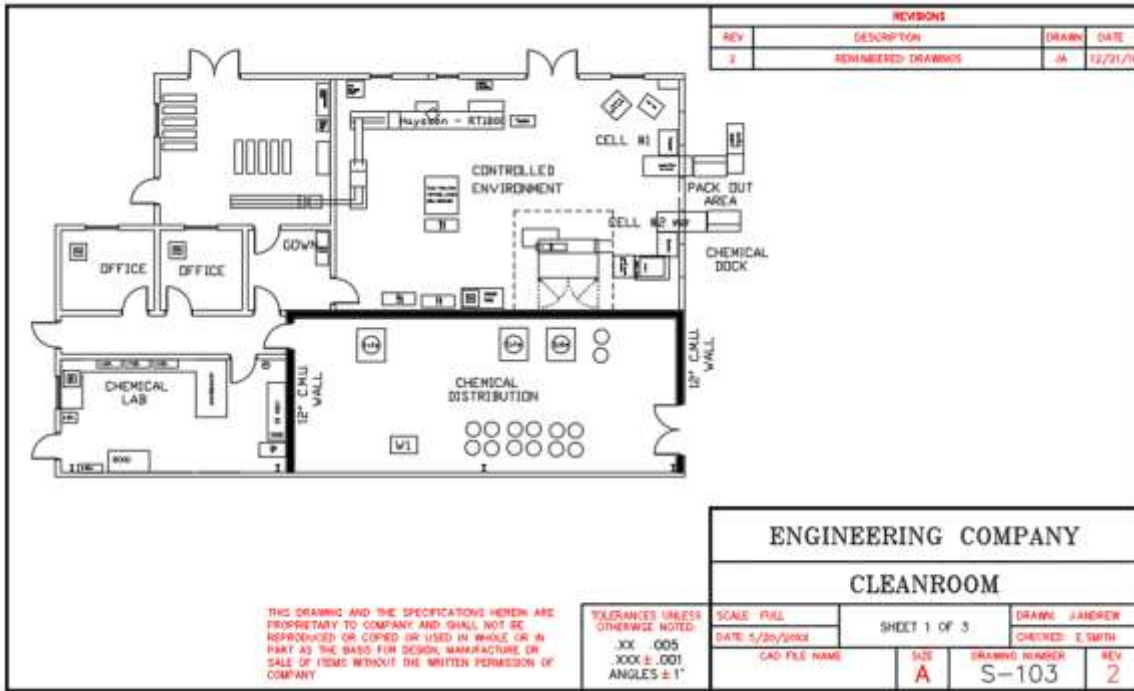
CONTENTS

- 1. AutoCAD Drawing Examples**
- 2. AutoCAD Toolbars**
- 3. Common AutoCAD Commands**
- 4. Flat Plate With Four Holes**
- 5. Generic Hub**
- 6. Bill of Materials**
- 7. Dimensional & Geometrical Tolerances**
- 8. Model Space and Layout**
- 9. Drawing Check List**
- 10. 2-D AutoCAD Command List**
- 11.Related Links**

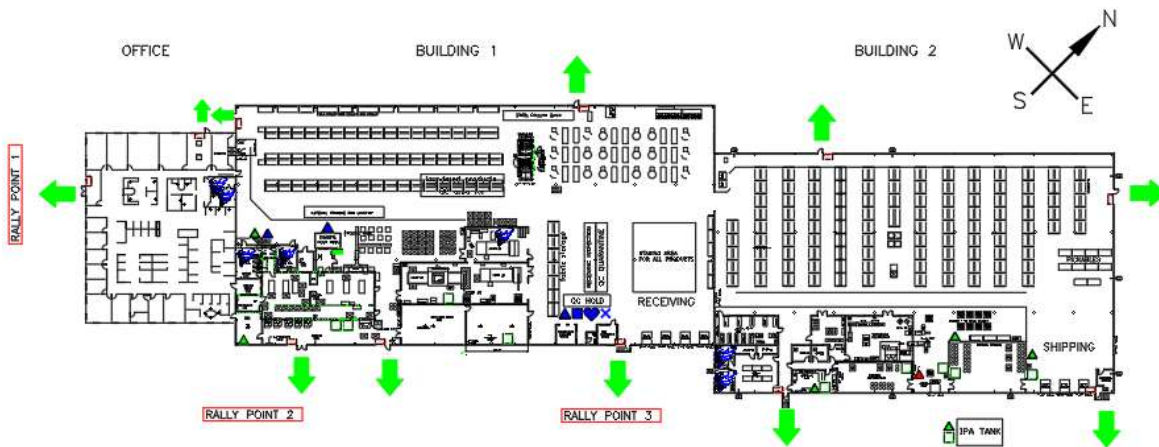
The AutoCAD and AutoCAD LT release 2021 manual has over 806 pages.

This lesson has only 98 pages but the most used drawing methods are described, enabling the reader to make all types of two dimensional engineering diagrams and drawings.

1. AutoCAD DRAWING EXAMPLES

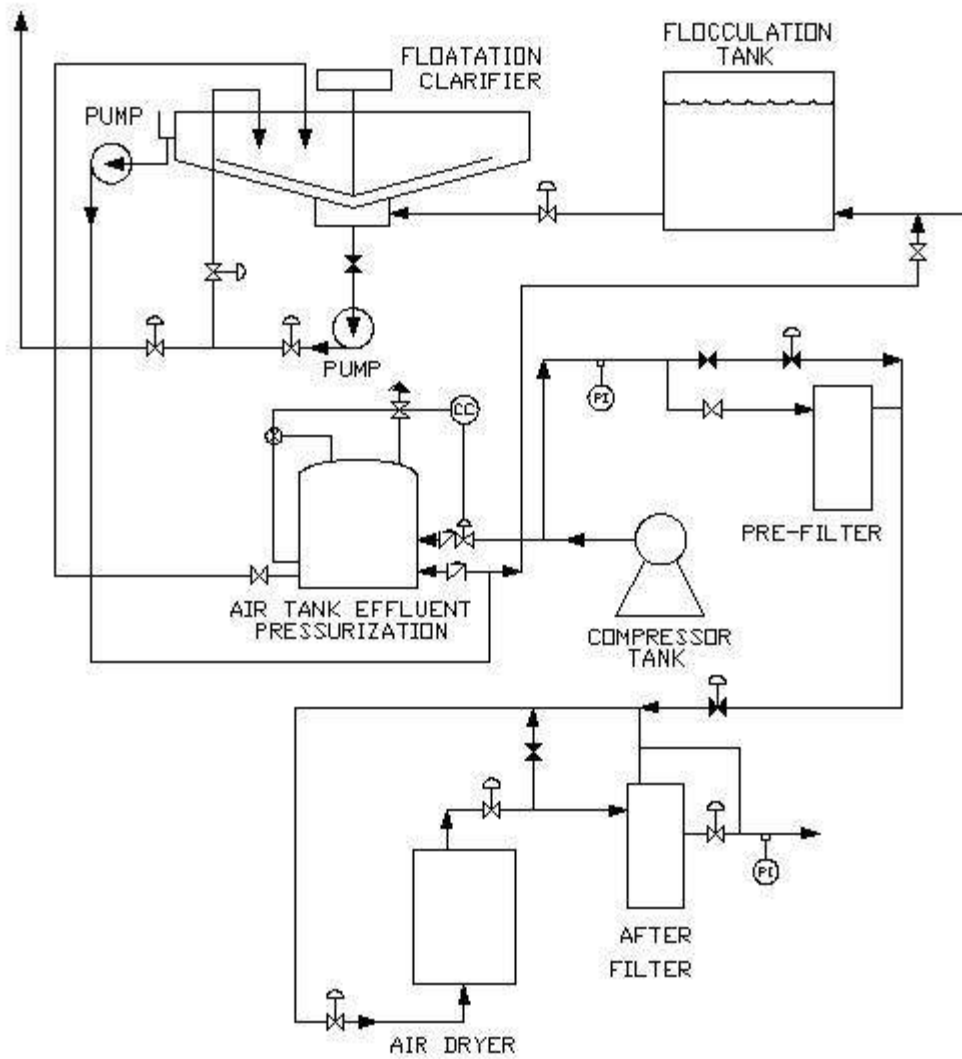


Building Plan



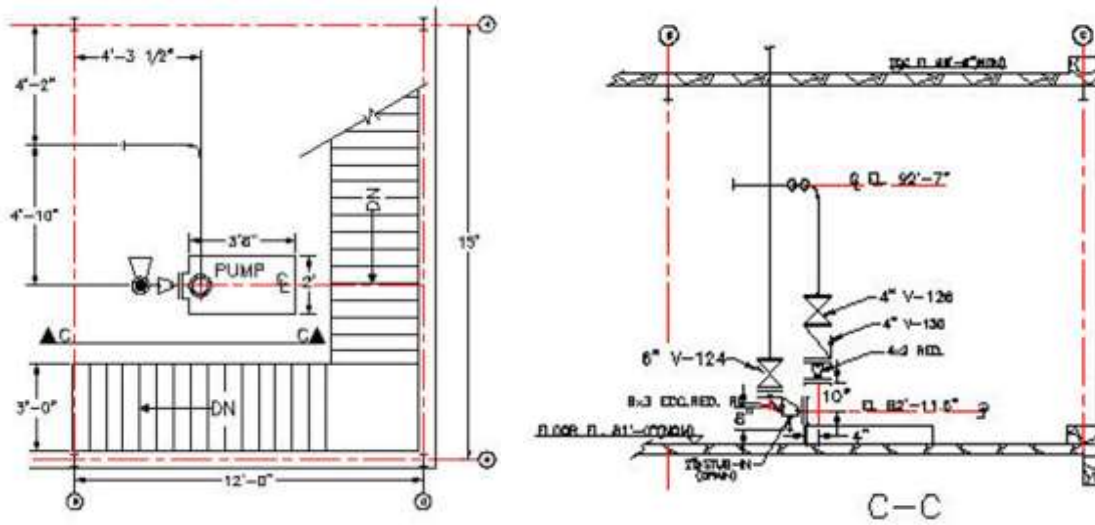
Building Plan

AutoCAD DRAWING EXAMPLES

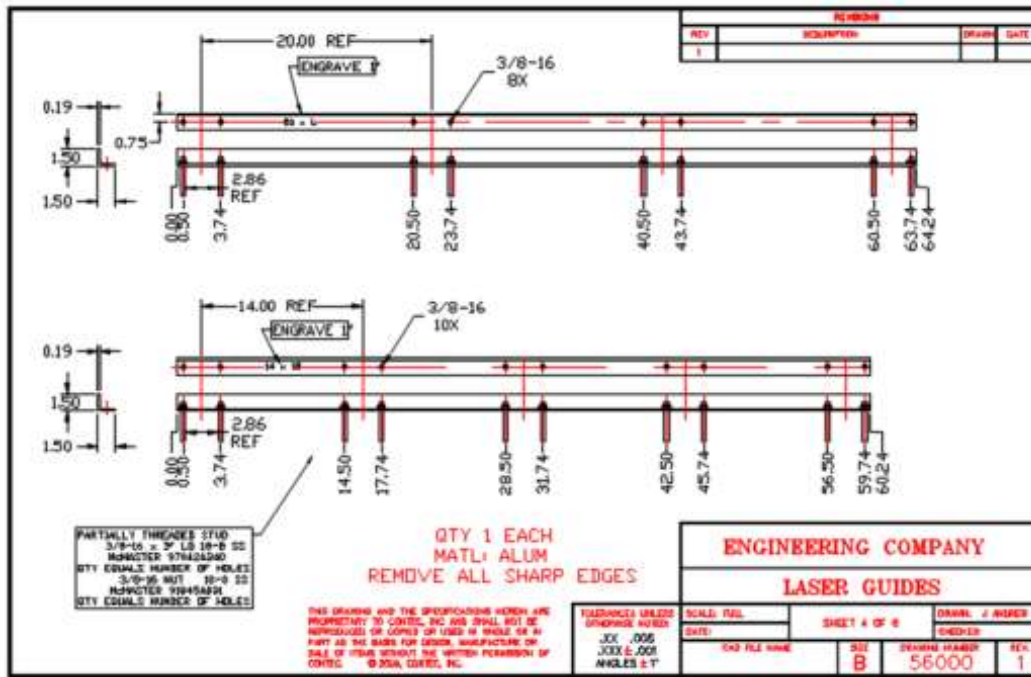


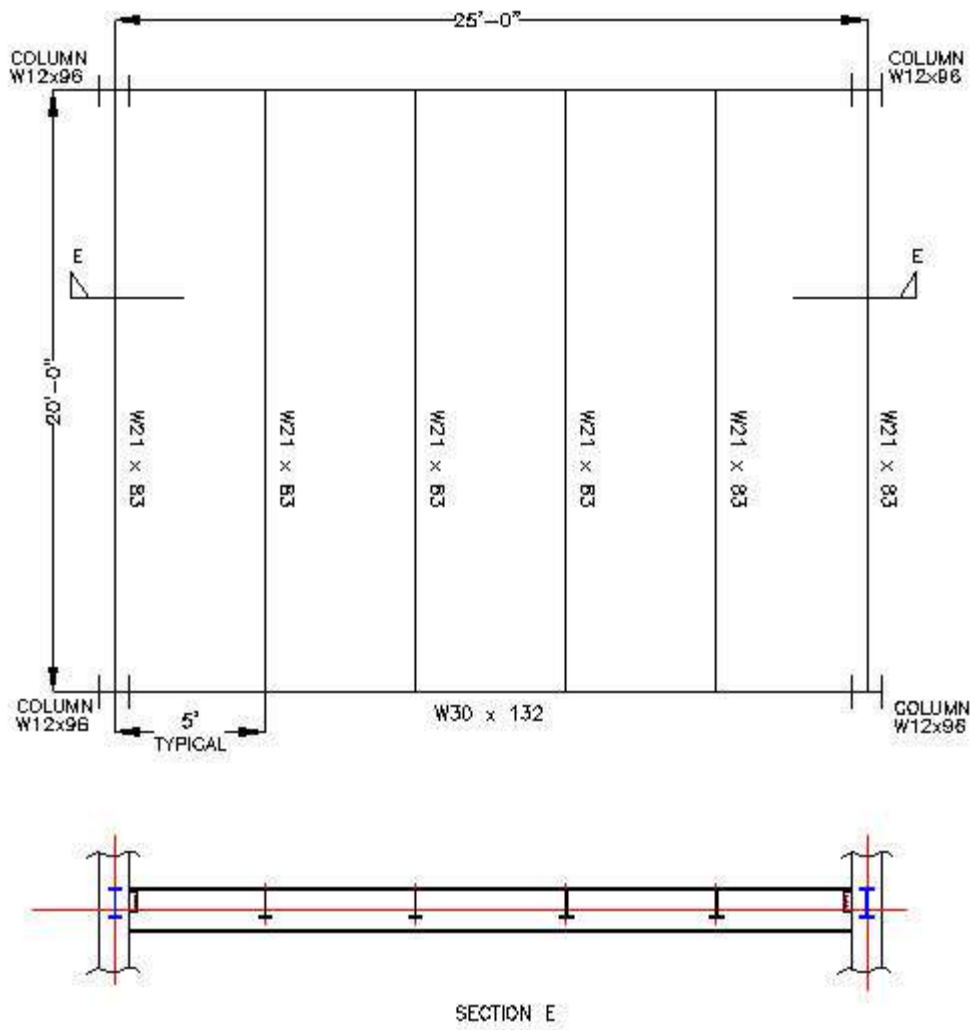
FLOW DIAGRAM

The flow diagram above was created using AutoCAD.



The piping equipment layout above is an AutoCAD drawing.

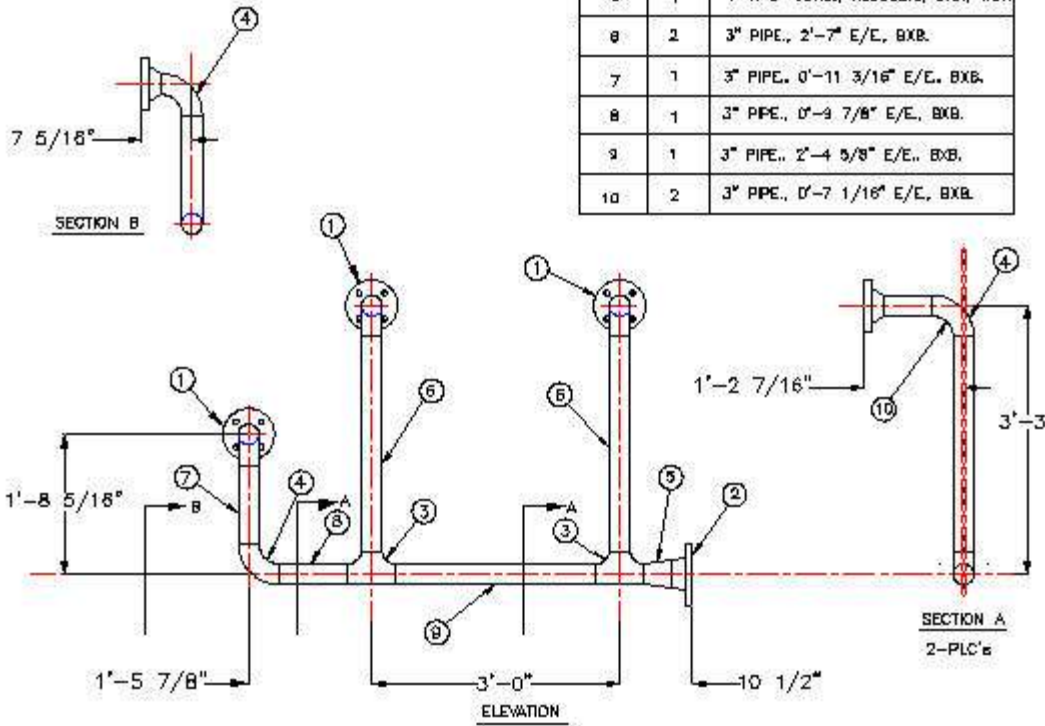




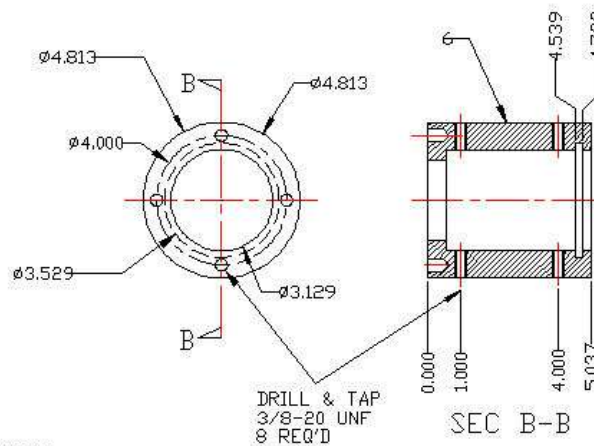
STEEL FLOOR

The steel floor above was designed using AutoCAD.

ITEM	QTY.	DESCRIPTION
1	3	3" FLANGE, WN., RF., 150#.
2	1	4" FLANGE, WN., RF., 150#.
3	2	3" TEE, STD., WGT.
4	3	3" ELBOW, 90°, LR STD., WGT.
5	1	4" X 3" CONNC. REDUCER, STD., WGT.
6	2	3" PIPE, 2'-7" E/E, BXB.
7	1	3" PIPE, 0'-11 3/16" E/E, BXB.
8	1	3" PIPE, 0'-4 7/8" E/E, BXB.
9	1	3" PIPE, 2'-4 5/8" E/E, BXB.
10	2	3" PIPE, 0'-7 1/16" E/E, BXB.

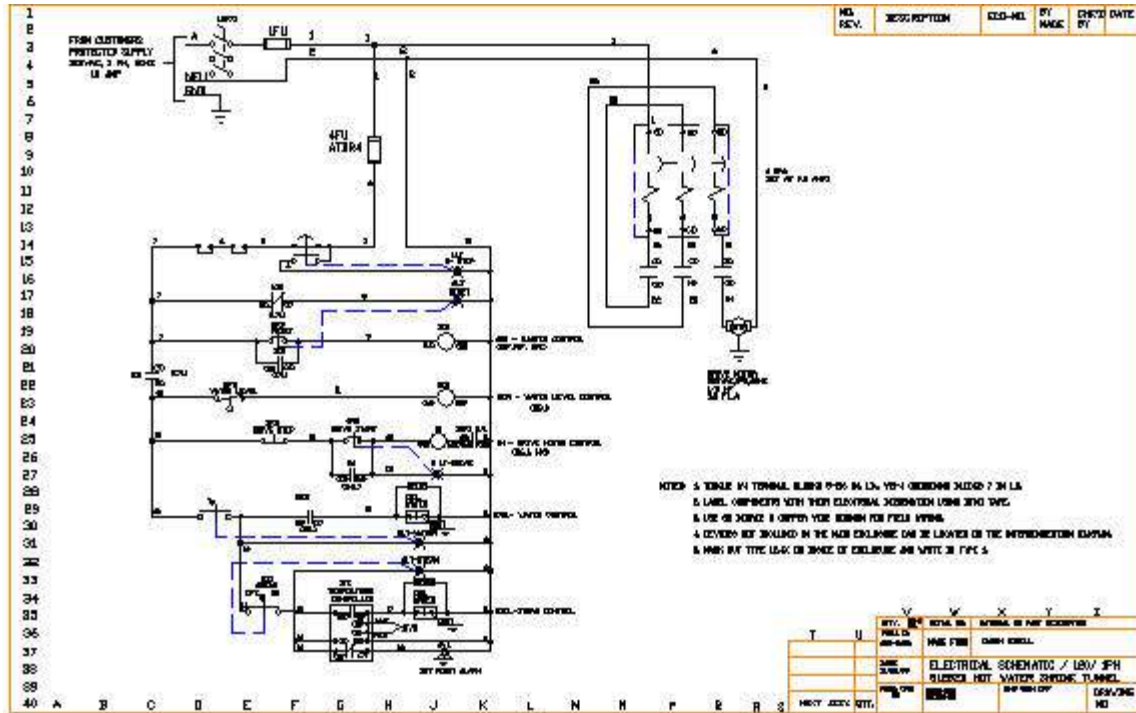


PROCESS PIPING SPOOL



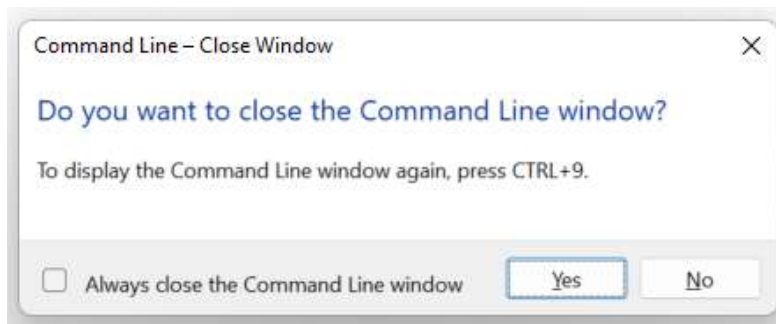
- NOTES:
 1. MATERIAL: 304 SS
 2. TOLERANCE ON ALL DIMENSIONS IS +/-0.002

MACHINED PART EXAMPLE



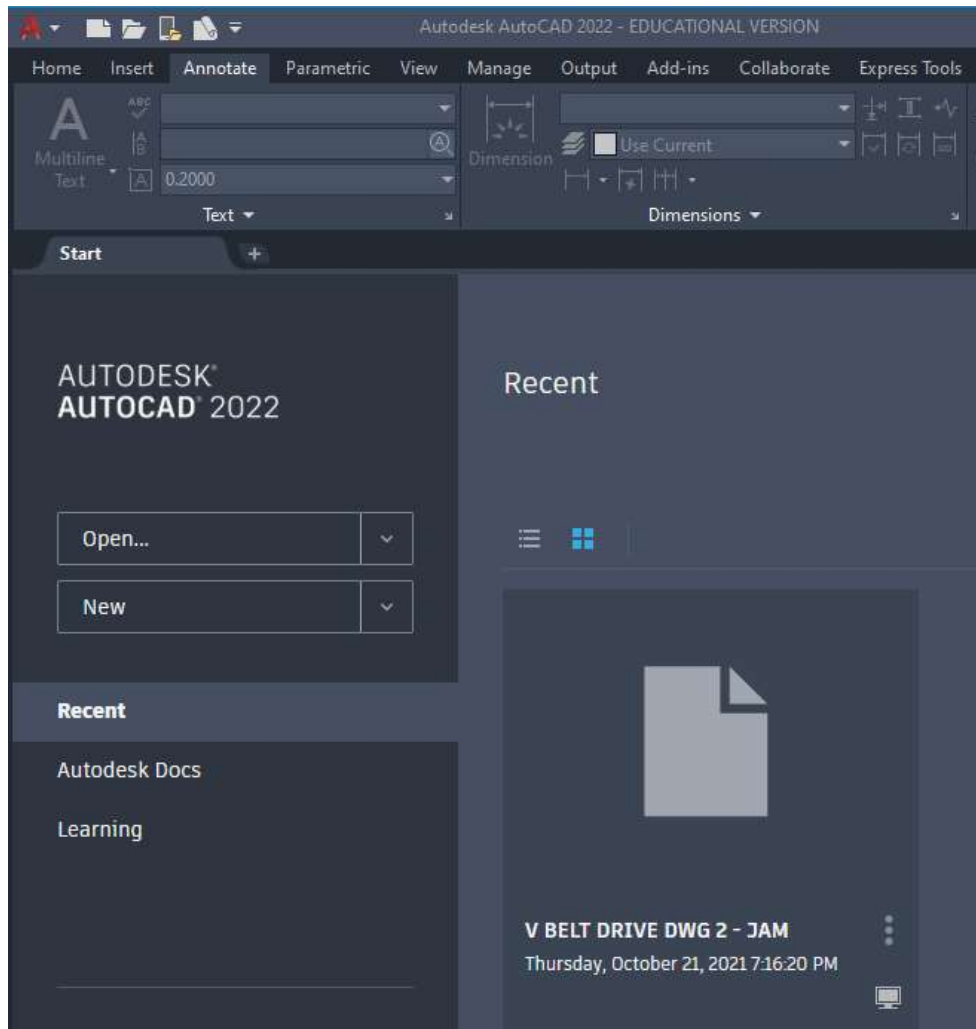
ELECTRICAL CONTROL DIAGRAM

The electrical control diagram above is an AutoCAD drawing.

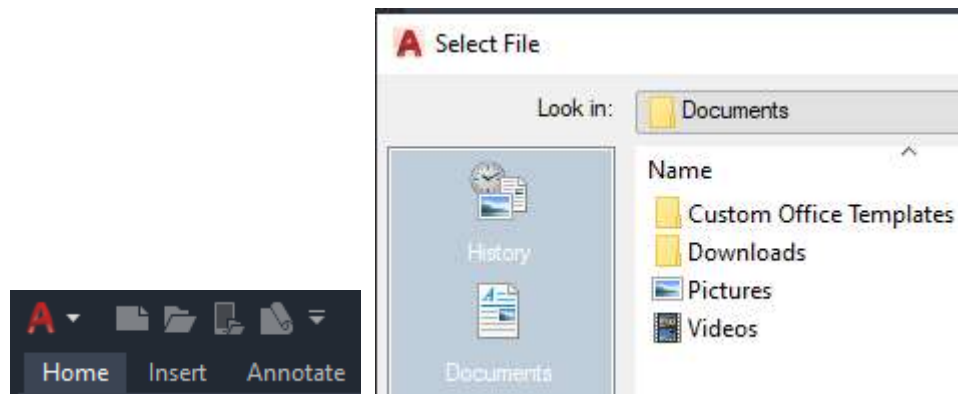


Toggle "Command Line" with: Ctrl key and 9

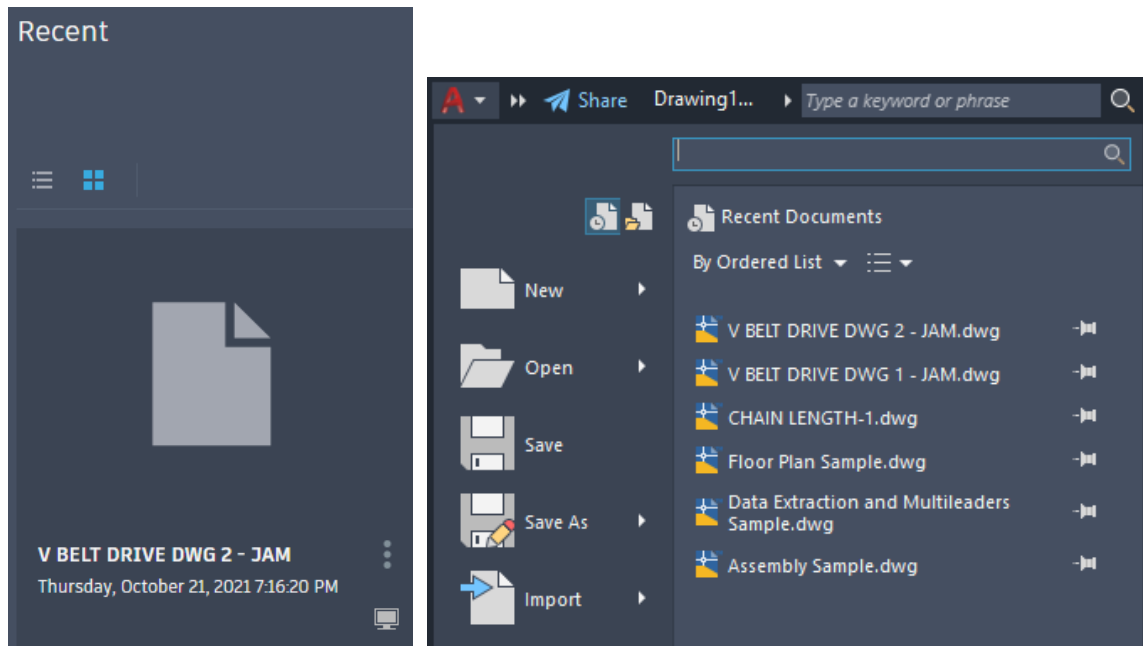
AUTOCAD 2019 to 2022



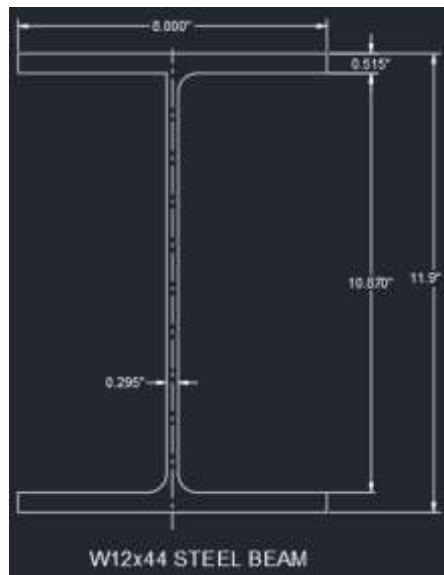
Open AutoCAD



Select "Open Folder" > Select a Folder

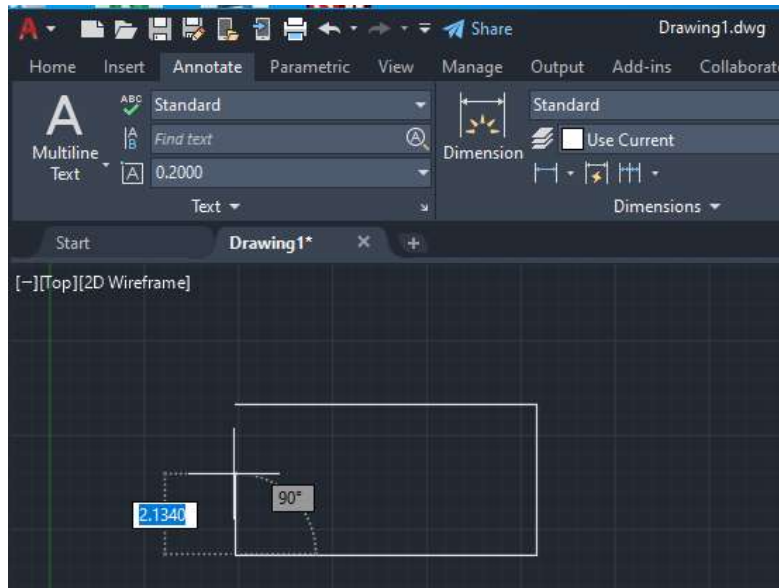


Select "Recent" for a list of new drawings.



Recent drawing selected and opened.

RECTANGLE

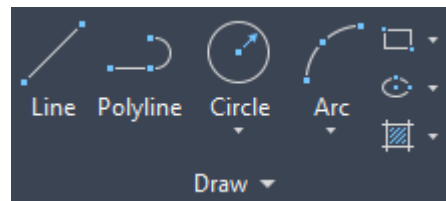


Draw an 8" by 4" Rectangle

Press Function Key F8 to draw horizontal or vertical lines.

Type **L** for Line command or select the "Line" tool from the toolbar.

Pick start point > drag line > pick line end point.



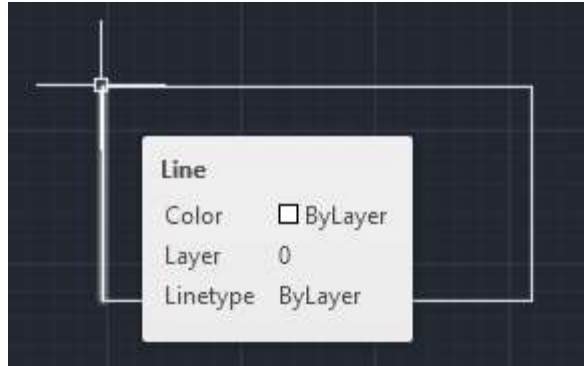
Type **L** > Space bar or

Select Line tool > Enter

Press function key F8 for horizontal and vertical lines.

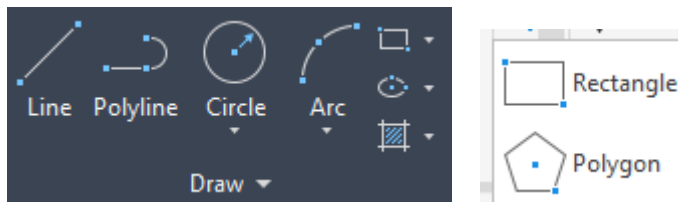
Pick top line start point left end >

Drag line right > Type 8 > Enter.



Drag down > Type 4 > Drag left > Type 8 > drag up to top left corner.

Draw a new rectangle with the rectangle tool.

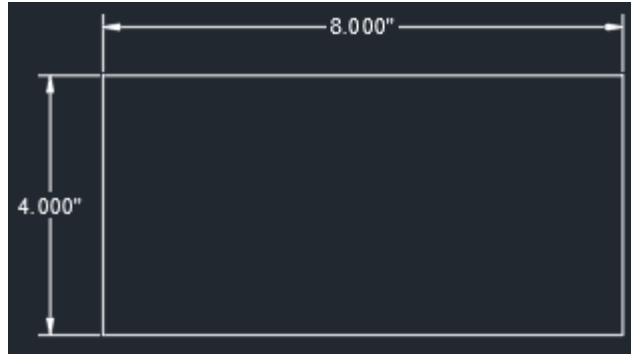


Select > Rectangle Tool > Rectangle > Pick a point and drag a rectangle > Type 8,4 > Enter

Rectangle dimensions are 8.000" long and 4.000" wide.



Pick automatic dimension tool >



Pick left side > Pick dimension location >

Pick top side > Pick dimension location.

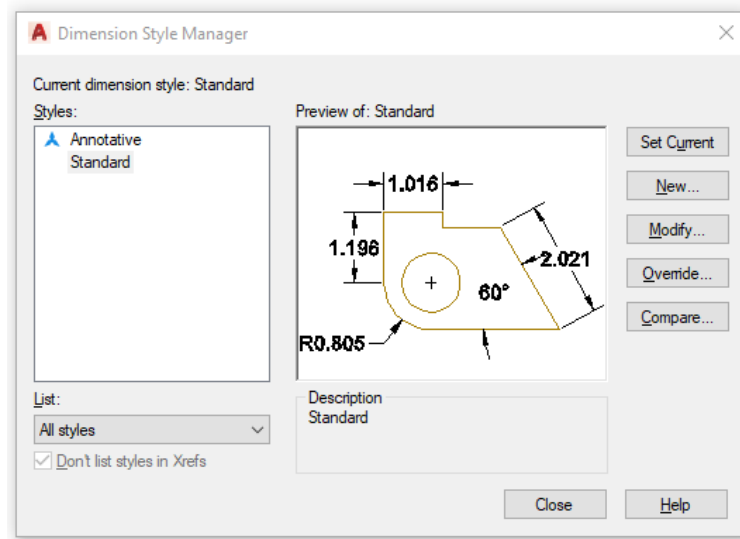
Dimensions 4.000 and 8.000 inches are added to the drawing.

GRID



Press function key F7 to add or remove display grid.

DIMENSION UNITS

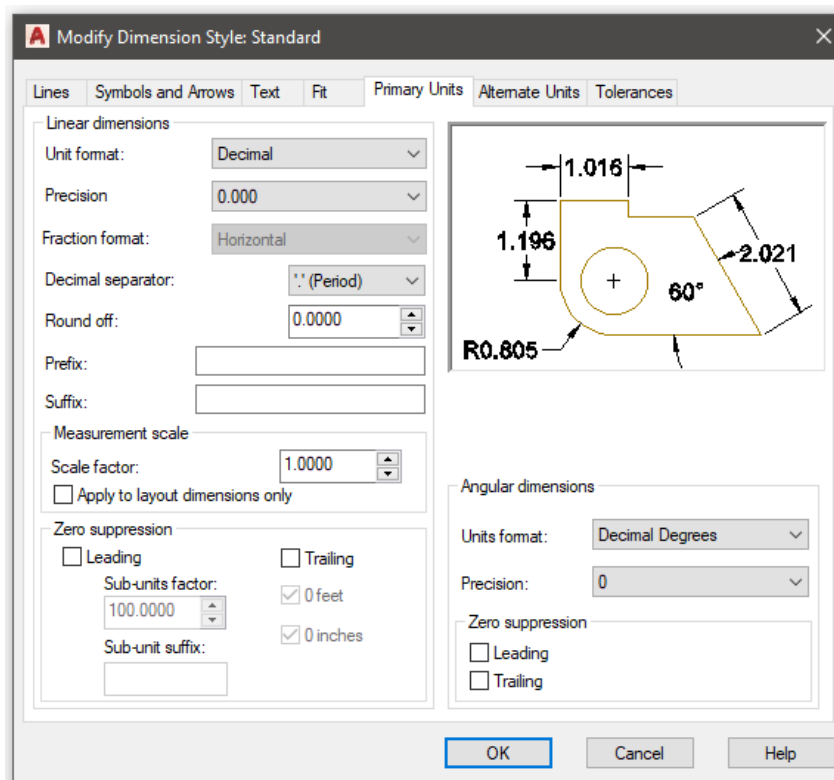


Type D > Enter

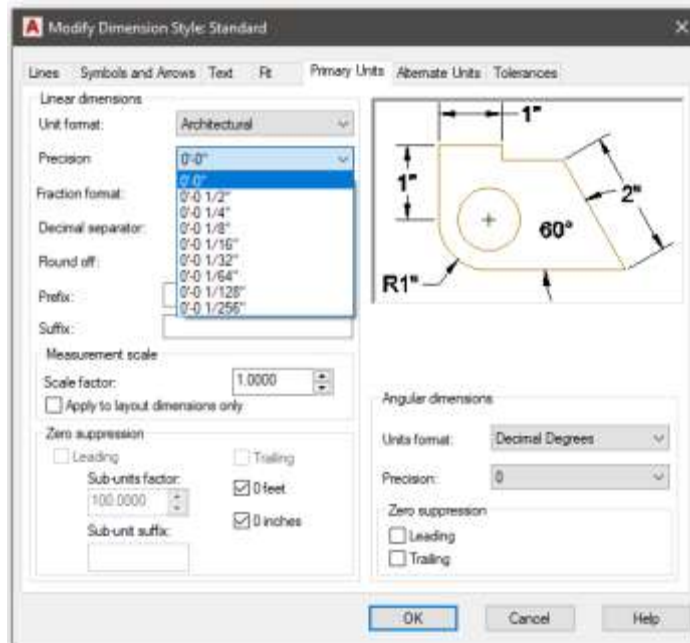
or

Type D on the command line for the "Dimension Style Manager".

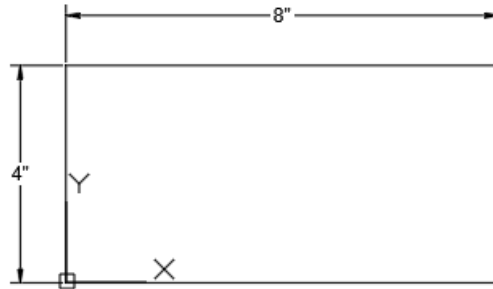
Select "Modify".



Select Unit Format "Architectural" drop down menu.

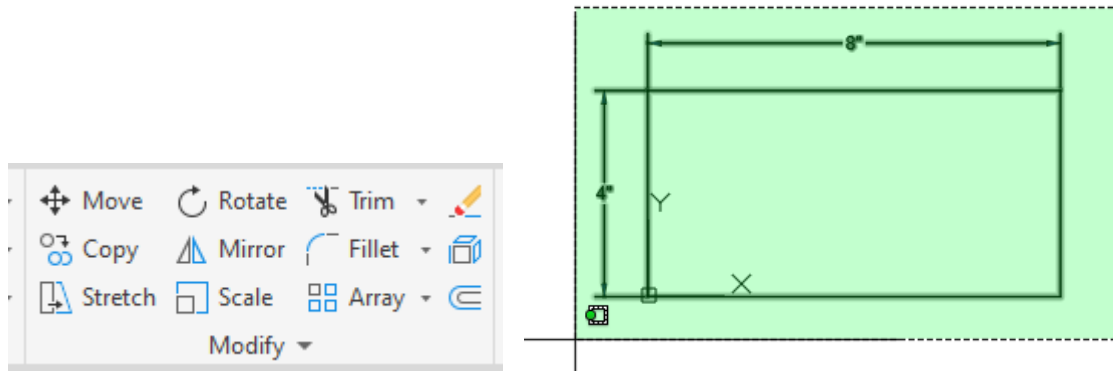


Select Fraction Format feet and inches 0'-0"

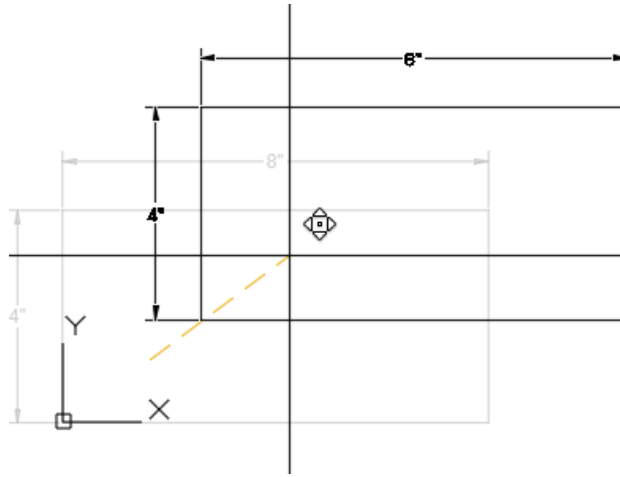


Dimension units are now changed to Architectural.

MOVE



Select Move or
Type M > Enter > Pick point at top right > Select the rectangle > Enter



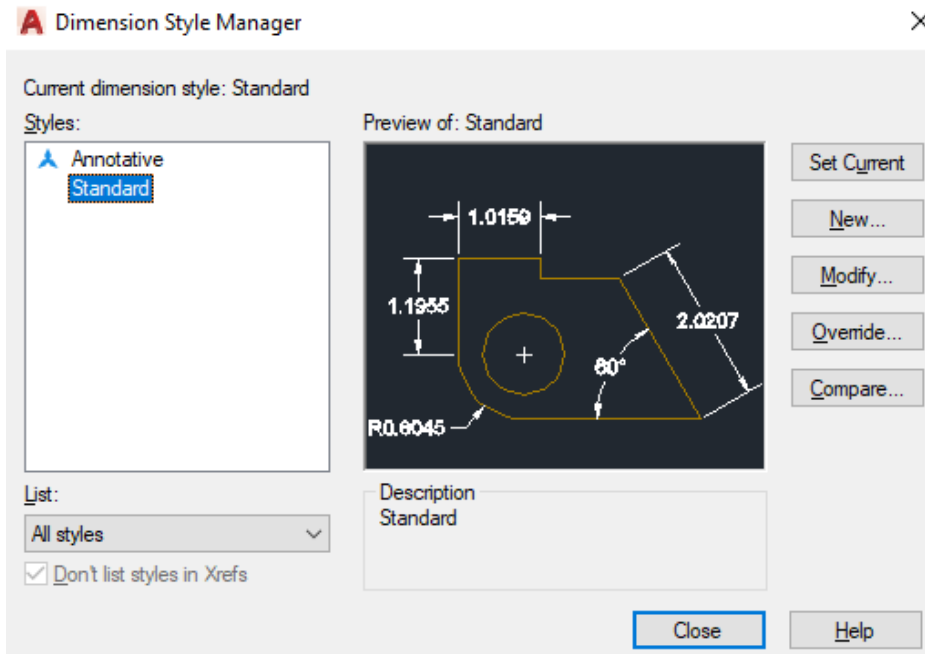
Pick a base point > Drag rectangle to new location >
Pick new location point > Enter.

ZOOM VIEW

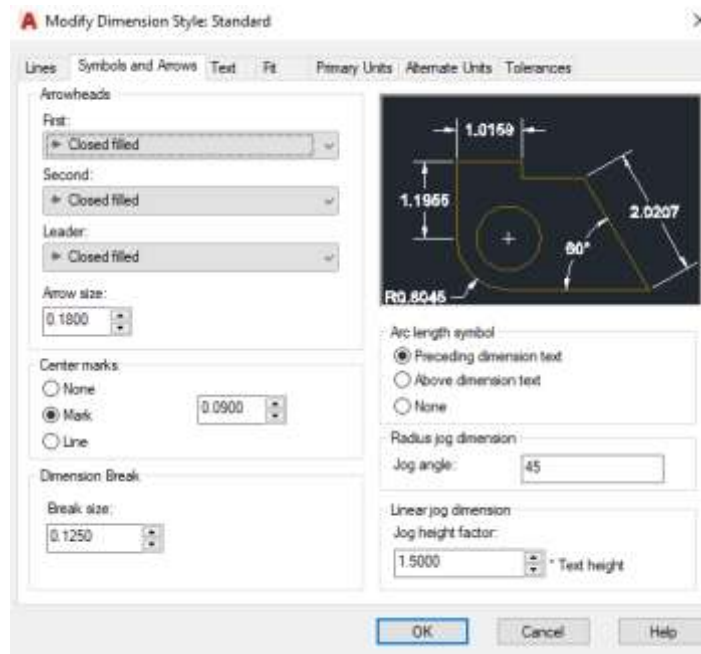
Roll mouse wheel toward you to Zoom Out.
Opposite to Zoom in.

PAN

Hold mouse wheel down and drag to move drawing.



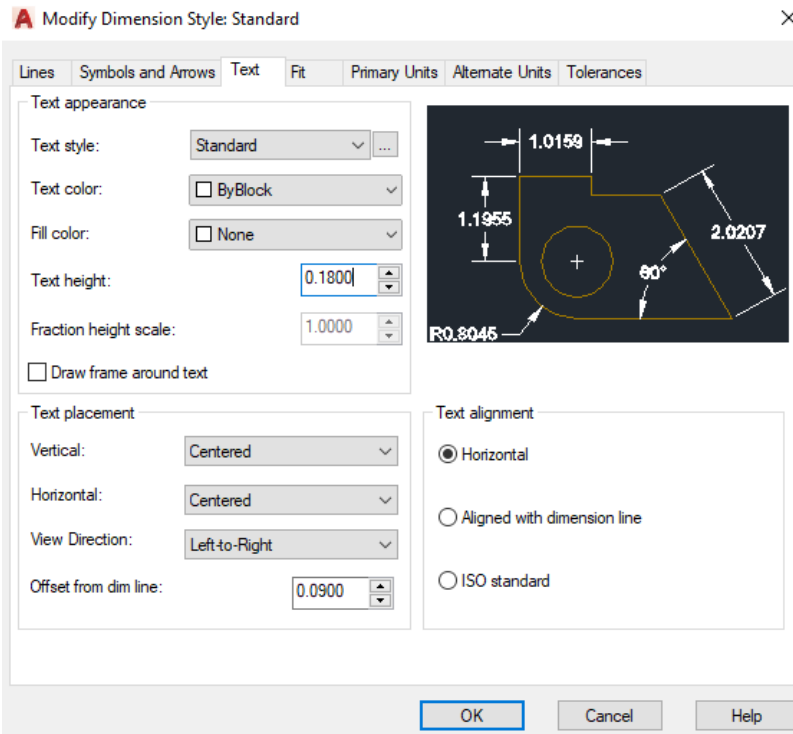
Type D > Enter > Select > Modify



Arrow size:

0.25

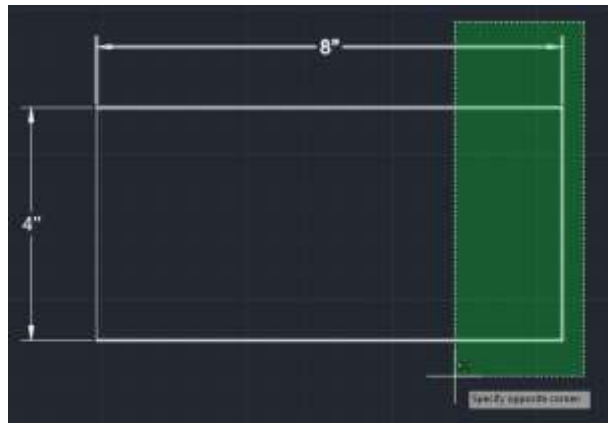
Edit arrow size >



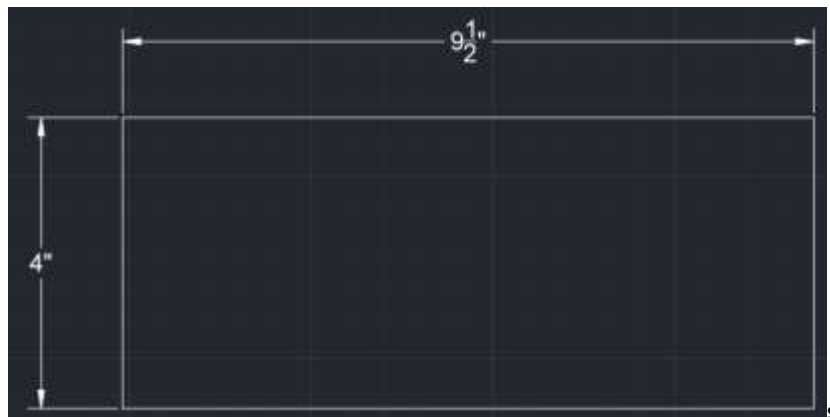
Edit text size >



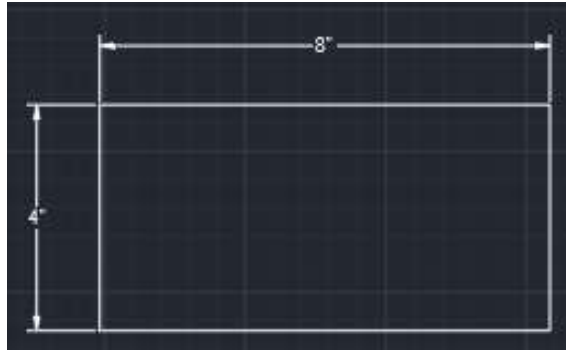
Rectangle is 4" x 8".



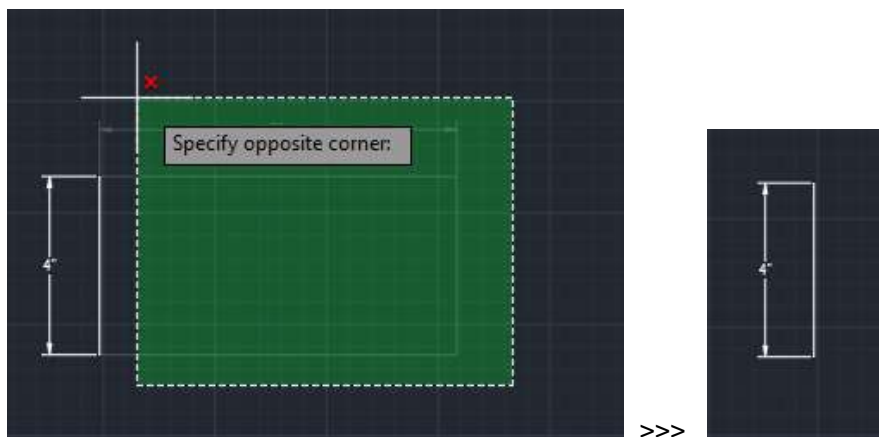
Stretch Command > Type S > Enter or Space bar >
Pick top right > Release mouse button > Drag bottom left >
Pick any base point > drag right >
Type > 1.5 > Space Bar



The rectangle length is "Stretched" > 8" to 9 1/2" > 1.5 inches longer.



Erase an Area Example



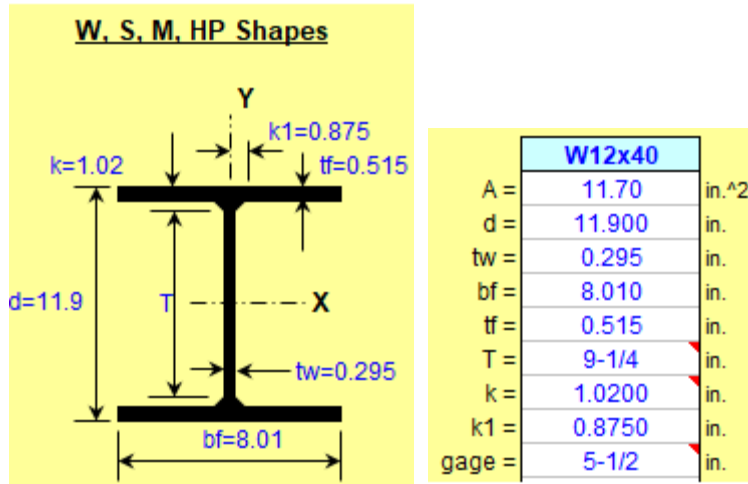
Drag select area to be erased:

Drawing after erase.

Type > E > Space Bar > Pick bottom right >

Drag to top left > Space Bar or Enter > Green area is erased.

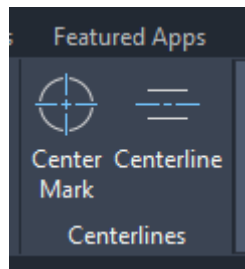
4" dimension remains.



Draw W12x40 steel beam section.

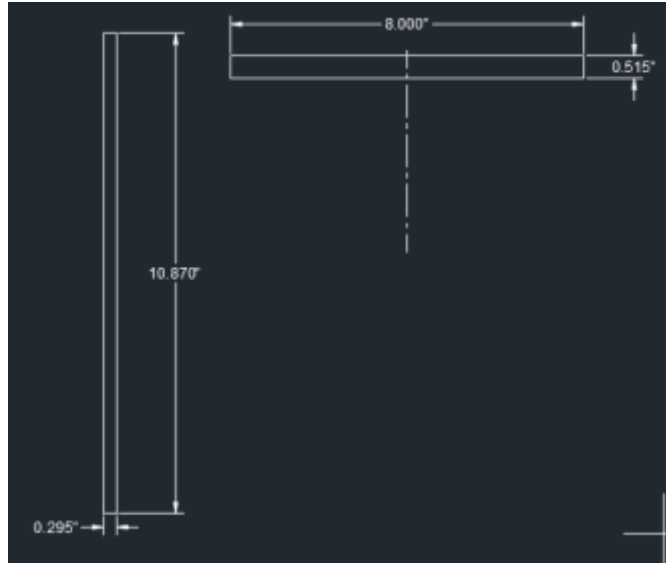


Draw beam flange 8.000" x 0.515 with Rectangle Tool.



Select Centerline tool >

Pick beam flange side > Pick opposite side > Enter

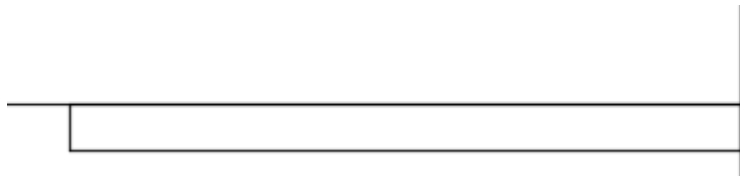


Draw beam web 10.870" x 0.295" with Rectangle Tool.

Beam section height is 11.9"

Web Height is 10.87, thickness is 0.290".

$$11.9 - .515 - .515 = 10.87"$$

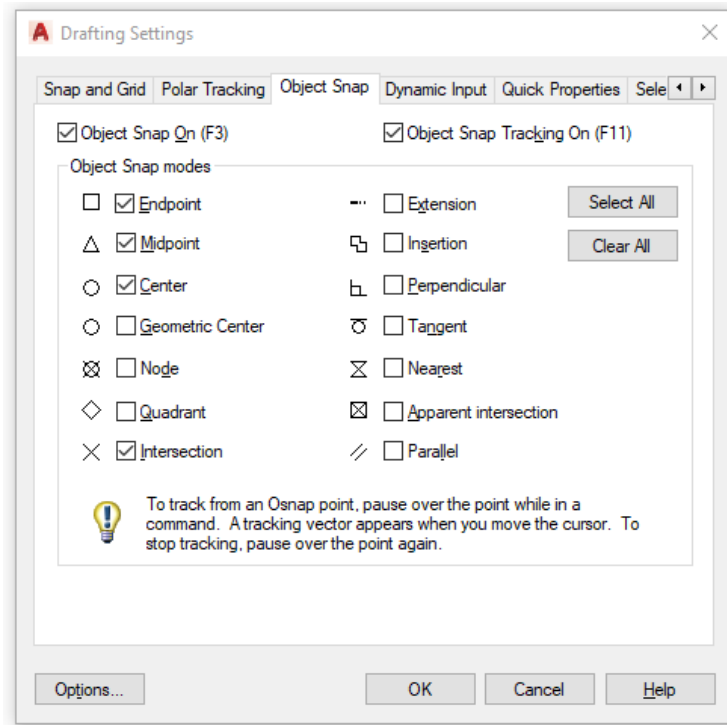


Select > Rectangle Tool > Rectangle >

Pick a point and drag a rectangle > Type 8,0.515> Enter

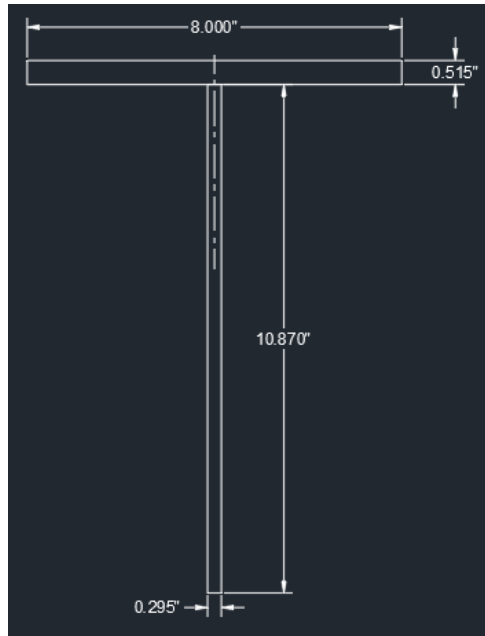
Rectangle dimensions are 8.000" long and 0.515" wide.

OBJECT SNAP



Type OS > Enter

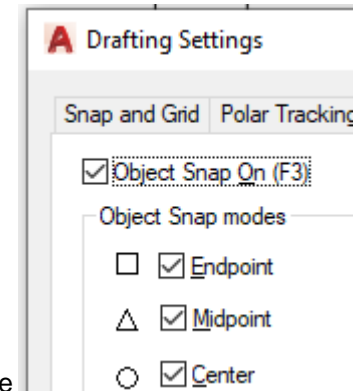
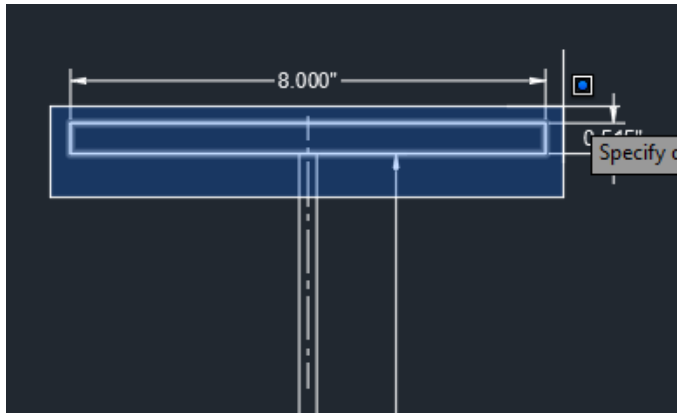
Set > Endpoint > Midpoint > Center > Intersection > Apparent intersection > OK



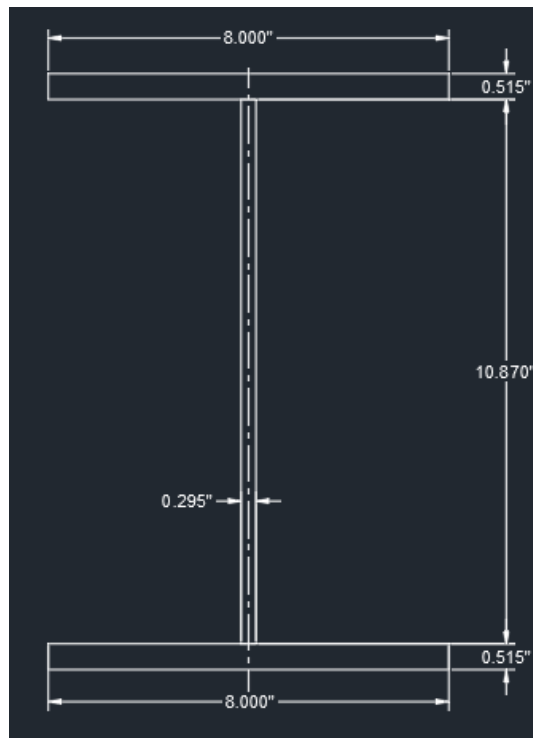
Move web to beam top flange as shown above.

Type M > Space bar > Select web > Pick web top center >

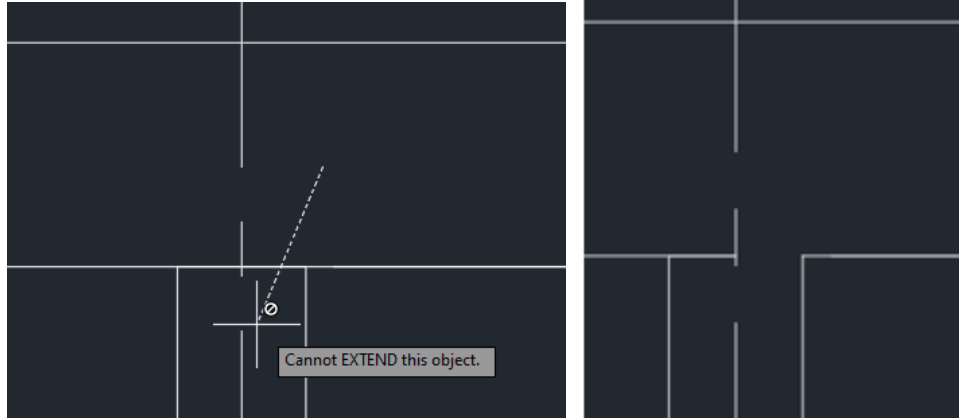
Drag to flange bottom center.



Object Snap type OS > Enter > Object Snap On
Mirror type MI > Select top flange >
Pick web vertical line center >
Drag right > OK



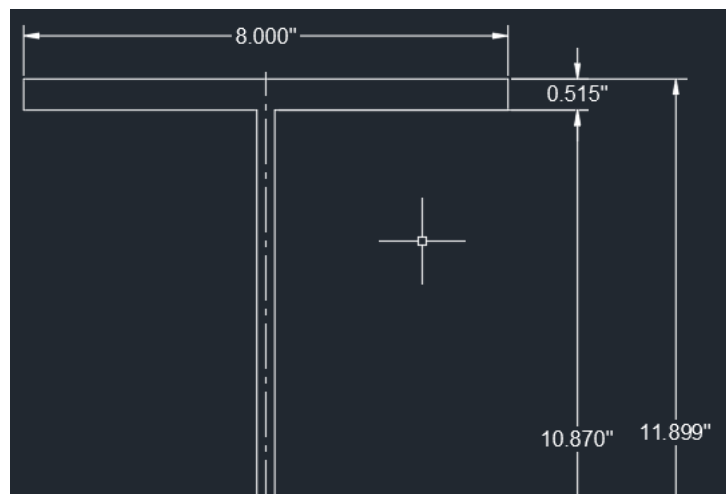
Bottom flange is a mirror copy of top flange.



Trim line and lines to be cut.
Type Trim or TR > Space bar >
Select lines to be trimmed >

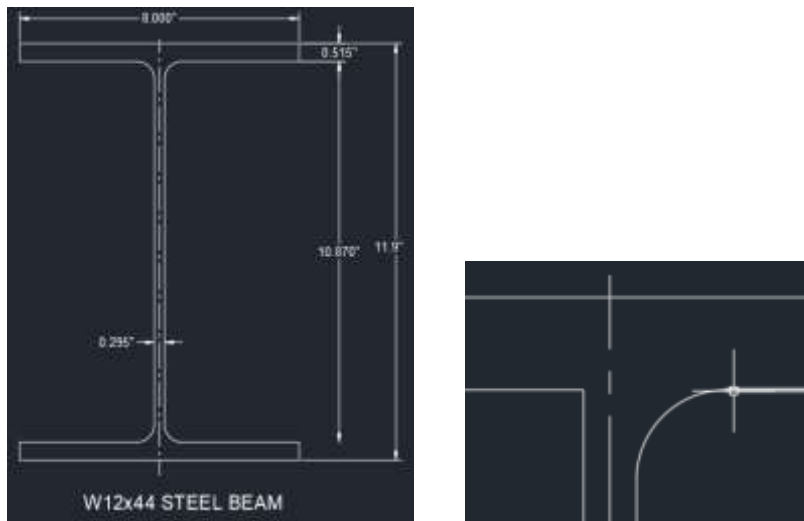


Lines have been trimmed.



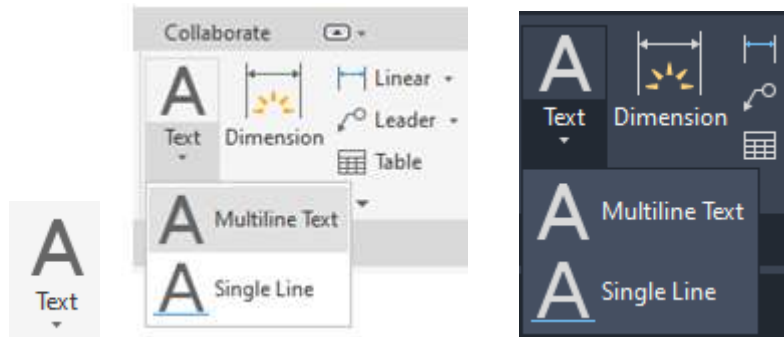
Explode 11.899" dimension X > Select dimension > Enter.
Edit ED > Enter > Pick 11.899" dimension > Type 11.9" > Enter

FILLET

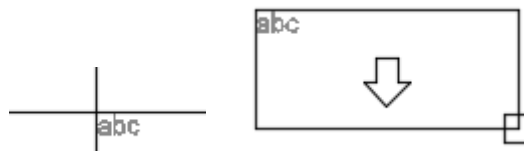


Type F > Space Bar > Type R > Type 0.5 > Space Bar >
Pick a line > Pick joining line > Space Bar OR Enter.
Fillet with radius 0.5" is created.

TEXT



Select > A Text > A Multiline Text >



Pick upper left point > Drag a text box to lower left > Pick point

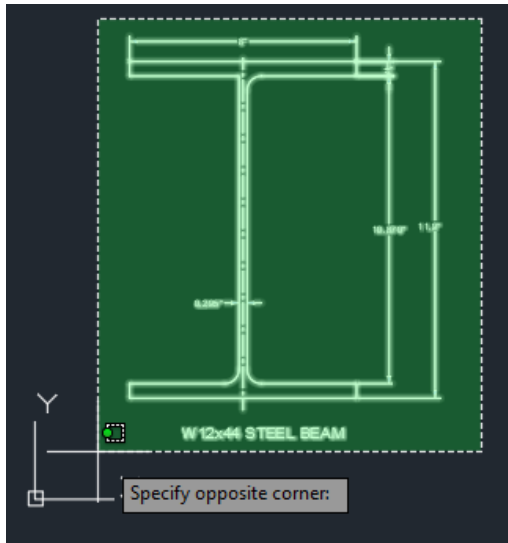
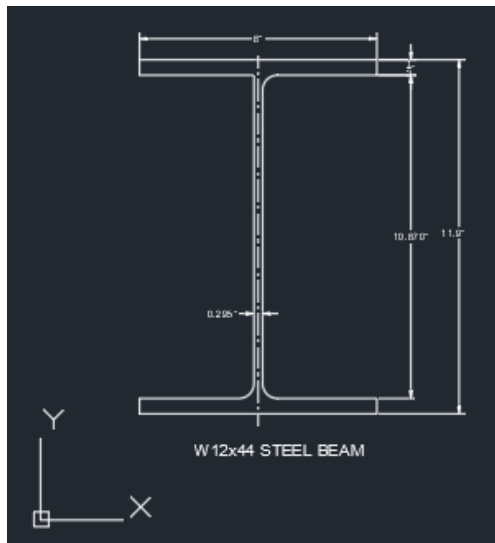


Type text

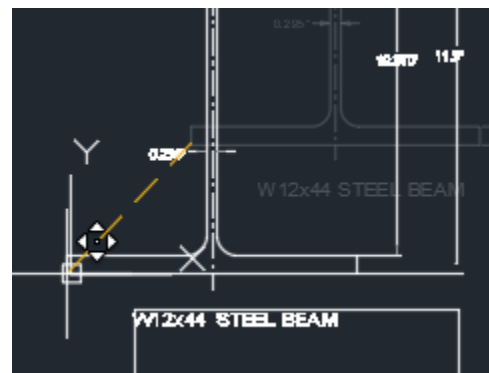
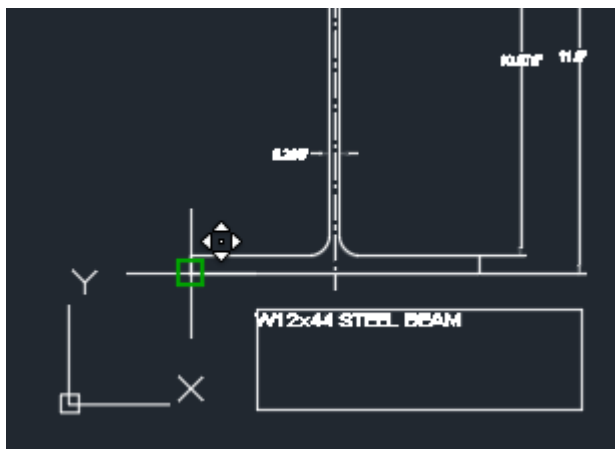


Drag a rectangle over text to select it > Type > Scale >
Pick a base point > Type > 4 > Enter
Type M for Move > Enter > Pick a point in text > Drag text to new location.

MOVE OBJECT TO X-Y ORIGIN



Type M > Select object >

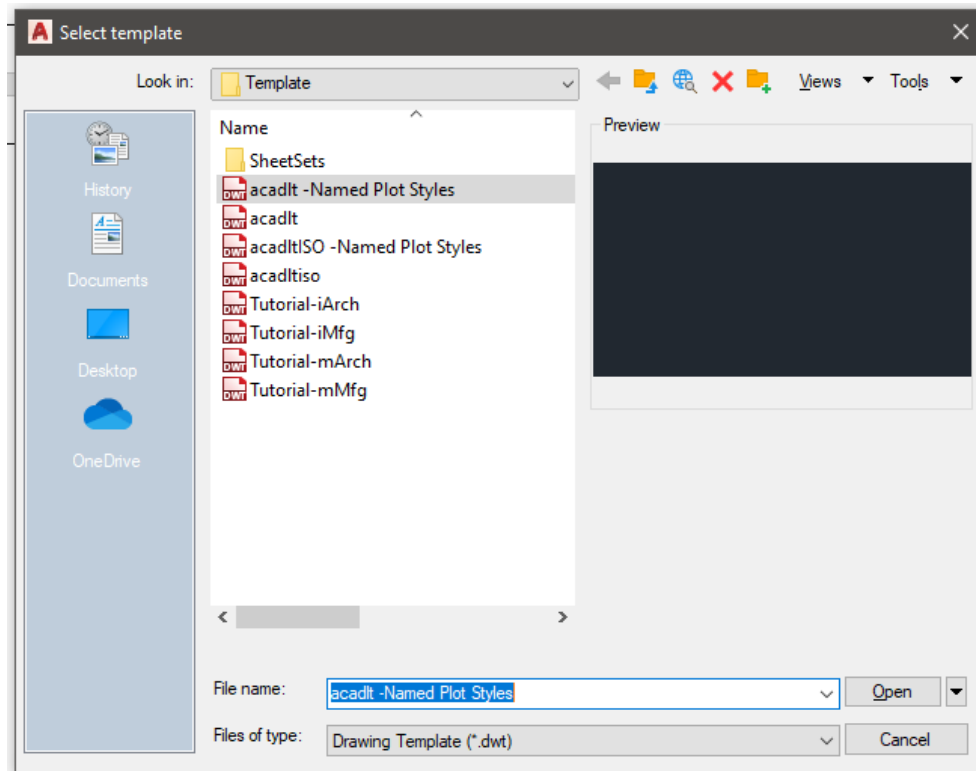


Pick point on object > Drag mouse pointer to X-Y Origin > Pick Origin.

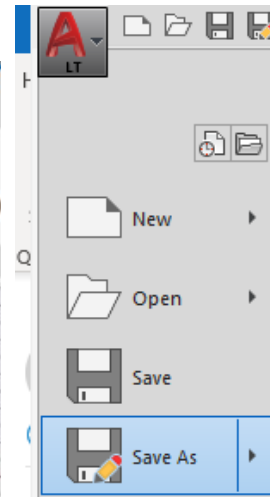
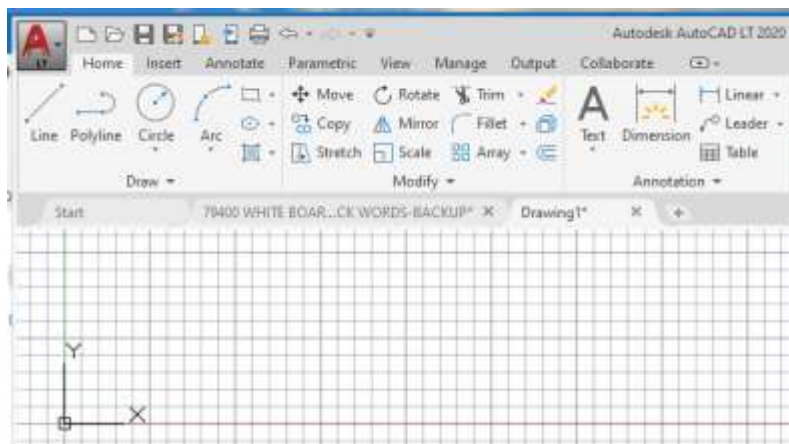
START A NEW DRAWING



Select top left **A** > Template >

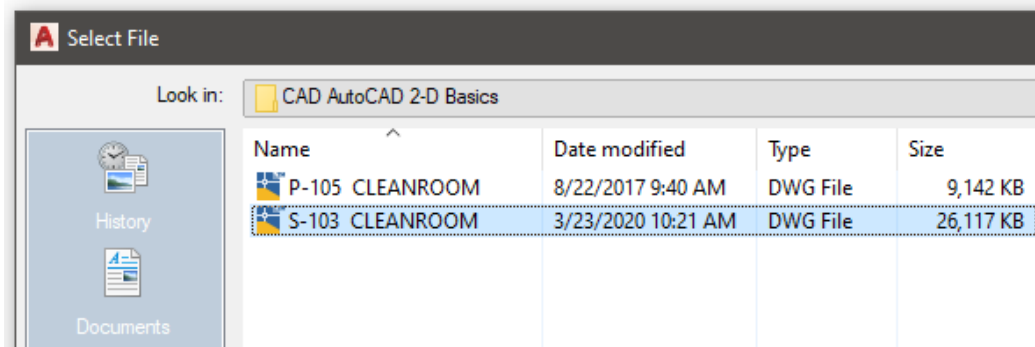


Select **Open** >

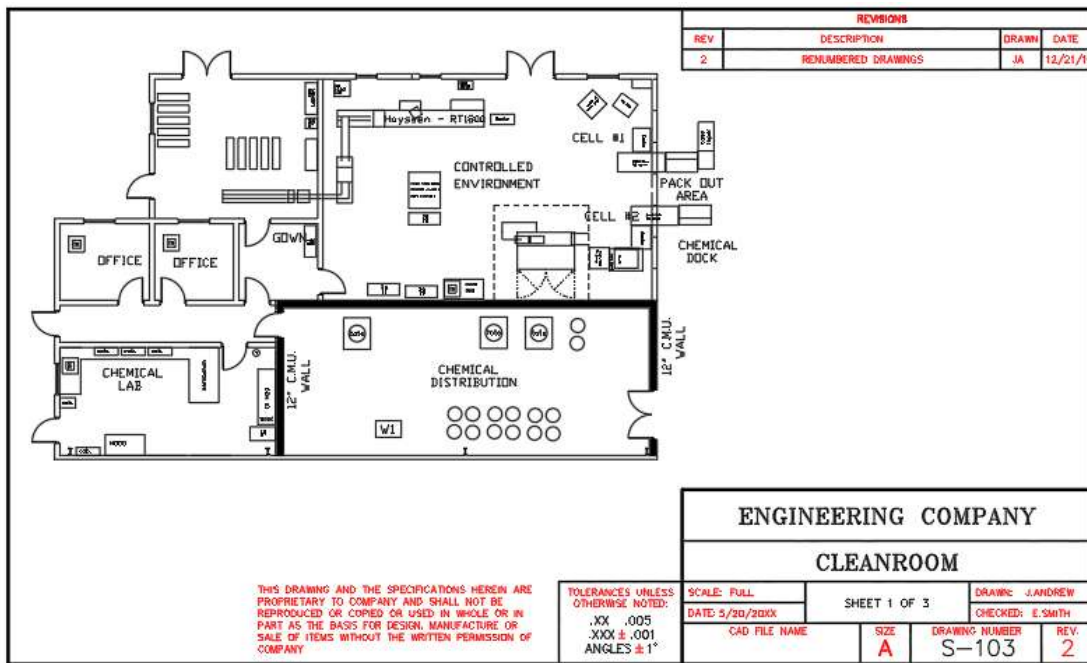


Save As > Drawing with no template.

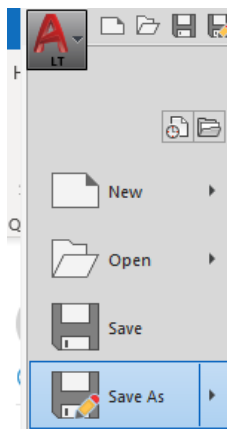
OPEN AN EXISTING DRAWING



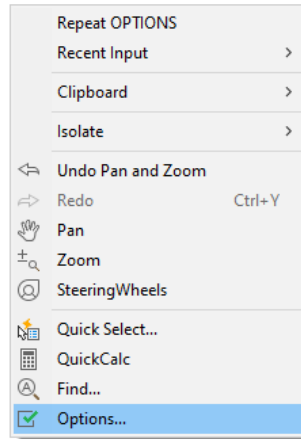
Select Drawing > S-103 CLEANROOM



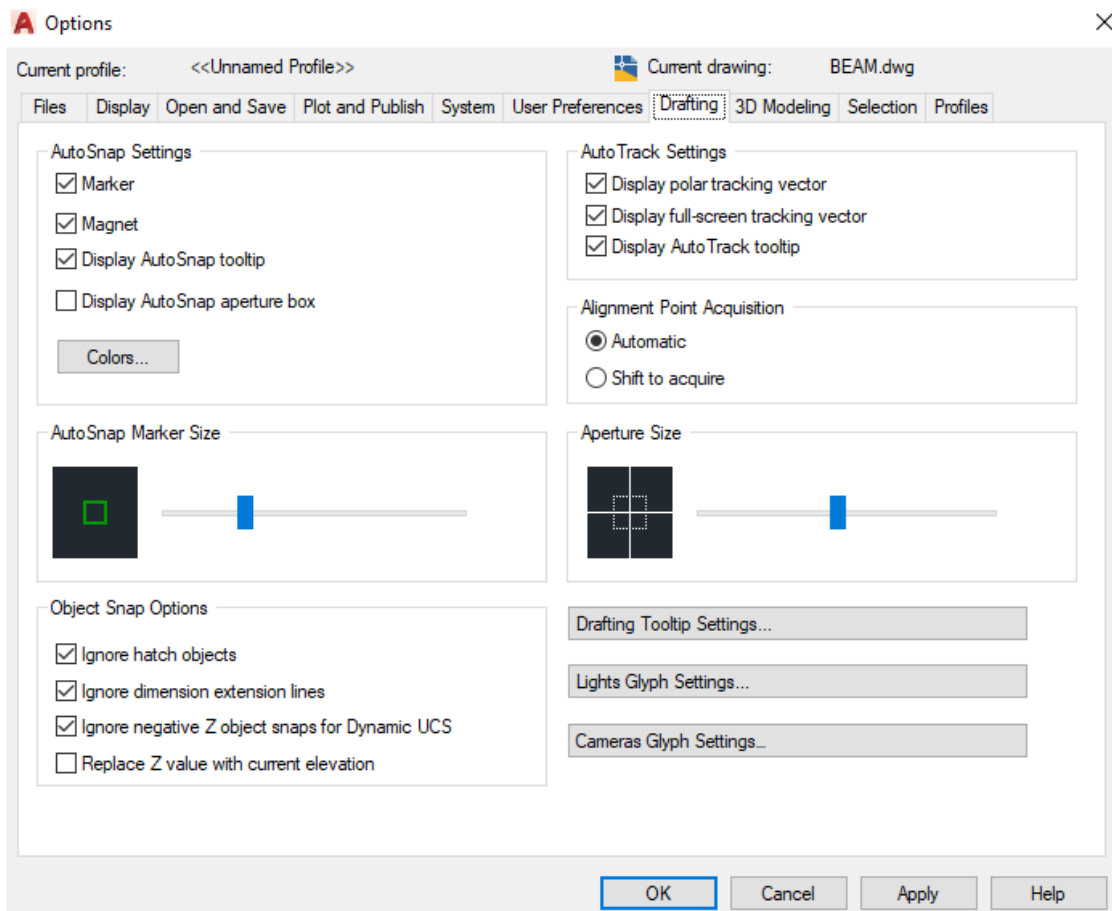
Save As > Drawing with template.



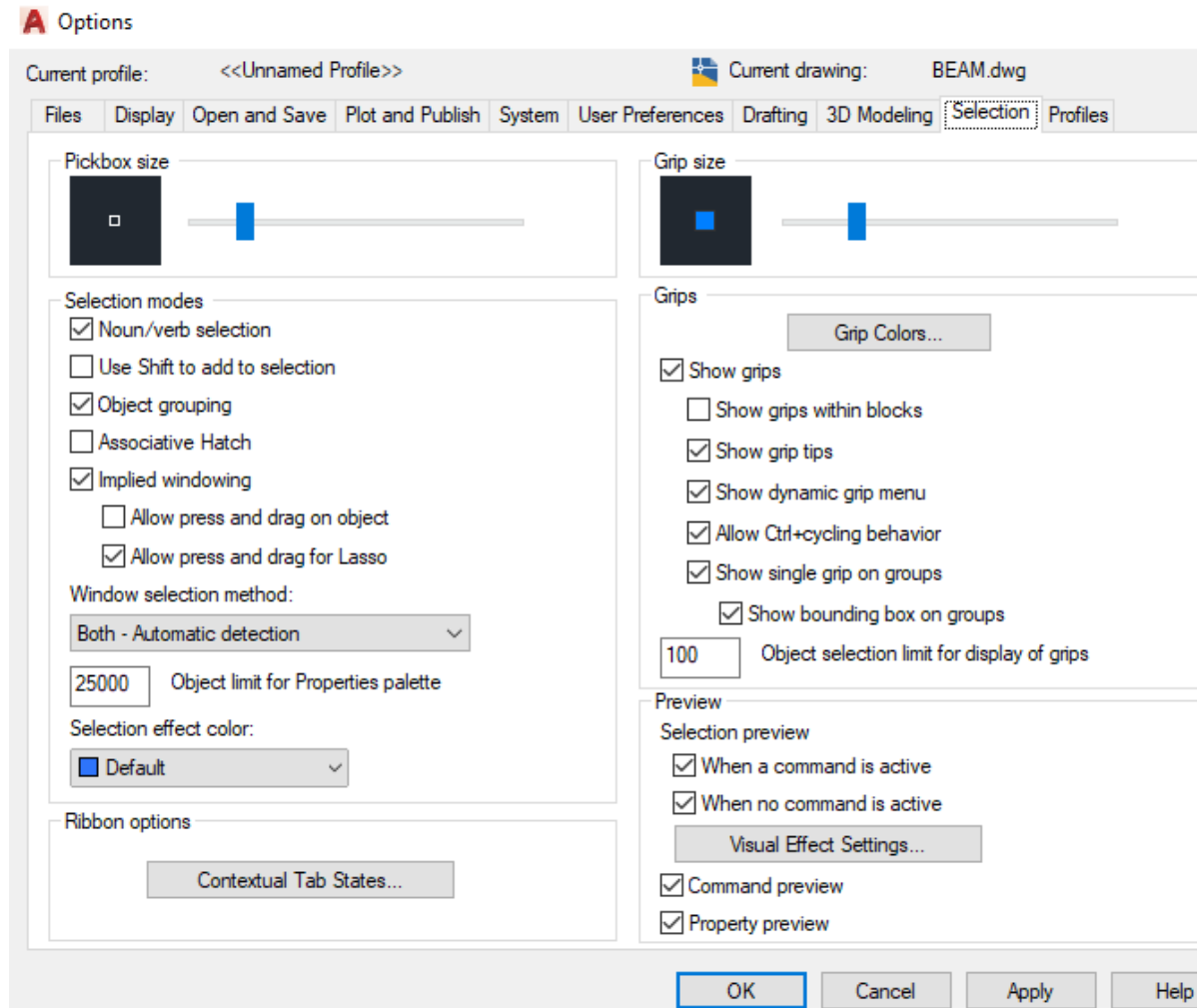
Mouse right click in the drawing area.



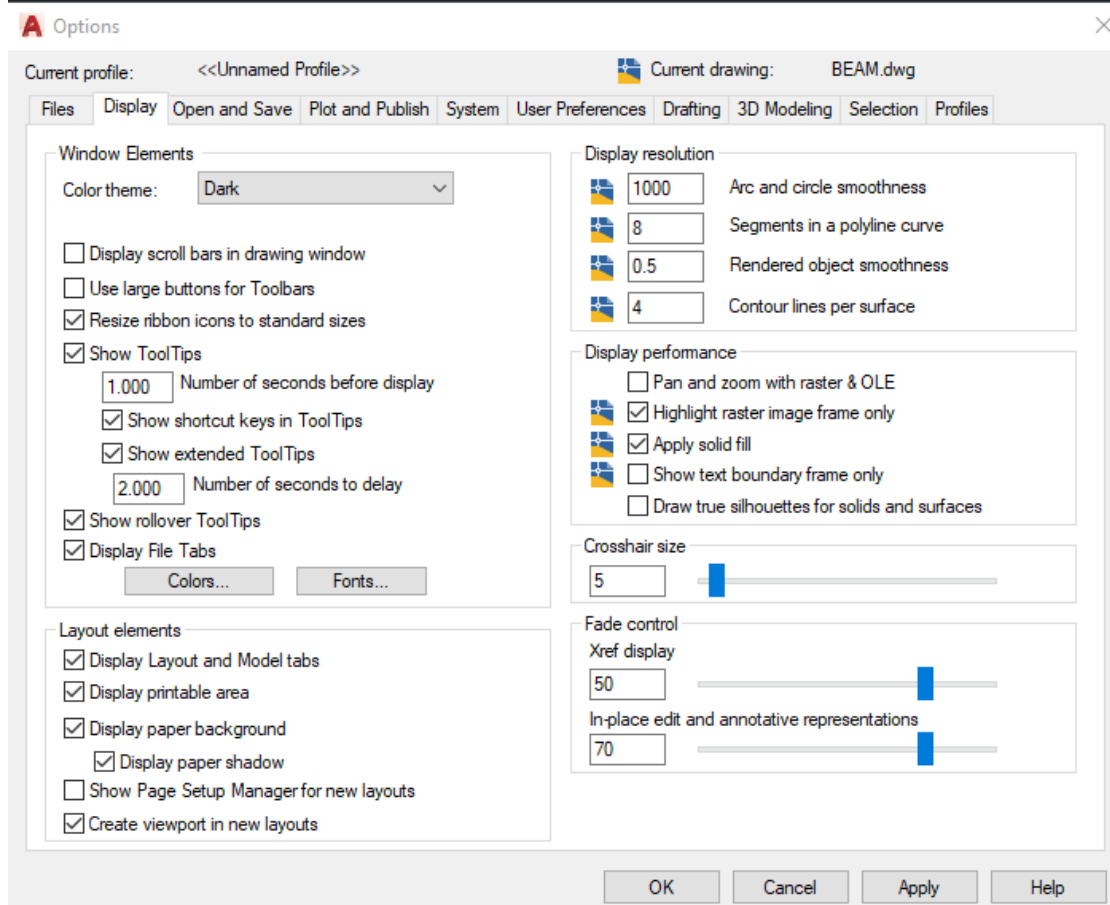
Select > Options



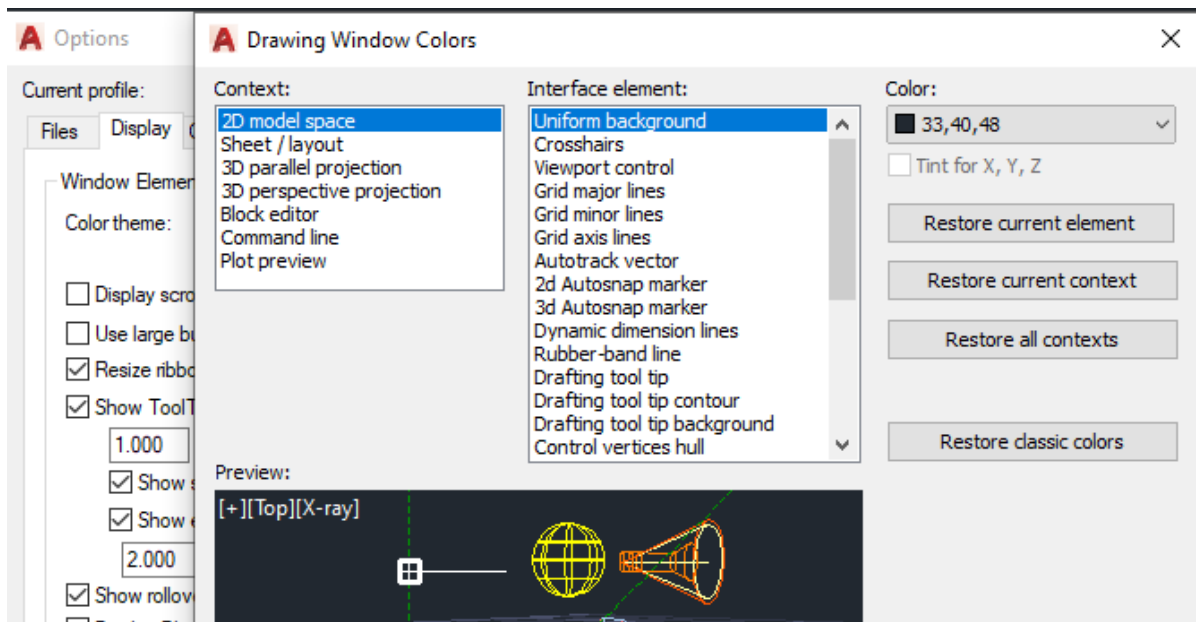
Select > "Drafting".



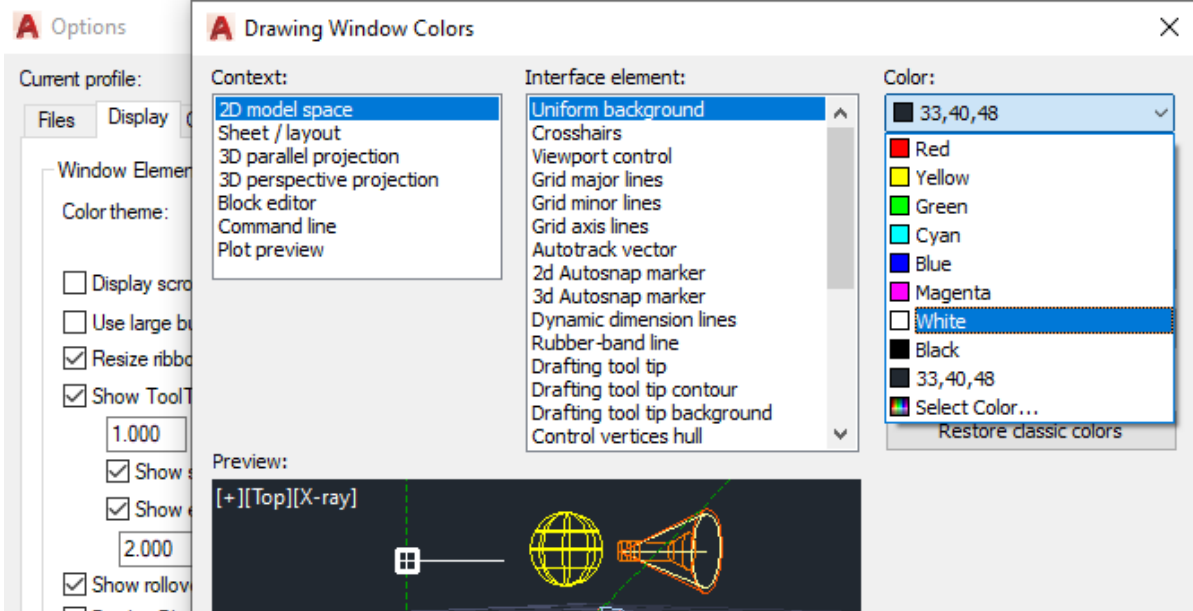
Select > "Selection".



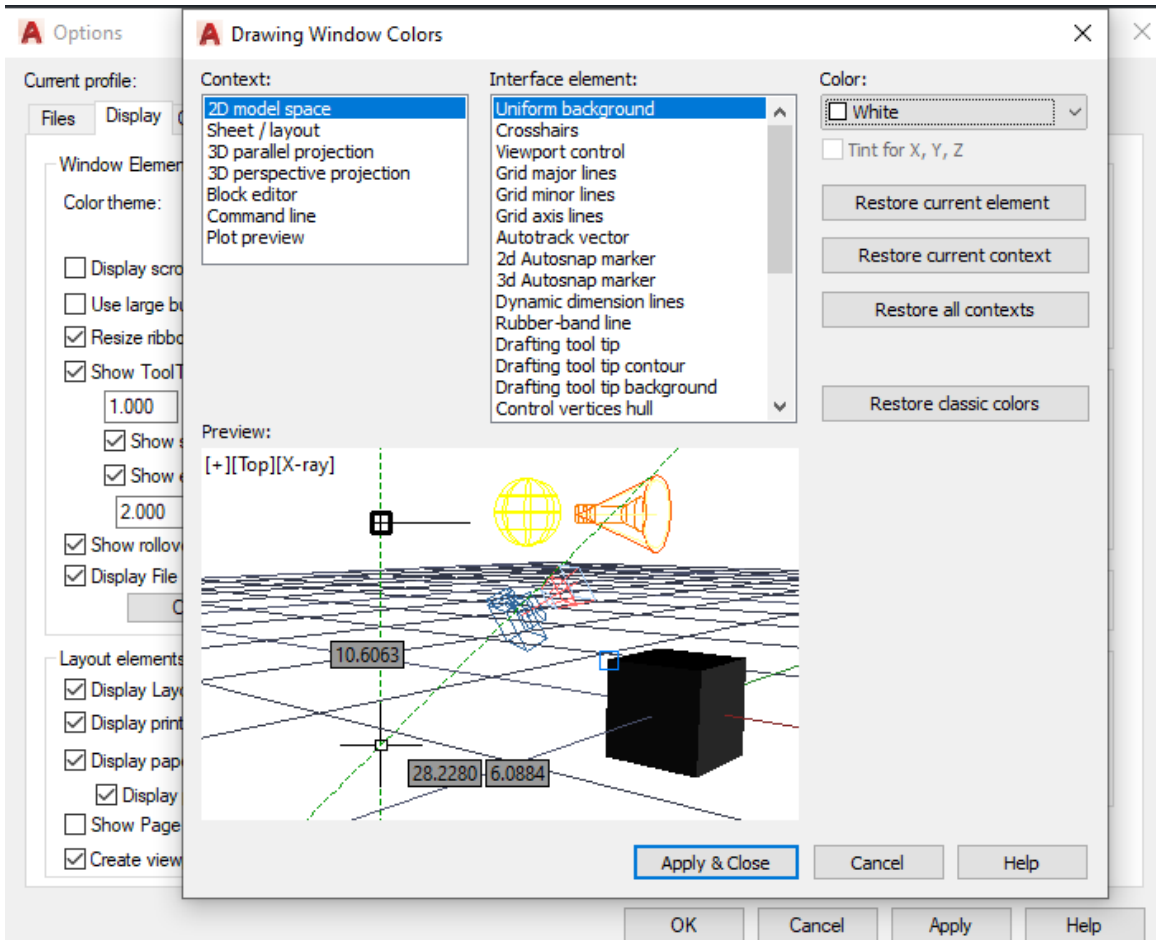
Select "Display" > Colors... >



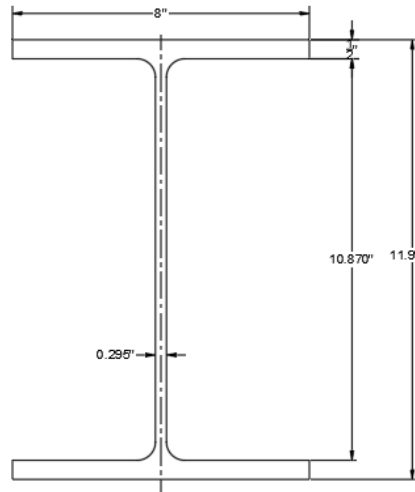
Select "Color" > Drop down menu >



Select "White" >

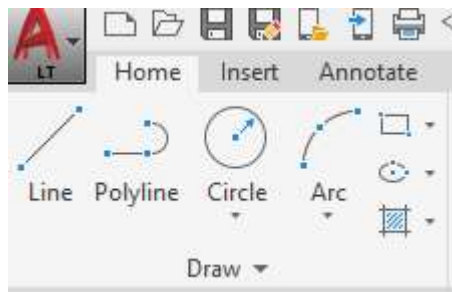


Select "Apply and Close" > OK

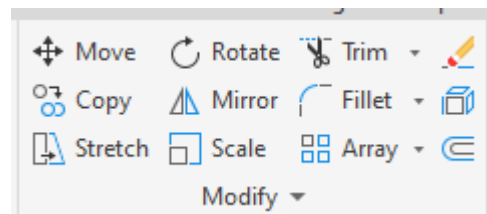


W12x44 STEEL BEAM
AutoCAD background is now white.

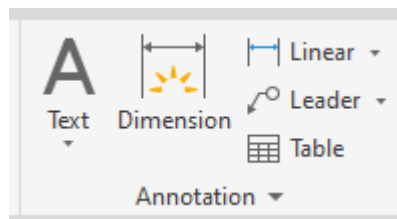
2. AutoCAD 2019 to 2022 TOOLBARS



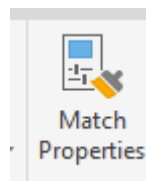
Draw



Modify



Annotation



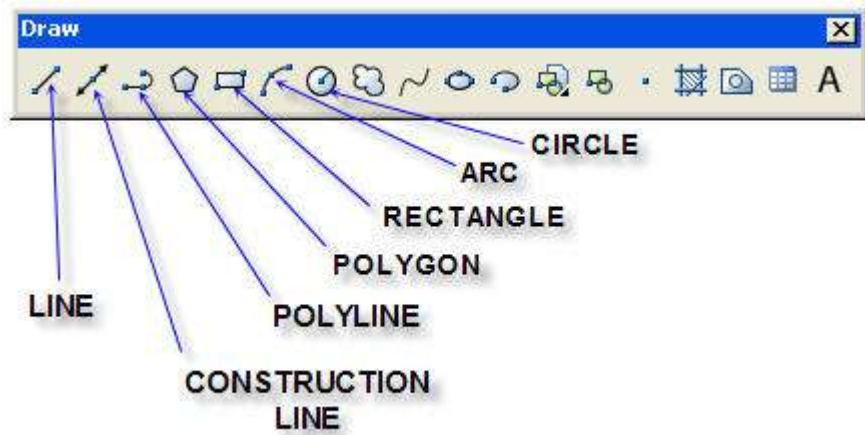
Match Properties

These four "Toolbars" are commonly used to make AutoCAD drawings.

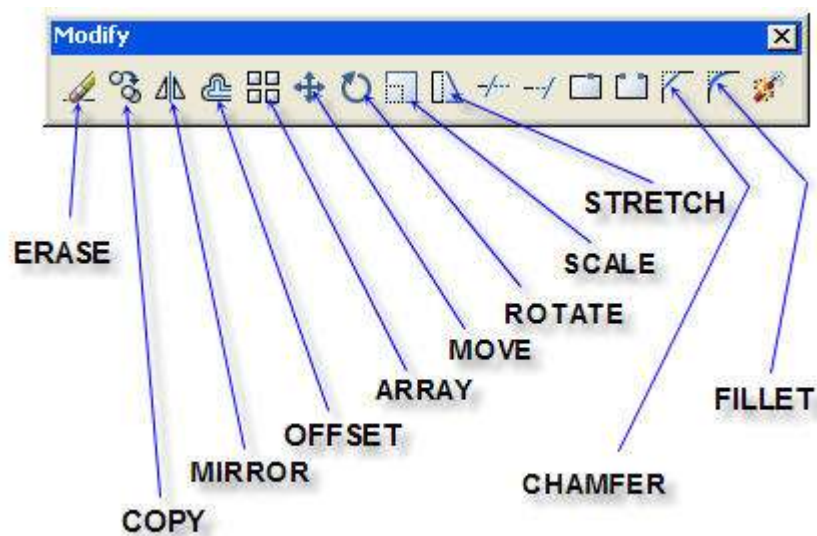
3. AutoCAD 2010 to 2018 TOOLBARS

Docking Toolbars

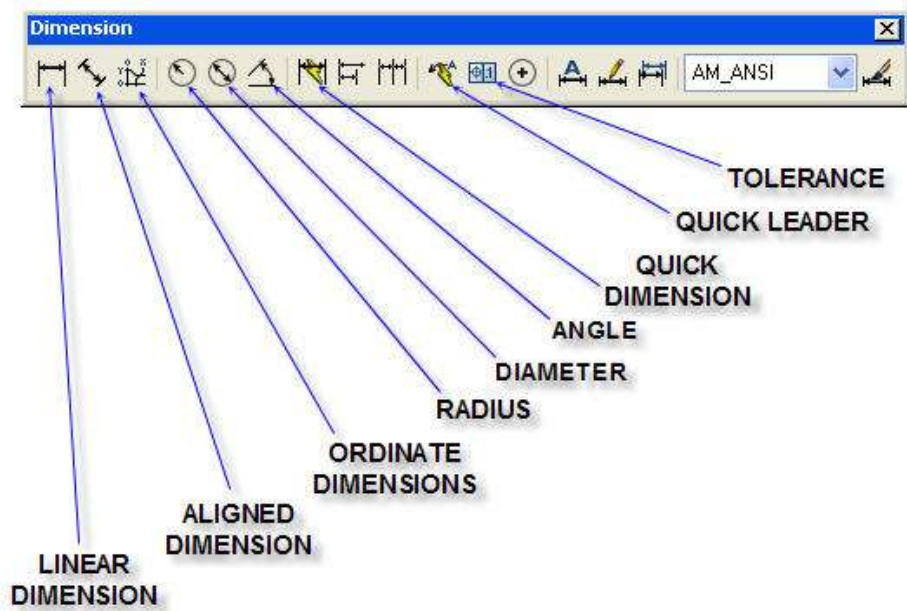
Dock a toolbar by picking its handle and dragging to the left or right side of the display area as shown. Toolbars may be docked at the bottom and top of the display but should be avoided because this practice reduces the already limited height.



Draw



Modify



Dimensions

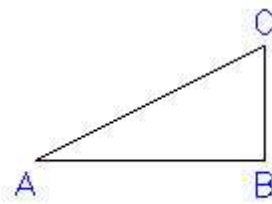
4. COMMON AutoCAD COMMANDS



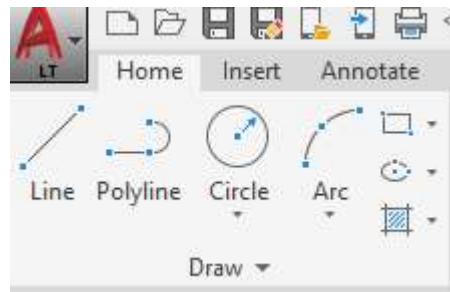
The **LTSCALE** command is used to change the **Line Type Scale** as illustrated above.

This is the end of Section 2. AutoCAD Toolbars

LINE COMMAND



L = LINE



Press function key F8 to draw horizontal or vertical lines.

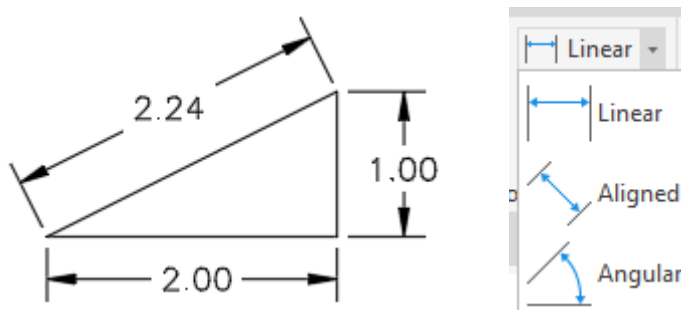
Type **L** for Line command or select the "Line" tool from the toolbar.

Pick start point > drag line > pick line end point.



DIMENSIONS

Select Dimension tool > Pick line > Place dimension



Toggle function key **f8** at the top of your keyboard until the command line is, **<Ortho on>**

Pick any point and drag a short distance to the right.

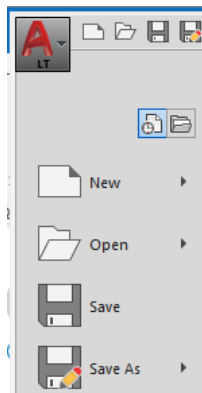
Type the distance **2**.

Drag a short distance up.

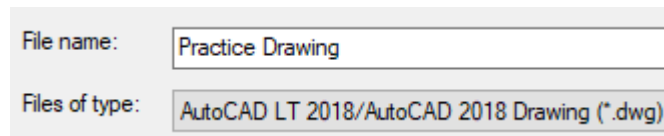
Type the distance **1**.

Select Linear tool > Pick two ends of angled line > Place dimension

SAVE A DRAWING



Select drop-down menu: **A > Save As >**



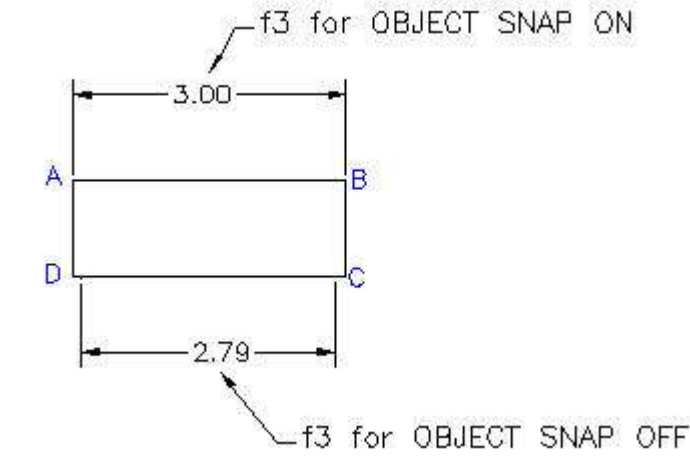
The **Save Drawing As** dialog box above opens.

Type the **File name** of the drawing > Practice Drawing >

Select an existing folder or create a new folder.

Select: **Save**.

OBJECT SNAP



The upper dimension above was placed by picking near to corners **A & B** and snapping automatically to the ends of line **A-B** with snap turned on followed by picking the location of the 3.00.

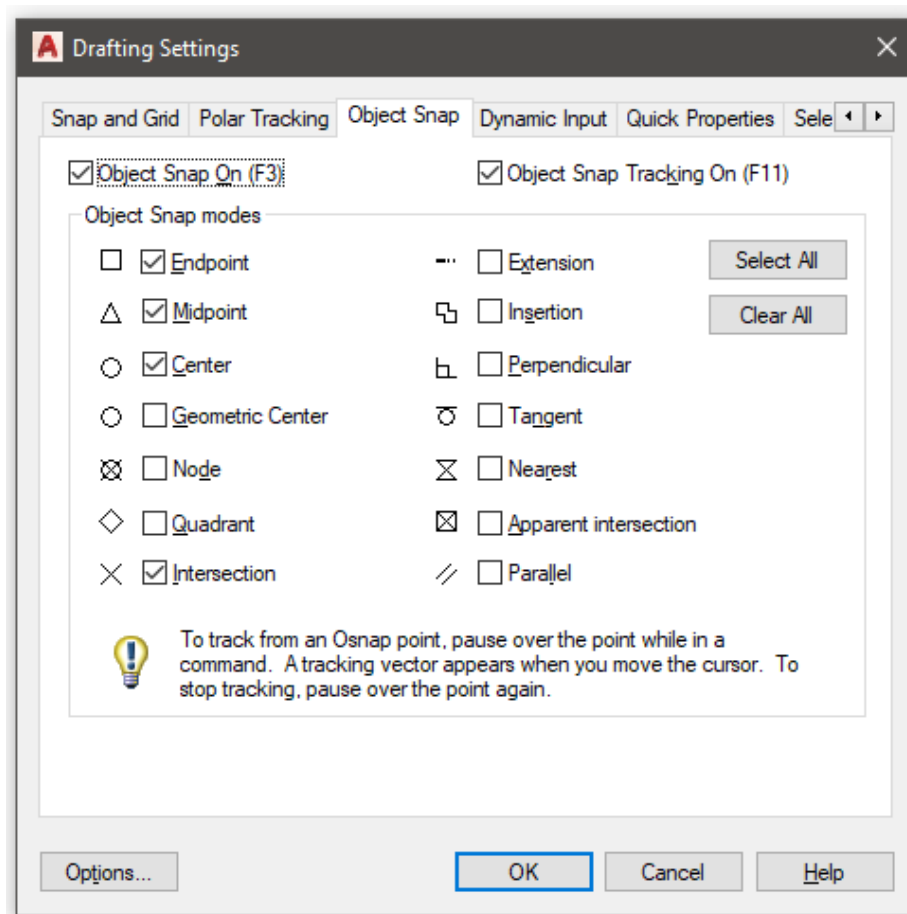
Select the function key **f3** to toggle Object Snap ON and Off

When adding the dimension to line **A-B** the mouse pointer will snap to the line intersections at **A** and **B** when the mouse pointer hovers near A or B.

The mouse pointer picks selected points near the line intersections at **C** and **D** when Object Snap is **OFF** as shown above.

The lower dimension above was placed by picking near to corners **D & C** with snap turned off followed by picking the location of the 2.79.

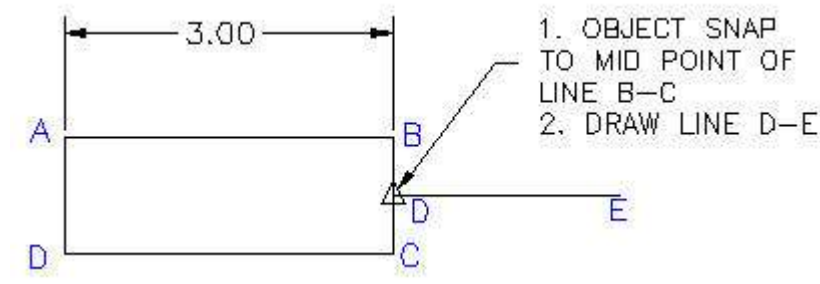
OBJECT SNAP DIALOG BOX



Type OS > Enter to open this dialog box.

Check > Endpoint > Midpoint > Center > Intersection > Apparent intersection > OK

Function key F3 also activates the Object Snap command.



The line D-E was drawn by picking near to D the mid-point of line B-C and dragging to point E. Because Object Snap was ON the line snapped to D.

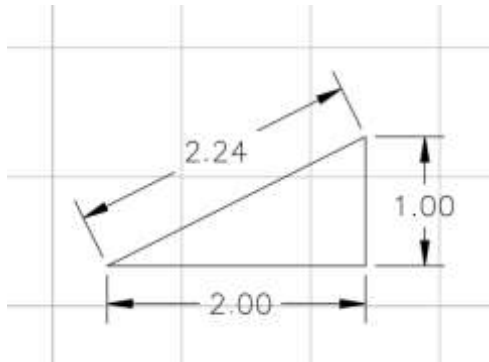
GRID

Command: GRID

GRID Specify grid spacing(X) or [ON OFF Snap Major Adaptive Limits Follow Aspect] <1.0000>: |

Type Grid or press F7 for Grid.

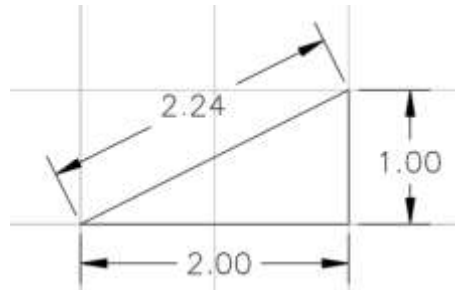
GRID Snap Control > F9 toggle Snap to ON or OFF



Grid On

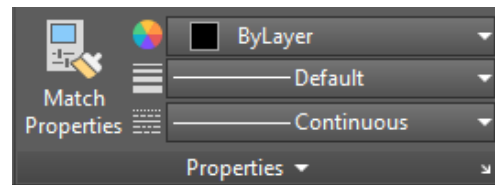
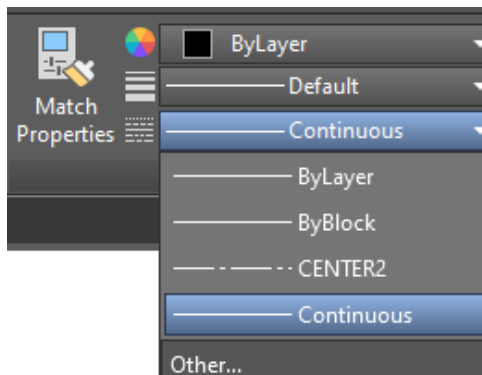


Type > M > Select Triangle > Pick corner > Enter



Drag triangle corner point to a grid corner.

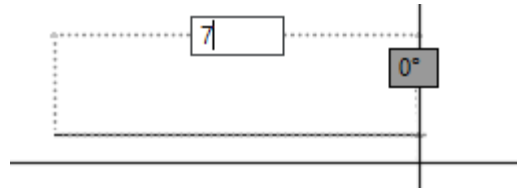
LAYERS and LINE TYPES



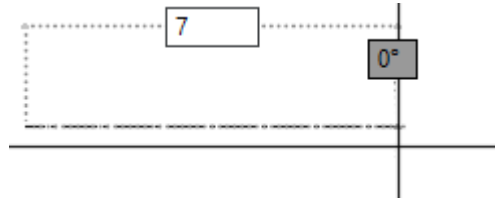
Properties

Select > Default > Drop Down Menu

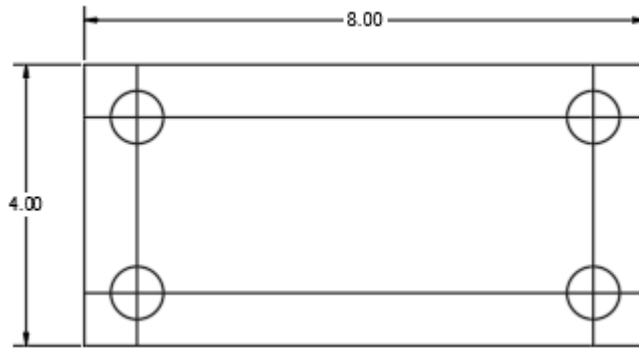
Select > Continuos



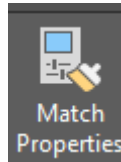
Select > By Layer > CONTINUOUS > L > Pick a point > Drag right > Type 7 > Enter



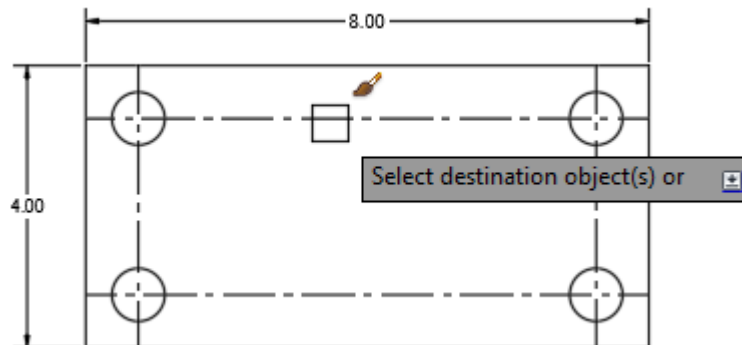
Select > By Layer > CENTER > L > Pick a point > Drag right > Type 7 > Enter



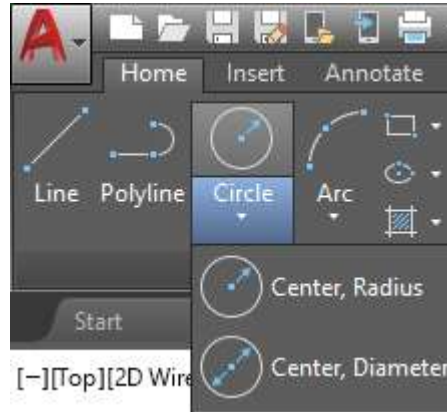
Changing Center Lines



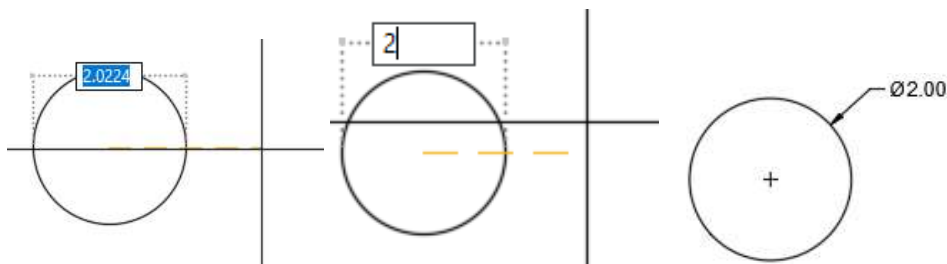
Select > Match Properties > Pick > Center Line > Pick lines to be changed



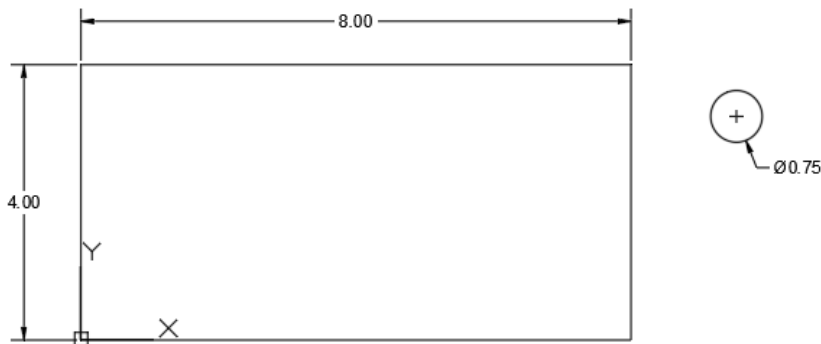
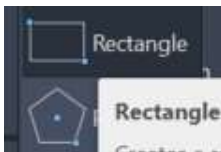
CIRCLE



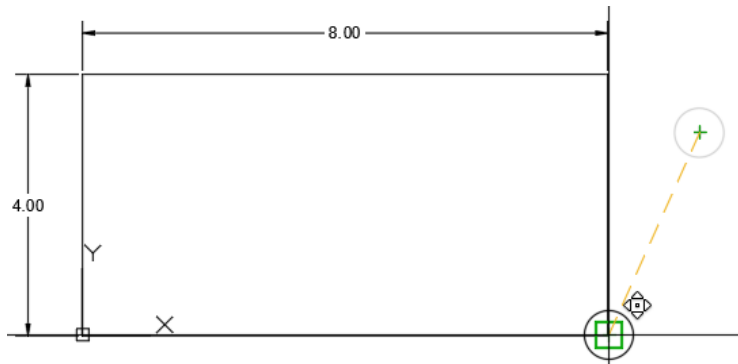
Select > Circle Tool > Center Diameter > Pick a point in the display area.
 GRID Control > F9 toggle Snap to ON or OFF



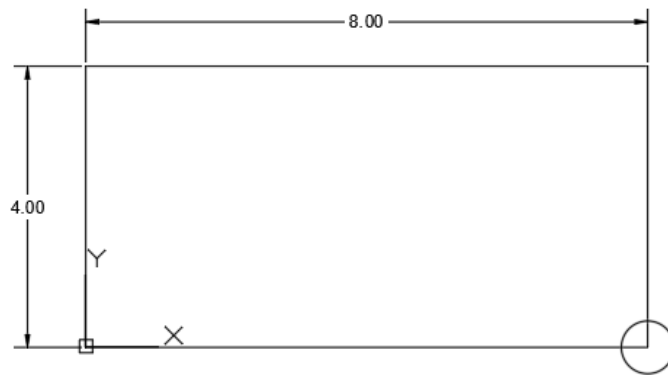
Drag Mouse > Pick a point > Type > 2 > Enter Dimension Tool > 2.00



Select Rectangle Tool > Type > 0,0 > Enter > 4,8 > Enter Draw 0.75 diameter circle.
 Rectangle corner is at the "Origin 0,0"



Type M > Select circle center > Enter > Drag circle to a corner.



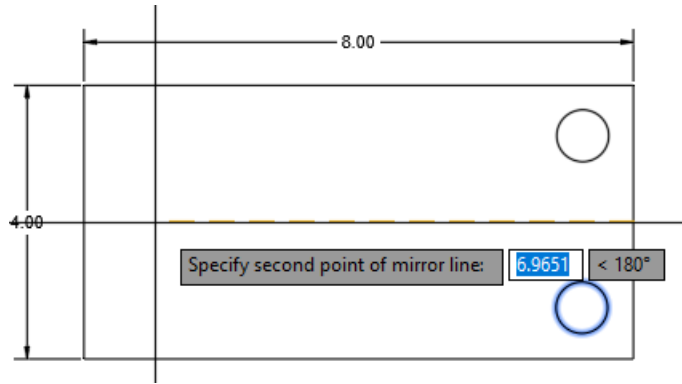
Circle is at a corner of the rectangle.

Press function key F8 > Toggle > Ortho on



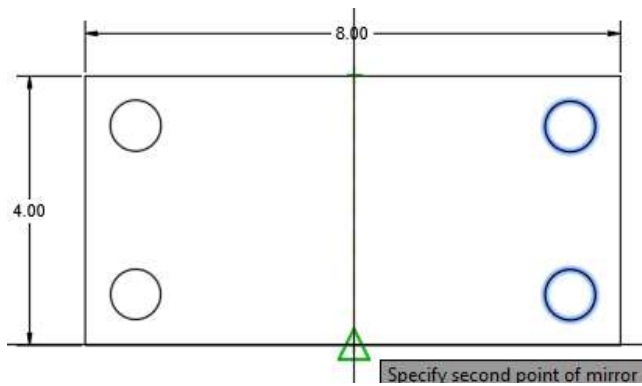
Type M > Select circle > Drag left > Type 0.75 > Enter

Type M > Select circle > Drag up > Type 0.75 > Enter



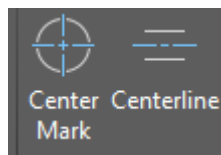
Mirror bottom right circle.

Type MI > select bottom right circle > Enter > Pick center of side > Drag left > Enter

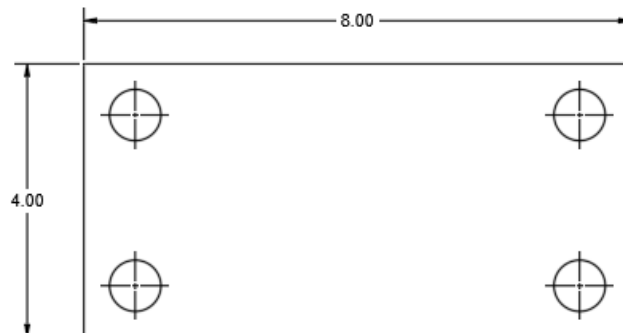


Mirror two right circles.

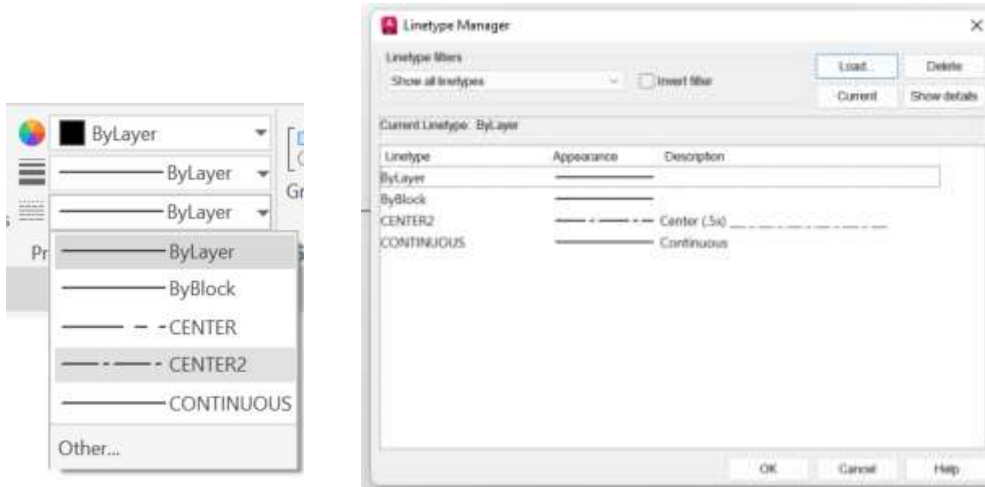
Type MI > Drag select two right circles > Enter > Pick center of top side > Drag down > Enter



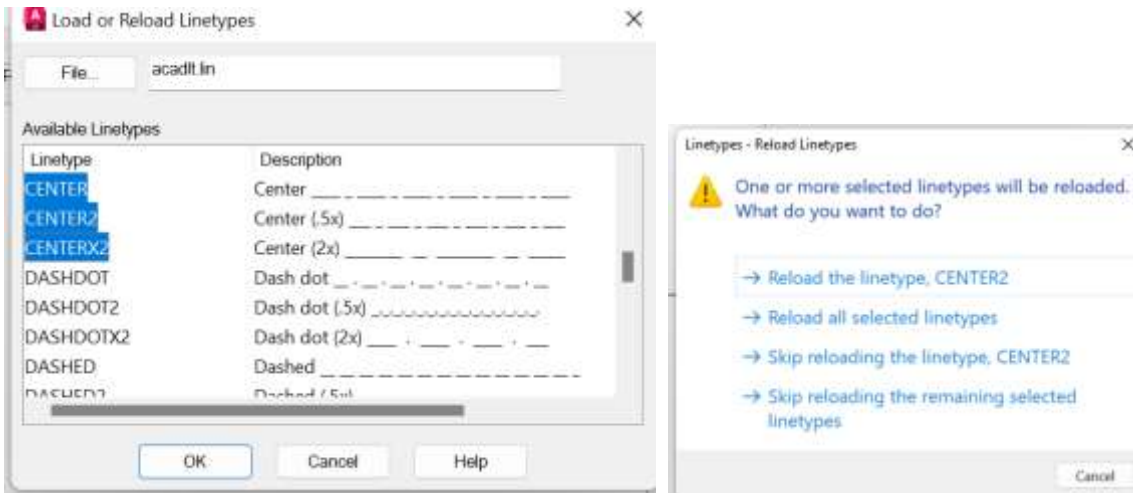
Center Mark Tool



LINETYPES



Select either "By Layer" > Select "Other" > Linetype Manager opens > Select "Load"



Ctrl key > Pick desired linetypes > OK

Select > Reload all selected linetypes > OK



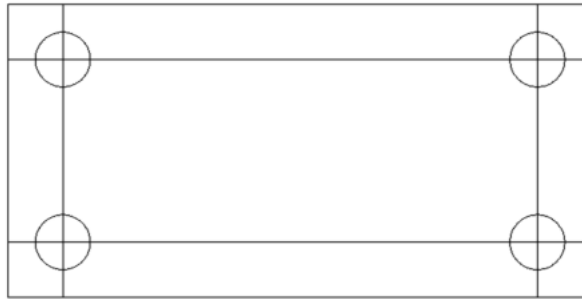
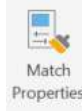
Select a centerline > in the Command Line
Type LTSCALE in the Command Line.



Enter a new scale factor > 3 > OK



New centerline scale.

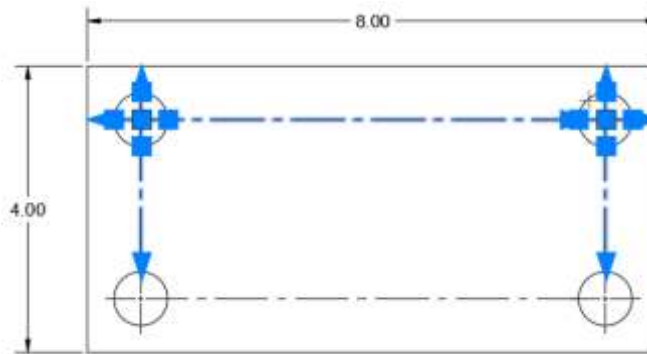


Select "Match Properties" > Pick the centerline above



Paint lines to be changed to centerlines.

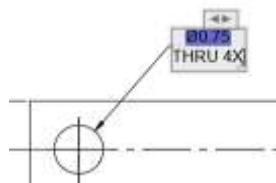
Select > Annotate > Select > Center Mark > Pick each circle.



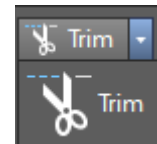
Drag circle center lines as shown.



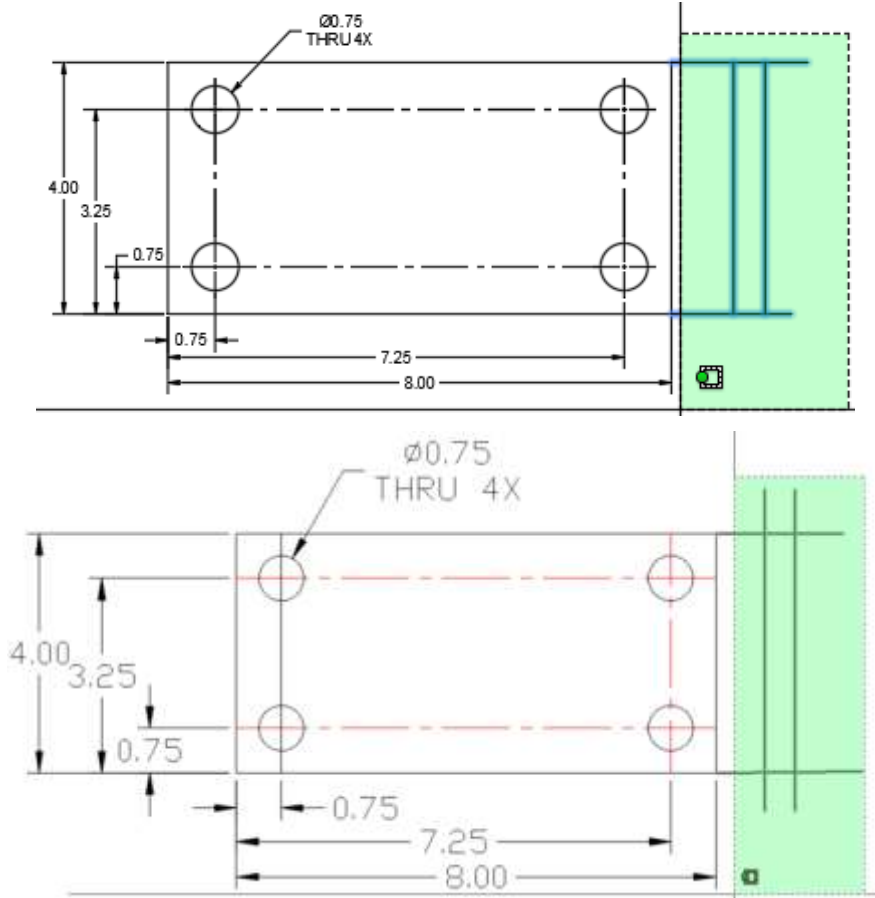
Select > Dimension Tool



Pick > Circle > Enter



Select > Trim tool



TRIM

Draw upper and lower horizontal extension lines.

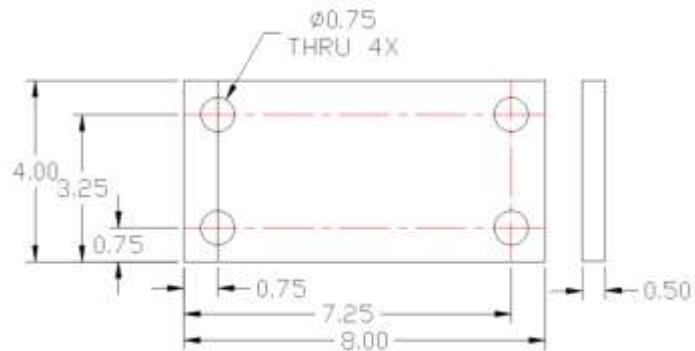
Add one vertical line between these two lines.

Make a copy of the vertical line (OFFSET 0.5 inches)

Select lines to be trimmed > Drag select Green Area > Type "Trim" in Command Line > Enter >

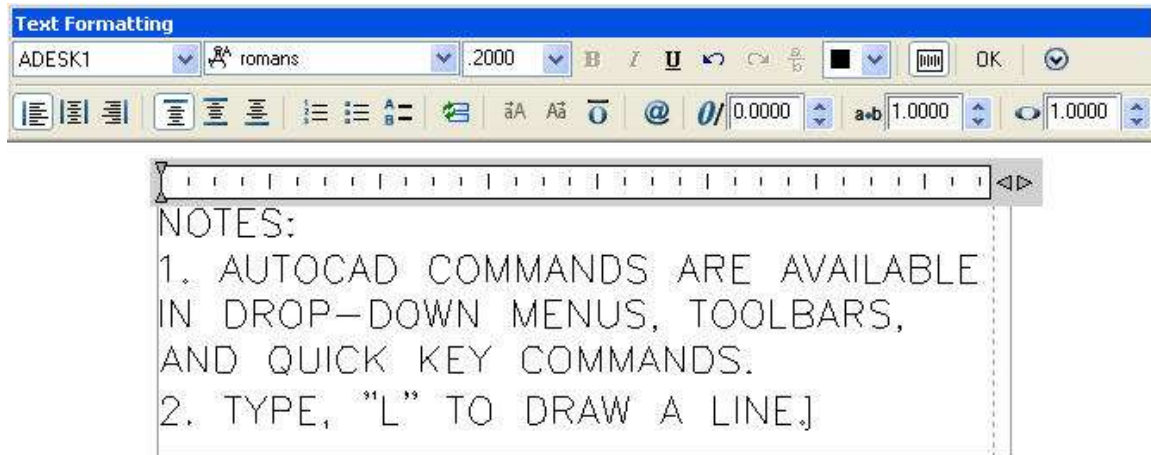
Pick lines to be trimmed.

Right side view of part is created by orthographic projection.



Fully dimensioned part with right side view lines after trimmed.

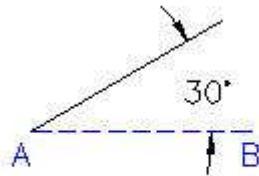
NOTE



Type **T** for Text in the **Command Line** at display bottom, and the, "Text Formatting" dialog box above will open.

Enter text as shown in the example above.

ROTATE



RO = ROTATE

Select the **Rotate** icon on the **Modify** toolbar **or type RO**

Pick line **A-B**.

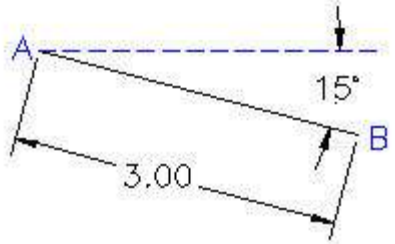
Pick base point at end **A**.

Type angle, **30**.

Press the **Spacebar** to end the command.

+30 rotates 30 degrees anticlockwise.

-30 rotates 30 degrees clockwise.



To Draw a Line at an Angle

Type **L** for Line.

Pick any point **A** and drag a short distance to the right as shown by the dashed line above.

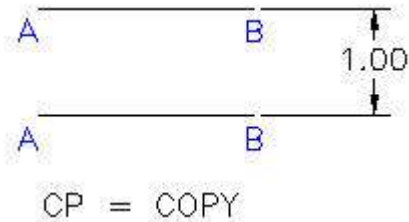
Type: **3 @ <-15**

Press the **Spacebar** to end the command.

f8, Ortho may be on or off.

3 is distance, **@** makes A the start point, **<** indicates angle, **-15** means 15 degrees anti-clockwise from horizontal.

COPY



Select the **Copy** icon on the **Modify** toolbar **or type CP**

Pick line **A-B**.

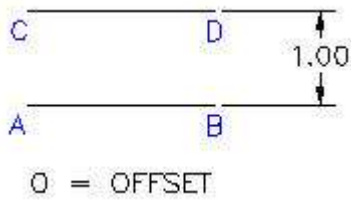
Press the **Spacebar**.

Drag to a point above line **A-B**.

Type distance **1**.

Press the **Spacebar** to end the command.

OFFSET



Select the **Offset** icon on the **Modify** toolbar **or type O**

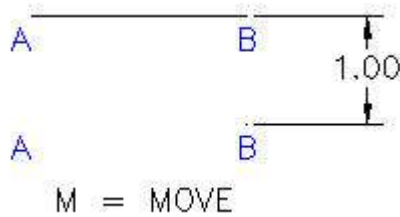
Type the distance **1**.

Pick line **A-B**.

Pick any point above A-B

Press the **Spacebar** to end the command.

MOVE



Select the **Move** icon on the **Modify** toolbar or type M

Pick line **A-B**.

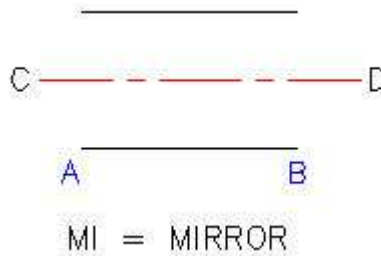
Press the **Spacebar**.

Drag to a point above line **A-B**.

Type distance **1**.

Press the **Spacebar** to end the command.

MIRROR



Select the **Mirror** icon on the **Modify** toolbar or type Mi

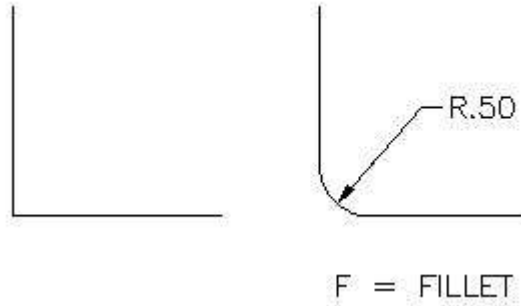
Pick line **A-B**.

Press the **Spacebar**.

Pick the **C** end of the centerline and drag to the **B** end.

Press the **Spacebar** to end the command.

FILLET & CHAMFER



Select the **Fillet** icon on the **Modify** toolbar **or type F**

Type **R** for Radius.

Type **.5**.

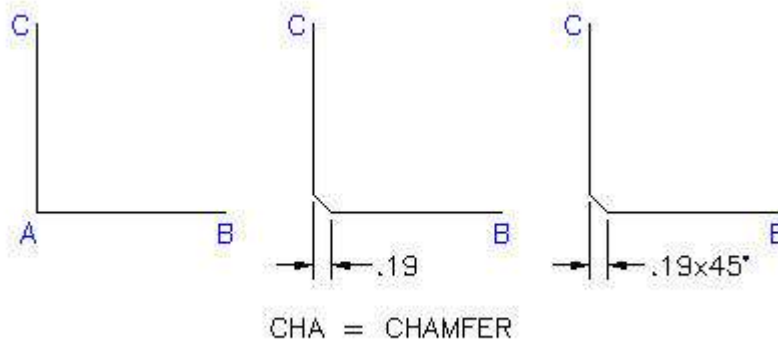
Pick the horizontal and vertical lines.

Press the **Spacebar** to end the command.

Select the **Radius Dimension** icon on the **Dimension** toolbar

Pick a point on the fillet.

Pick the location of the **R.50** dimension as shown above.



Select the **Chamfer** icon on the **Modify** toolbar **or type CHA**

Type, **D** for first chamfer distance.

Press the **Spacebar**.

Type, **.19**.

Press the **Spacebar**.

Press the **Spacebar**. (second chamfer distance is same as first)

Pick line **A-B**.

Pick line **B-C**.

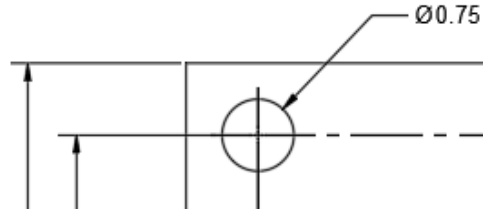
Pick the **Linear Dimension** icon on the **Dimension Toolbar**.

Pick the corners of the **0.19** inch chamfer as shown above.

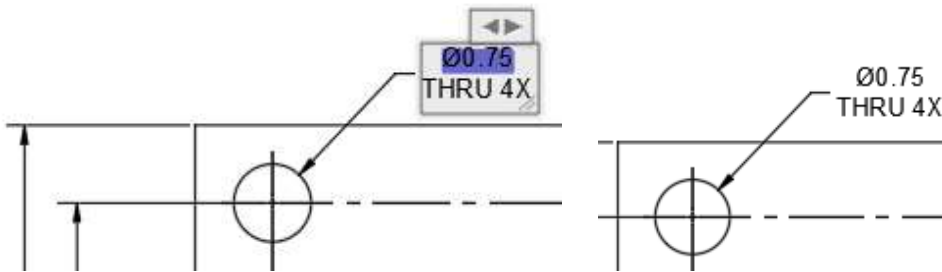
Pick the chamfer dimension location.

Press the **Spacebar**.

EDIT DIMENSIONS



Type > ED > Enter > Pick > Dimension > Edit dimension



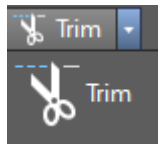
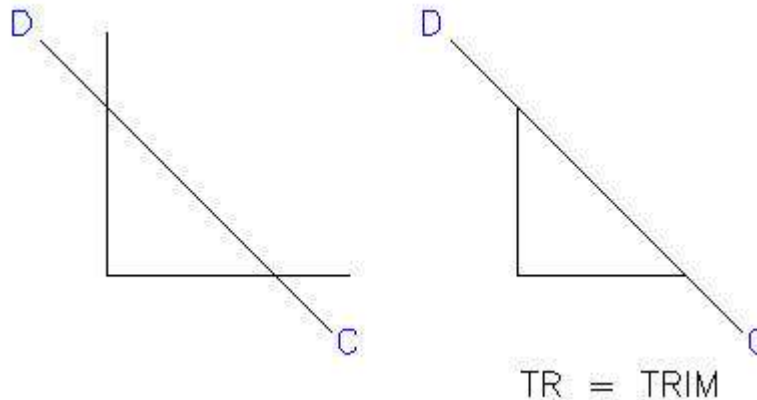
Type, **ED** for Edit.

Pick the **0.19** dimension.....see the "Text Formatting" dialog above.

Type, **x 45%%D** as shown here.

Select, **OK**.

TRIM



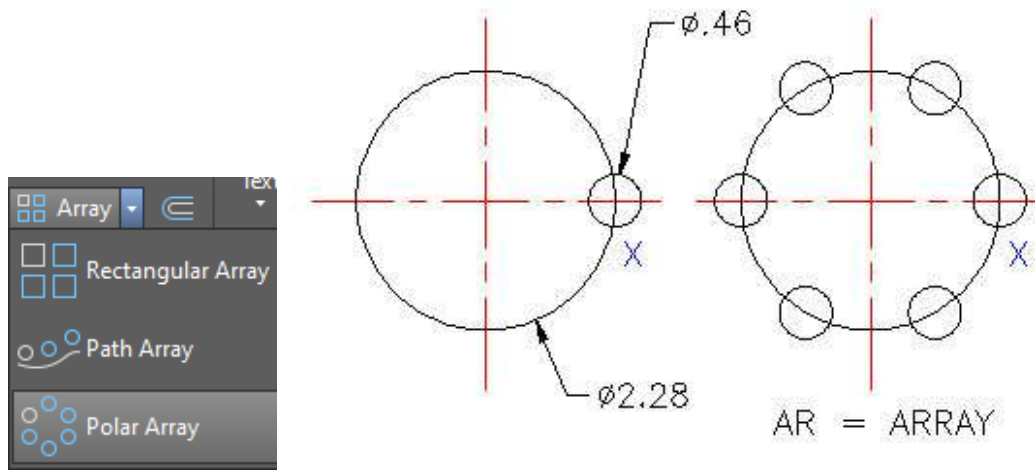
Select the **Trim** icon or type TR

Pick "Cut Line" **D-C**.

Pick the ends of the vertical and horizontal lines to be removed.

Press the **Spacebar** to end the command.

ARRAY - POLAR



Select the **Array** icon **or type AR**

Pick **Polar Array**.

Pick **Select objects**.

Select the 0.46 inch diameter circle in the drawing.

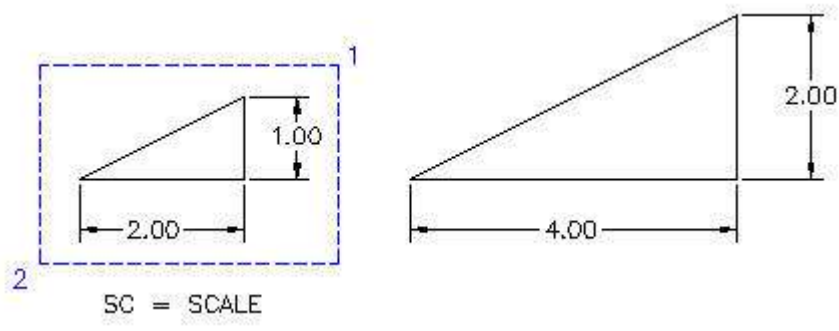
Pick **Center Point**.

Select the center point of the 2.28 inch diameter bolt circle on the drawing.

Pick **OK**.

NOTE: Rectangular array may be selected.

SCALE



Select the **Scale** icon on the **Modify** toolbar **or type AR**

Select Objects by picking point **1** and drag to point **2**, as illustrated above.

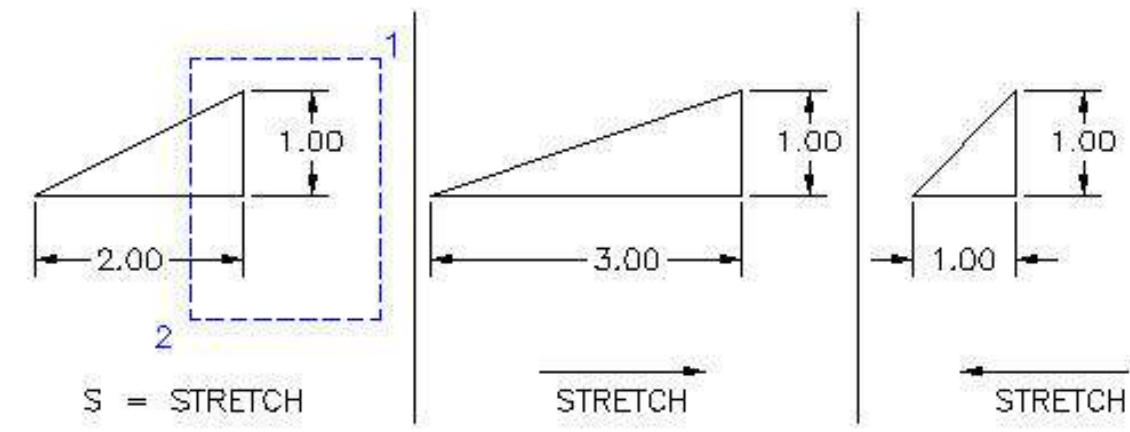
Press the **Spacebar**.

Specify base point by picking the lower left corner of the triangle.

Specify scale factor by typing **2**, for example.

Press the **Spacebar** to end the command.

STRETCH



Select the **Stretch** icon on the **Modify** toolbar **or type S**

The Stretch command is used to extend or reduce the length of a drawn object.

Select Objects by picking point **1** and drag to point **2**, as illustrated above.

This type of selection is called a **Crossing Window**.

Press the **Spacebar**.

Toggle function key **f8** at the top of your keyboard until you see, **Ortho on**.

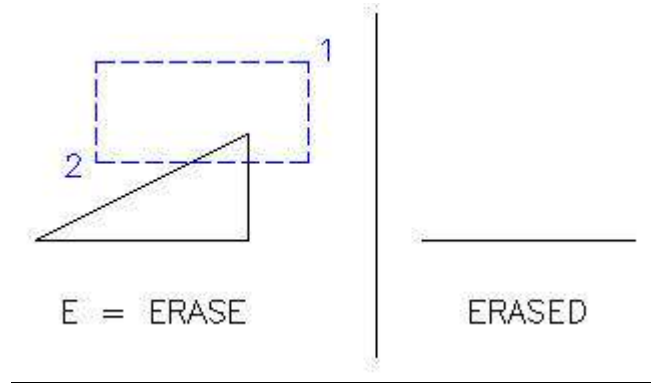
Specify base point by picking near to the **2**.

Drag to the right and type distance **1** **or drag to left and type distance 1**.

The 2.00 inch dimension becomes 3.00 inches **or the 2.00 dimension becomes 1.00 inch**.

Press the **Spacebar** to end the command.

ERASE



Select the **Erase** icon on the **Modify** toolbar **or type E**

Select Objects by picking point **1** and drag to point **2**, as illustrated above.

This type of selection is called a **Crossing Window**.

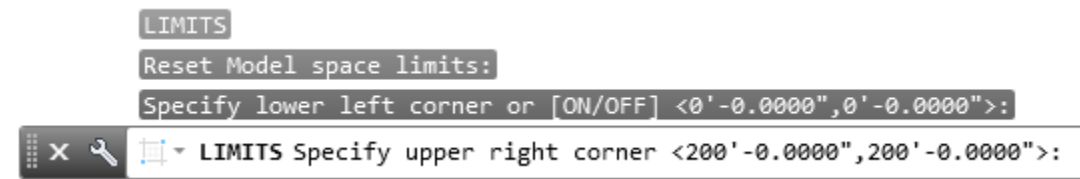
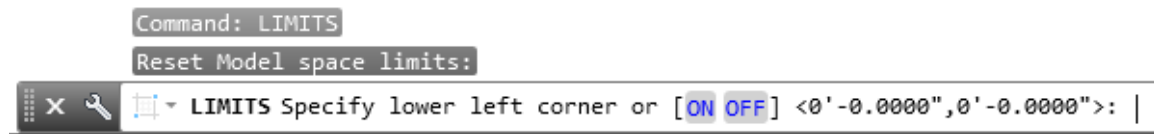
Press the **Spacebar**.

LIMITS - TO CHANGE DRAWING AREA



Select > View > UCS Icon

Type > Limits > Enter > 0,0 > Enter



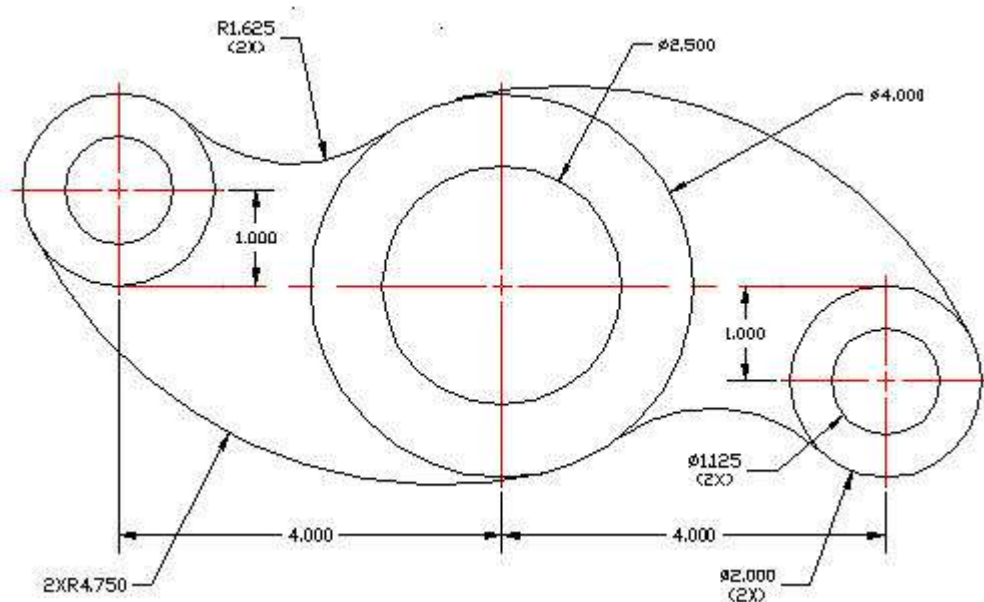
To Increase the Display Area

Type **Limits** on the command line.

Press the **SPACE BAR** to accept: < 0, 0 > as the lower left corner of the display.

Type: **200', c 200'** on the command line to change the upper right corner.

DIMENSION SCALE



To Increase the Dimension Scale

Type **DIMSCALE** on the command line.

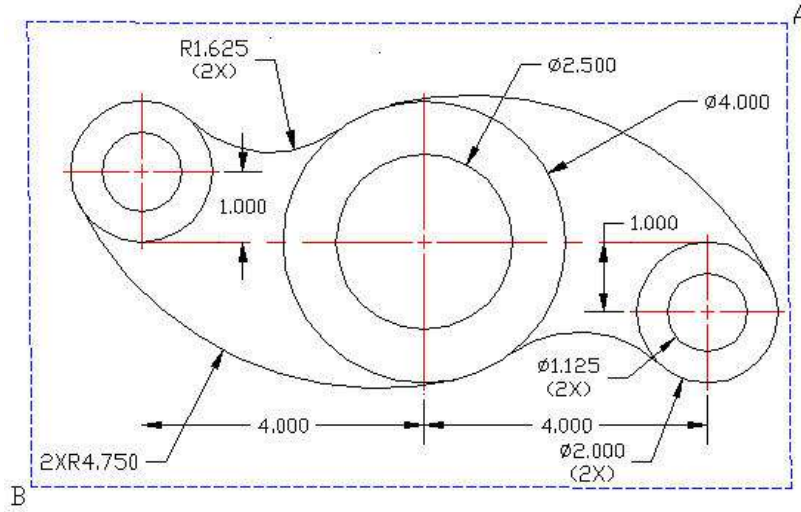
The present Dimension Scale is 1.0.

Change it by typing: **1.5**

Select the whole drawing by dragging the mouse pointer from A to B as shown below.

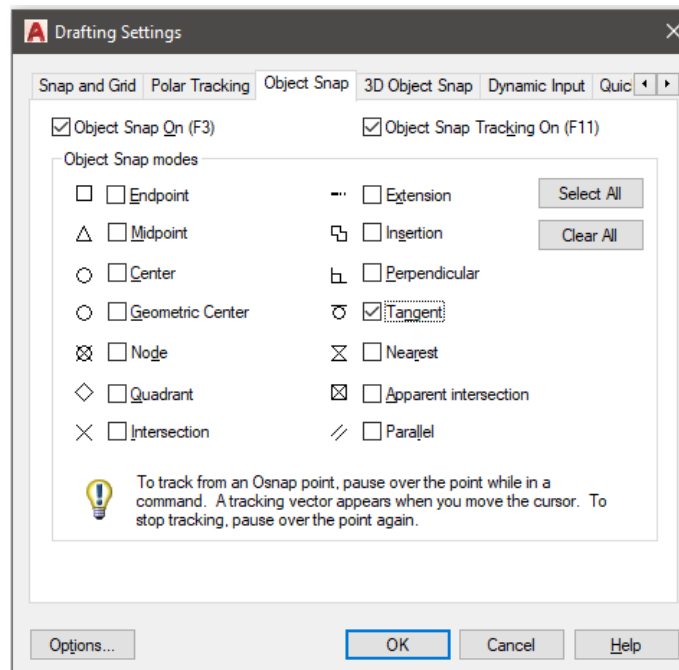
Pick drop-down menu: Dimension > Update.

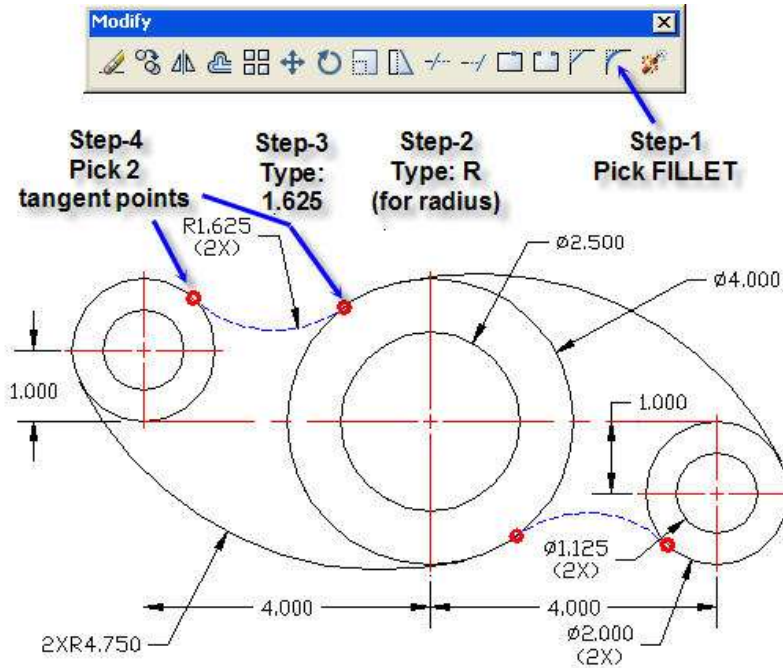
Press the **SPACE BAR** to end this command.



The dimension scale has been increased from 1.0 to 1.5 above.

TANGENT





Drawing the 2 Fillets with 1.625" Radius in the figure below.

Type OS for Object Snap on the command line.

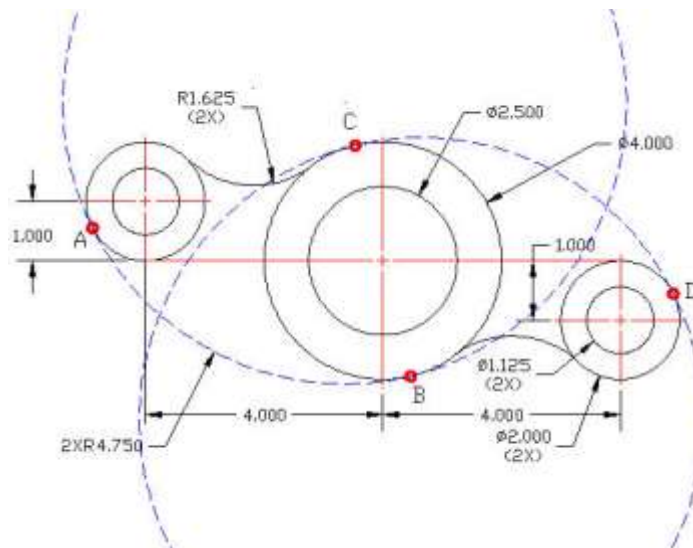
Step-1: Pick Clear All

Step-2: Pick the Tangent check box > OK

Drawing the 2 Fillets with 1.625" Radius (Continued)

Draw the two 1.625 inch radii by following the 4 steps illustrated above.

CIRCLE > TAN, TAN, RADIUS



Drawing the 2 Arcs with 4.750" Radius

Pick drop-down menu: **Draw > Circle > Tan, Tan, Radius**

Pick point A

Pick point B

Type: **4.75** for Radius

Pick point C

Pick point D

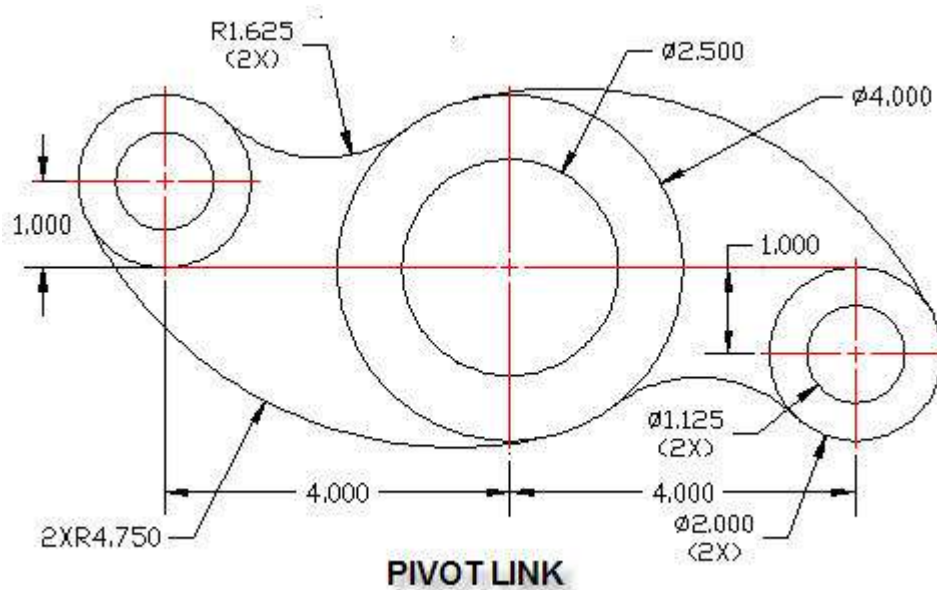
Trim the 4.750" diameter circles

Type: **TR** for Trim

Pick the 4.000" dia and 2.000" dia circles

Pick any point in the top half of the 4.750" radius circle

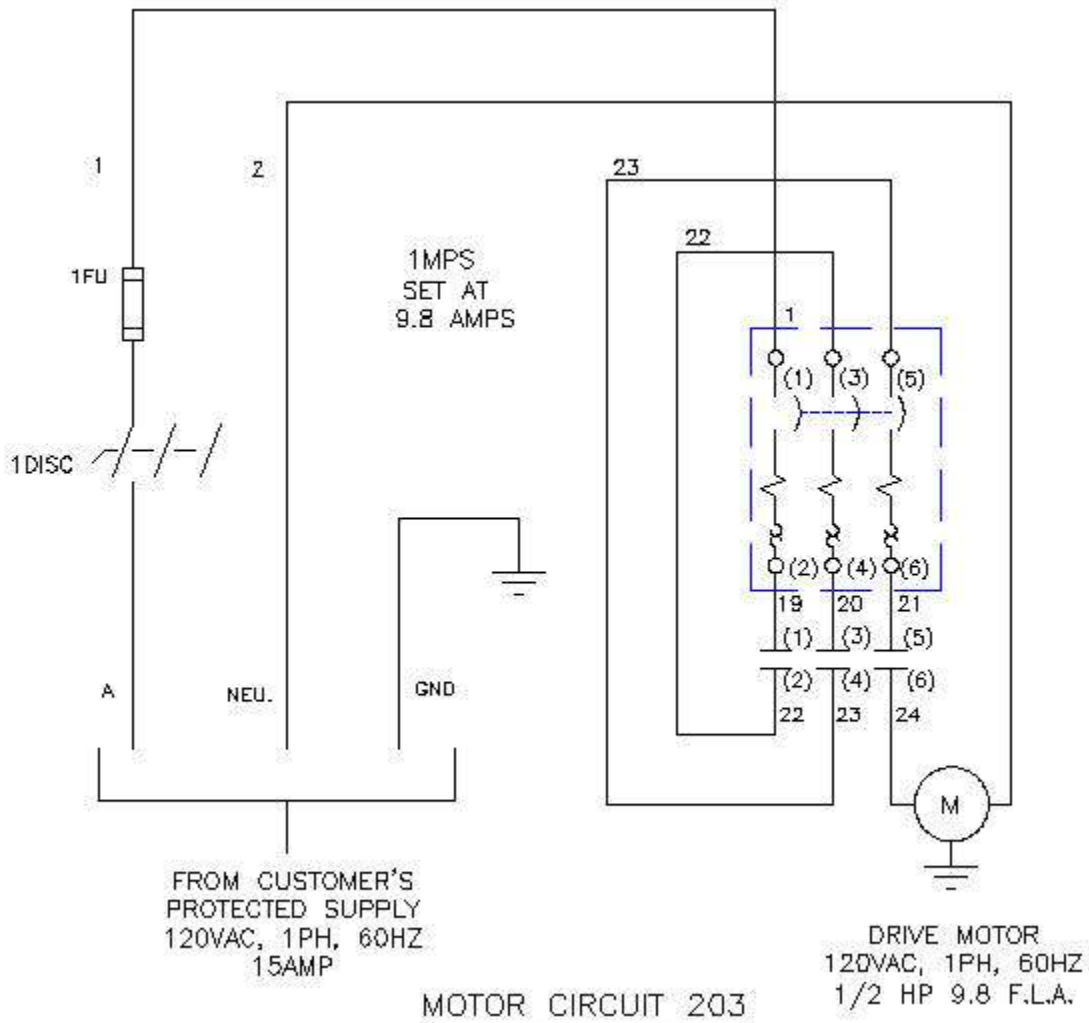
Repeat the Trim command on the bottom 4.750" radius circle



Finished Pivot Link drawing with title.

WBLOCKS

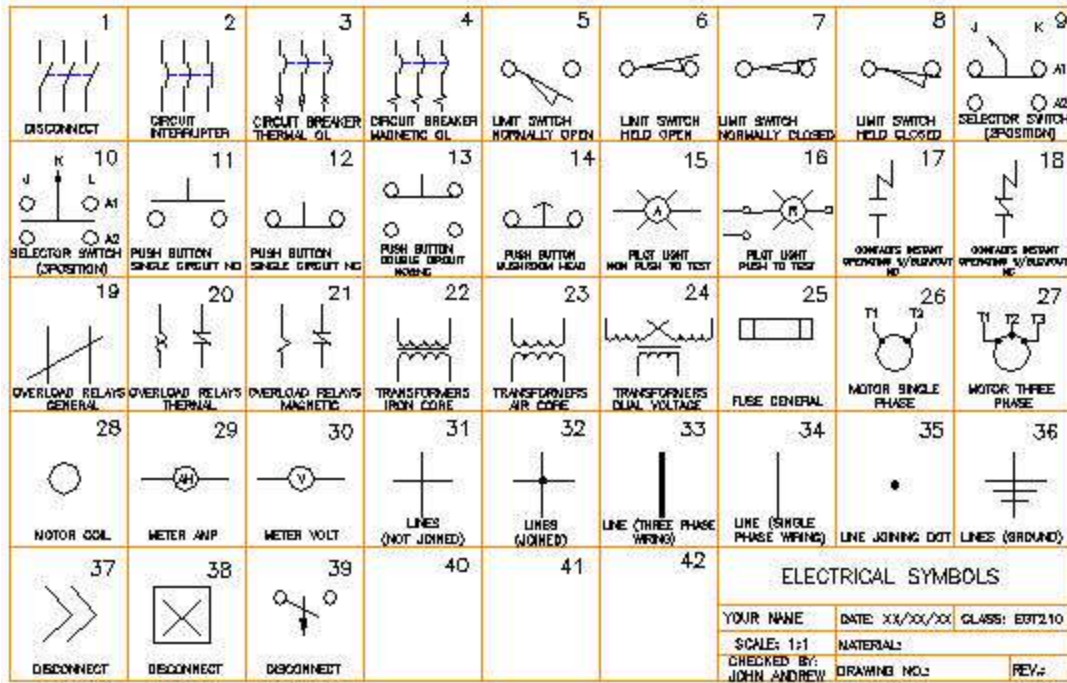
Write Blocks



Make the electrical circuit drawing above.

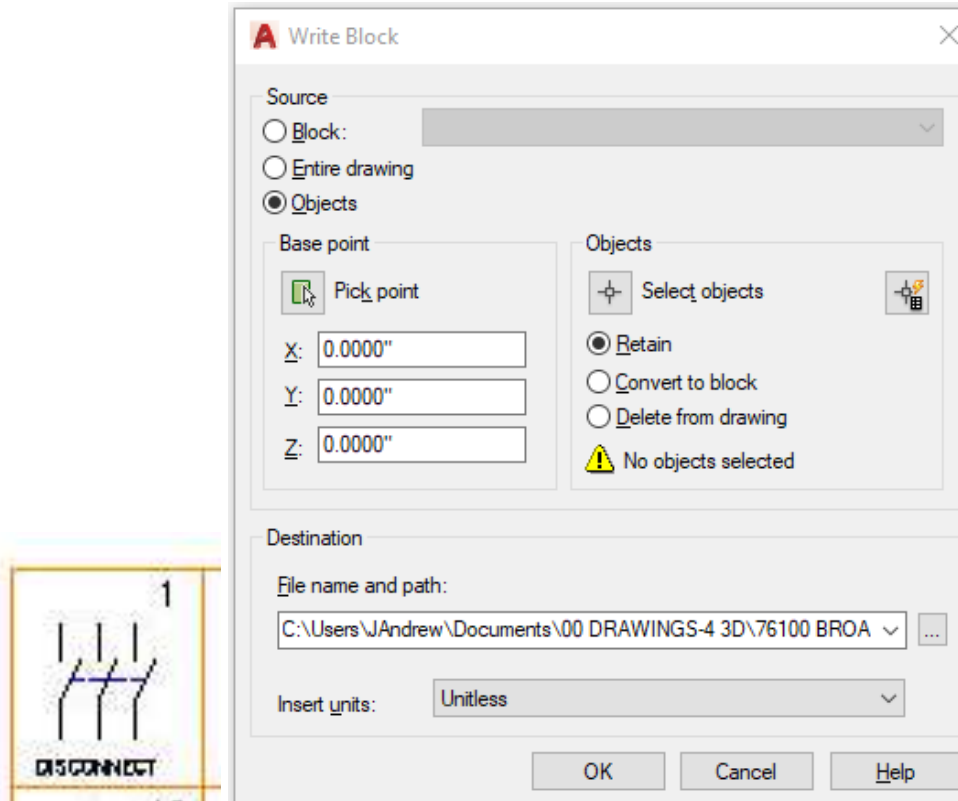
"Motor Circuit 203" using WBLOCKS.

See "Electrical Symbols" below.

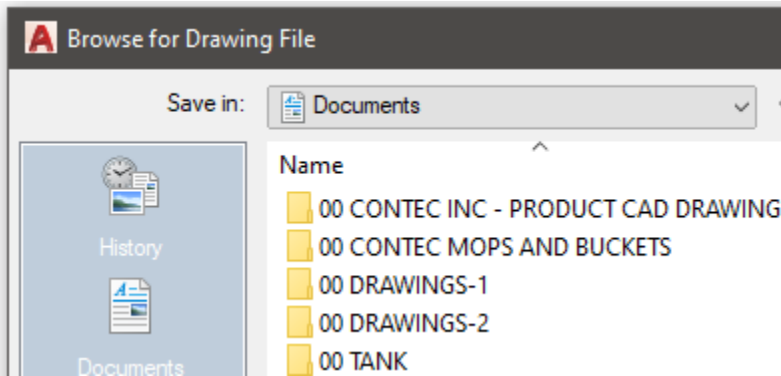


ELECTRICAL SYMBOLS

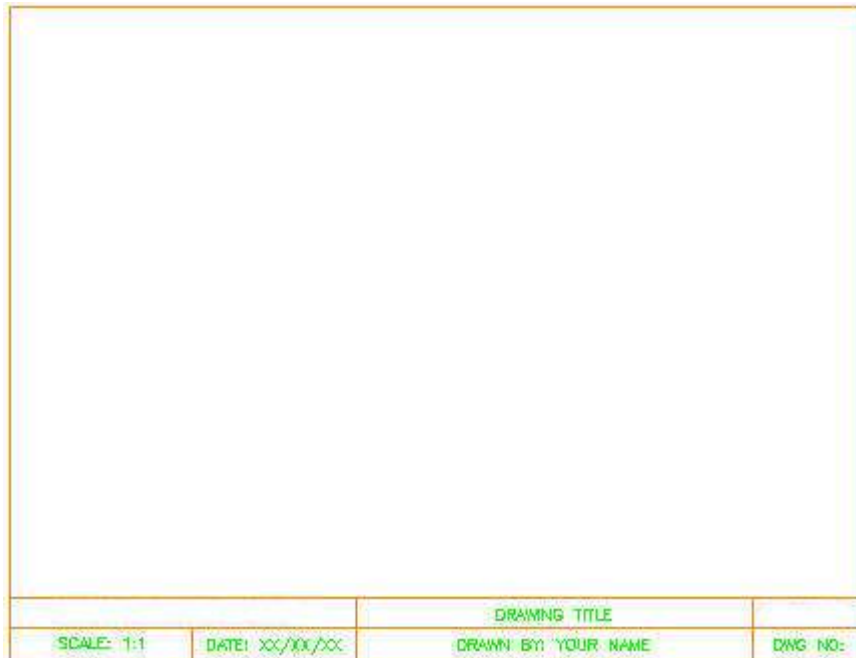
1. Open any AutoCAD drawing. The electrical symbols drawing above is for illustration purposes.



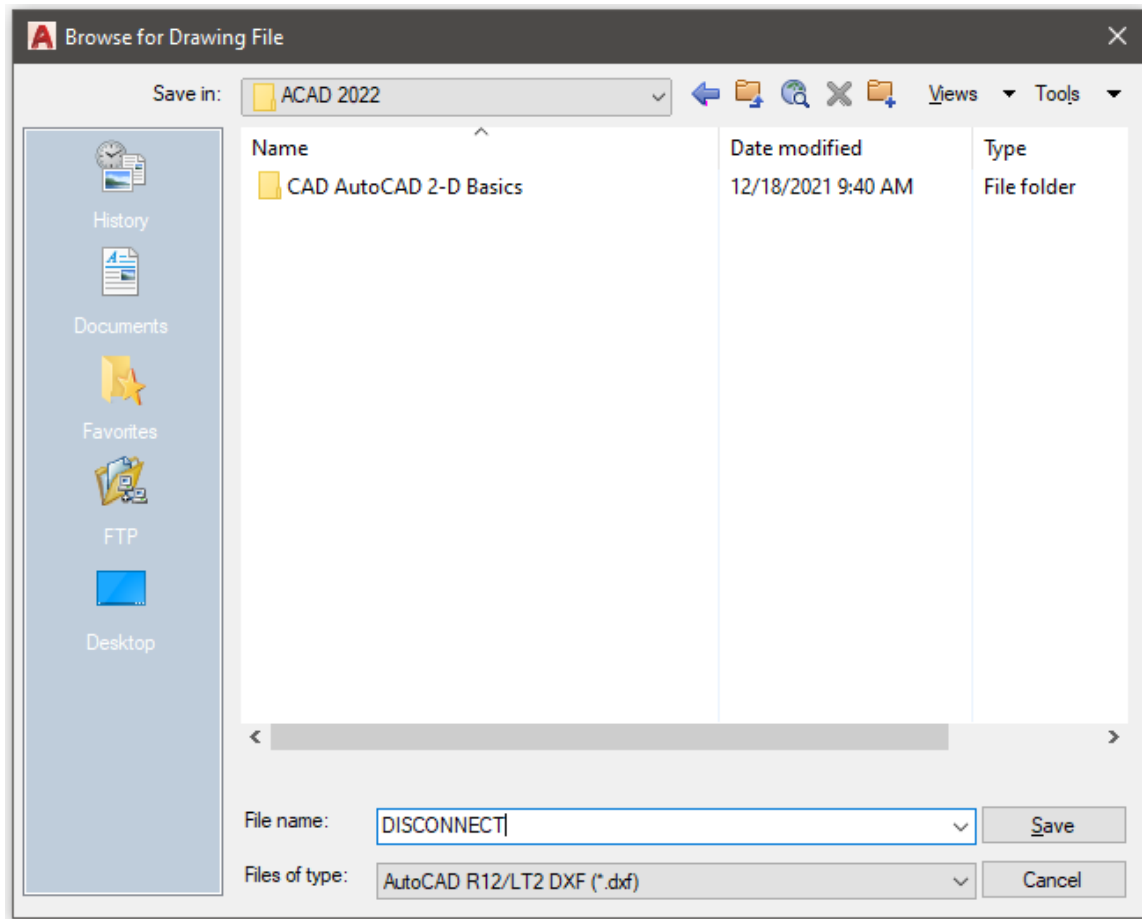
2. Type: WBLOCK in the Command line in AutoCAD.
3. Pick: Pick point in the Write Block dialog box.
4. Pick any point inside the Disconnect symbol.
5. Pick: Select objects in the Write Block dialog box.
6. Select the highlighted area.



7. Pick the Browse button with 3 dots.
8. Browse until you find the drawing folder.
9. Change the title from: new block.dwg to DISCONNECT.dwg
10. Press OK.



11. Open a blank title-block template as shown above.

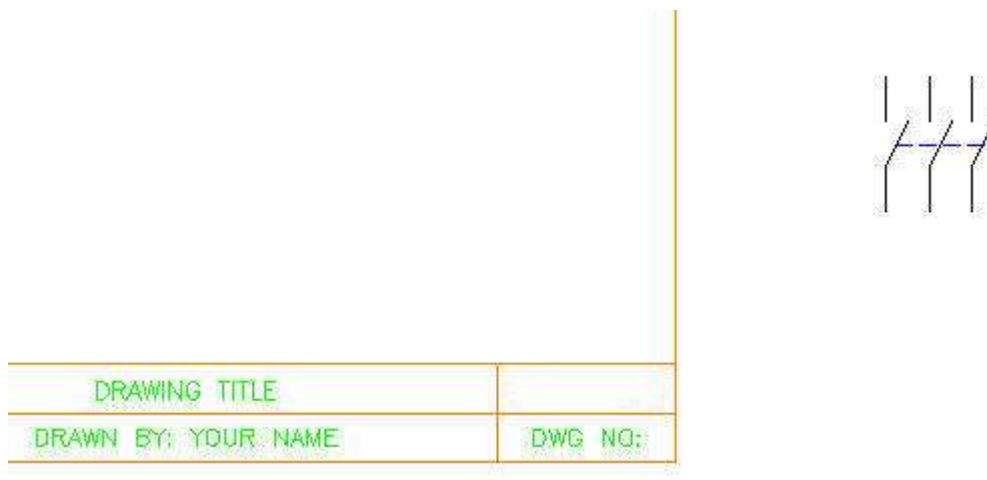


12. Select drop-down menu: Insert > Block.

13. The Insert dialog box shown above will open.

14. Pick: Browse and select the DISCONNECT drawing file in your folder.

15. Pick: OK.



- 16. Pick a point for the DISCONNECT block on the right side of the template as shown above.
- 17. Repeat the above steps for each Electrical Symbol needed to complete the: MOTOR CIRUIT 203 drawing.

This is the end of Section 3. Common AutoCAD Commands

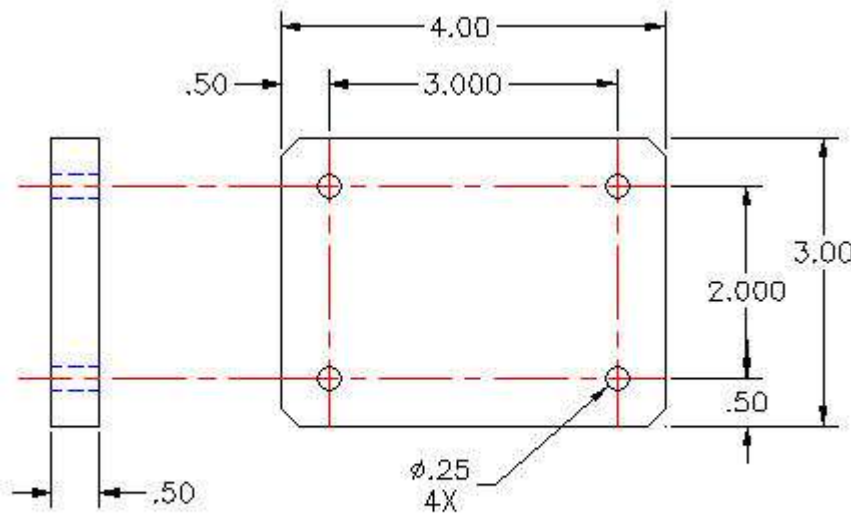
FLAT PLATE WITH FOUR HOLES



The 3-dimensional solid model of a flat plate shown above could be drawn with all versions of AutoCAD from release 14 to 2007.

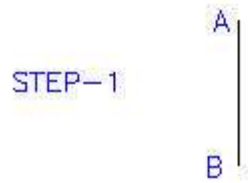
EXAMPLE-1

The flat plate with four holes illustrated above has been chosen to demonstrate some of the most common AutoCAD commands used to make many types of engineering drawings.



PART M2297 REV B

The dimensioned front and side views of the example part are shown above.



Step-1 Draw Line A-B

Open AutoCAD.

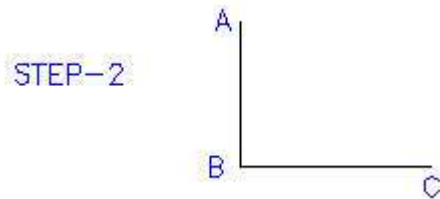
Type on the command: **L** for Line.

Toggle function key **F8** at the top of your keyboard until you see, **Ortho on**.

Pick any point **A** and drag down a short distance.

Type the distance **1.5**

Press the **Spacebar** or **Enter Key** to end the command.



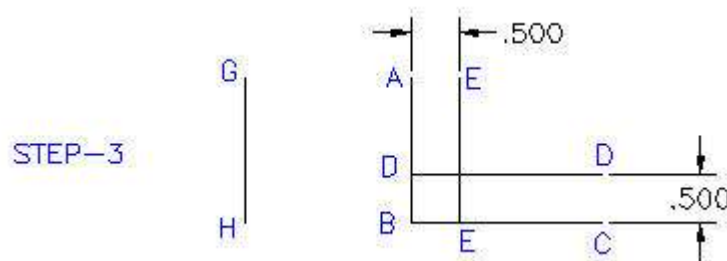
Step-2 Draw Line B-C

Press the **Spacebar** or **Enter Key** to repeat the last command.

Pick any point **B** and drag to the right a short distance.

Type the distance **2**.

Press the **Spacebar** or **Enter Key** to end the command.



Step-3 Draw Offset Lines D-D & E-E

Pick line B-C.

Type **O**, for Offset on the command line.

Type the distance **.5**.

Pick line B-C.

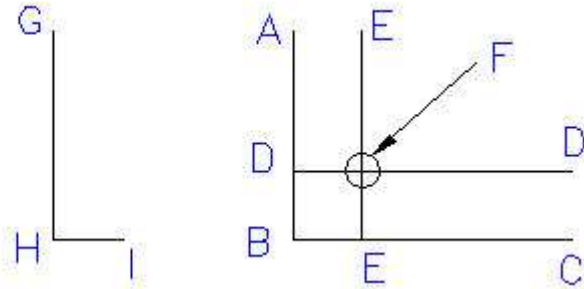
Pick a point above line B-C.

Pick line A-B.

Pick a point to the right of line A-B.

Press the **Spacebar**.

STEP-4



Step-4 Draw Circle F

Type **C** for Circle on the command Line.

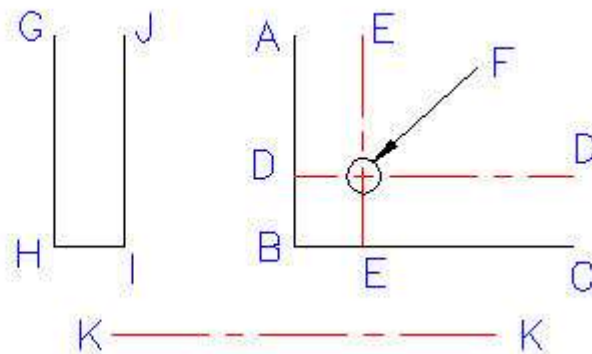
Pick the intersection of lines D-D and E-E.

Type **D** for Diameter.

Type **.5**.

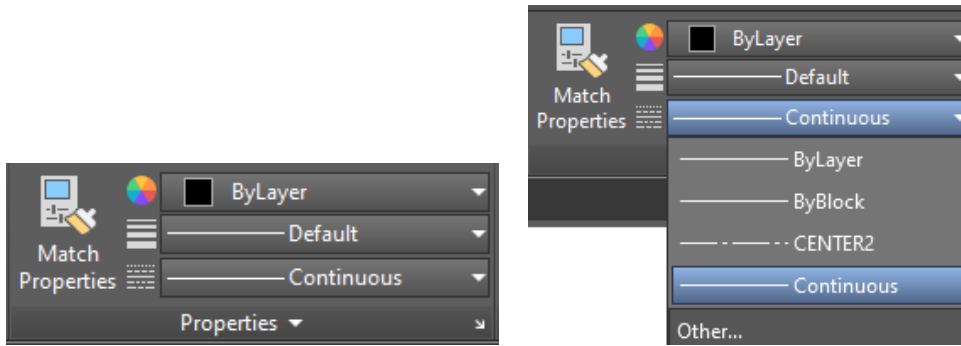
Press the **Spacebar**.

STEP-5



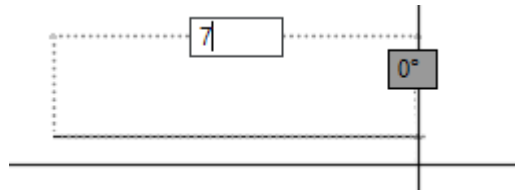
Step-5 Change Lines D-D & E-E to Centerlines

LAYERS and LINE TYPES – ACAD 2019

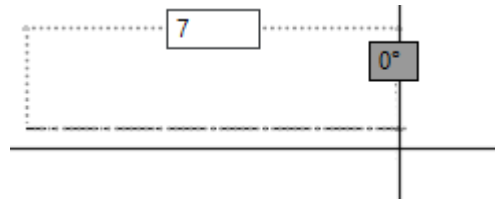


Select > Continuous drop down menu > Other > Enter

CENTER 2 > OK



Select > By Layer > CONTINUOS > L > Pick a point > Drag right > Type 7 > Enter



Select > By Layer > CENTER > L > Pick a point > Drag right > Type 7 > Enter

Select the Center Line Layer, pictured here.

Type L on the command Line.

Draw center line K-K any length.

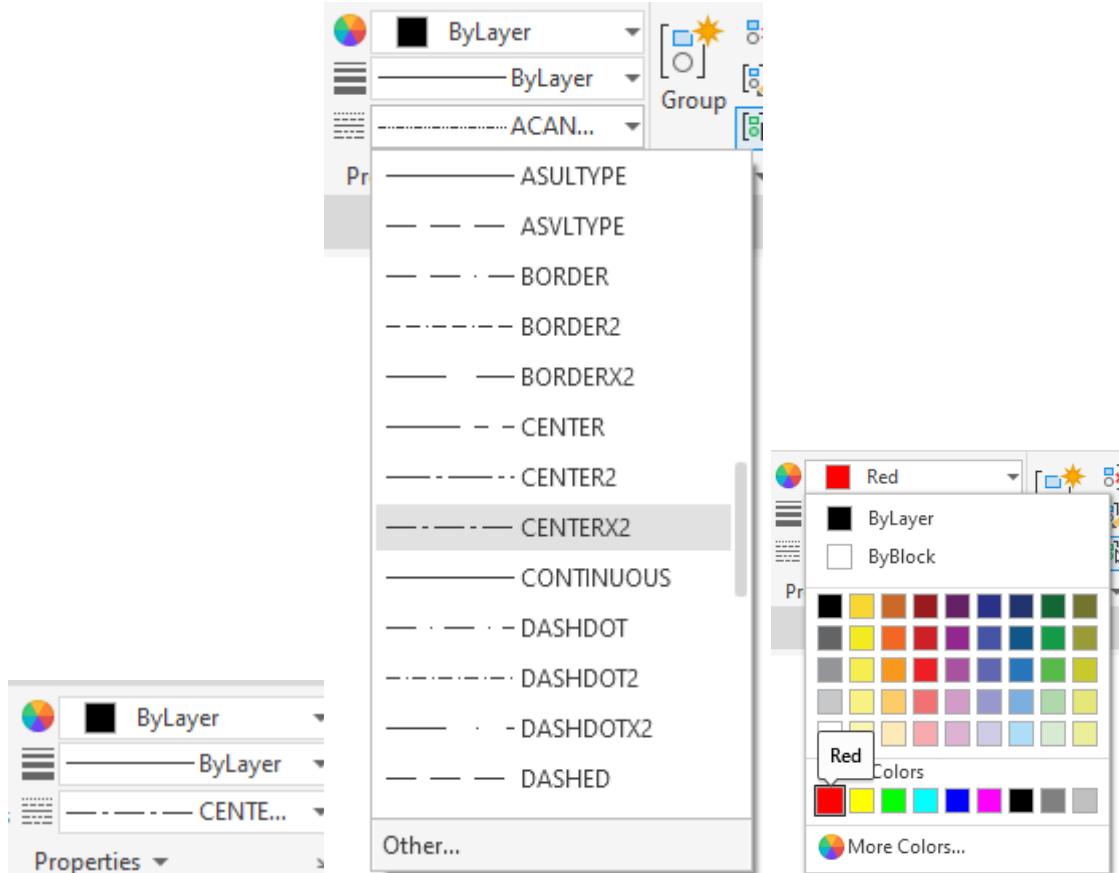
Pick the "Paint Brush" icon shown above.

Pick any point on center line K-K.

Pick lines D-D & E-E.

Press the Spacebar.

LAYERS and LINE TYPES – ACAD 2020



Select > BYLAYER > Drag drop down to > CENTERX2 > OK

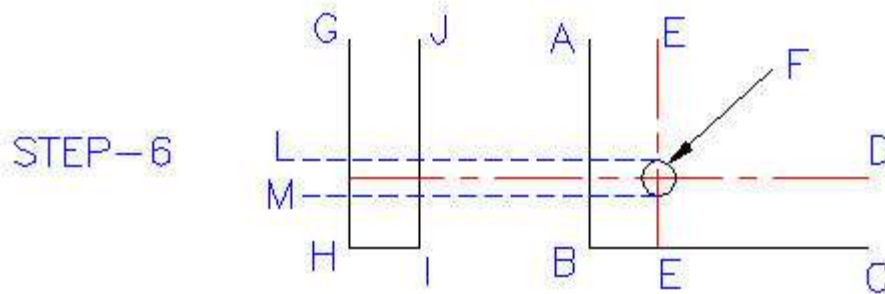


Select > Bylayer > Drag drop down to > CENTERX2 > OK

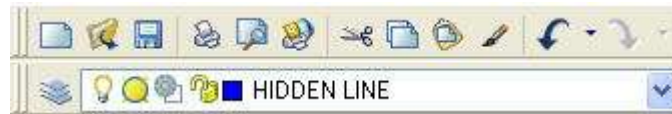
Select > Red

Type > L > Pick a point > Drag right > Pick a point > Enter





Step-6 Draw Hidden Lines L & M



Select the **Hidden Line Layer**, pictured here.

Type **L** on the command Line.

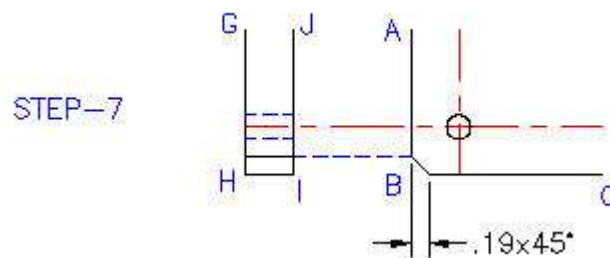
Pick the intersection of line E-E and the top of circle F.

Draw hidden line **L** any length.

Pick the intersection of line E-E and the bottom of circle F.

Draw hidden line **M** any length.

Press the **Spacebar**.



Step-7 Trim Hidden Lines L & M

Type **TR** for Trim on the command Line.

Select Lines H-G and J-I.

Pick a point on line L that extends to the right of line J-I.

Continue picking line segments that need to be trimmed off.

Press the **Spacebar**.

Select the **Chamfer** icon on the **Modify** toolbar

Type, **D** for first chamfer distance.

Press the **Spacebar**.

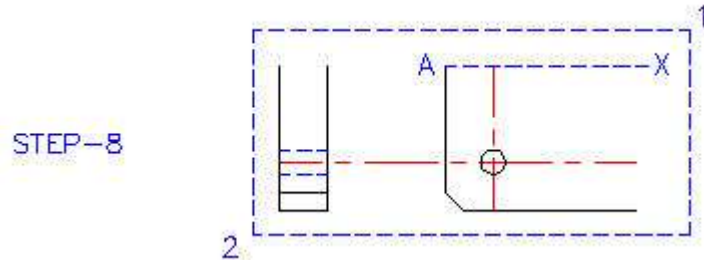
Type, **.188**.

Press the **Spacebar**.

Press the **Spacebar**. (second chamfer distance is same as first)

Pick line **A-B**.

Pick line **B-C**.



Step-8 Mirror About Imaginary Line A-X

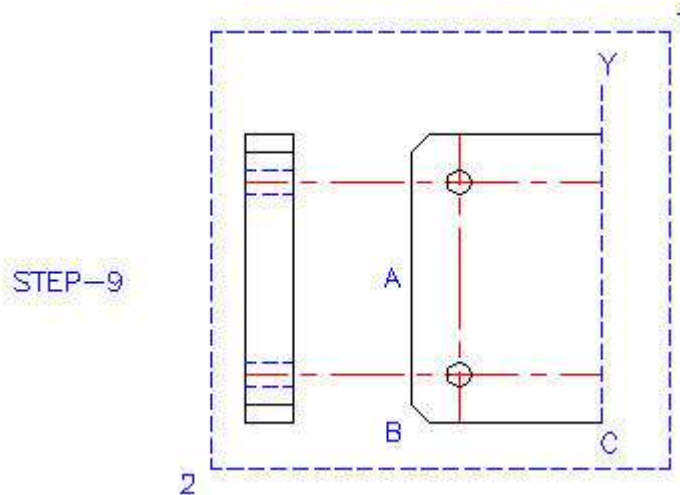
Toggle function key **f8** until you see, **Ortho on**.

Type **MI** for Mirror on the command Line.

Select the object by picking point 1 and dragging to point 2.

Select point **A** and drag to the right and pick any point **X**.

Press the **Spacebar**.



Step-9 Mirror About Imaginary Line A-X

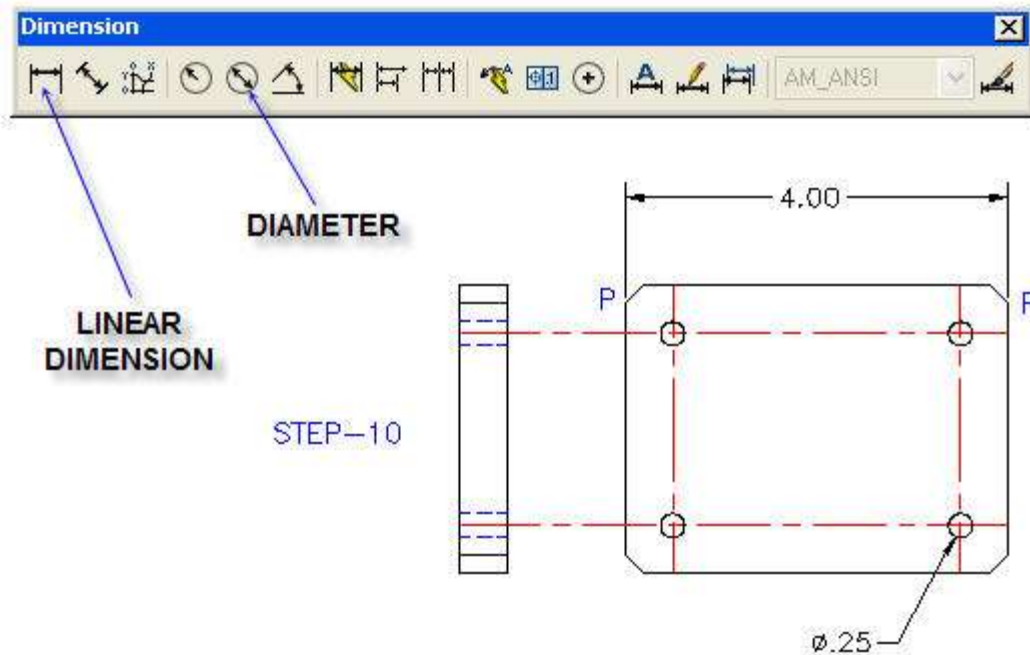
Toggle function key **f8** until you see, **Ortho on**.

Type **MI** for Mirror on the command Line.

Select the object by picking point 1 and dragging to point 2.

Select point **C** and drag to the right and pick any point **Y**.

Press the **Spacebar**.



STEP 10 Add Dimensions to Complete the Drawing

Pick the **Linear Dimension** icon on the Dimension **Toolbar**.

Pick corners **P** and **R**.

Pick the dimension location at **4.000**.

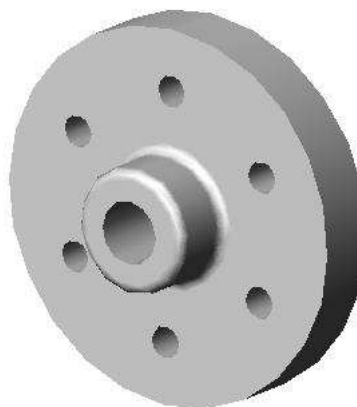
Pick the **Diameter** icon.

Pick a point **on** the circle.

Pick the dimension location at **$\phi.250$** .

Press the **Spacebar**.

This is the end of Section 4. Flat Plate With Four Holes



GENERIC HUB

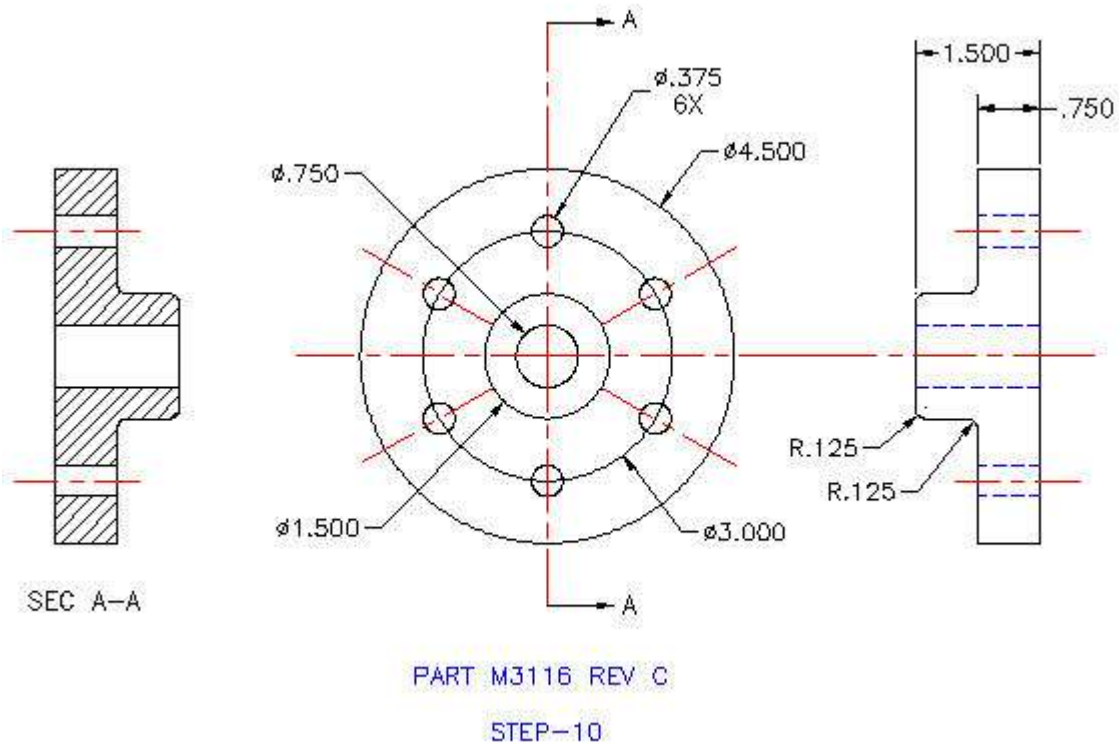
The 3-dimensional solid model of the part shown above could be drawn with all versions of

AutoCAD from release 14 to 2007.

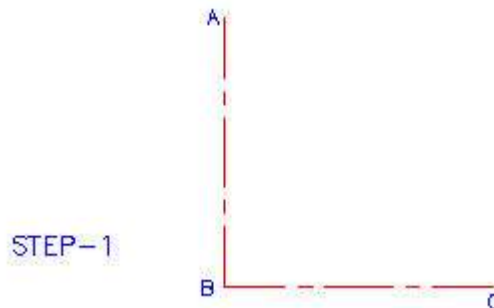
EXAMPLE-2

The AutoCAD commands used to make a dimensioned engineering drawing of generic hub pictured above demonstrate more AutoCAD commands.

Open any version of AutoCAD.



The dimensioned front, side, and section views of the hub are shown above.



Step-1 Draw Lines A-B and B-C

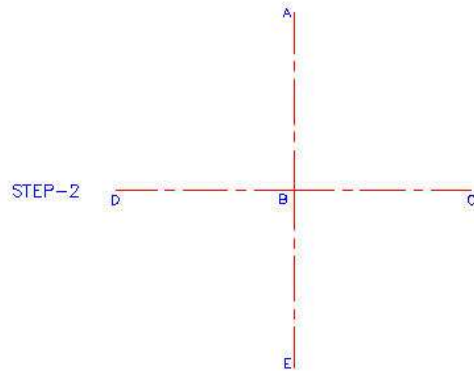
Type the command: **L** for Line.

Toggle function key **f8** at the top of your keyboard until you see, **Ortho on**.

Pick any point **A** and drag down a short distance.

Type the distance **5**.

Press the **Spacebar** or **Enter Key** to end the command.



Step-2 Draw Center Lines A-E & C-D

Toggle function key **f8** until you see, **Ortho on**.

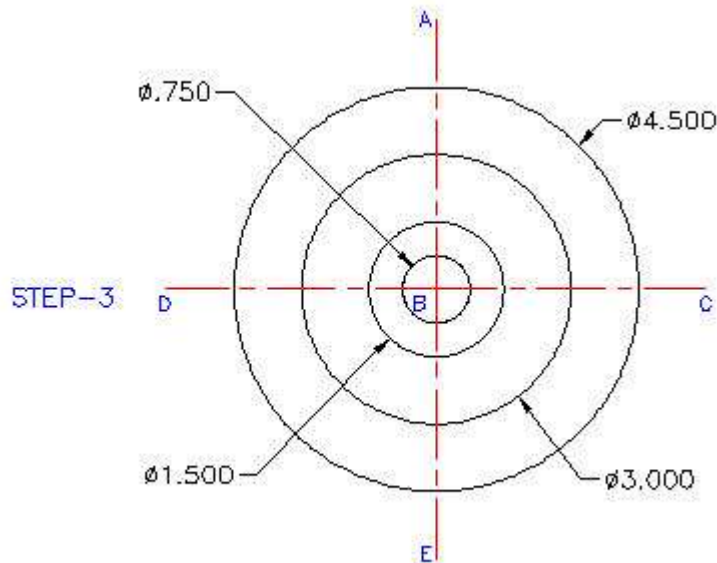
Type **MI** for Mirror on the command Line.

Select line **A-B**.

Select point **B** and drag to the right and pick point **C**.

Repeat above steps to obtain line **B-D** by mirroring line **B-C** about line **A-B**.

Press the **Spacebar**.



Step-3 Draw 4 Circles

Type **C** for Circle on the command Line.

Pick center point at B.

Type **D** for Diameter.

Type **.75** for the inner circle.

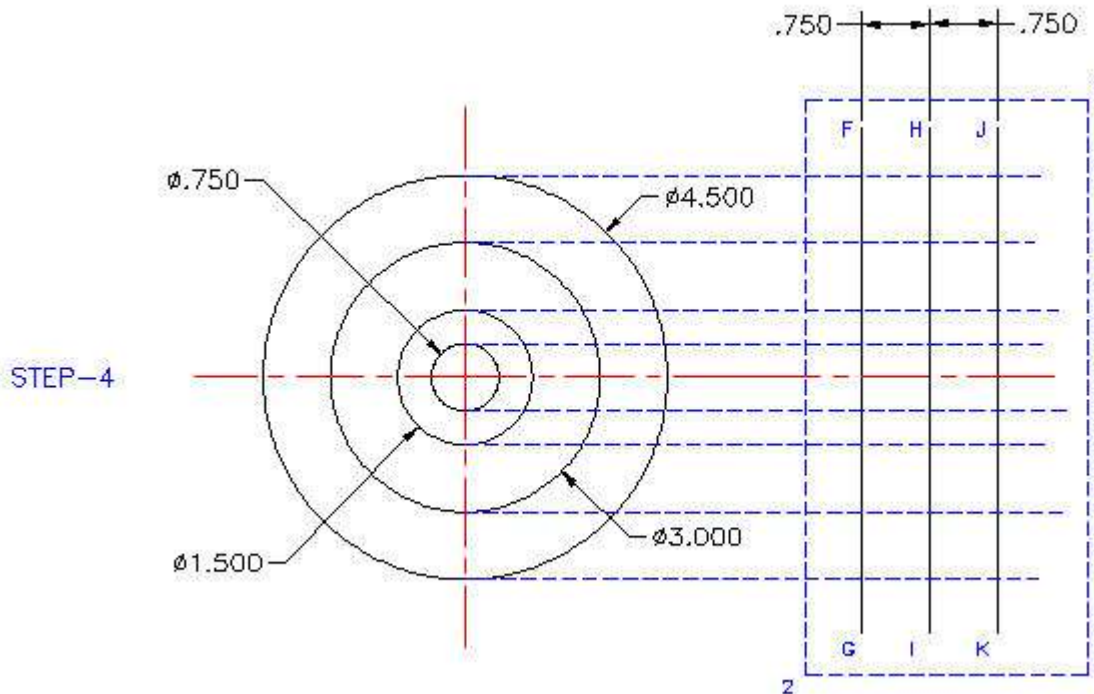
Press the **Spacebar** or **Enter Key** to repeat the command.

Type **D** for Diameter.

Type **1.5** for the next circle.

Press the **Spacebar**.

Repeat the above 3 steps for the remaining **3** and **4.5** diameter circles.



Step-4 Project the Side View Construction Lines

Type **L** for Line on the command Line.

Pick point **F** above the 4.5 diameter circle and drag down to **G** below the circle.

Press the **Spacebar** to exit the Line command.

Type **O** for Offset on the command Line.

Type distance, **.75**.

Pick line **F-G**.

Pick any point to the right side of line **F-G** to create line **H-I**.

Pick line **H-I**.

Pick any point to the right side of line **H-I** to create line **H-J**.

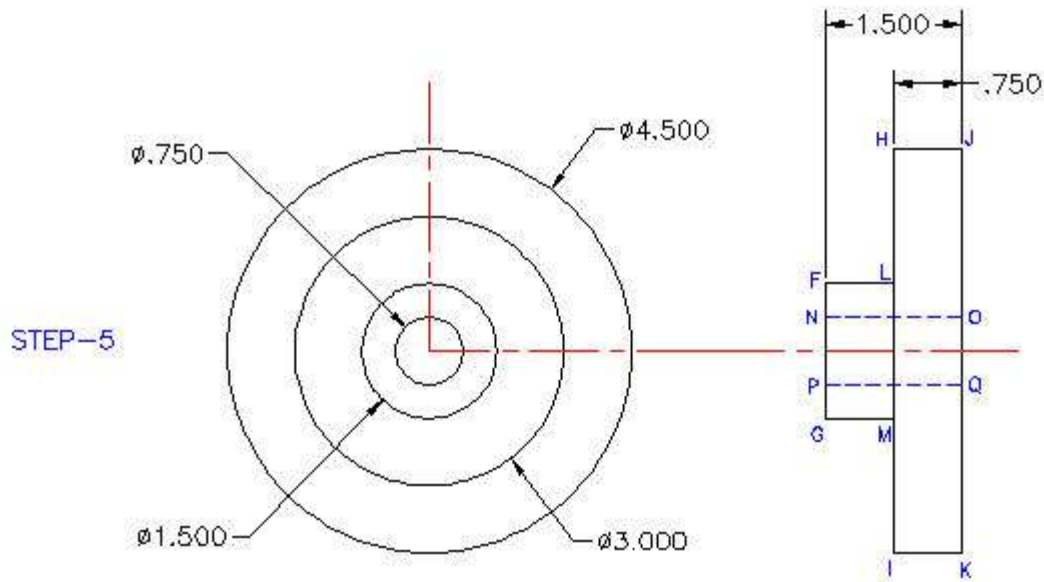
Press the **Spacebar**.

Type **TR** for Trim on the command Line.

Select the object by picking point 1 and dragging to point 2.

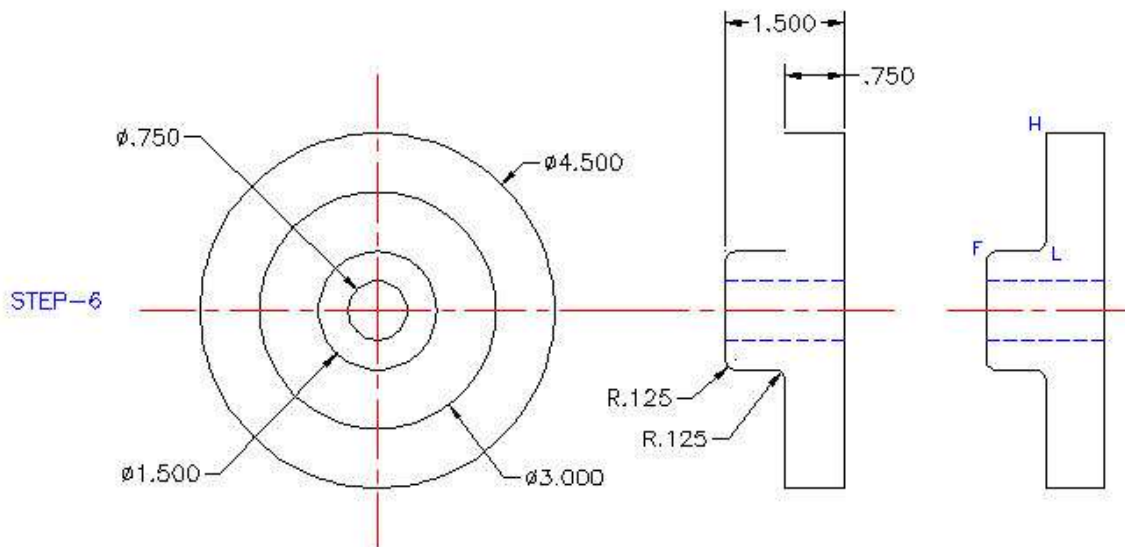
Pick line segments to be removed by trimming to obtain the side view below.

Press the **Spacebar**.



Step-5 Trim the Side View Construction Lines

Trim the side view construction lines as shown above.



Step-6 Add Fillets to the Side View

Type **F** for Fillet on the command Line.

Type **R** for radius.

Type **.125** for the dimension.

Pick 2 lines intersecting at a corner to form a fillet.

Press the **Spacebar** to repeat the Fillet command at each rounded corner.

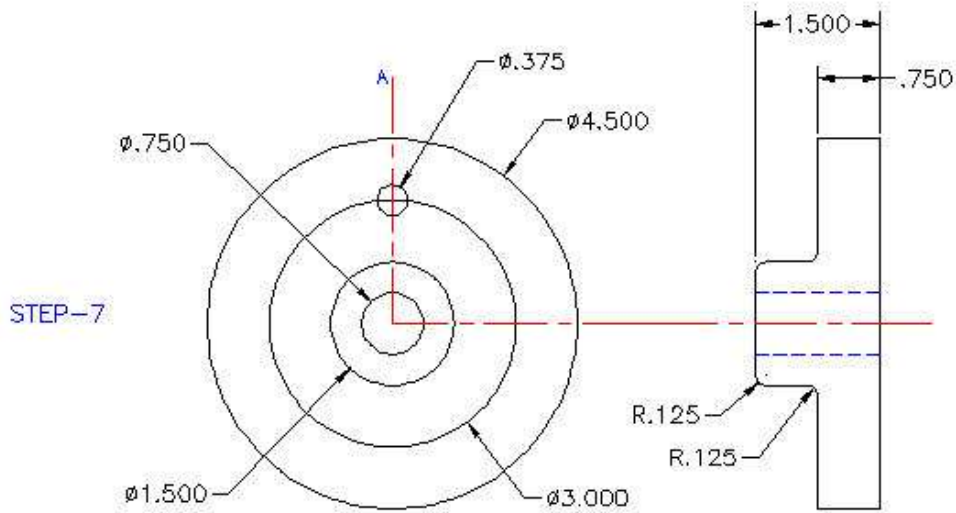
Type **L** for Line on the command Line.

Pick point **H** and drag down to **L**.

Type **F** for Fillet on the command Line.

Pick lines **L-F** and **L-H** intersecting at **L** to form a fillet.

Press the **Spacebar** to exit the command.



Step-7 Add the first 0.375 inch Diameter Bolt Hole

Type **C** for Circle on the command Line.

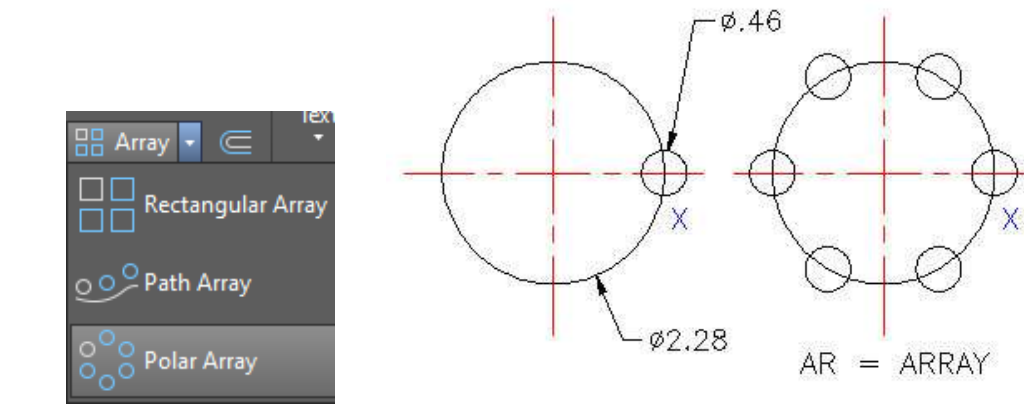
Pick the center point for the .375 inch diameter circle shown above.

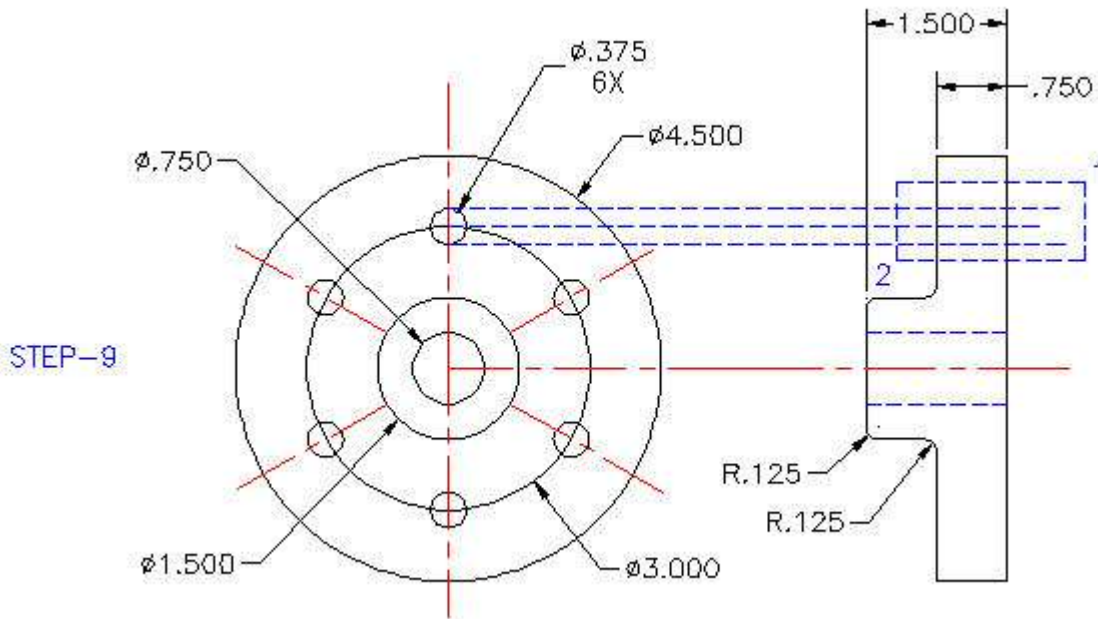
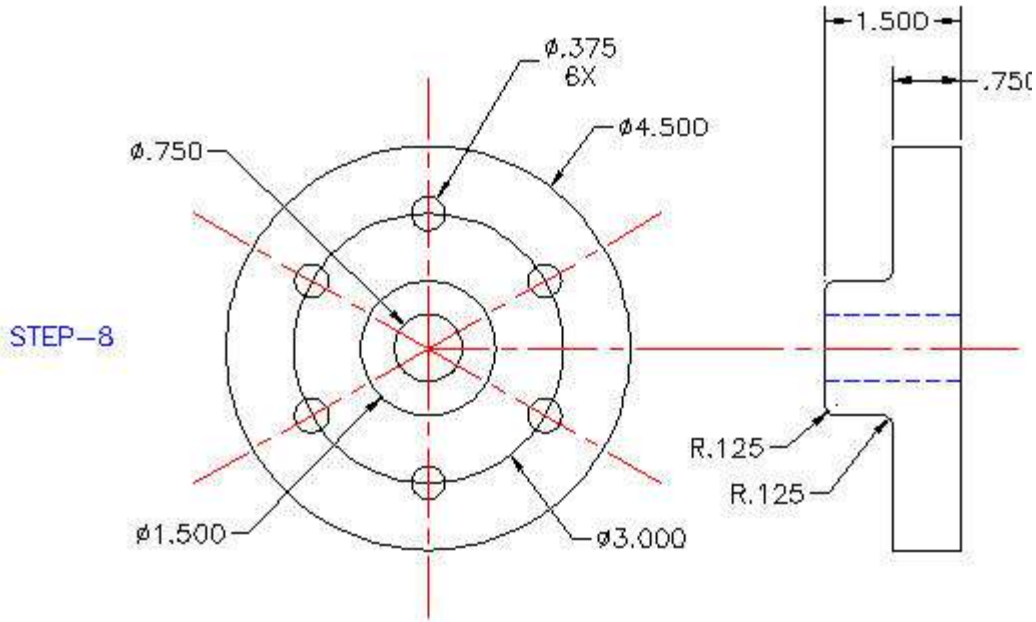
Type **D** for diameter.

Type **.375** for the dimension.

Press the **Spacebar** to exit the command.

ARRAY





Step-9 Project the first 0.375 inch Diameter Bolt Hole to the Side View

Type **L** for Line on the command Line.

Pick the center point for the .375 inch diameter circle shown above.

Drag the center line beyond the side view of the hub as shown above.

Press the **Spacebar** to repeat the command.

Draw the upper and lower, hole projection lines.

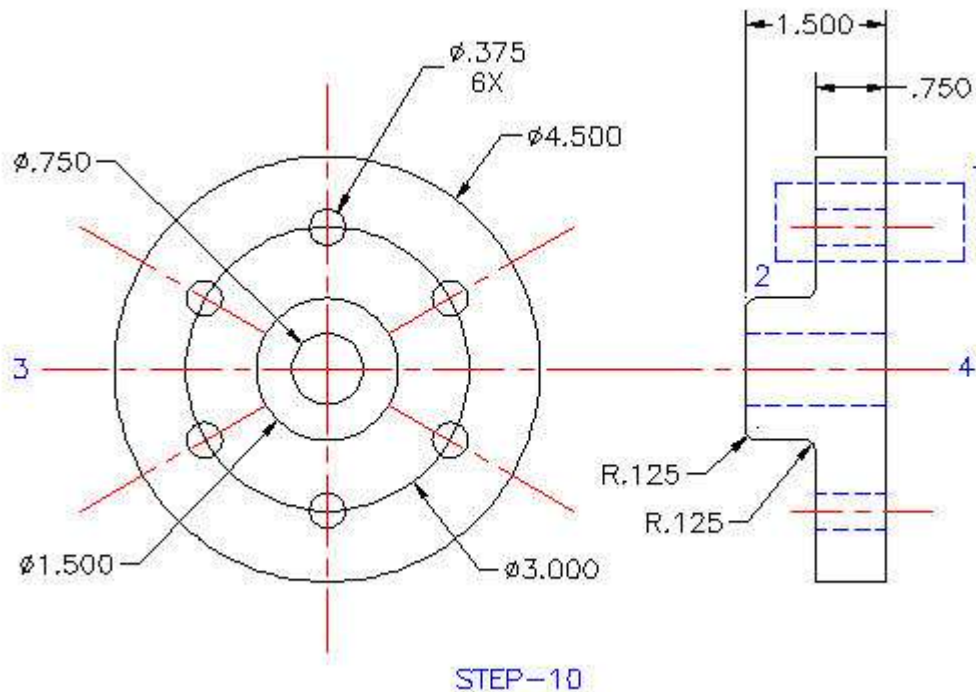
Press the **Spacebar**.

Type **TR** for Trim on the command Line.

Select the object by picking point 1 and dragging to point 2.

Pick line segments to be removed by trimming to obtain the side view below.

Press the **Spacebar**.



Step-10 Mirror Upper Hole About Center Line 3-4

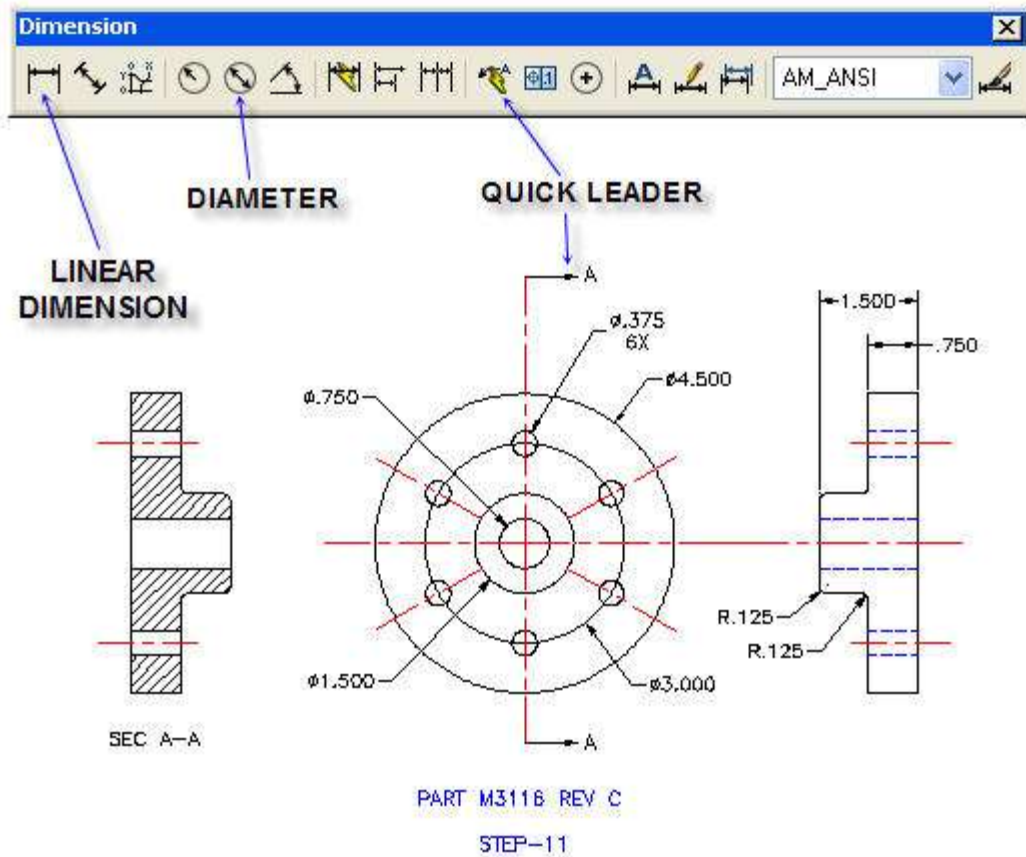
Toggle function key **f8** until you see, **Ortho on**.

Type **MI** for Mirror on the command Line.

Select the object by picking point 1 and dragging to point 2.

Select point 3 and drag to the right and pick point 4.

Press the **Spacebar**.



Step-11 Add Dimensions to Complete the Drawing

Pick the **Linear Dimension** icon on the **Dimension Toolbar**.

Pick side view upper corners to obtain the **1.500** & **0.750** dimensions.

Pick the **Diameter** icon.

Pick a point on one circle and place the diameter dimension.

Repeat the step above to dimension the remaining diameters.

Pick the **Quick Leader** icon and draw the Section A-A view arrows.

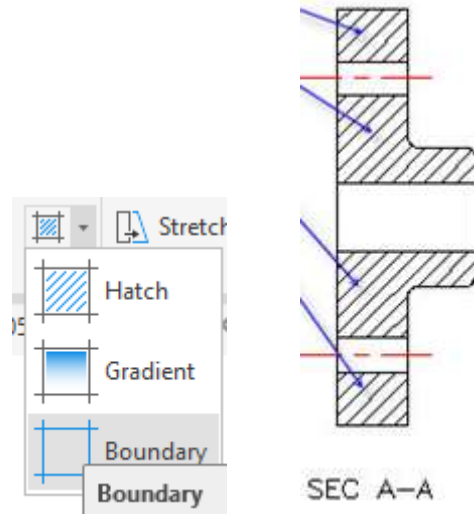
Press the **Spacebar**.

HATCH

```
Pick internal point or [Select objects/Undo/seTtings]:
>>Enter new value for ORTHOMODE <1>:
Resuming HATCH command.
```

```
[Hatch Icon] ▾ HATCH Pick internal point or [Select objects Undo
seTtings]: |
```


Type > H > Pick internal point > OK



Step-12 Add Hatching to the Sectioned Side View

Type > H > Pick internal point > OK

Pick **OK**.

This is the end of Section 5. Generic Hub

BILL OF MATERIALS

BILL OF MATERIALS			
ITEM	QTY	DESCRIPTION	MATERIAL
1	2	BASE	DELTRIN
2	2	FOAM GUIDE	ALUM 6061
3	4	TOP PLATE	DELTRIN
4	4	GUIDE	316 SS
5	4	PIN	316 SS
6	4	SCREW 1/4-20	316 SS

REV	DESCRIPTION	DRAWN	DATE
1			

BILL OF MATERIALS			
ITEM	QTY	DESCRIPTION	MATERIAL
1	2	BASE	WHITE DELRIN
2	2	FOAM GUIDE	ALUM 6061
3	2	TOP PLATE	WHITE DELRIN
4	4	SHCS #6-32 2 1/4" LG McMASTER 92185A175	316 SS
5	4	ROLL ROD 1/4"DIA 1 FT LG McMASTER 8934K26	304 SS
6	4	SPRING PIN 1/8"DIA x 3/4"LG McMASTER 92373A126	18-8 SS
7	4	GUIDE ROLL	WHITE DELRIN

QUANTITIES IN BOM ARE FOR 2 ASSEMBLIES

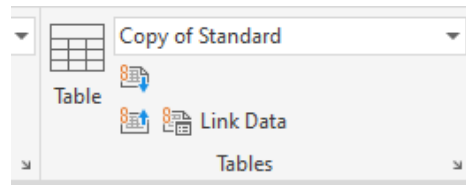
ENGINEERING COMPANY
SWAB FOAM GUIDE

SCALE: Full
DATE: 2/13/2021
SHEET 1 OF 11
DRAWN: J ANDREW
CHECKED: E SMITH
SIZE: A
DRAWING NUMBER: 5200
REV: 1

TOLERANCES UNLESS OTHERWISE NOTED:
.XX .005
.XXX ± .001
ANGLES ± 1°

THIS DRAWING AND THE SPECIFICATIONS HEREIN ARE PROPRIETARY TO COMPANY, INC AND SHALL NOT BE REPRODUCED OR COPIED OR USED IN WHOLE OR IN PART AS THE BASIS FOR DESIGN, MANUFACTURE OR SALE OF ITEMS WITHOUT THE WRITTEN PERMISSION OF COMPANY.

NEW FOAM GUIDE ROLL & BKT SUB ASSY



Select > Annotate > Table

Insert Table

Table style: Copy of Standard

Insert options: Start from empty table

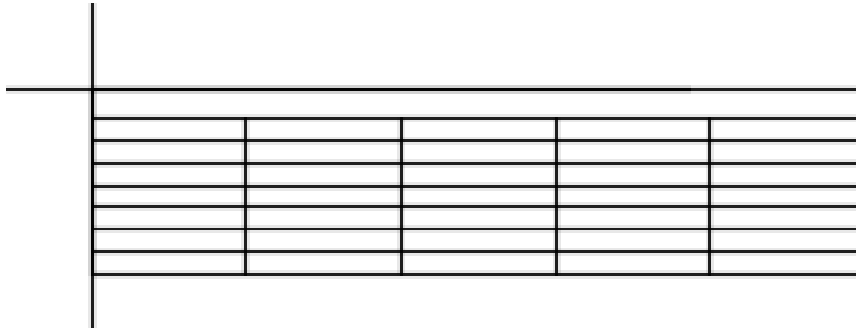
Column & row settings: Columns: 5, Column width: 2.5000, Data rows: 6, Row height: 1 Line(s)

Set cell styles: First row cell style: Title, Second row cell style: Header, All other row cell styles: Data

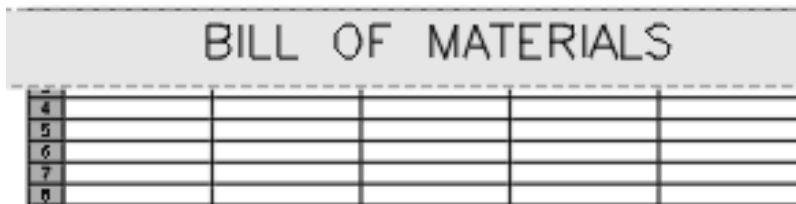
Title		
Header	Header	Header
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data
Data	Data	Data

Buttons: OK, Cancel, Help

Select > **Table** > **Columns** > **5 Rows** > **6** > **OK**

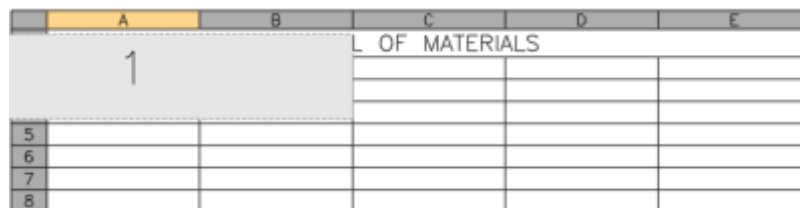


Pick top left table location point > Enter



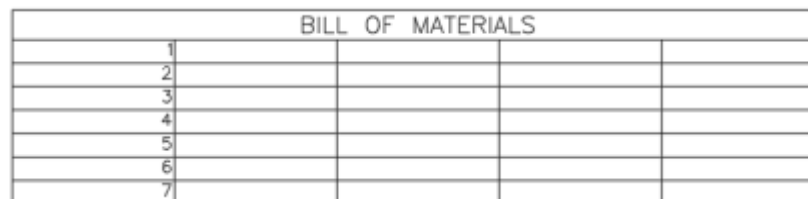
BILL OF MATERIALS				
4				
5				
6				
7				
8				

Type > **BILL OF MATERIALS** > Enter



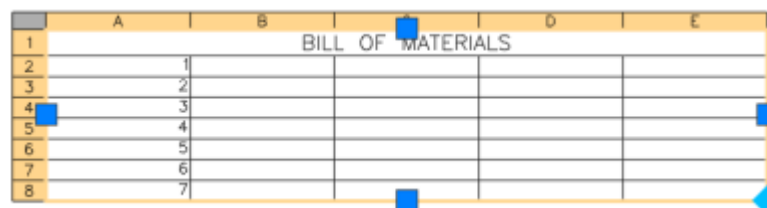
	A	B	C	D	E
	1		L OF MATERIALS		
5					
6					
7					
8					

Type > **1** > Enter



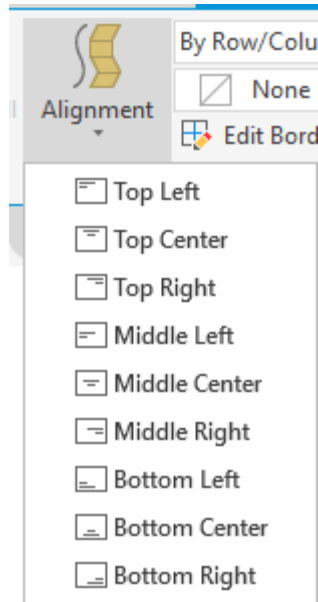
BILL OF MATERIALS				
1				
2				
3				
4				
5				
6				
7				

Type > **2-3-4-5-6** > Enter



	A	B	C	D	E
1			BILL OF MATERIALS		
2		1			
3		2			
4		3			
5		4			
6		5			
7		6			
8		7			

Select table top left corner.



Select > Alignment toolbar > Middle Center

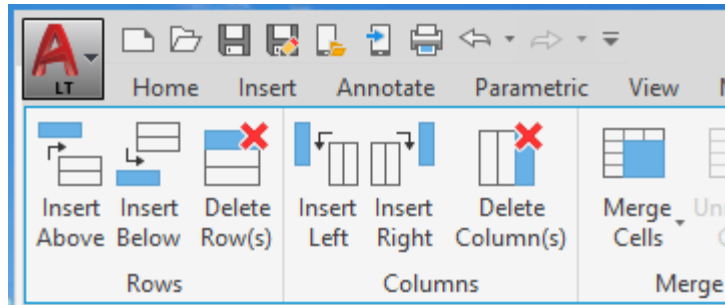
	A	B	C	D	E
1	BILL OF MATERIALS				
2	1				
3	2				
4	3				
5	4				
6	5				
7	6				
8	7				

Pick table "A" column > Enter

BILL OF MATERIALS				
ITEM	QTY	DESCRIPTION	MATERIAL	
1				
2				
3				
4				
5				
6				

	A	B	C	D	E
1	BILL OF MATERIALS				
2	ITEM	QTY	DESCRIPTION	MATERIAL	
3	1				
4	2				
5	3				
6	4				
7	5				
8	6				

Pick table column "C" >



Select > Annotate > Pick top left corner of table > Delete Columns(s) > Column E > Enter

BILL OF MATERIALS			
ITEM	QTY	DESCRIPTION	MATERIAL
1			
2			
3			
4			
5			
6			

Column E is deleted.

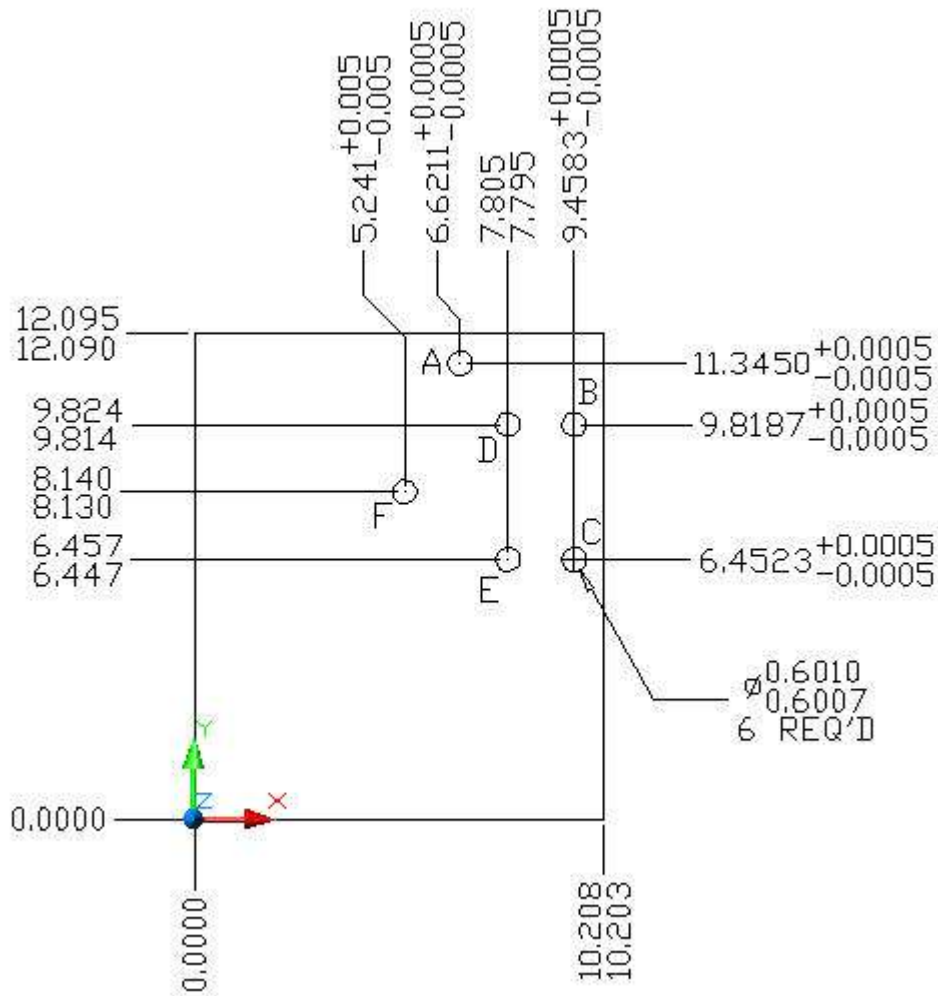
	A	B	C	D
1	BILL OF MATERIALS			
2	ITEM	QTY	DESCRIPTION	MATERIAL
3	1			
4	2			
5	3			
6	4			
7	5			
8	6			

Pick table cell B3 > Type 2 > Enter

BILL OF MATERIALS			
ITEM	QTY	DESCRIPTION	MATERIAL
1	2	BASE	DELTRIN
2	2	FOAM GUIDE	ALUM 6061
3	4	TOP PLATE	DELTRIN
4	4	GUIDE	316 SS
5	4	PIN	316 SS
6	4	SCREW 1/4-20	316 SS

Complete Bill of Materials

DIMENSIONS & GEOMETRIC TOLERANCES



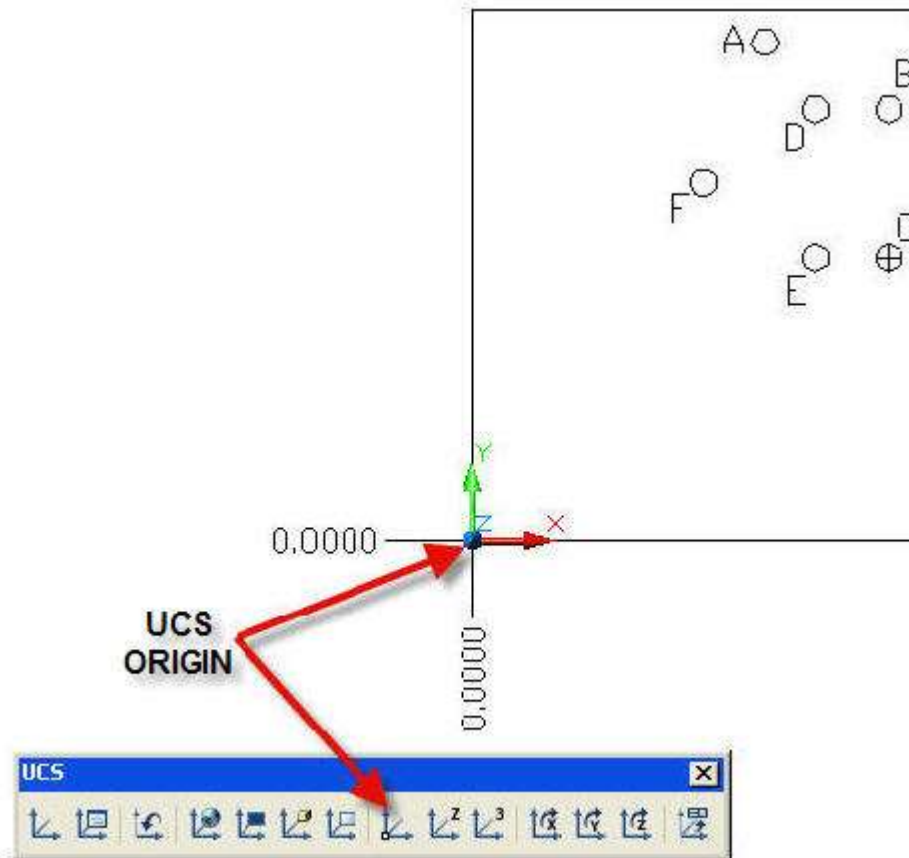
JIG PLATE EXAMPLE



1. Draw the jig plate outline:
Select > Rectangle > Pick a point > Drag a rectangle > 10.203,12.090 > Enter

2. Add locator holes: A, B, and C, having a 0.6000 nominal diameter. Adding +/-0.0005 inch tolerance is described below.
3. Now draw the support holes: D, E, and F also of 0.6000 nominal diameter.
4. Move the **“User Coordinate System” (UCS)** origin to the bottom left corner of the jig plate base, as seen above, by selecting the following commands:

Old versions of AutoCAD
Pick drop down menu: **View > Toolbar > UCS**
D

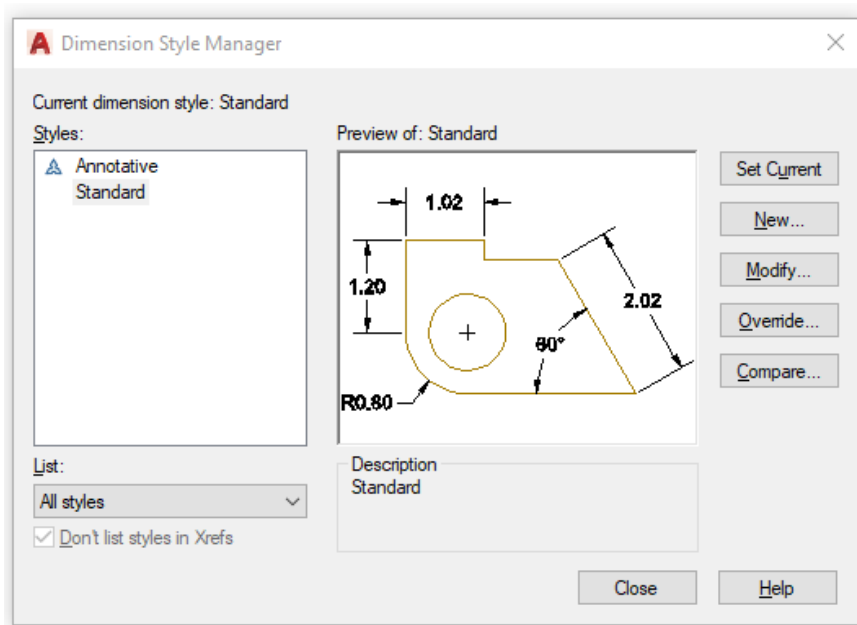


5. Locate a new position for the User Coordinate System UCS origin.

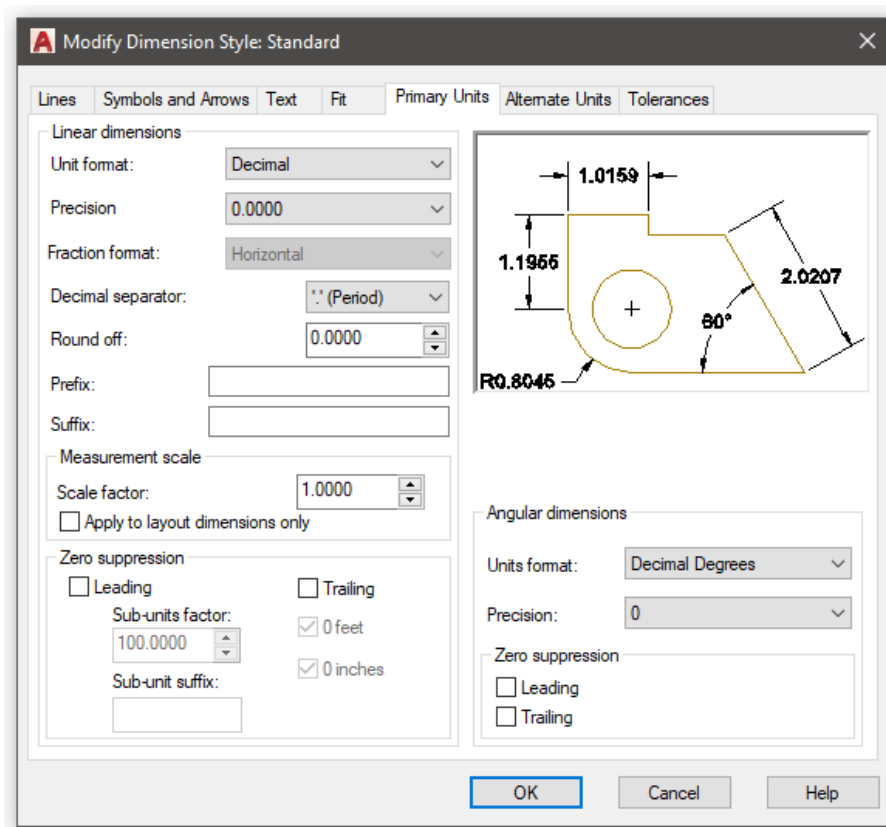
Type > UCS > Pick bottom left corner of plate.

The normal location of the origin is at $x = 0$ and $y = 0$ at the bottom left corner of the display.

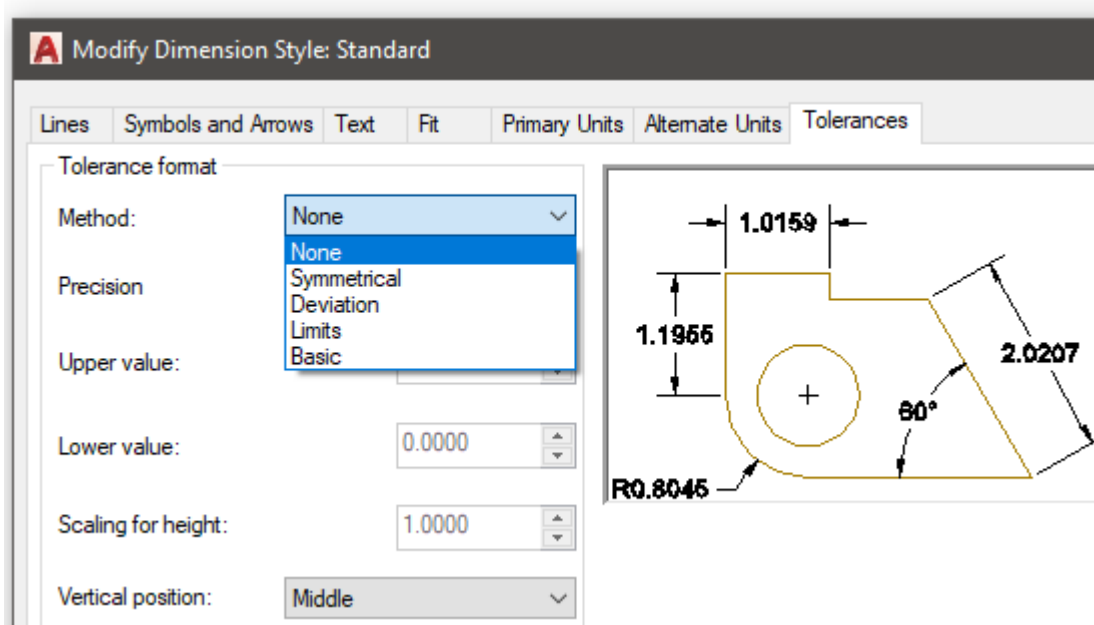
Type > D > Enter



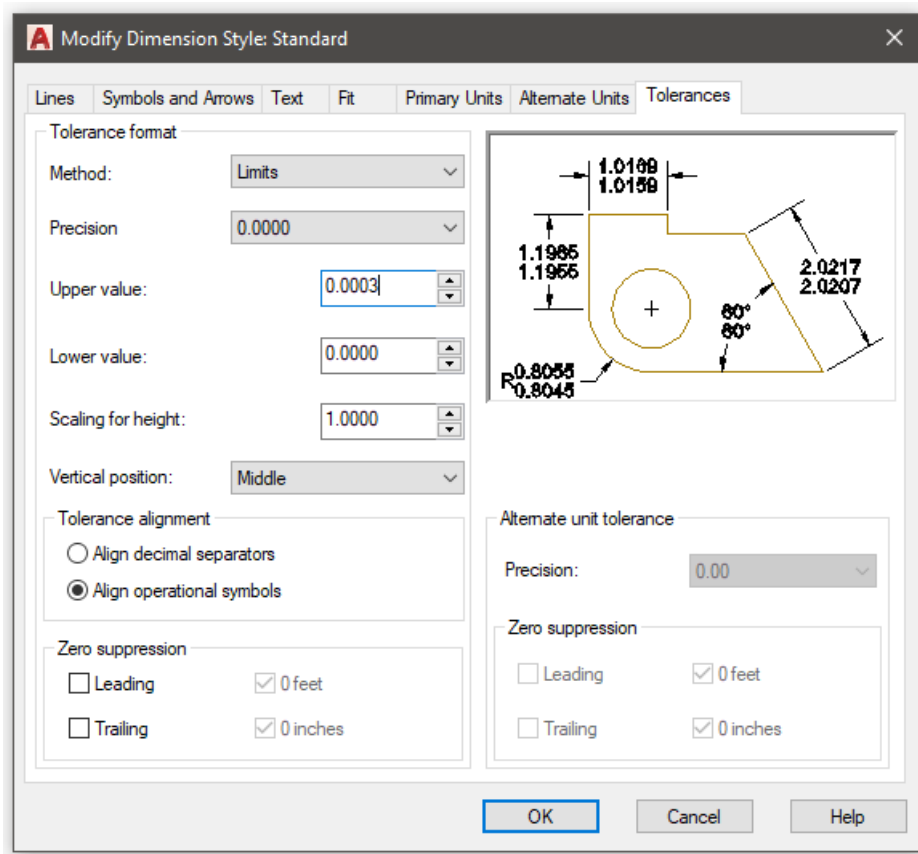
Pick > Modify



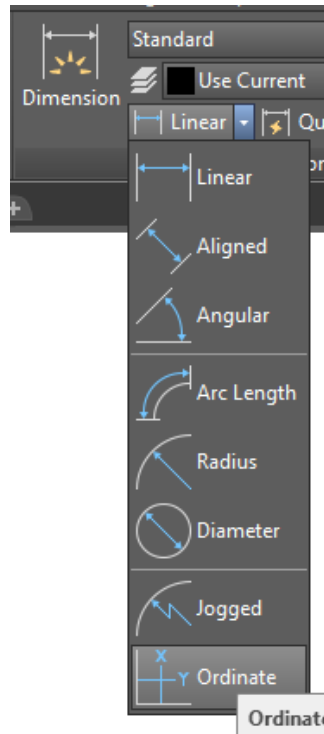
Set Primary Units to Decimal > 0.0000



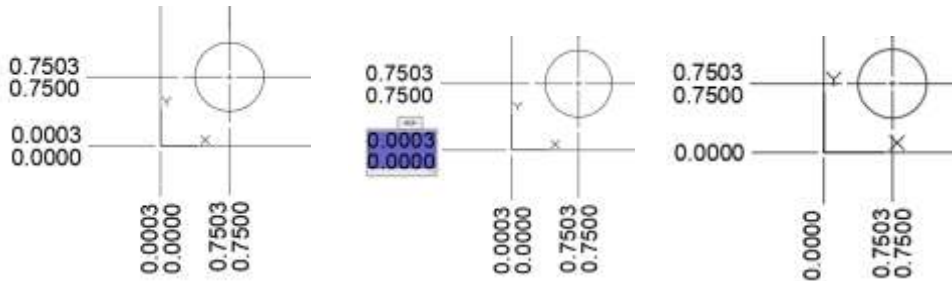
Select > Tolerances > None > Limits



Type Upper Value > 0.0003 > OK > Close

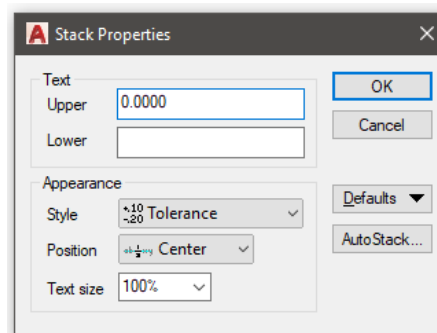


Select > Linear > Ordinate



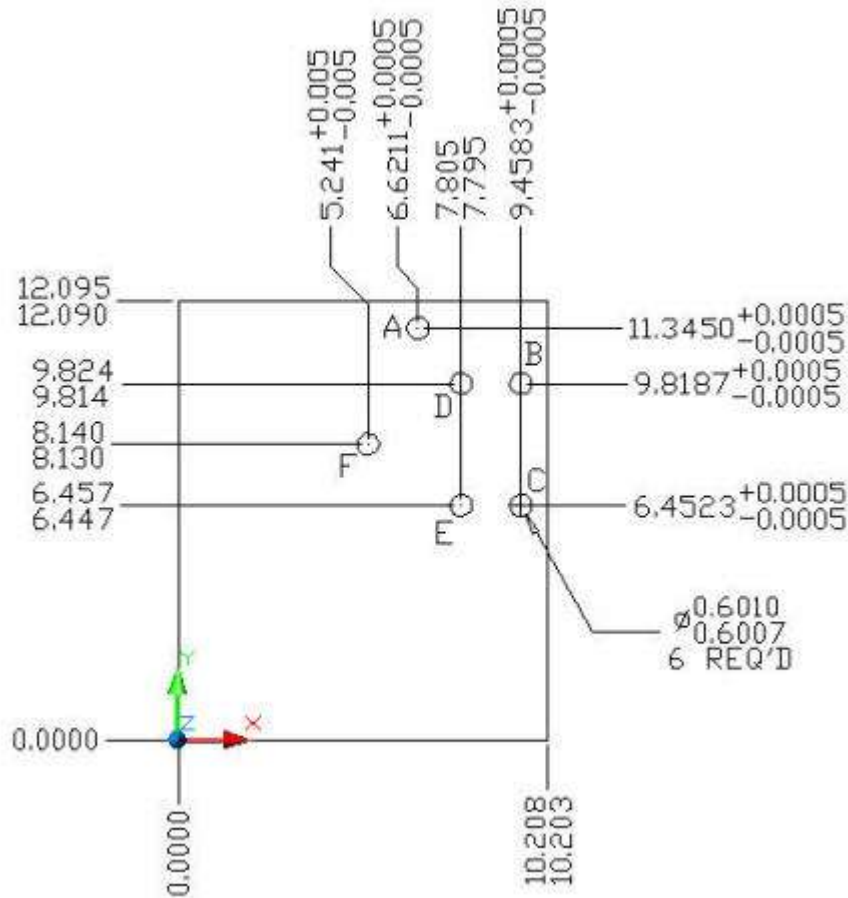
Origin dimensions must be edited.

Type > ED > Select upper/lower dimensions > Edit > 0.0000 > Enter or OK



6. Set the Support Hole Dimension Units and **Limit Dimensions** shown above

7. Dimension the A, B, C, D and F hole diameters with Limit tolerances of +0.0005 and 0.0000.



8. Dimension the A, B and C positions in the jig plate with Deviation tolerances of +0.0005 and -0.0005.

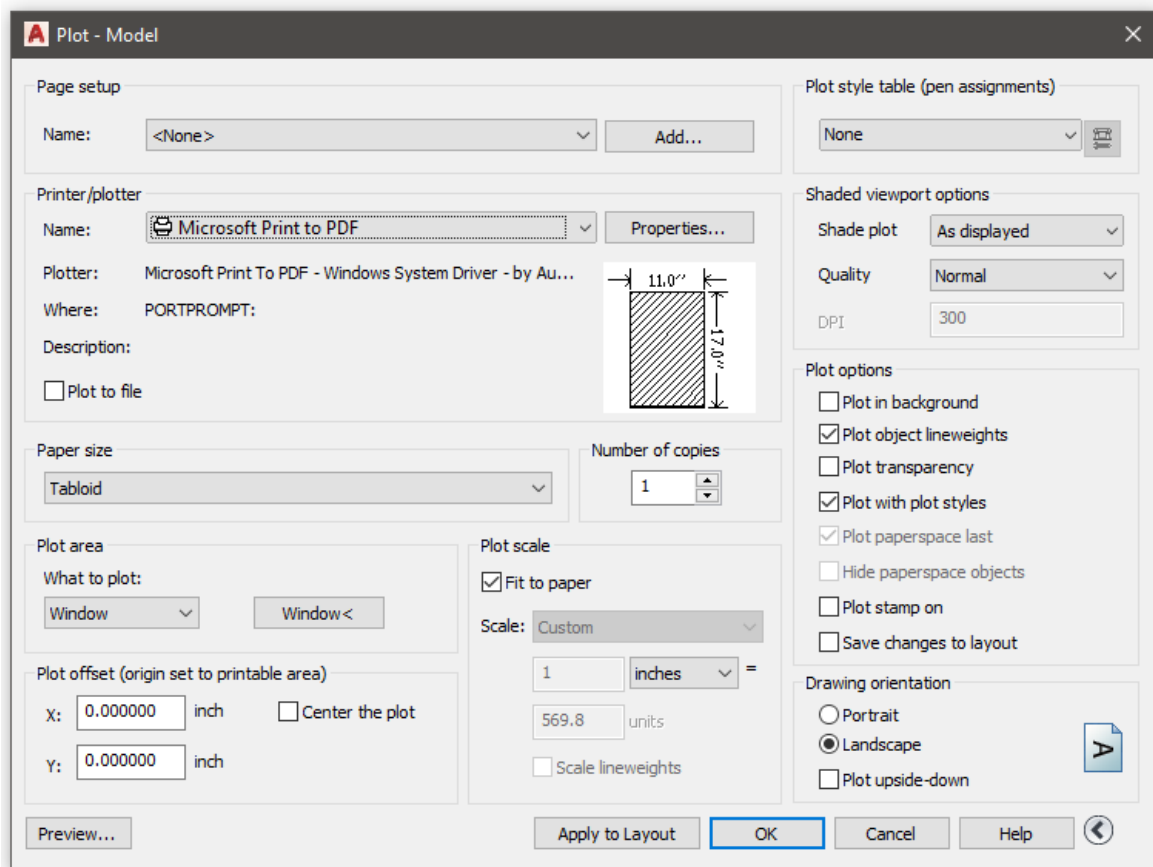
9. Dimension the D, E and F positions in the jig plate with Limit tolerances of +0.0010 and -0.0000.

On the AutoCAD command line type: **D** for dimension > **M** for modify > Primary units > 0.0000 > **Tolerances** > Method > Deviation > Upper value > 0.0003 > Lower value > 0.0000.

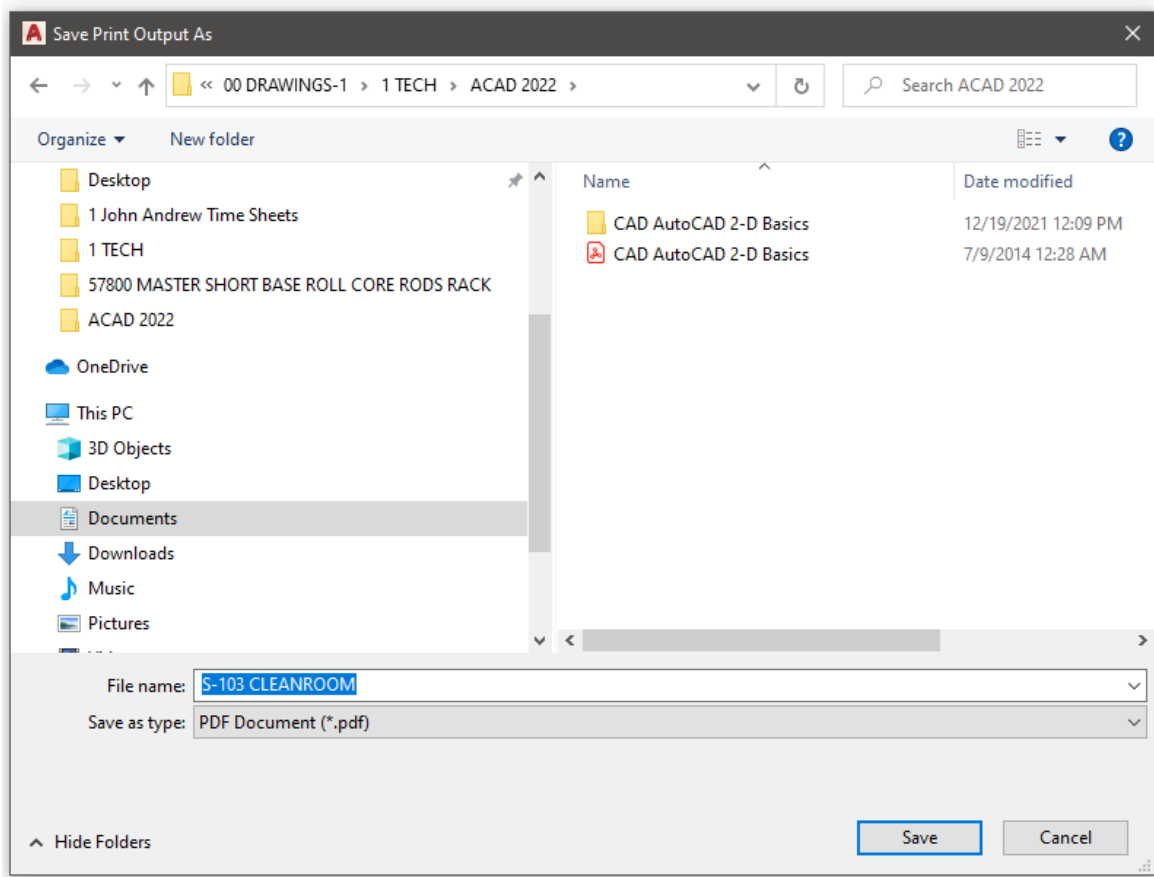
PRINTING A DRAWING IN MODEL SPACE

Select drop-down menus: **File > Print >**

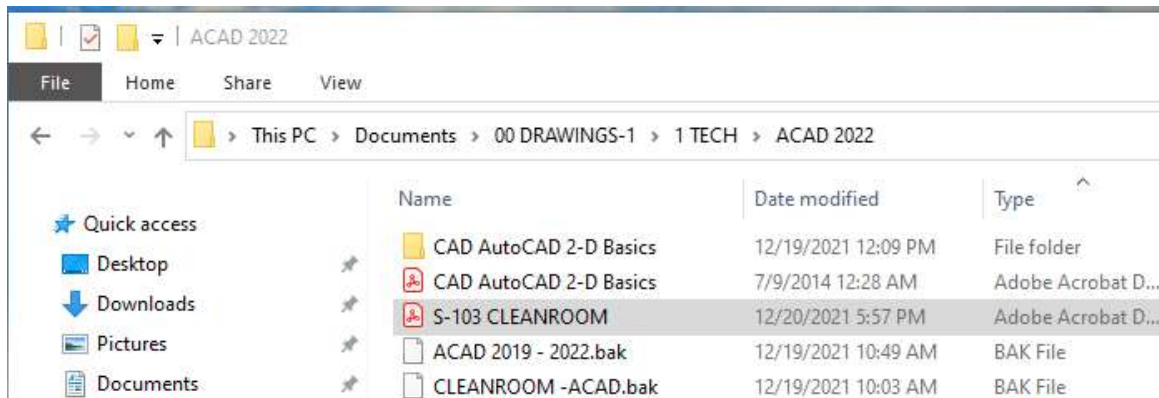
The, "Plot – Model" dialog box opens as pictured below.

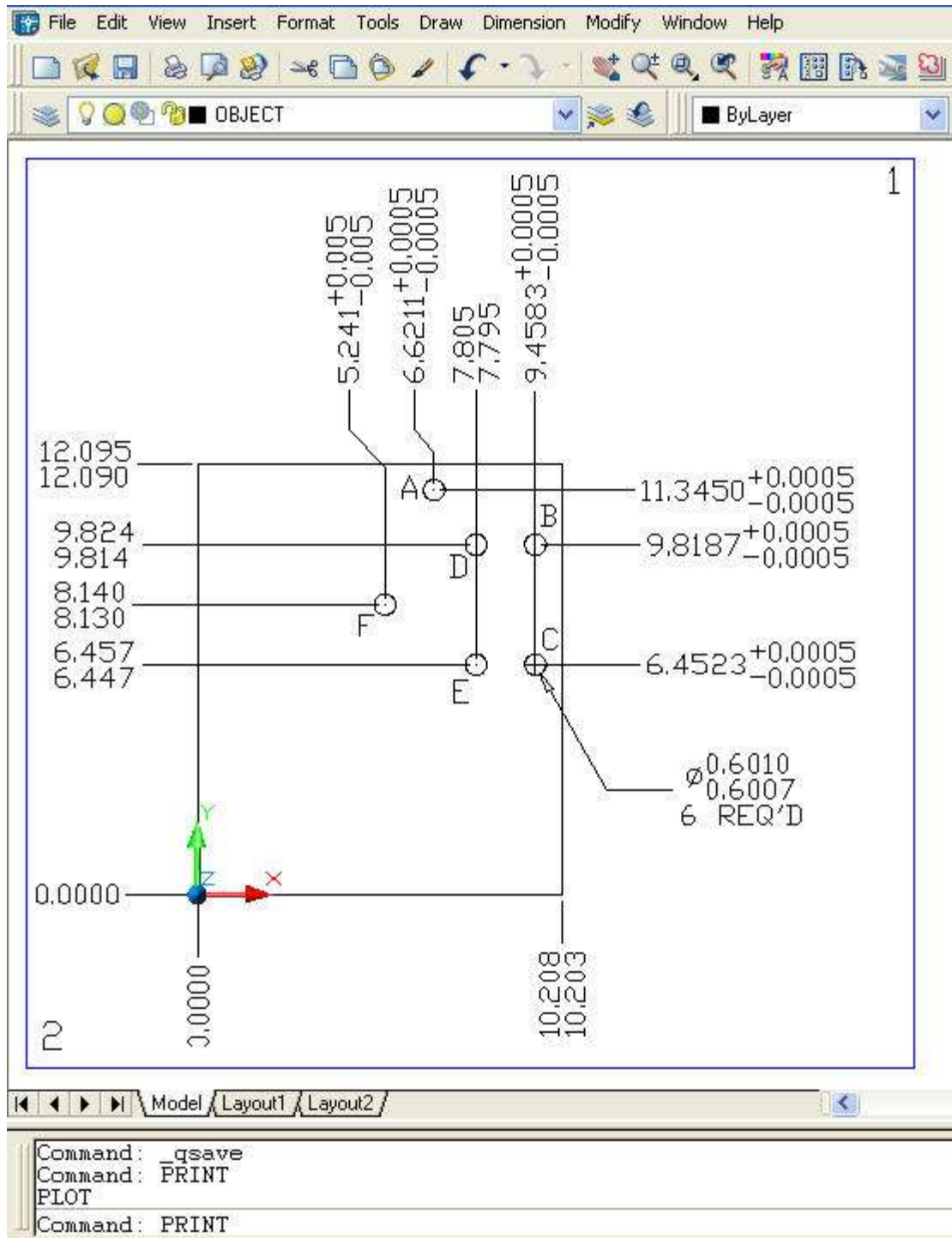


Set Paper Size > Tabloid (11" x 17") > OK

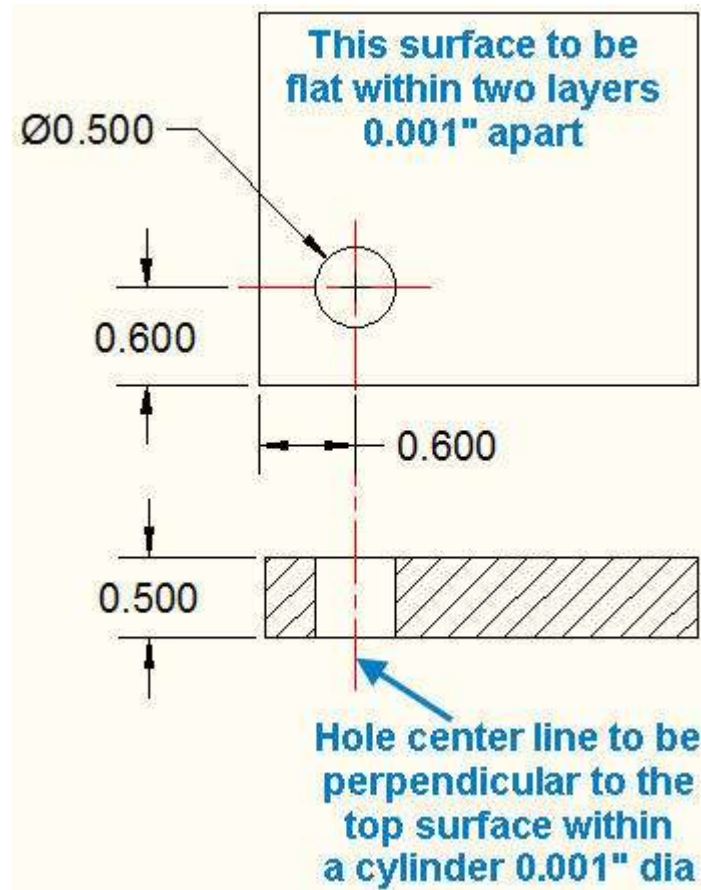


Type > Drawing File Name > S-103 CLEANROOM > Save



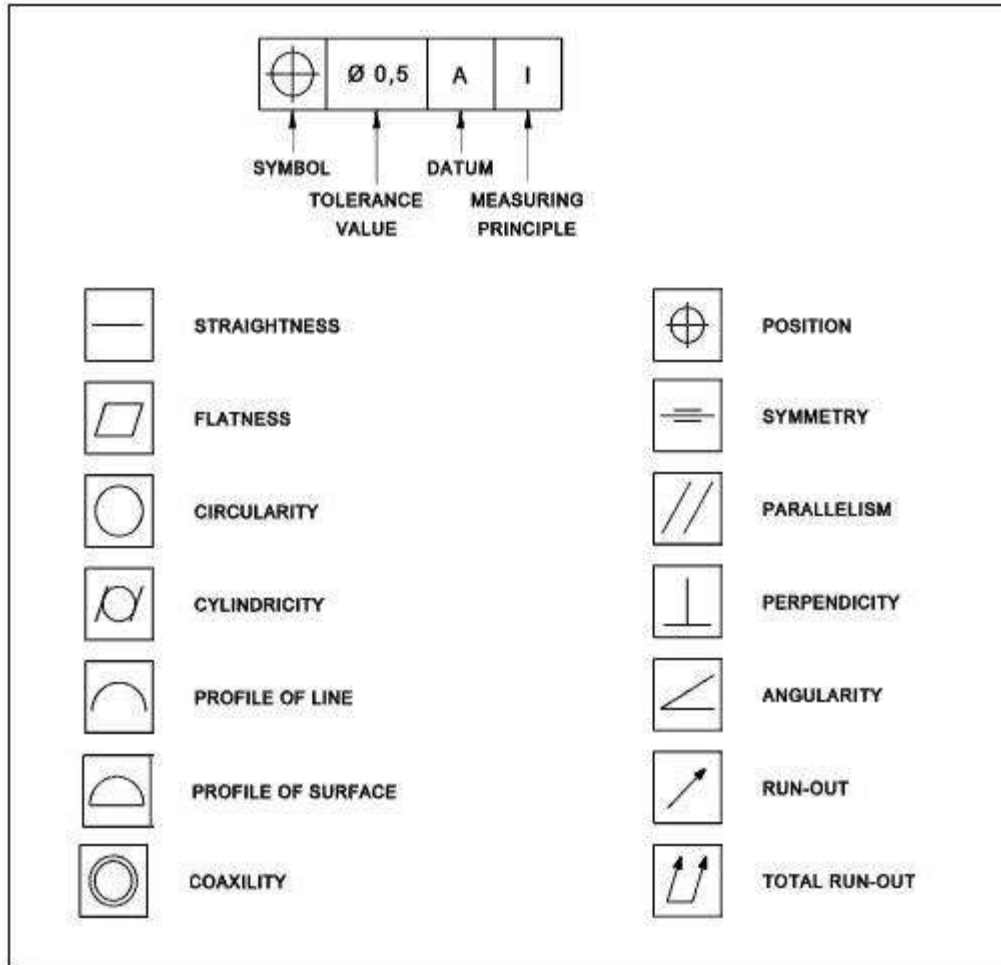


AUTOCAD GEOMETRIC TOLERANCE MENUS



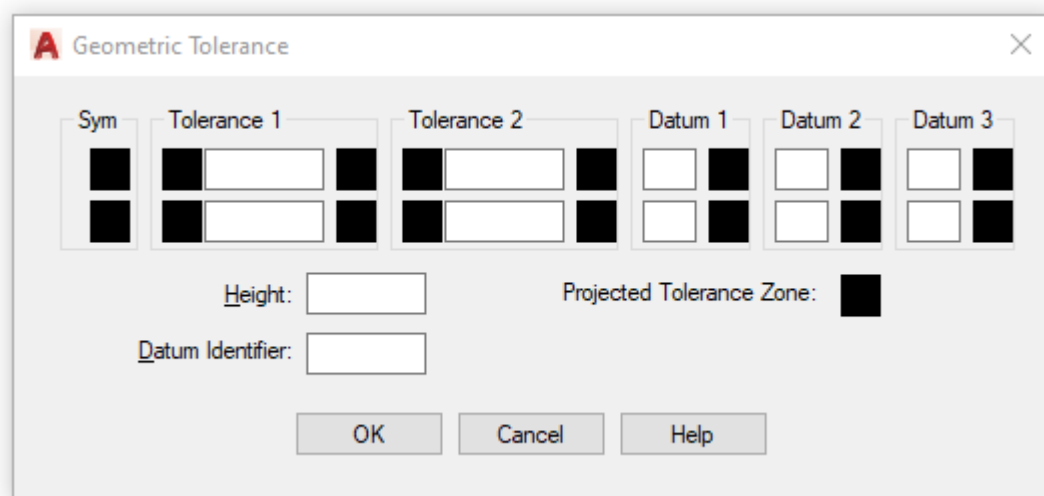
OBJECTIVE

A flat top surface and a perpendicular hole are required in the metal plate above.

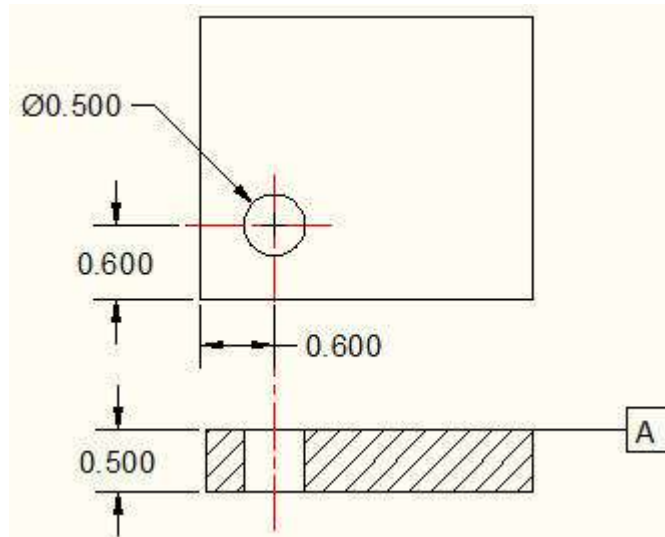


The standard symbols for, “**Geometric Tolerances**” shown above are generated by AutoCAD.

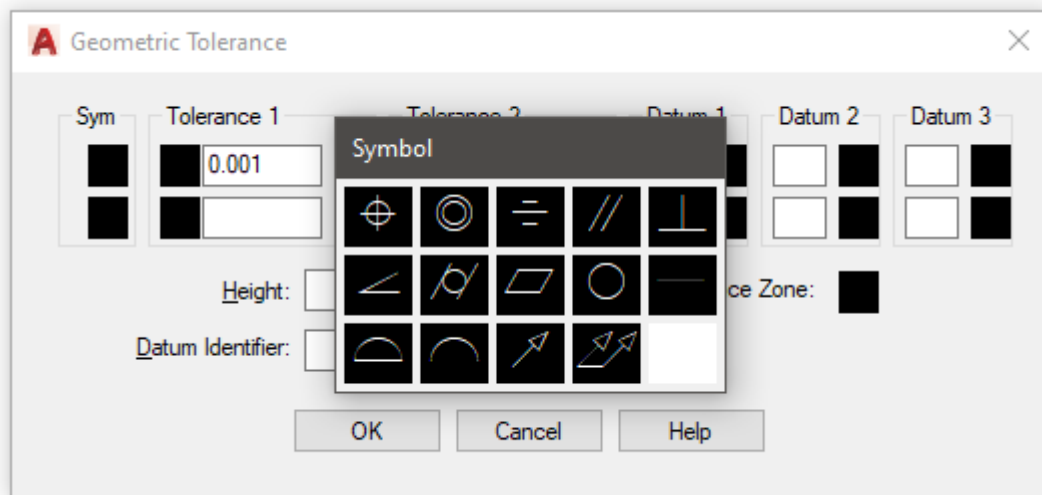
Type > Tolerance > **Geometric Tolerances dialog box opens as below:**



2. Type **A** as Datum Identifier as above.



2. Place the **A in a box**, at datum plane, drawn by you as shown above.



4. Pick drop down menu again: **Dimension > Tolerance ...**

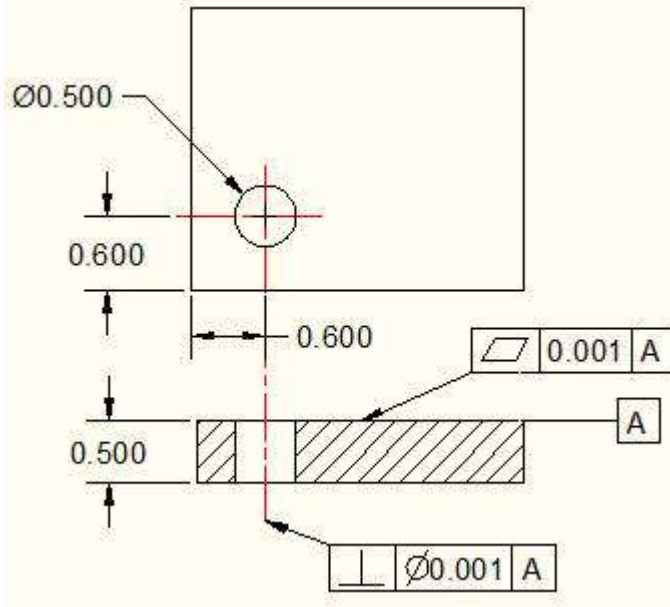
5. Pick the Perpendicular to symbol: **Symbol > \perp**

6. Select the diameter symbol: **\varnothing**

7. Type the tolerance: **0.001**

8. Type **A** under "Datum 1, as shown above.

9. Create the “Flatness” symbol also following the steps above.



CREATE LEADER ARROW

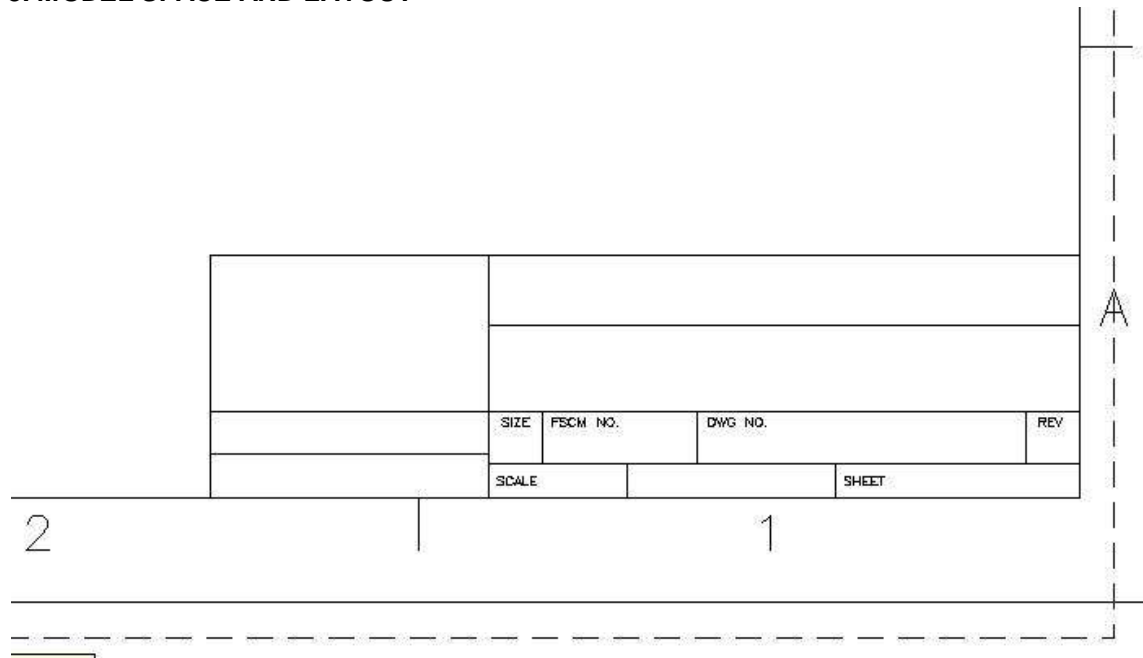
10. Type, “LEAD” on the command line > Pick arrow point location > Drag to Geometric tolerance box corner to create a leader arrow.

NOTE:

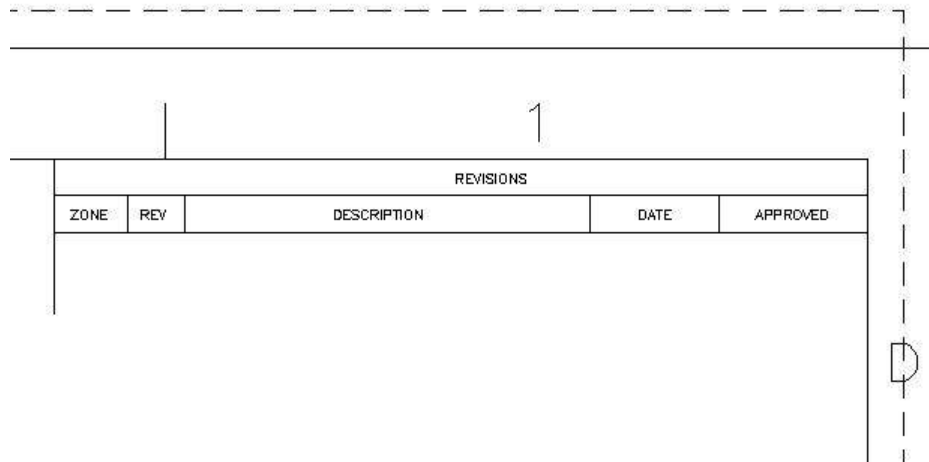
It is recommended that you use the **MLEADER** command to create multiline text with leader arrows.

This is the end of Section 7. Dimensional Tolerance.

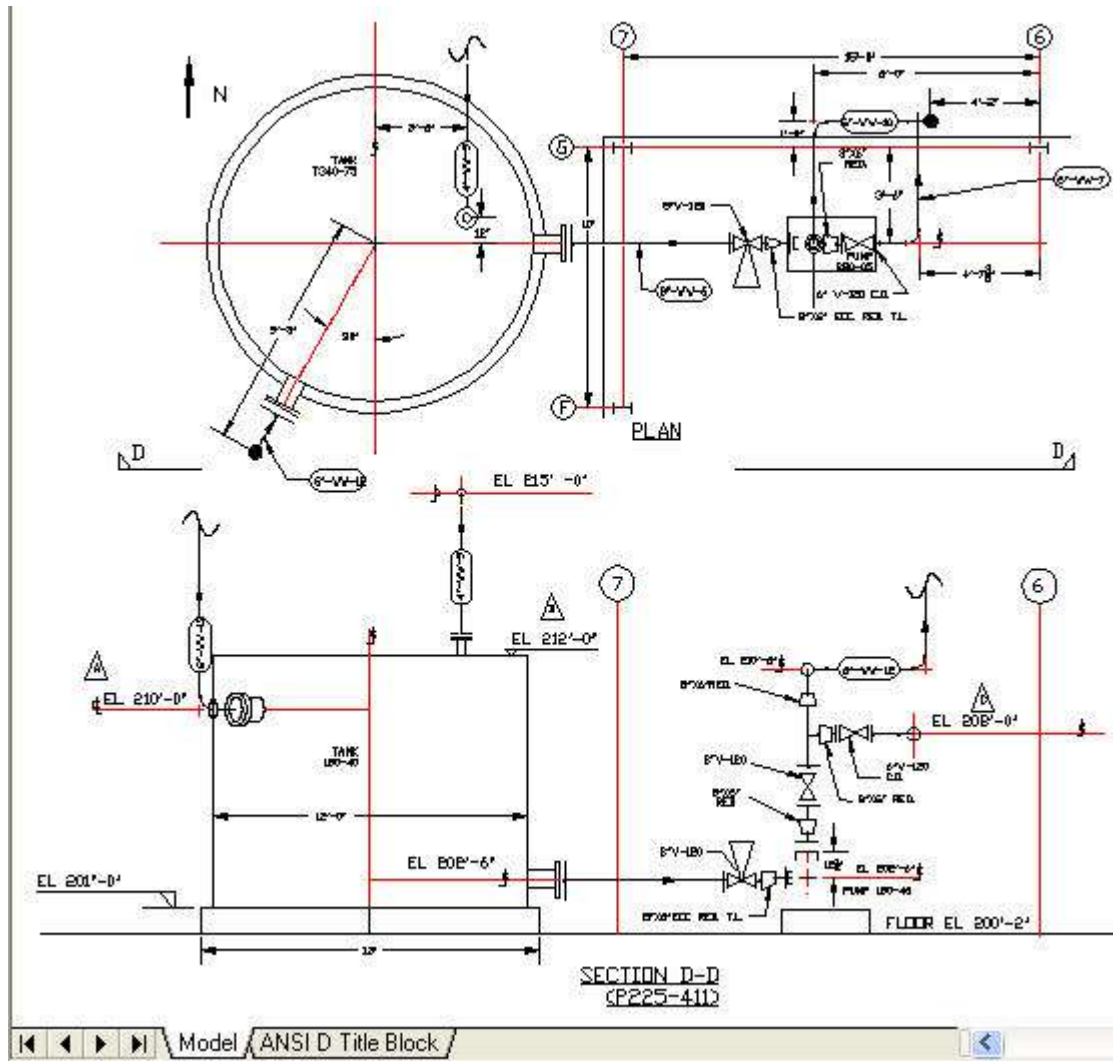
8. MODEL SPACE AND LAYOUT



The ANSI D template title block is magnified in the view above.



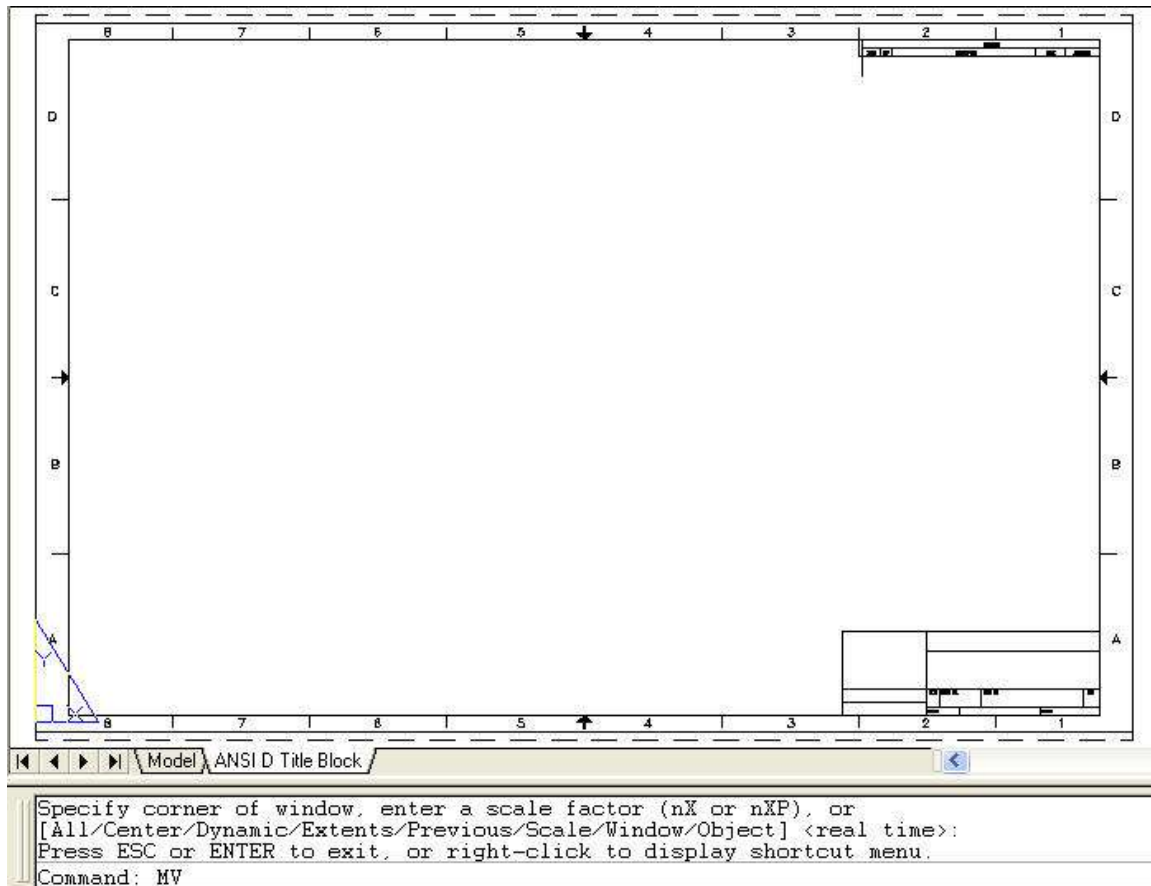
The template revision list is located in the top right corner shown here.



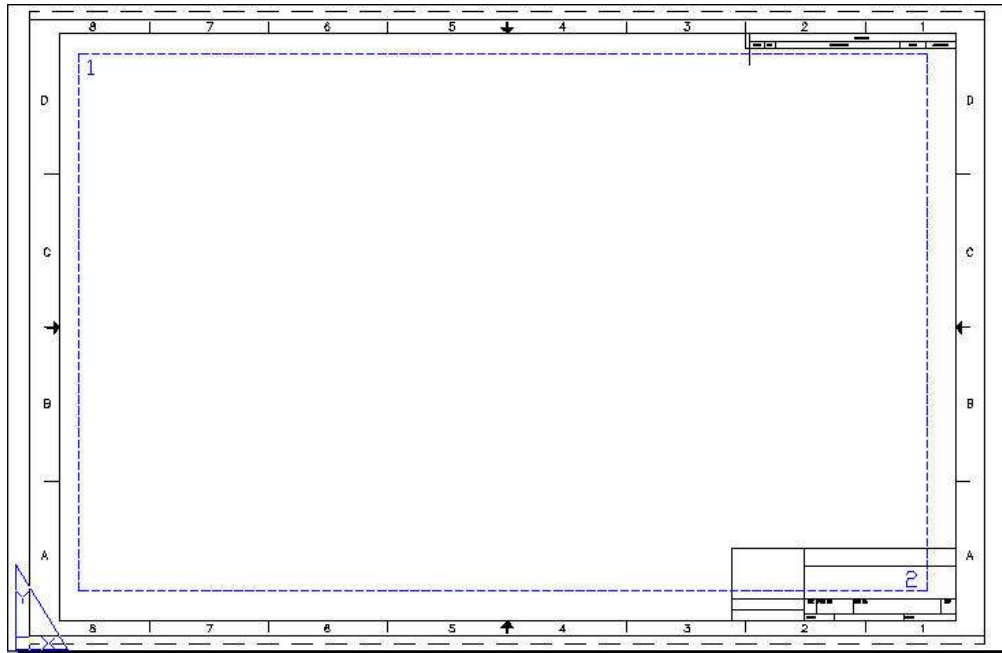
Hydraulic Circuit Diagram Example

The Hydraulic Circuit diagram above is drawn in the **Model** tab area, as seen above.

The **ANSI D** template title block came from the **Select template** dialog box described above and is in the **Layout** area and **ANSI D Title Block** replaces the name normally found on the **Layout** tab.



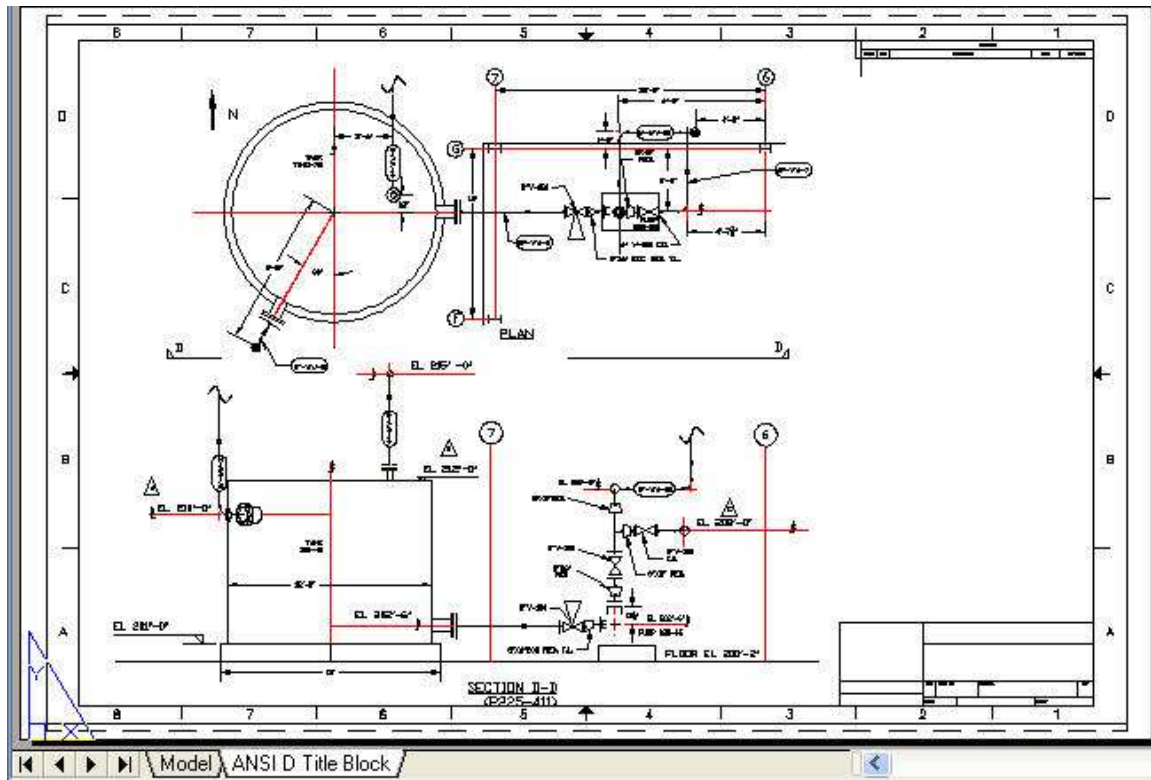
The **ANSI D** template title block came from the **Select template** dialog box and is in the **Layout** tab area as seen above.



Keeping in the **Layout** tab area, type **MV** for Model View on the command line.

Pick point **1** with the cursor and drag to point **2**.

The drawing in the **Model View** is scaled and copied into the **Layout** area automatically as illustrated below.



The process piping drawing above is automatically scaled by AutoCAD to fit inside the “**Layout**” selected area as shown above. The drawing scale is changed but dimensions do not change when a drawing or diagram is transferred from Model View to the Layout area.

DO NOT DRAW OR DIMENSION IN LAYOUT

Do not add lines or dimensions to a drawing in the Layout area!

Any changes to a drawing should be done in Model Space because the drawings in Layout have been scaled and dimensions added will not be correct.

This is the end of Section 8. Model Space and Layout

DRAWING CHECK LIST

Every completed drawing should be checked by the originator and an independent Checker before being released to a Client. See the sample drawing check list below:

Name: _____ **Date:** _____

- | | |
|--|-------------------------|
| <p>A DWG NO: _____</p> <p>B DIMENSIONS:</p> <ul style="list-style-type: none">1 Units: Inch or Metric2 Overall Height, Width, Length3 Zero Datum's4 Hole Locations, Sizes & Shapes5 All Features Dimensioned6 Decimal Places are a Minimum7 Dimensions are Not Crowded8 Tolerances are Indicated <p>C VIEWS:</p> <ul style="list-style-type: none">9 Orthographic Projection10 Outline of Parts are 0.7 Line weight11 All Other Lines are 0.3 Line weight12 Section & Detail Views Required? <p>D NOTES:</p> <ul style="list-style-type: none">13 Material?14 Pre-finish Treatment?14 Finish?16 Hardware, Nearside or Far Side? | <p>COMMENTS:</p> |
|--|-------------------------|

- 17 How Many: Holes, Hardware?
- 18 Shipping Instructions Required?
- 19 Description of All Revisions

E FORMAT:

- 20 Template: "A", "B", "C", "D", "E"
- 21 Part Name and Number
- 22 Revision Letter or Number
- 23 Drawn By: Initials & Date
- 24 Scale
- 25 Drawing File Path

F ANALYSIS:

- 26 Fit: Interference With Adjacent Parts
- 27 Form:
- 28 Function
- 29 Tolerances
- 30 Manufacturer's Part Selection

This is the end of Section 9. Drawing Check List

10. 2-D AutoCAD COMMAND LIST

A list of the most commonly used AutoCAD commands are given below.

CHK

Toolbars	Key	View > Toolbars > Views > UCS > Solids
Object Snap	F3 OS	OSnap On / Off
Isometric Plane	F5	Top > Right > Left
Grid	F7	On / Off
Ortho	F8	On / Off
Snap	F9	On / Off
Arc	A	Start + Peak + End or Start + Center + Angle
Array	AR	Rectangular or Polar, Offsets, Select Objects
Boundary Hatch	BH	Scale (1.00)> Angle (0) > Pick Point > OK
Hatch Edit	HE	Pick Hatch > Edit
Chamfer	CHA	Dist 1 > Dist 2
Circle	C	Pick Center > D > (2.5)
Copy	CP	Select Objects > Pick Base Point > Pick End Point
Dimension Style	D	Dim Manager > Modify > Primary Units > Text > Fit

Ellipse	EL	Pick Major Diameter > Pick Minor Radius
Extend	EX	Pick Boundary > Pick Lines to Extend
Fillet	F	R > (0) > Pick 2 Lines to Intersect
Isometric	SNAP LIMIT	S > I > (.5) > OK, F5 > Top Plane > Right > Left
Drawing Limits	S	Specify Lower Left Corner > Upper Right Corner
Line	L	Pick Start > Pick End
Mirror	MI	Pick Objects > Pick 2 Points on Mirror Line
Move	M	Select Objects > Pick Start > Pick End
Offset	O	Enter offset > Pick Line or Object > Pick Side
Object Snap	OS	End Point > Center > Intersect
Stretch	S	Cross Select > Pick Start Point > End Point
Trim	TR	Select Cut Lines > Pick Lines to Trim
Undo	U	
Zoom All	Z > A	
Zoom Window	Z > W	Pick Window Corner > Pick Opposite Corner
Pan	P ctrl +	Pick Window Start > Pick Window End
Plot	P	Plot Device > Plot Settings > Pick Window
Text Single Line	DT	Pick Location > Text Height > Angle > (Type Text)
Text Multiline	T	Pick 1st Corner > Opp Corner > (Type Text)
Text Edit	ED	Pick Text > Edit
Text Symbols		(Degree= %%D) (Dia= %%C) (+/- = %%P)
Polyline	PL	Pick Start Point > Pick Several Points Along Line
Polyline Edit	PE	Pick Polyline > Width > (.2) > Fit > (Smooths Line)
Block	B	Select Objects > Base Point > Name > OK
Wblock	WB	Select Objects > Pick Point > Browse > Name > OK
Insert	I	Browse > Select Block
External		
Reference	XR	Xref-Manager > Attach > File Path to Drawing
Attribute		
Definition	ATT	Pick Insert Point > Tag > Prompt > Value > Wblock
Attribute Edit	ATE	Pick Attribute Block > Edit

This is the end of Section 10. 2-D AutoCAD Command List

11. RELATED LINKS

The AutoCAD home page at: www.autocad.com.

The United States Patent and Trademark Office Home Page at www.uspto.gov
Provides links to information about all aspects pertaining to invention patents.

This is the end of the AutoCAD 2-D Basics course.

Please attempt the quiz.