



PDHonline Course G209 (8 PDH)

AutoCAD 3-D Basics

Instructor: John R. Andrew, P.E.

2020

PDH Online | PDH Center

5272 Meadow Estates Drive
Fairfax, VA 22030-6658
Phone: 703-988-0088
www.PDHonline.com

An Approved Continuing Education Provider

AUTOCAD 3-D BASICS

Course Content

AutoCAD 3-D Basics has 8 PDH units and is divided into 10 sections. Step-by-step illustrated examples show how to use tools to create all types of three dimensional solid models of machine parts and assemblies.

Note that the first stage in creating many 3-dimensional drawings is a 2-dimensional drawing. This course begins with a recap of some of the most useful AutoCAD commands needed to: make, save, and print 2-dimensional drawings.

SECTIONS

- 1. 3D Solid Model Examples**
- 2. Menus, Toolbars, and Commands**
- 3. Dimension Controls**
- 4. User Coordinate System**
- 5. Simple 3D Solid Models**
- 6. Extrusions**
- 7. Commonly Used Commands**
- 8. Flat Plate With Five Holes**
- 9. Generic Hub**
- 10. Drawing Check List**
- 11. 3-D Command List**
- 12. Related Links**

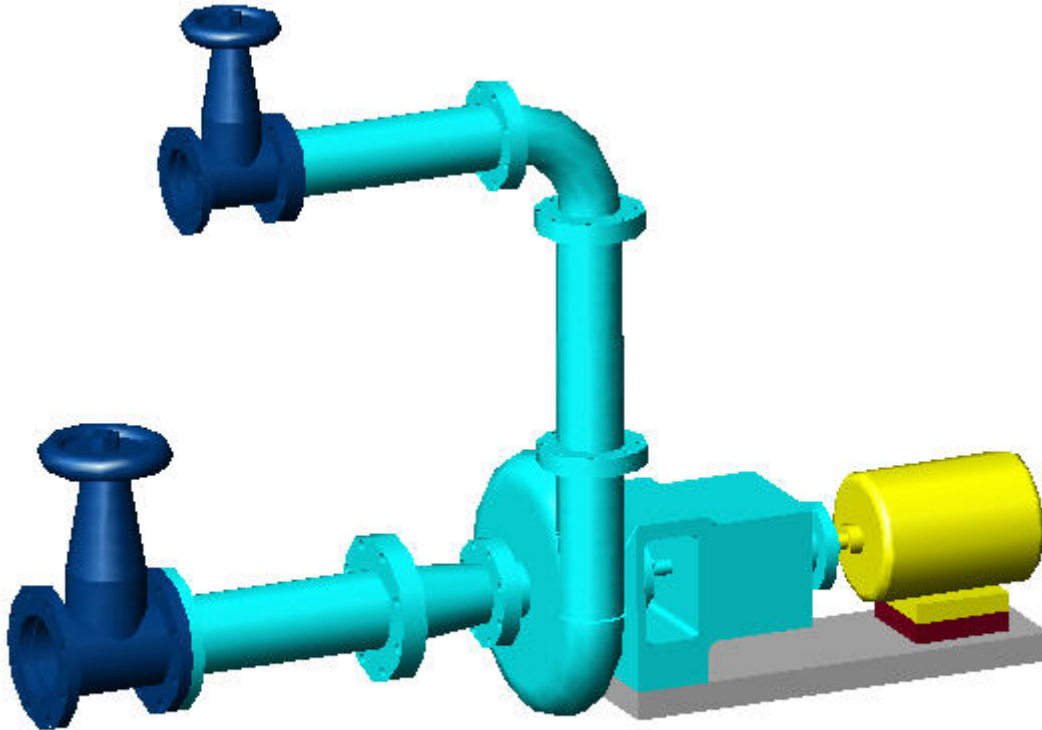
The techniques described here include all versions of AutoCAD released including releases 14 to 2012.

The AutoCAD release 2011 manual has over 1000 pages. This lesson has only 87 pages but the most used 3-D drawing methods are described, enabling the reader to make all types of three dimensional drawings. All versions of AutoCAD lite are used to create 2 dimensional drawings only.

AutoCAD Inventor is: 3D, parametric, and has many more advanced features than regular AutoCAD. Please see my course: G280, AutoCAD Inventor - Solid Modeling, Stress and Dynamic Analysis.

1. 3D SOLID MODEL EXAMPLES

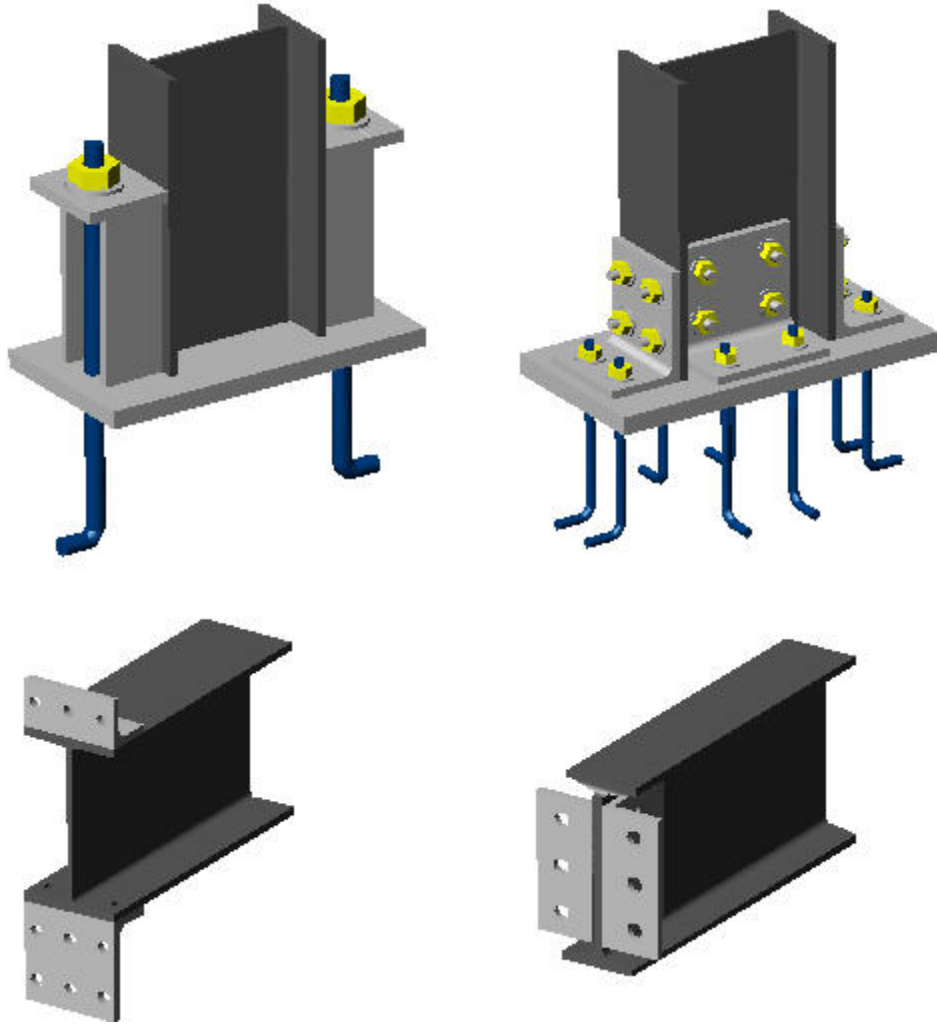
Three dimensional AutoCAD drawing commands and tools described in this lesson are applicable to the: Mechanical, Structural, Industrial, Chemical, Electrical, Manufacturing, and Civil engineering disciplines.



CENTRIFUGAL PUMP INSTALLATION

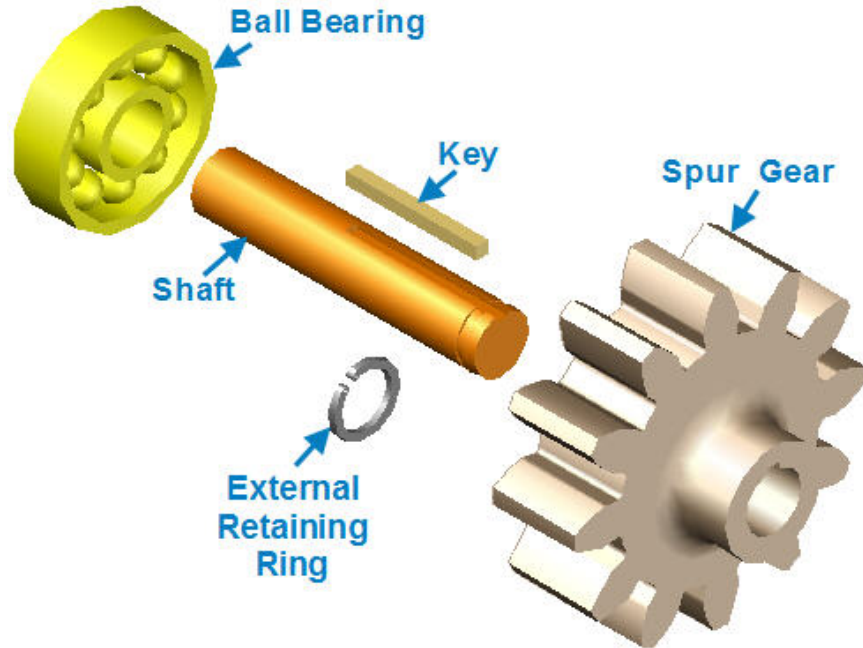
PUMP INSTALLATION

The pipe spool and pump installation above is 3D solid modeled in AutoCAD.



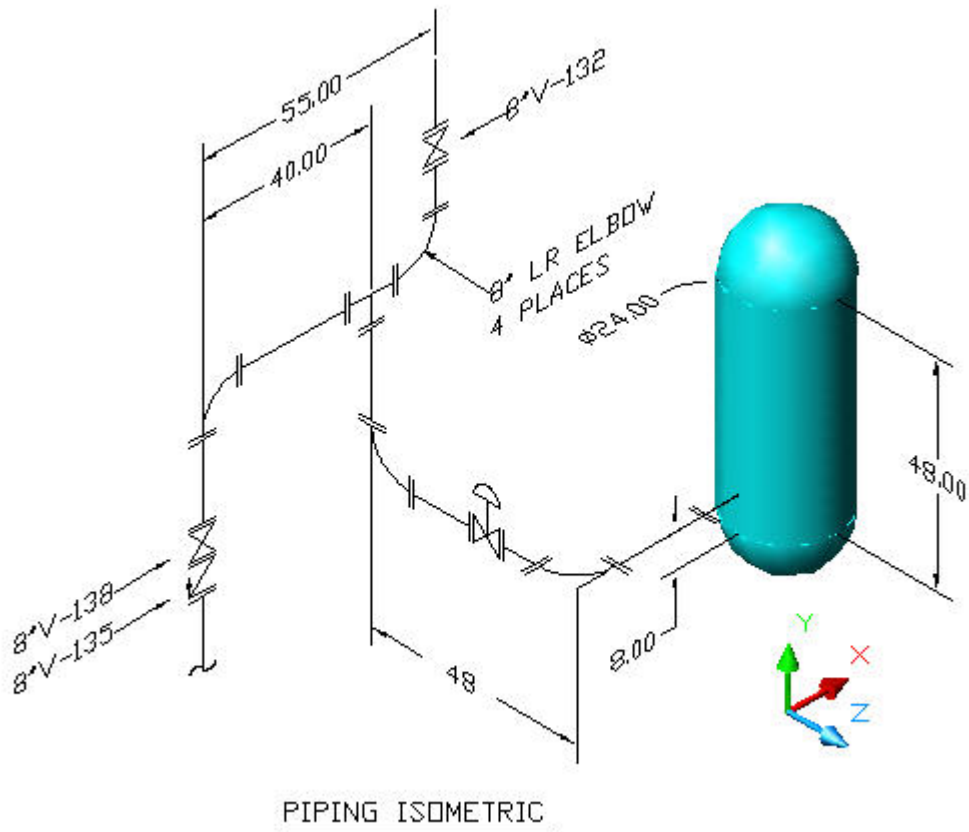
STRUCTURAL STEEL DETAILS

The 3D structural details help fabricators and construction crews to understand 2D drawings.



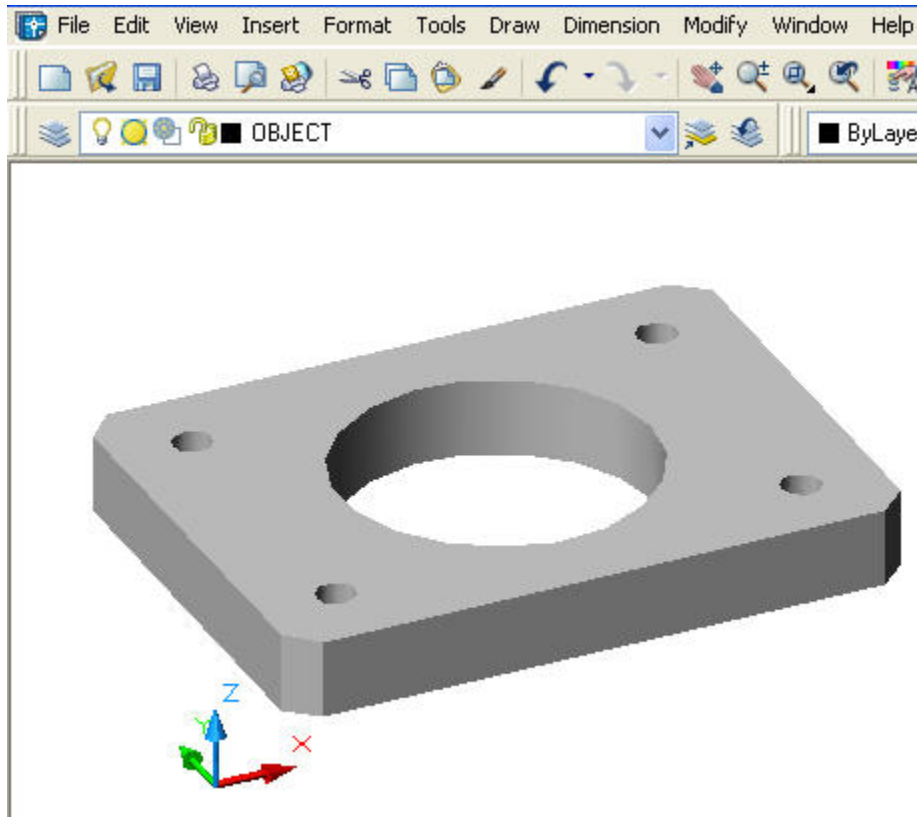
MACHINE PARTS

The machine parts above were created using AutoCAD solid modeling.



PIPING ISOMETRIC

The pipe spool design above is another AutoCAD drawing.



2. MENUS, TOOLBARS AND COMMANDS

Drop Down Menus

The drop-down menus along the top of the display are shown above and contain all of the 2D and 3D drawing commands under four headings:

Insert, Draw, Dimension, and Modify

Model View

The flat metal plate with 5 holes shown above is in the **Model View** area.

The **Toolbars**, described below also contain all AutoCAD commands.

Toolbars AutoCAD release 14 to 2005

To show the Toolbars shown below select: **View > Toolbars > Check: Draw > Modify > Dimension > Close.**

Toolbars AutoCAD release 2006 - 2012

To show Toolbars right click any Drop Down Menu or Toolbar to open a complete list of Toolbars and select: **Draw > Modify > Dimension > Solids > Solids Editing > View .**

Quick Key Commands

Quick Key commands are typed on the Command Line and include:

- L** for Line
- C** for Circle
- M** for move
- E** for Erase
- MI** for Mirror
- CP** for Copy

and are illustrated below.

FUNCTION KEYS

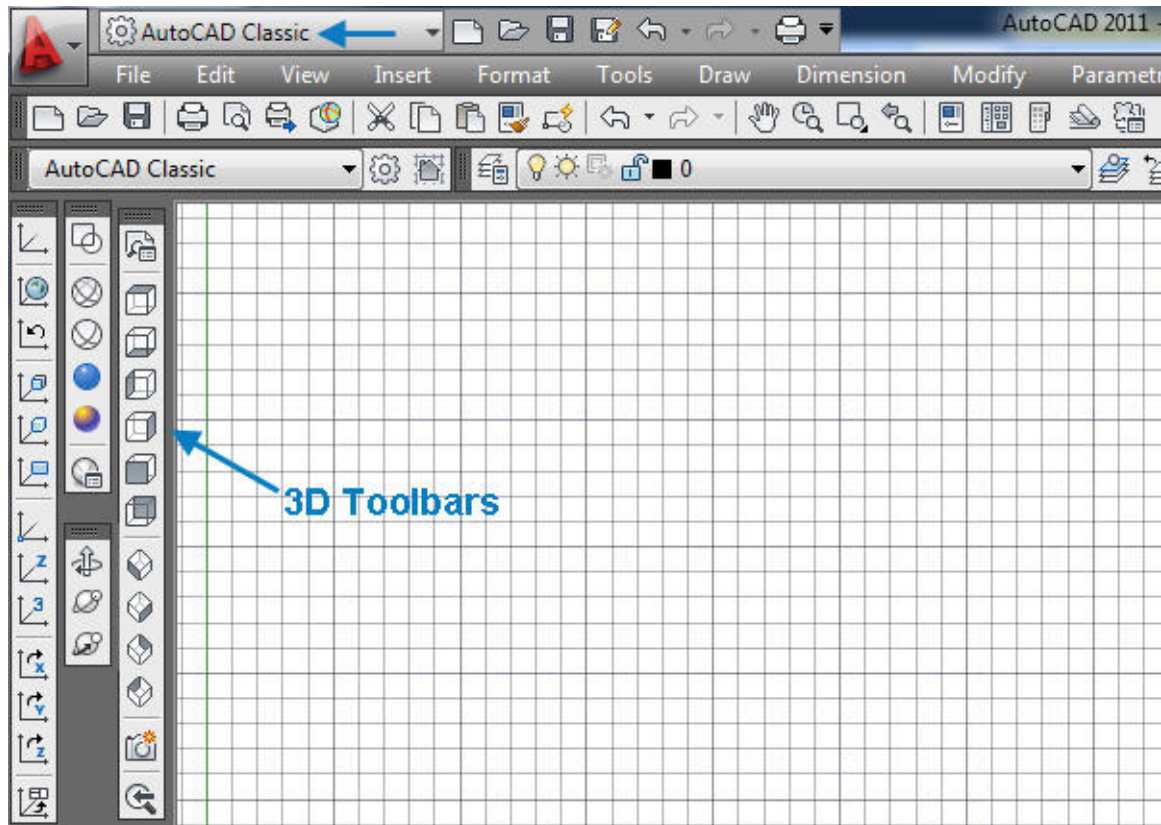
The function key **f3** is used to toggle Object Snap on and off.

The function key **f7** is used to toggle Grid on and off.

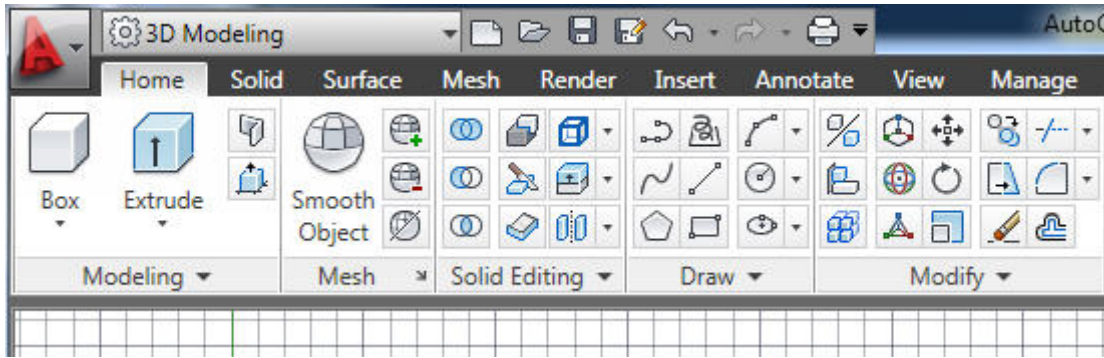
The function key **f8** is used to toggle Ortho on and off.

The function key **f9** is used to toggle Snap on and off to Grid points.

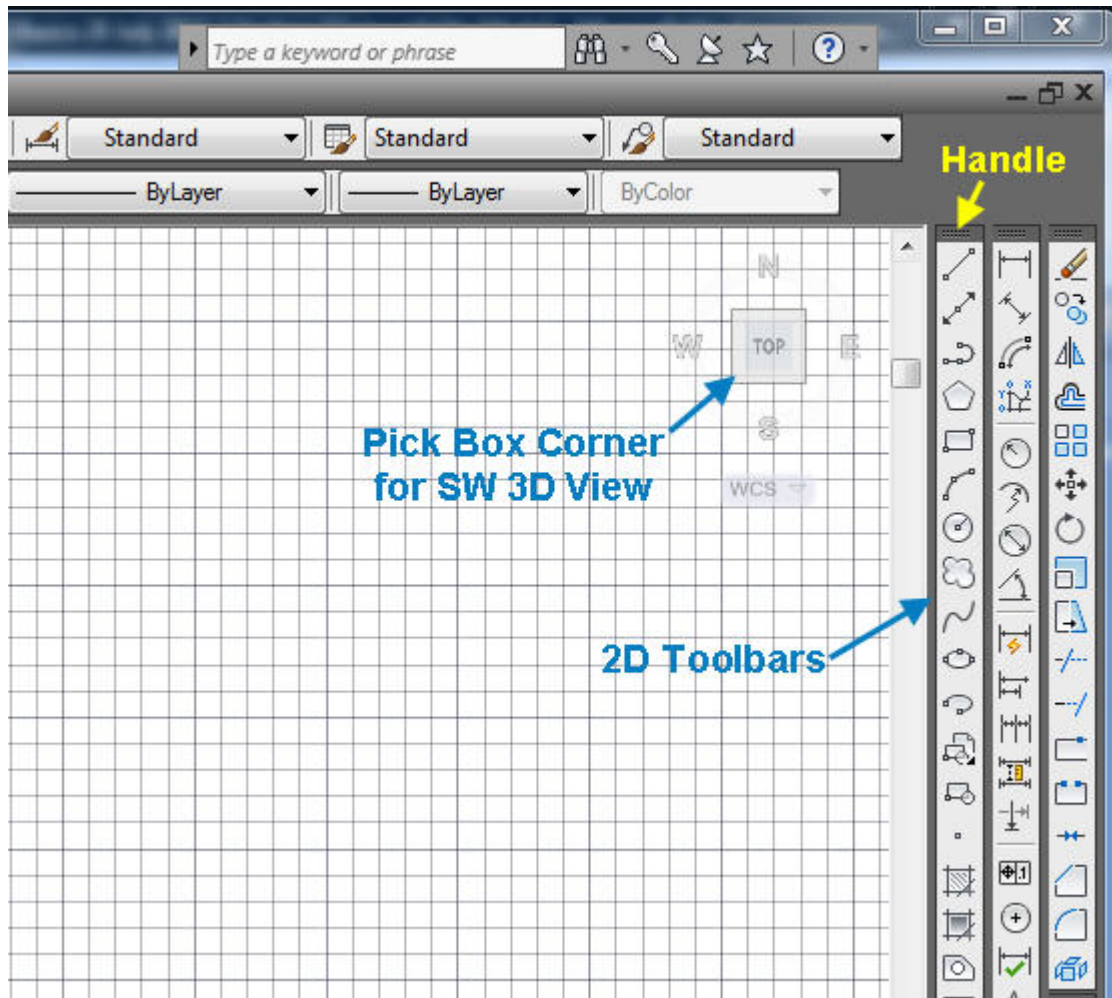
Work Spaces



Click on the “Workspace” drop down menu “AutoCAD Classic” and select the desired environment.



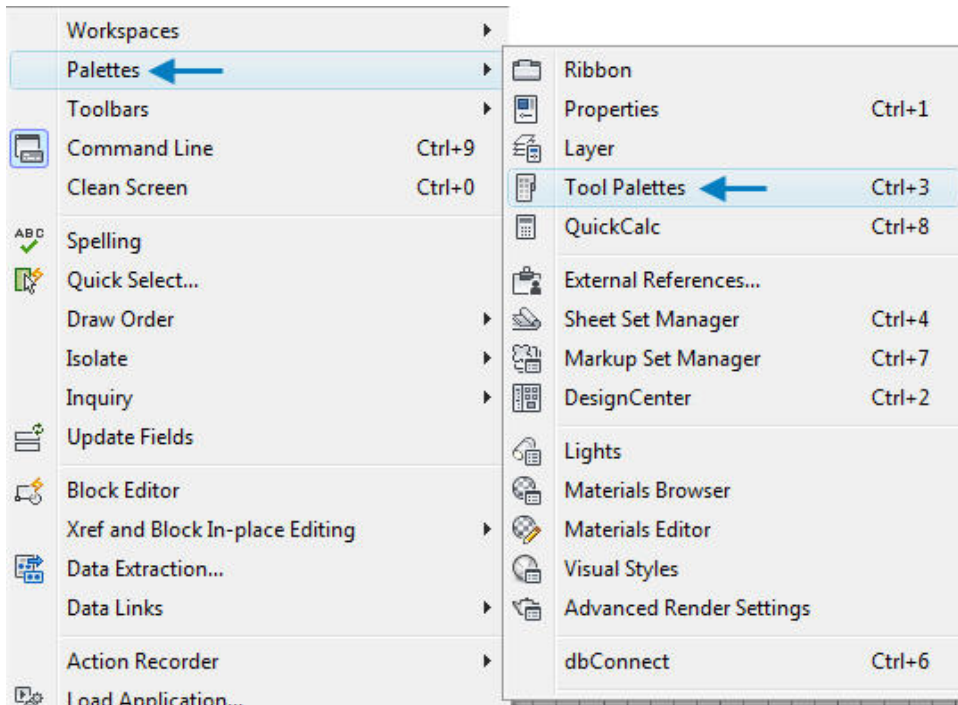
3D Modeling workspace has been selected above.



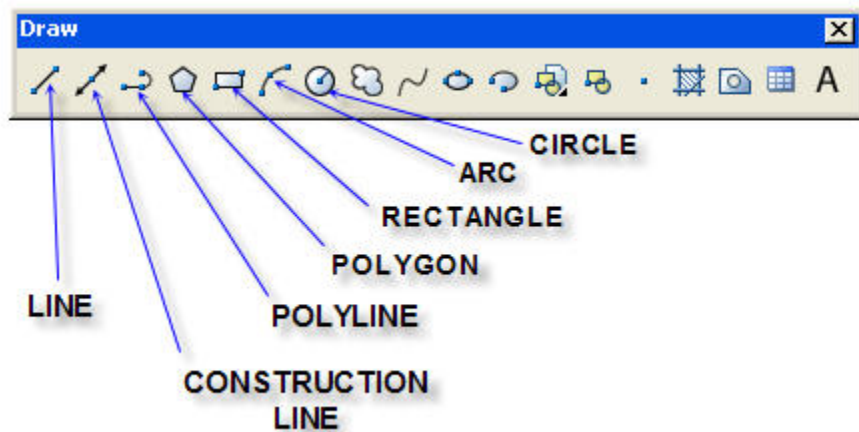
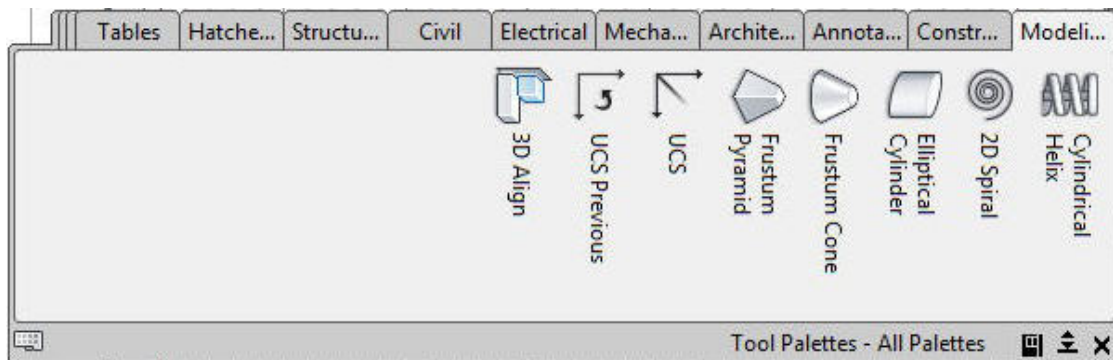
Right click on any toolbar > Pick desired new toolbar from the list of all toolbars.

Docking Toolbars

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



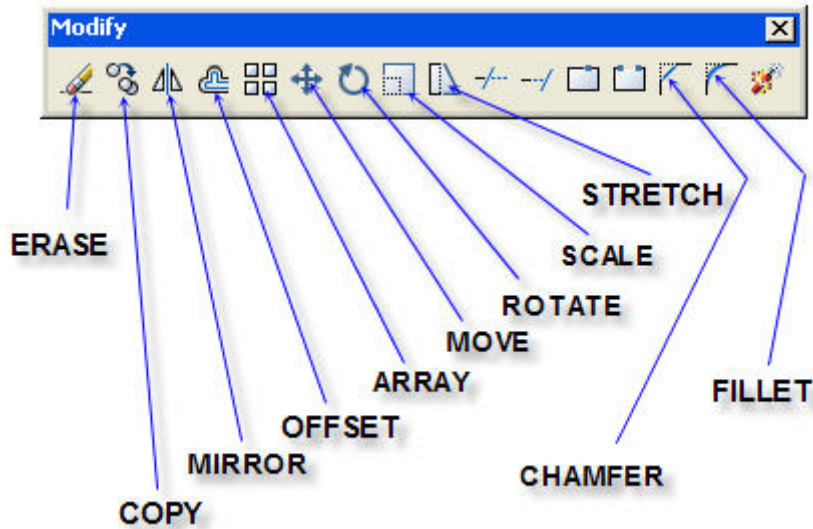
Pick drop-down menu: Tools > Palettes > Tool Palettes > All Palettes opens as shown below.



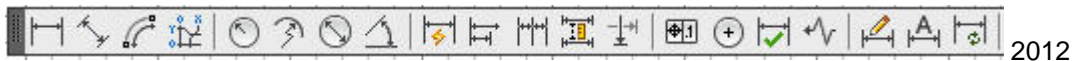
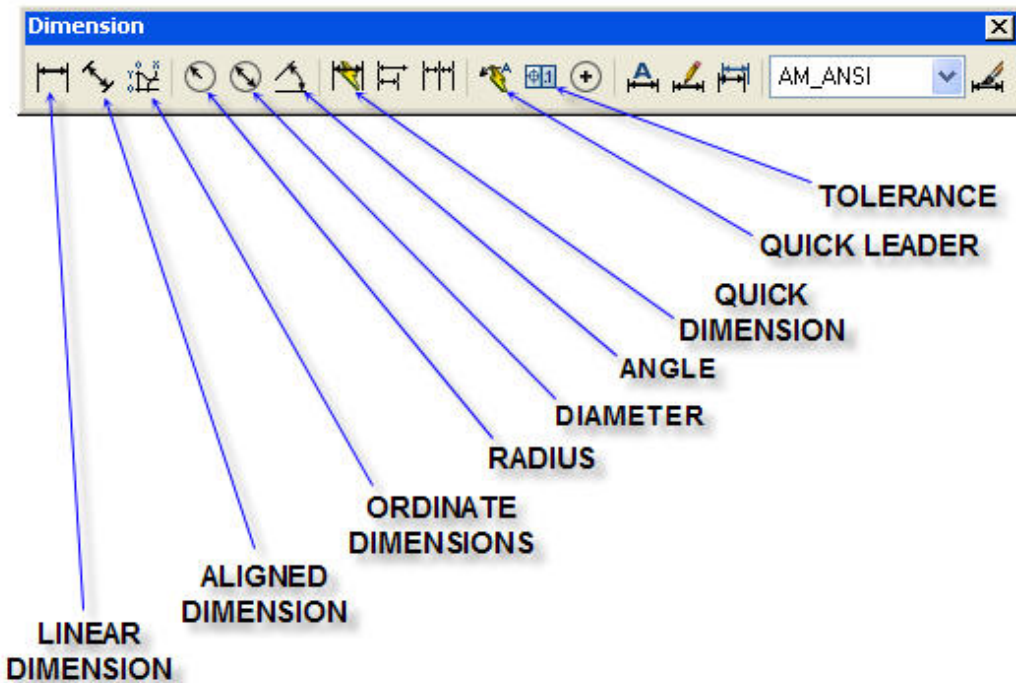
Draw toolbar 2000 with commands labeled above and Draw toolbar 2012 below.



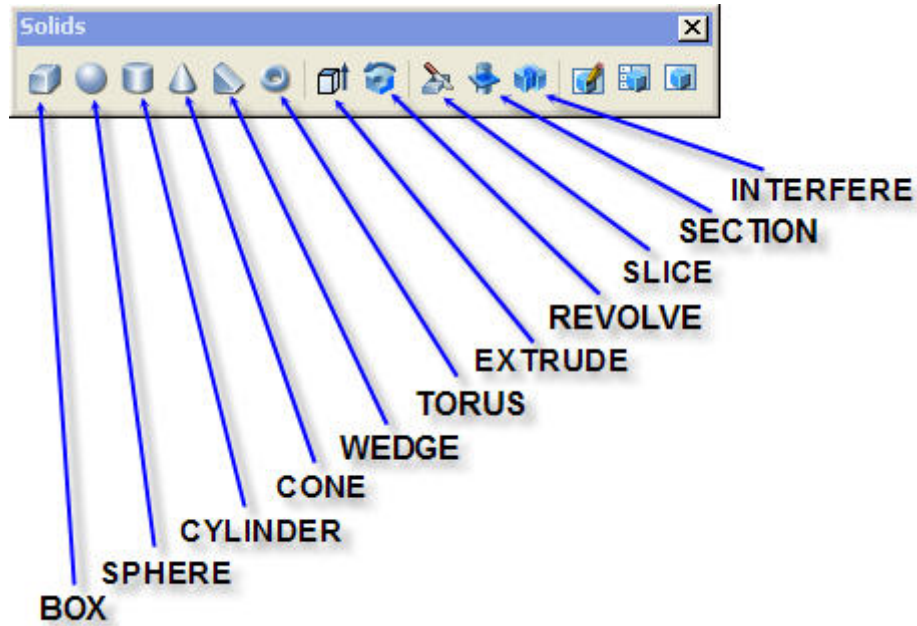
Each icon has a pop-out label describing its purpose. The icons of the most commonly used



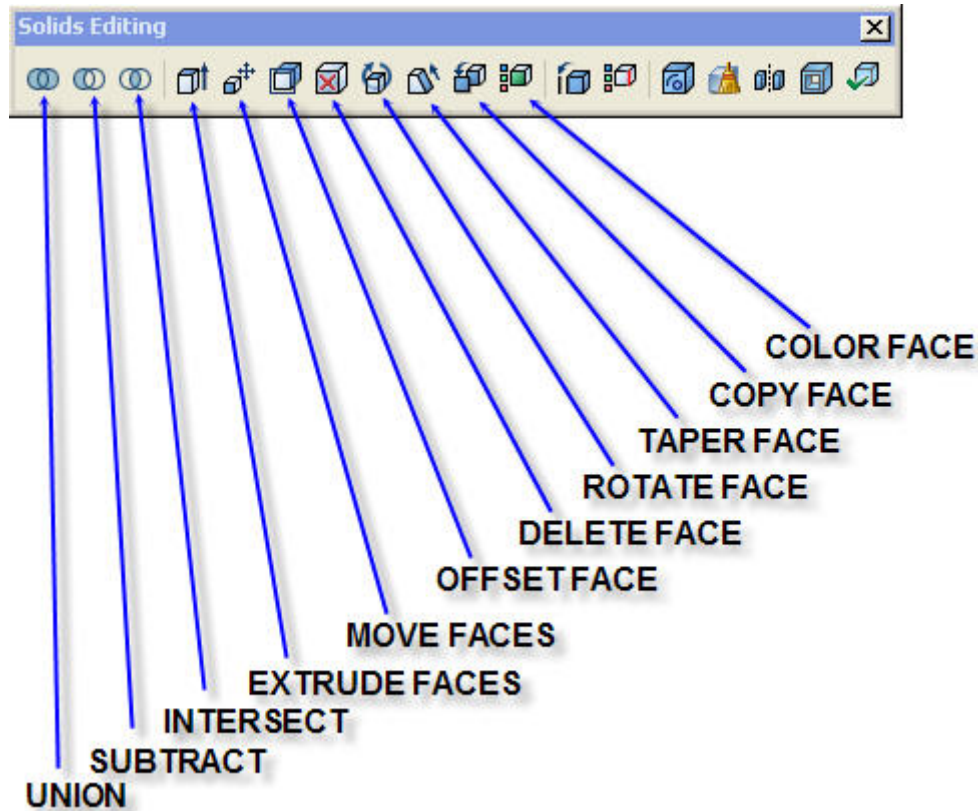
Modify toolbar commands are labeled above.



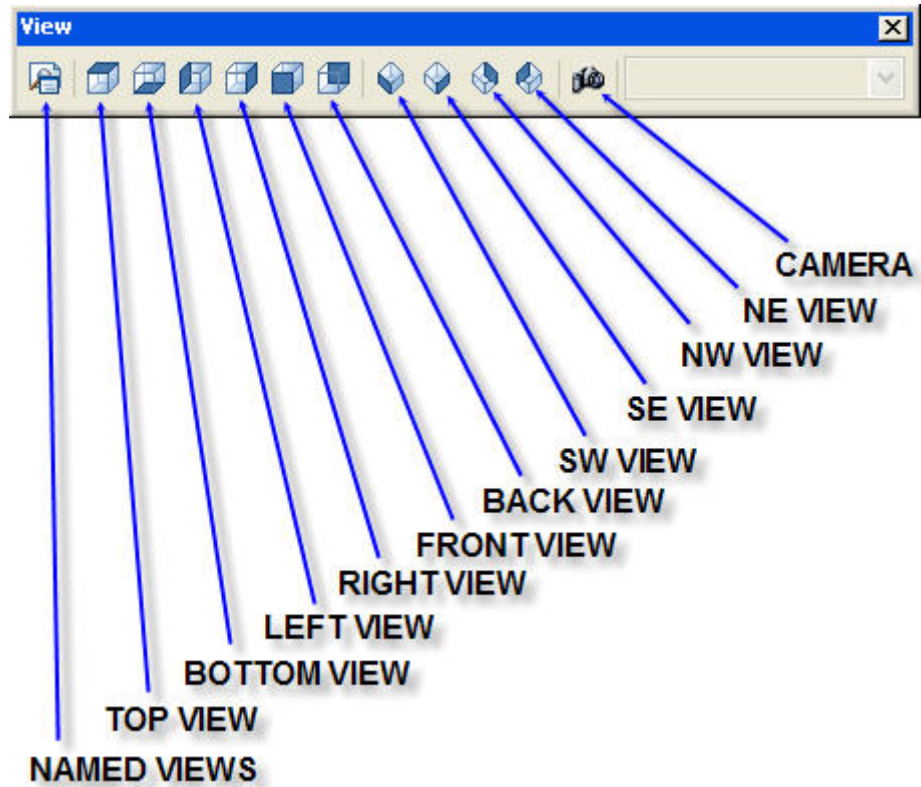
Dimension toolbar commands are labeled above.



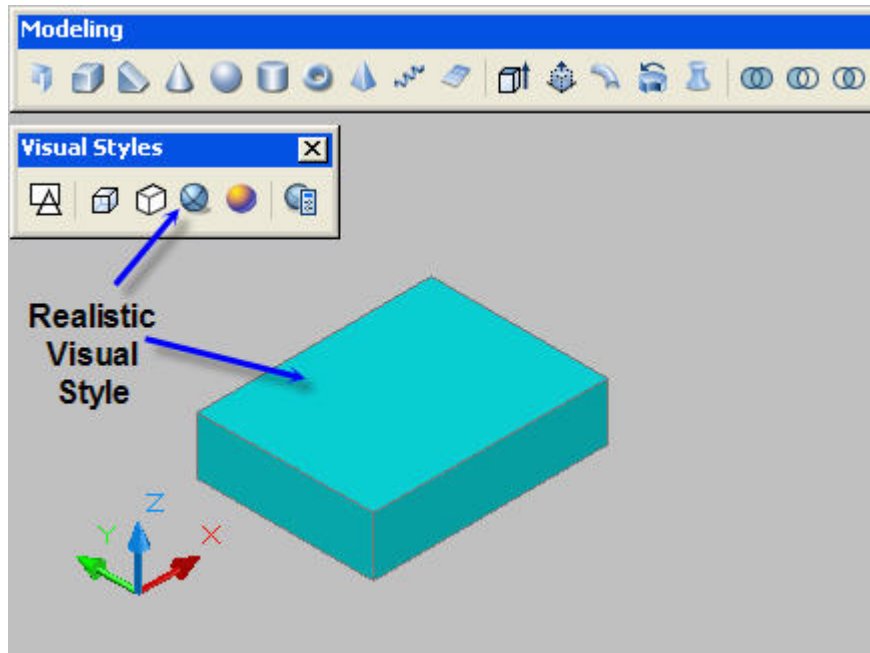
Solids toolbar commands are labeled above.



Solids Editing toolbar commands are labeled above.

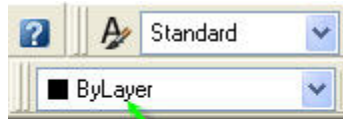


View toolbar commands are labeled above.



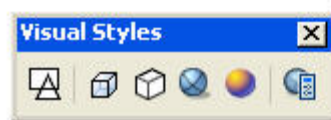
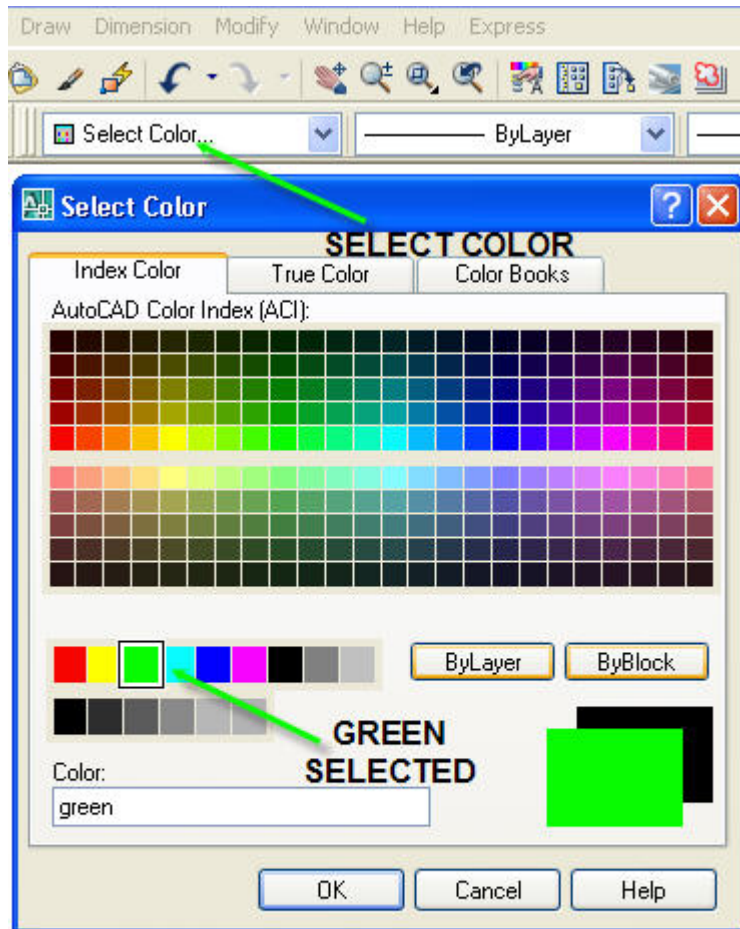
The **Visual Styles** toolbar is pictured above.
Select the object to change its Visual Style

Pick **Visual Style** “Realistic”



Select Color menu

Pick the **ByLayer** drop down menu to open the **Select Color** dialog box below.

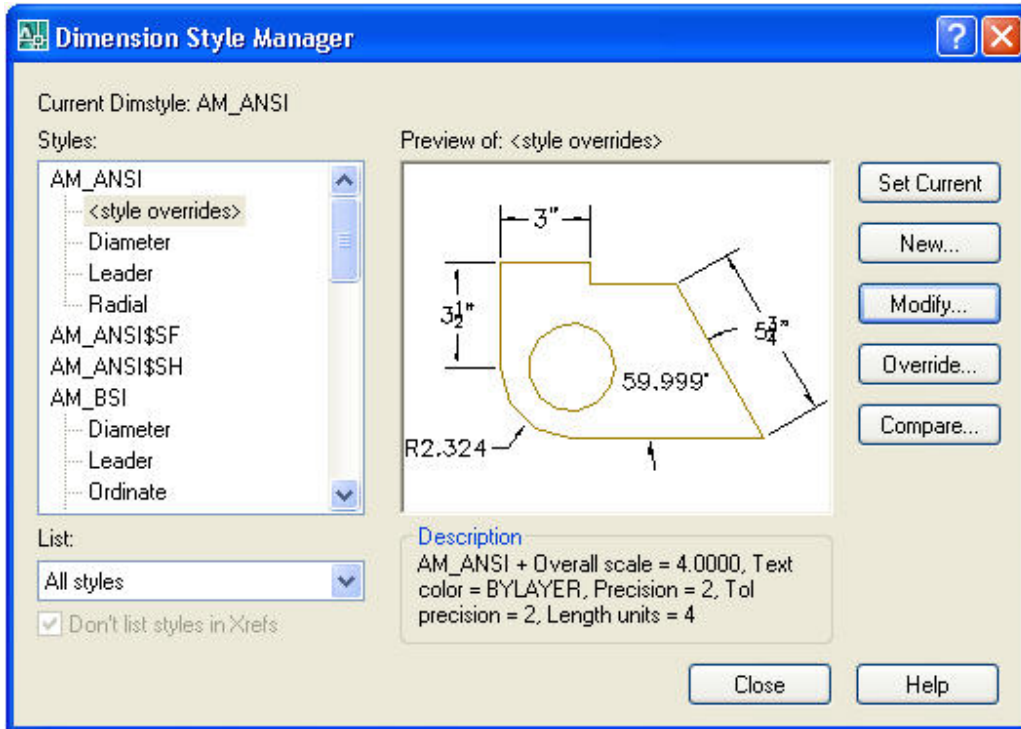


Change an objects color with the **Select Color** dialog box shown above.

Select the object > Select a color > OK

3. DIMENSION CONTROLS

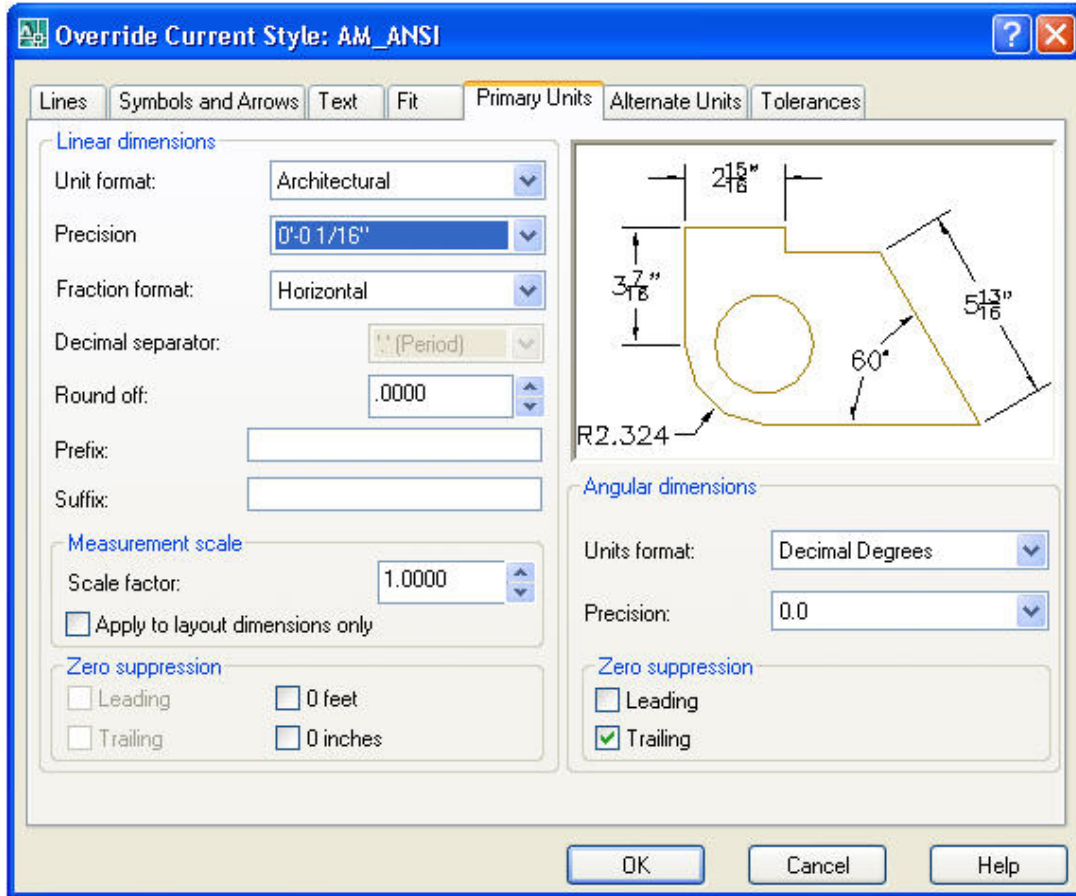
This dialog box below opens when: **D** for Dimension Style Manager is typed on the command line.



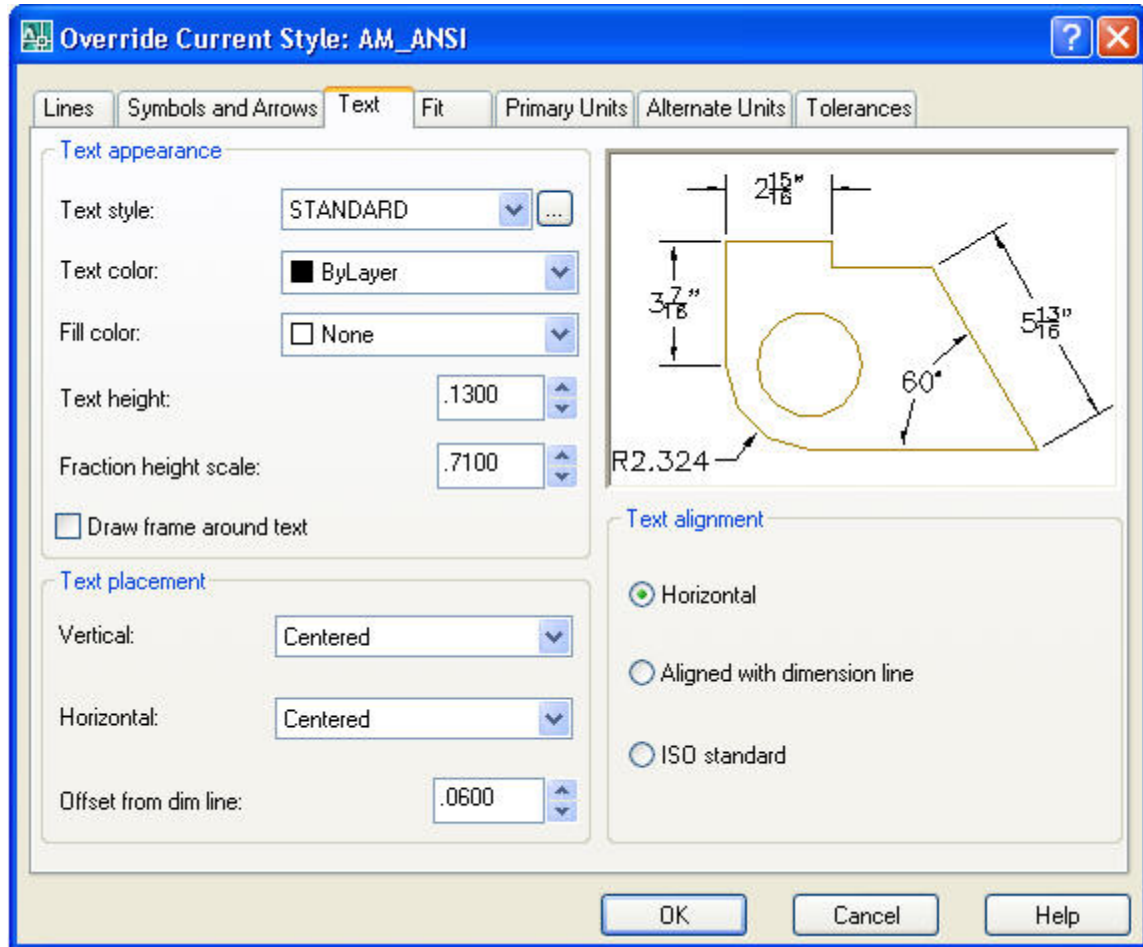
Note: All dimensions are in the X-Y plane.

Dimension features are adjusted with the **Dimension Style Manager** dialog box above.

Select the **Modify...** button to obtain the **Override Current Style** dialog below.

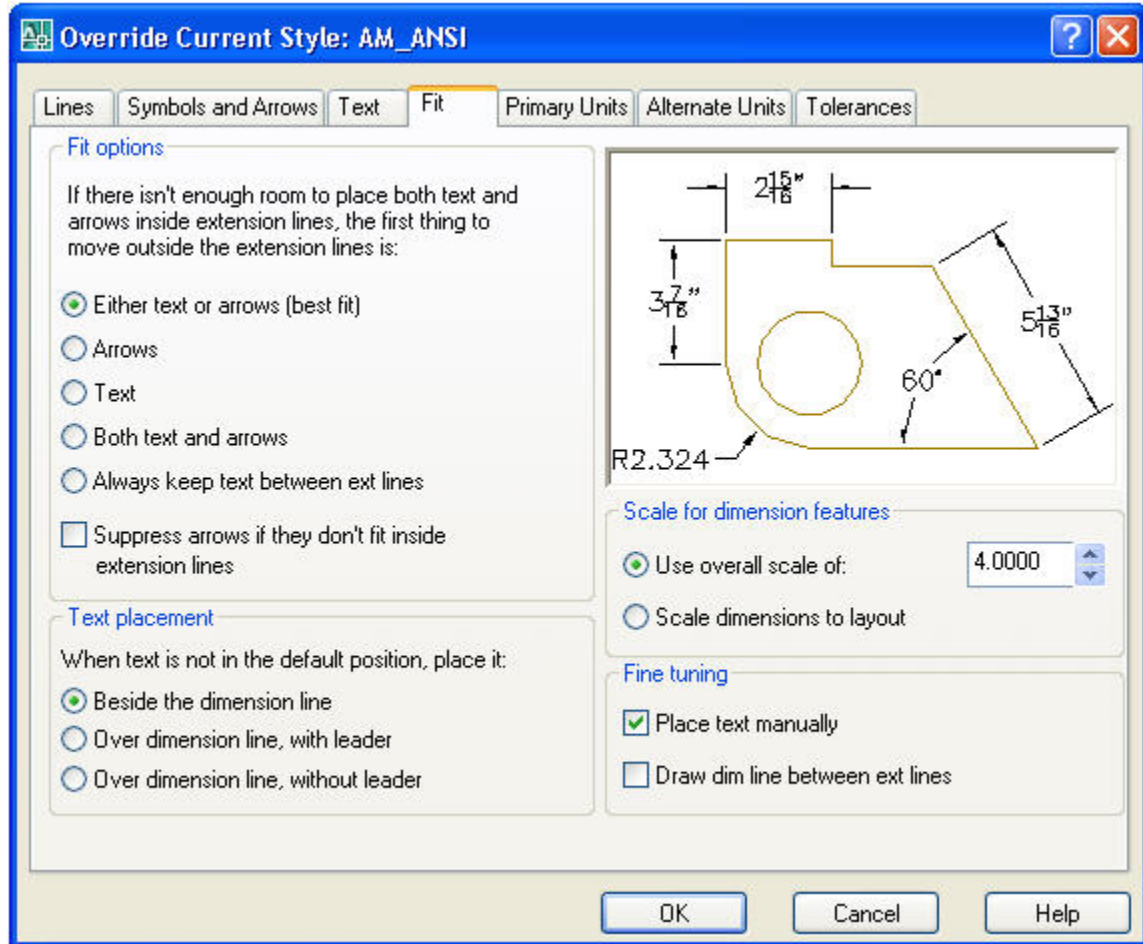


Primary Units Tab offers: Scientific, Decimal, Engineering, Architectural, and Fractional dimension styles.



Text Tab above offers: Text height: .1300 is the normal size when dimensions are inches. Text height: 3.3 when dimensions are millimeters.

Dimension size can be easily scaled by typing: **DIMSCALE** on the command line, > Type the new scale number > Select menu **Dimension > Update** > Select all dimensions needing to be updated to the new scale.



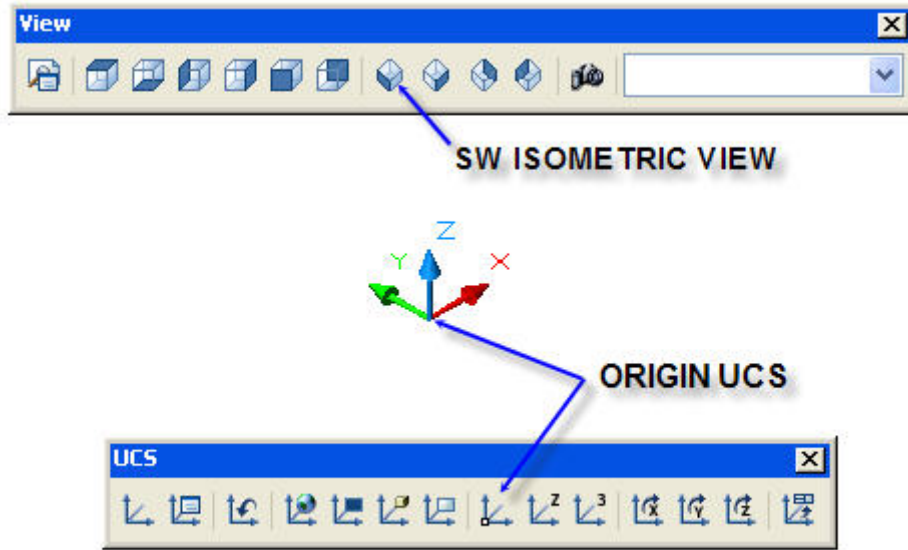
Fit Tab above offers: Text placement and Fine tuning.

Pick Beside the dimension line and

Place text manually.

4. USER COORDINATE SYSTEM (UCS)

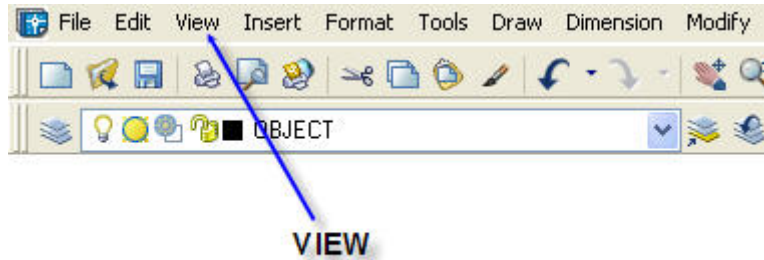
The User Coordinate System commands are used to establish the **X-Y, X-Z, and Y-Z plane orientations and the Isometric View.**



SW ISOMETRIC VIEW

Pick the **SW Isometric View** icon on the **View** toolbar as illustrated above.

Display the **X, Y, and Z Origin UCS** as shown above and described below.



USER COORDINATE SYSTEM (UCS)

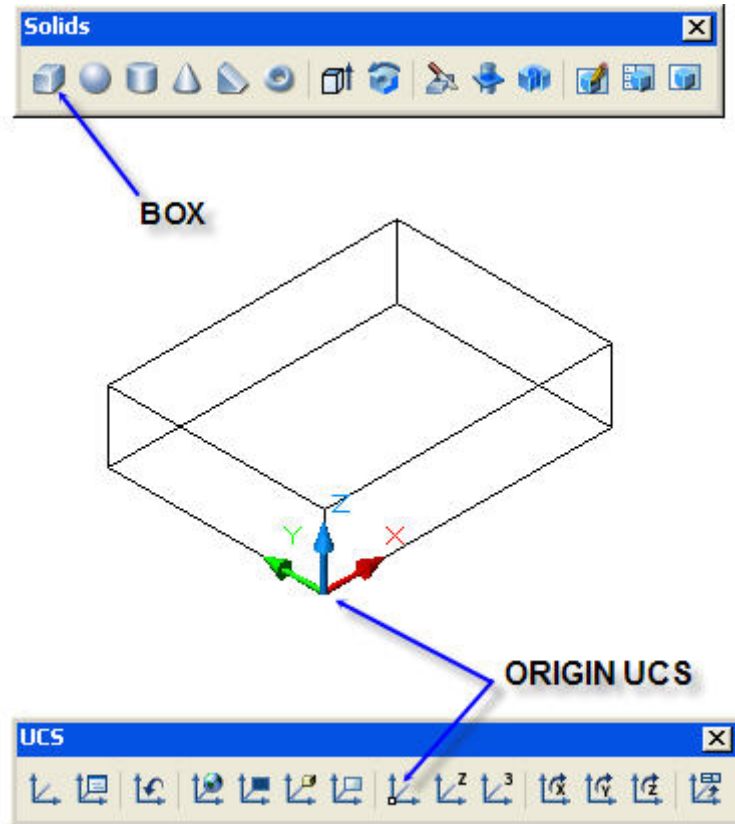
Select the **View** drop down menu shown above to view the **X, Y, and Z** coordinate origin.

Pick **Display > UCS Icon > ON**.

Repeat by picking **Display > UCS Icon > On > Origin**.

5. SIMPLE 3D SOLID MODELS

Box, Sphere, Cylinder, Cone, Wedge, and Taurus form basic 3D building blocks. Creating, modifying, and combining each of them is described below.



BOX

Pick the **Box** icon on the **Solids** toolbar.

Pick the location of the bottom corner of the box that is to be drawn, as shown above.

Type **L** for Length in the **X** direction > **4**.

Type **3** for Width in the **Y** direction.

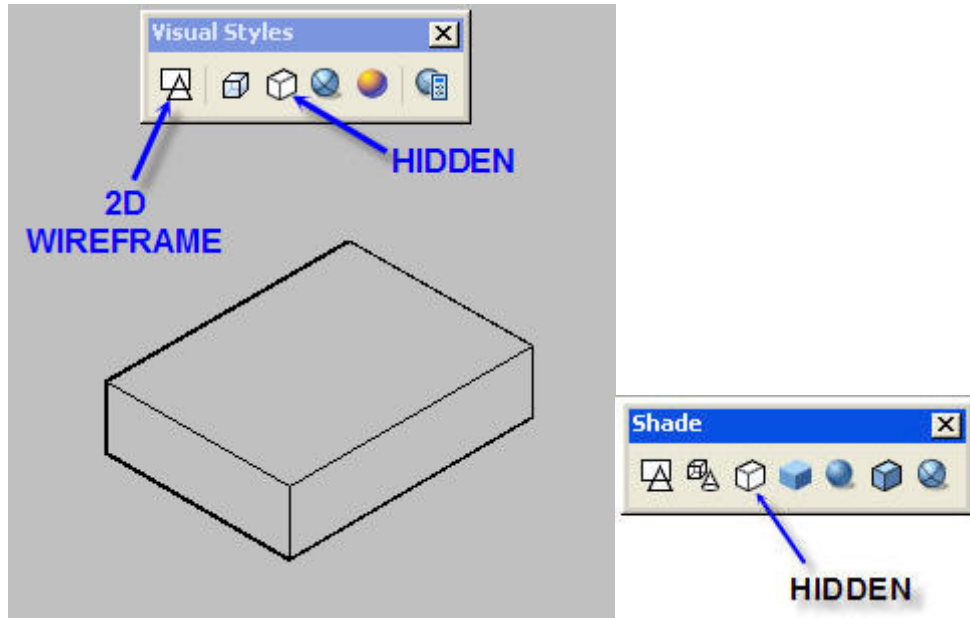
Type **1** for Height in the **Z** direction.

Press the **Space bar** to end this command.

PLACE UCS ORIGIN

Pick the **Origin UCS** icon as shown above.

Pick the bottom corner of the **Box** to place the **UCS Origin**.



HIDE HIDDEN LINES

Pick the **Hidden** icon on the **Styles** toolbar (**Shade** toolbar on older releases of AutoCAD) to view the solid model.

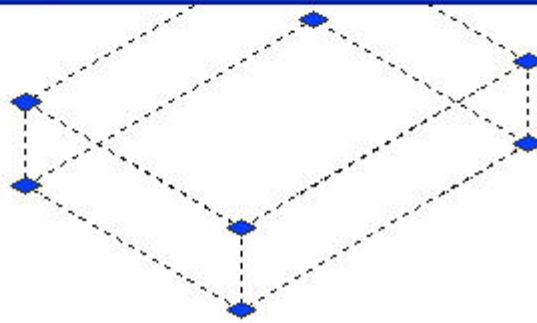
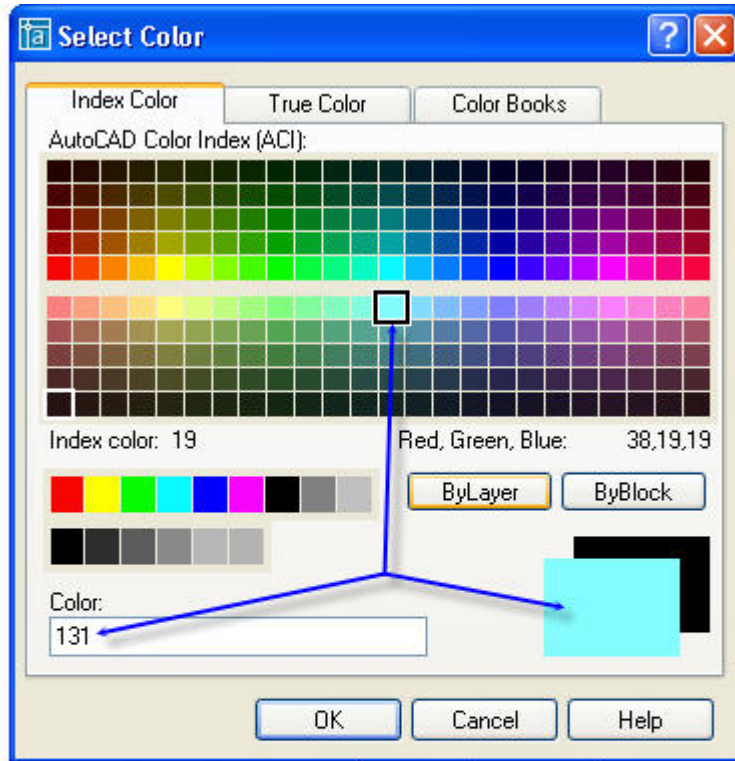
Pick the 2D Wireframe icon to restore the background.



COLOR SELECTOR

COLOR SELECTOR

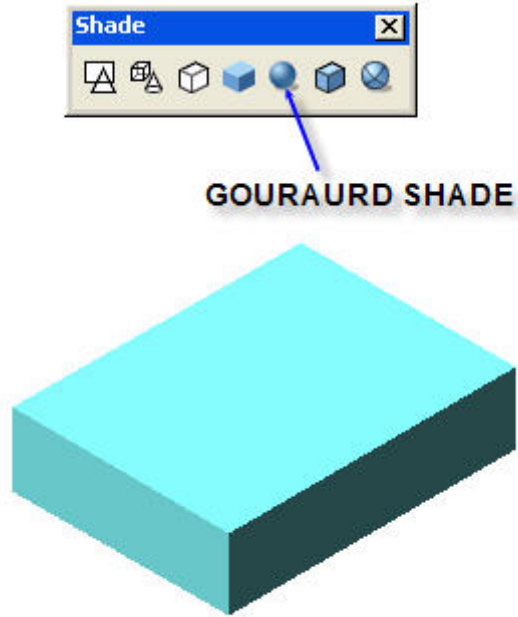
Pick the **Color Selector** drop-down menu as shown here.



SELECT COLOR

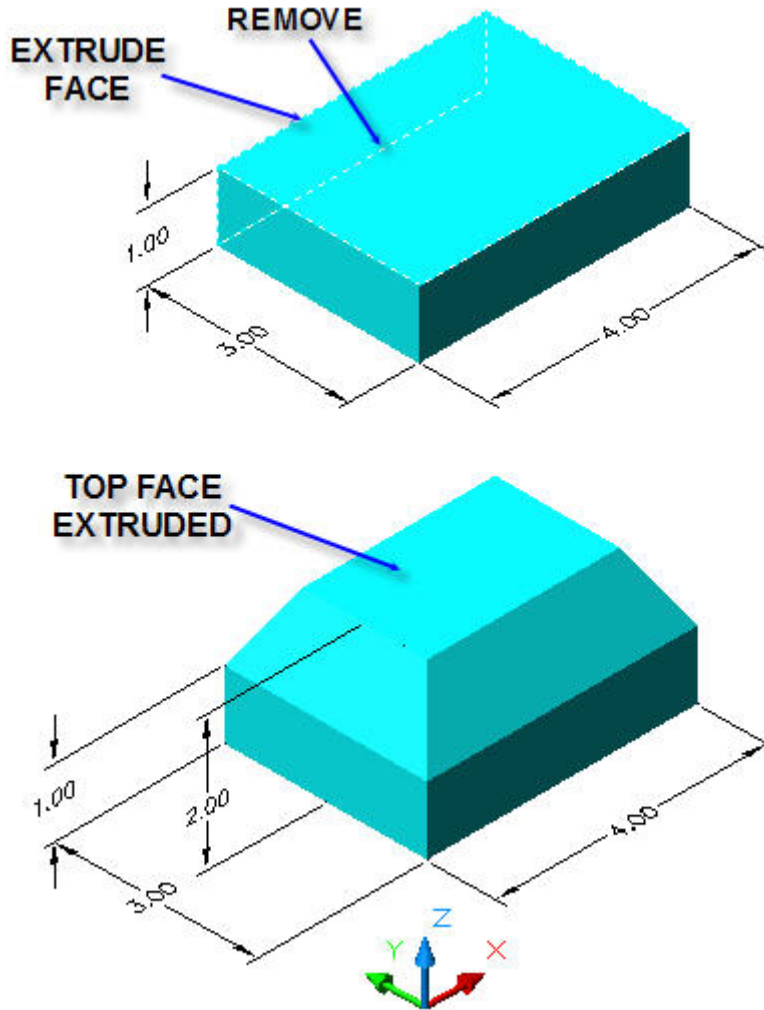
Pick the desired **Color** square in color pallet as shown above.

Press the **Space bar** to end this command.



SHADE

Pick the **Gouraud Shade** icon on the **Shade** toolbar and all items drawn in **Model View** will be shaded.



EXTRUDE FACE

Select the **Modify** drop down menu > **Solids Editing** > **Extrude faces**.

Pick the **Box** top surface edge.

Type **R** for Remove and pick the back bottom **Box** edge as shown above.

Press the **Space bar** to indicate that all surfaces have been selected or removed.

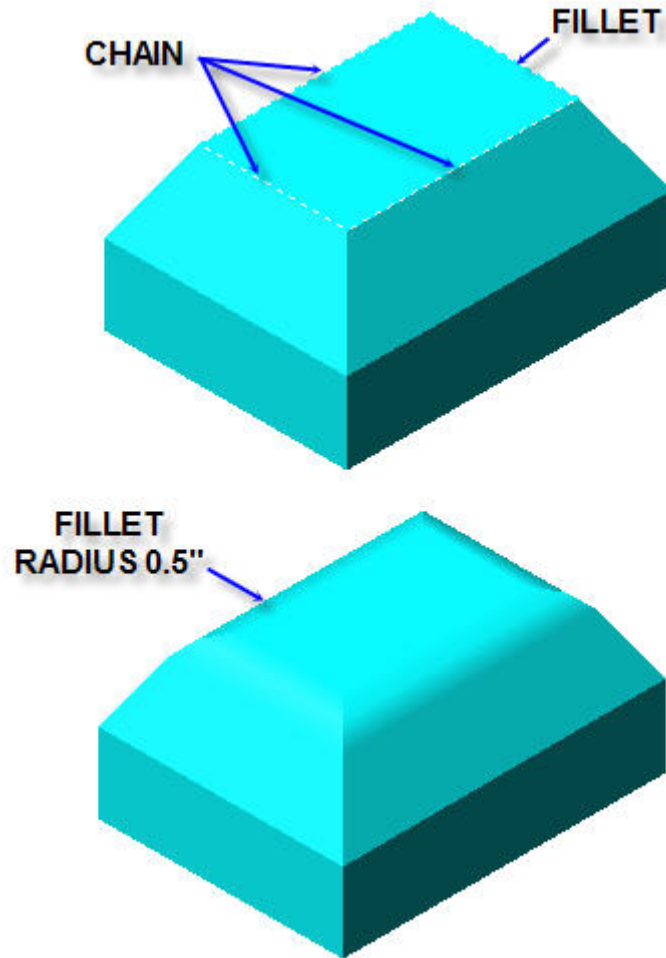
Type > **1.0** for extrusion height in the **Z** direction.

Type the angle of taper > **30** (degrees).

Press the **Space bar** to end this command.

See, “**ADD DIMENSIONS**” page 28.

Rev: 15 Mar 08



FILLET

Type **F** for Fillet.

Type **R** for Radius.

Type **> 0.5**

Pick the **Box** top surface edge where it shows Fillet above.

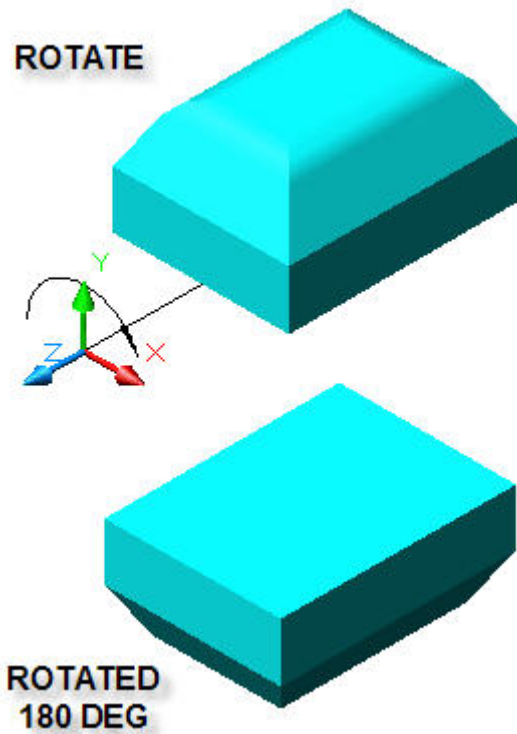
Press the **Space bar**.

Type **C** for Chain

Pick the 3 remaining top edges.

Press the **Space bar** to end this command.

Select the entire solid and restore the surface color as described in, "SELECT COLOR" above.



ROTATE BOX IN THE X-Y PLANE

Type **RO** for Rotate.

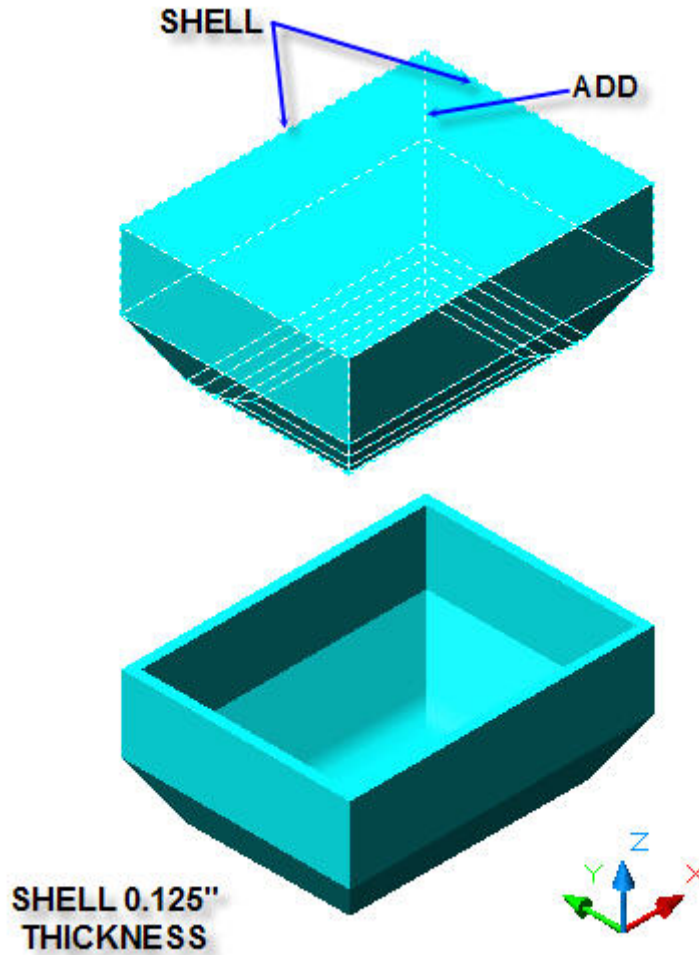
Select the **Box** shown at the top of the above figure.

Pick a **Base Point** about which rotation will take place in the **X-Y** plane.

Type **R** for Radius.

Type rotation angle > **180** (+180 anticlockwise, -180 clockwise)

Press the **Space bar** to end this command.



SHELL

Select the **Modify** drop down menu > **Solids Editing** > **Shell**.

Select the 3D Solid **Box**.

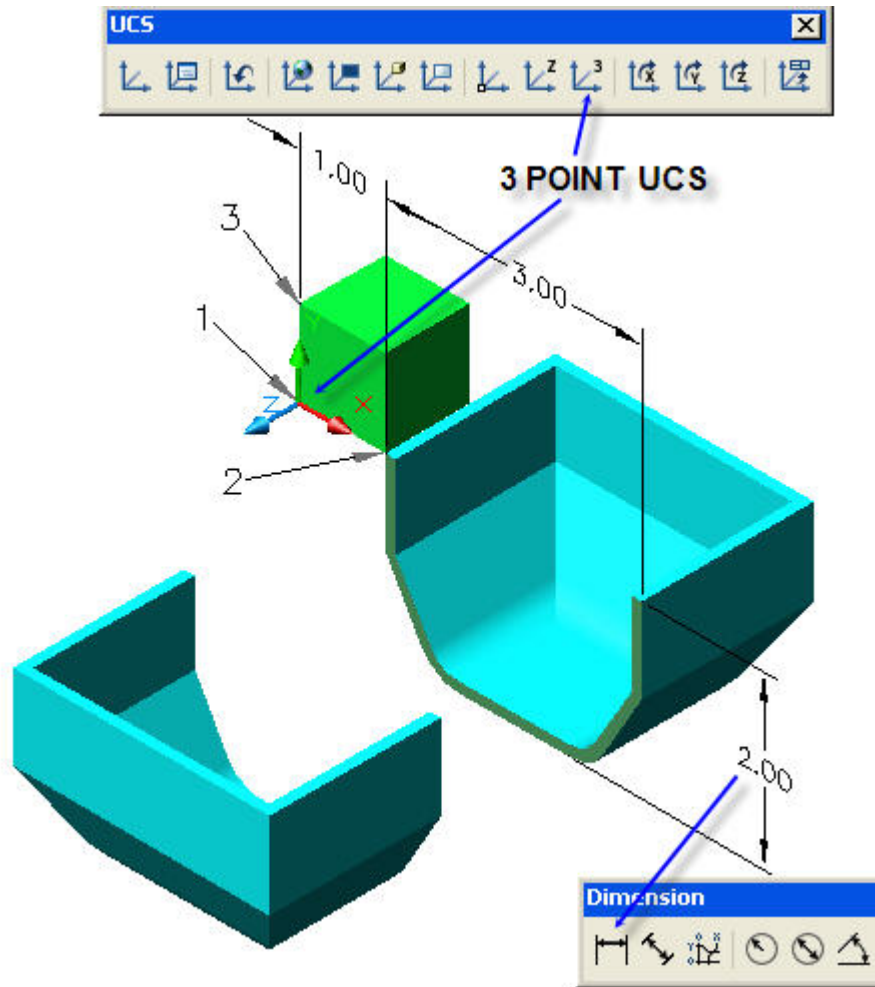
Select the top 2 edges as shown above to **Remove Faces**.

Type **A** to add faces and pick the far corner to restore faces removed by the previous step.

Press the **Space bar**.

Enter shell offset distance or thickness > **0.125**.

Press the **Space bar** to exit this command.



3 POINT UCS ORIGIN

Place the User Coordinate System, UCS.

Create the green box: Type: **BOX** on the command lines.

Pick the location of the lowest corner.

Type: **C** for Cube > **1** for dimension

Place the green box at the point on the blue box shell edge where it is to be sliced as shown above.

Pick the **3 Point UCS Origin** icon as indicated above.

Pick point **1** for **Origin**.

Pick point **2** for the **X axis**.

Pick point **3** for the **Y axis**.

SLICE

Type **SL** for Slice.

Pick cutting plane points **1**, **2**, and **3**.

Press the **Space bar**.

Type **B** for **Both sides** to remain after slicing.

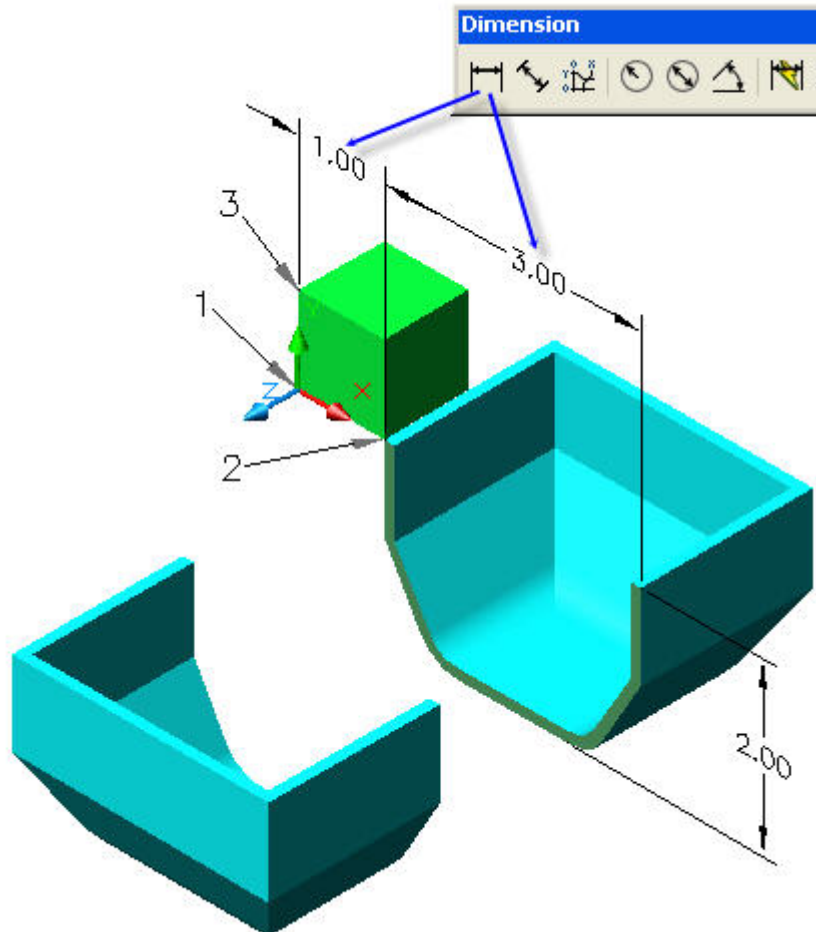
Press the **Space bar** to end this command.

Type **M** for Move.

Select the left end of the **Box**.

Pick any **Base point**.

Drag the mouse pointer so that the left end of the box moves to the location shown.



ADD DIMENSIONS

Note: All dimensions are in the X-Y plane on the left side of the green box.

Use 3 Point UCS to move the X-Y plane to the box right side to dimension the length of the box.

Type **OS** for **Object Snap** in the **Drafting Settings** dialog box pictured below.

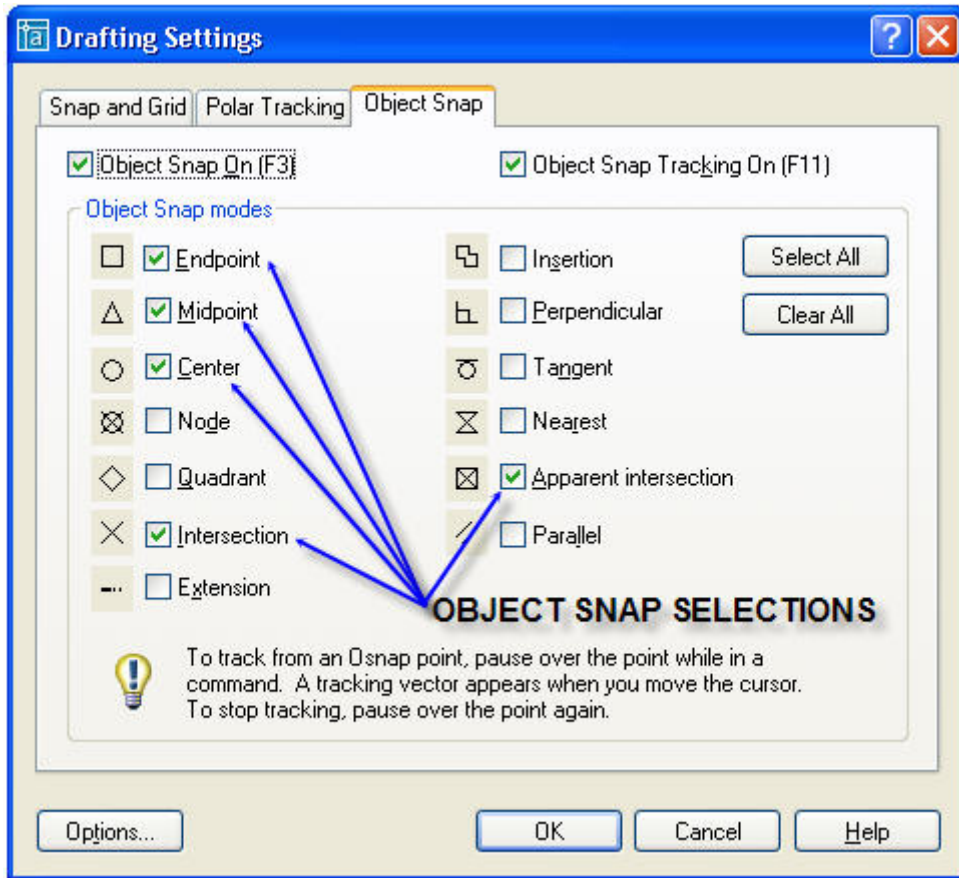
Now the mouse pointer will snap to: **Endpoint**, **Midpoint**, **Center**, and **Intersection**.

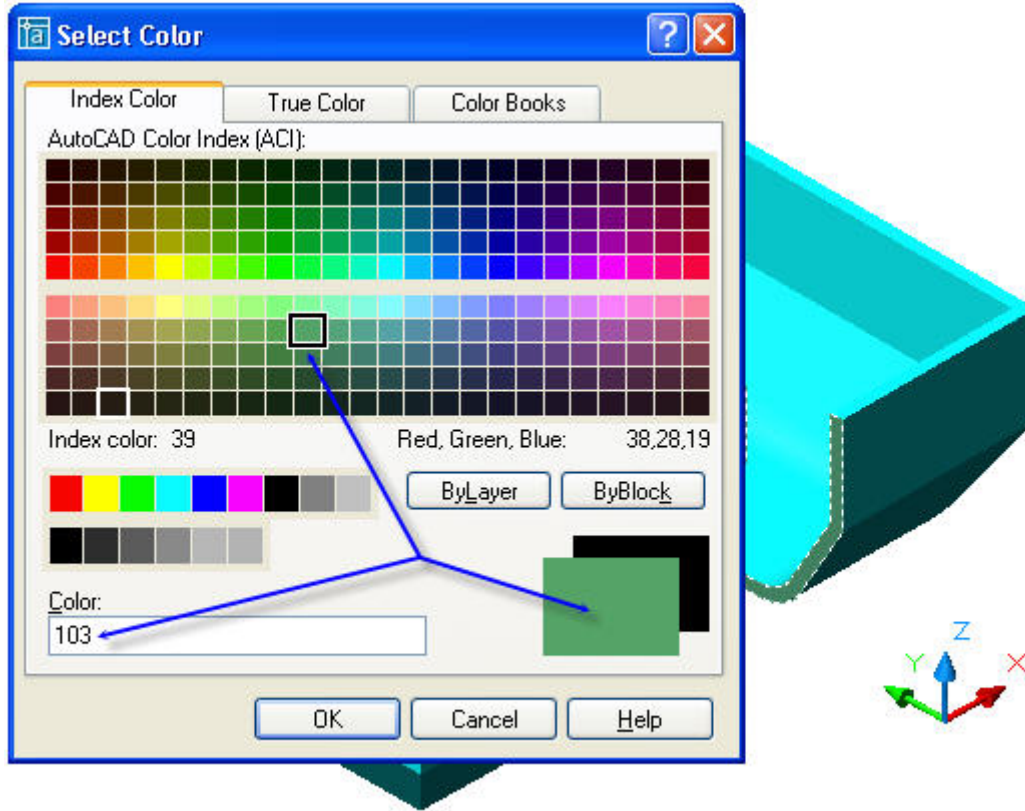
Toggle function key **f3** until **Object Snap** is **ON** within the command line.

Pick the Linear Dimension icon as shown above.

Pick each top corner of the green box,

Pick the 1.00 inch dimension location.





COLOR SELECTOR

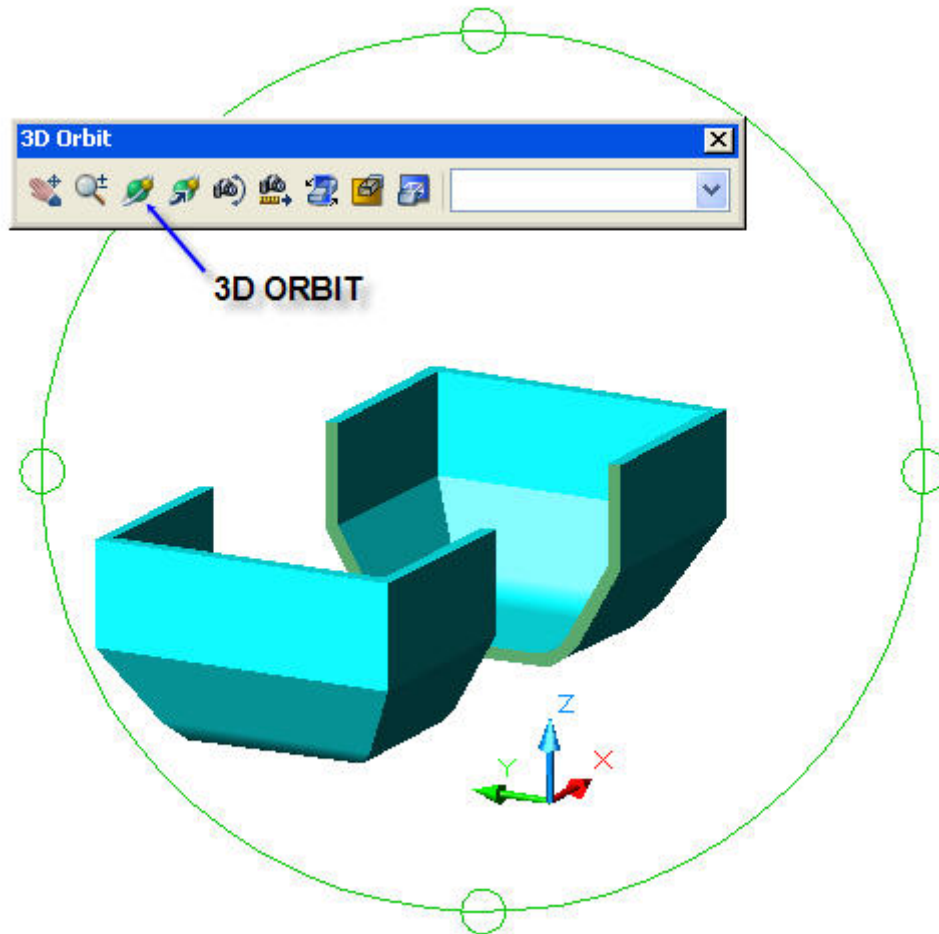
Select the object to be colored.

Pick the **Color Selector** drop-down menu.

SELECT COLOR

Pick the desired **Color** square in color pallet as shown above.

Press the **Space bar** or pick **OK** to end this command.



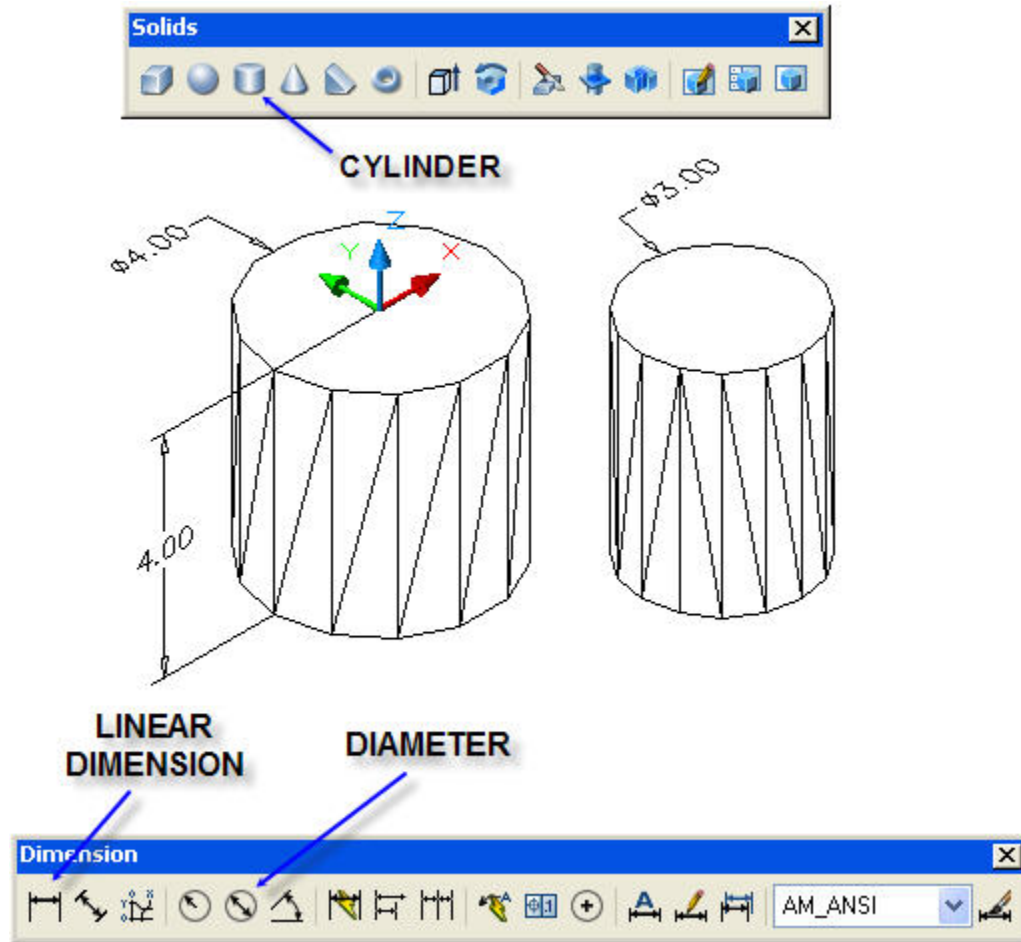
3D ORBIT

Right click on any toolbar > Pick “Orbit” from the list of all toolbars.

Select **3D Orbit** as shown above.

Drag the mouse pointer across the, “orbit circle” until the object has rotated to the desired orientation.

Press the **Space bar** or press **Enter** to end this command.



CYLINDER

Pick the **Cylinder** icon on the **Solids** toolbar.

Pick the location of the top center of the cylinder that as shown above.

Type **D** for Diameter > **4**.

Type **-4** for Height in the **negative Z** direction.

Press the **Space bar** to end this command.

Diameter Dimension

Note: All dimensions are in the X-Y plane.

See the "Add Dimensions" note above.

Pick the **Origin UCS** icon, **X**, **Y**, and **Z** arrows, and place them on top of the cylinder.

Place the **X-Y** plane on the top of the cylinder by the **3-Point UCS** method described under the Dimension topic described above.

Pick the **Diameter** icon as shown above.

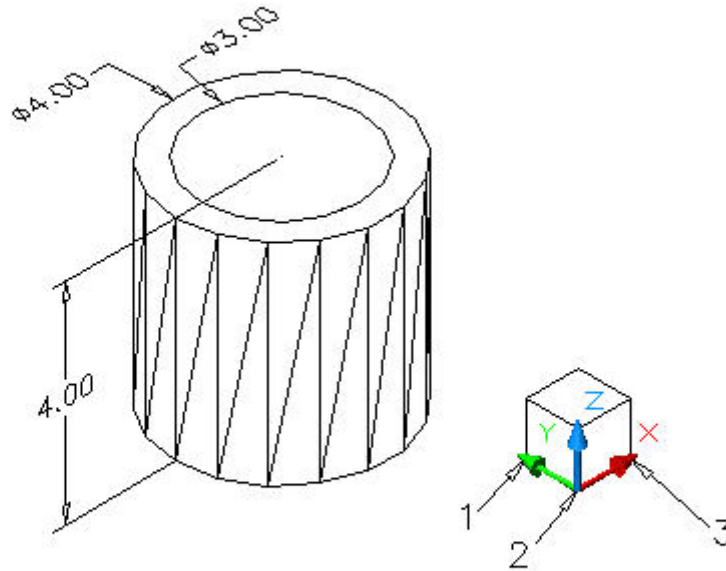
Pick any point on the edge of the circular top of the cylinder.

The 4.00 inch diameter dimension is now placed.

Height Dimension

Repeat the Dimension steps above to place the 4.00 inch height.

Press the **Space bar**.



MOVE THE 3.00 DIA CYLINDER TO THE CENTER OF THE 4.00 DIA CYLINDER

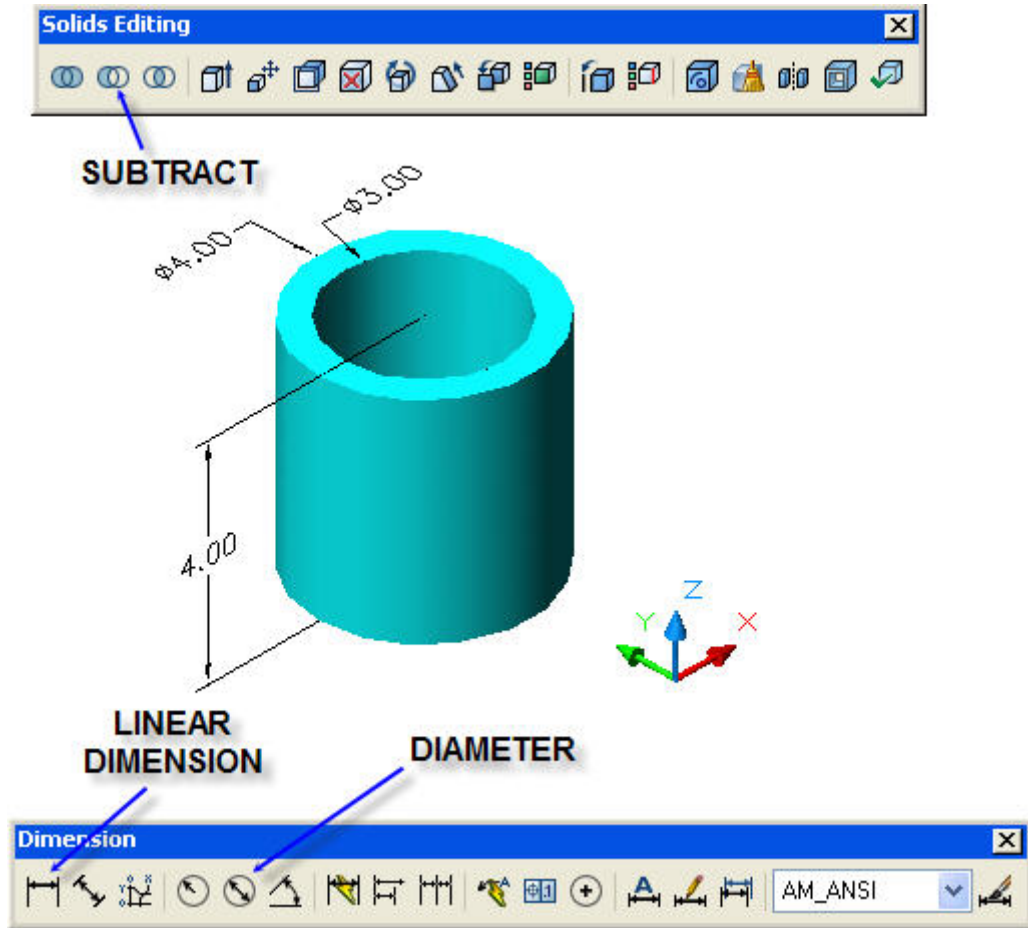
Type **M** for Move.

Select the 3.00 dia cylinder.

Press the **Spacebar**.

Pick the center of the 3.00 dia cylinder.

Drag the 3.00 dia cylinder to the center of the 4.00 dia cylinder as shown above.



SUBTRACT INNER CYLINDER FROM OUTER

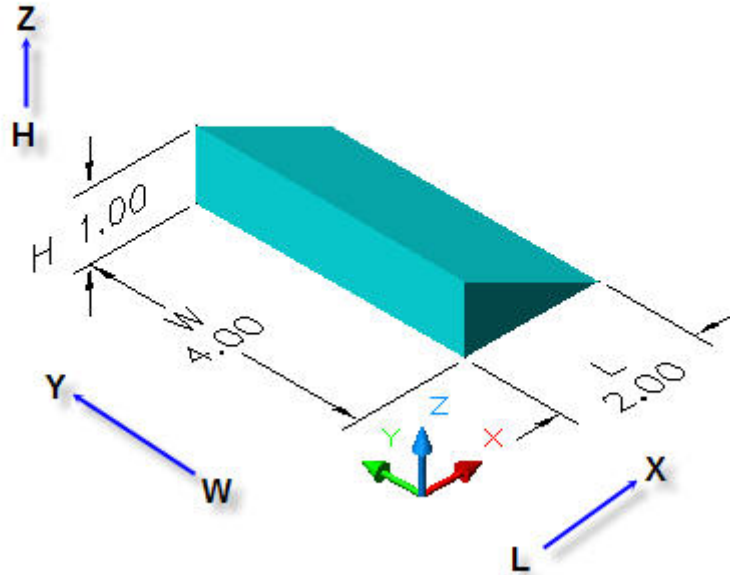
Type: **SU** for subtract on the command line.

Pick the outer cylinder.

Press the **Space bar**.

Pick the inner cylinder.

Press the **Space bar** to end this command.



CYLINDER

Pick the **Wedge** icon on the **Solids** toolbar.

Pick the location of the bottom corner of the wedge as shown above.

Type **L** for Length in the **X direction** > **2**.

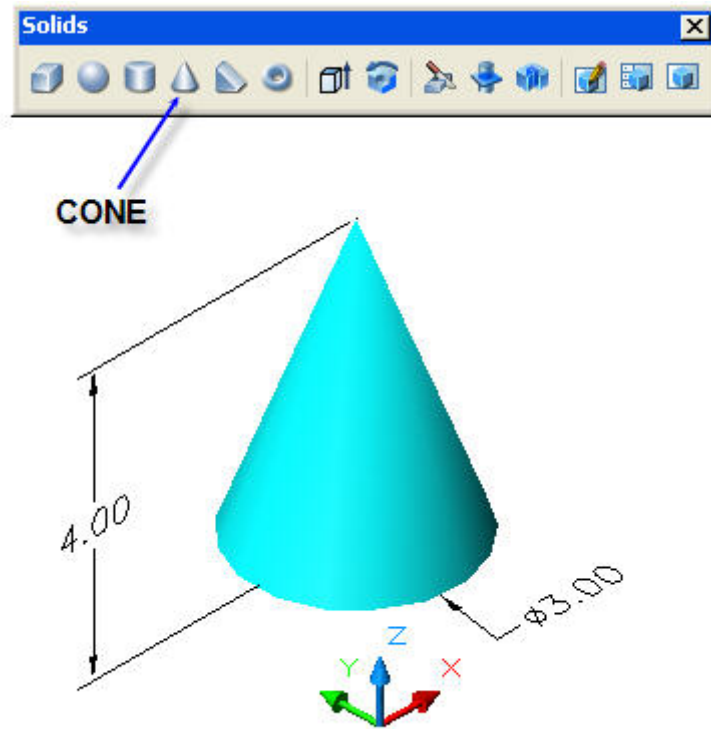
Type **4** for Width in the **Y** direction.

Type **1** for Height in the **Z** direction.

Press the **Space bar** to end this command.

Dimensions

See the "Add Dimensions" note for cylinders above.



CONE

Pick the **Cone** icon on the **Solids** toolbar.

Pick the location of the bottom center of the cone as shown above.

Type **D** for Diameter > **3**.

Type **4** for Height in the **Z** direction.

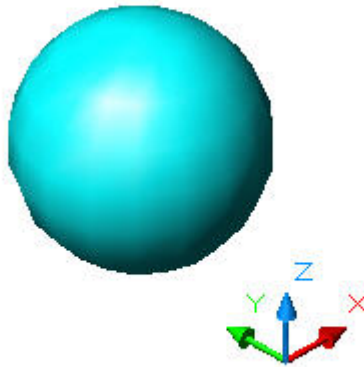
Press the **Space bar** to end this command.

Dimensions

See the "Add Dimensions" note for cylinders above.



SPHERE



SPHERE

Pick the **Sphere** icon on the **Solids** toolbar.

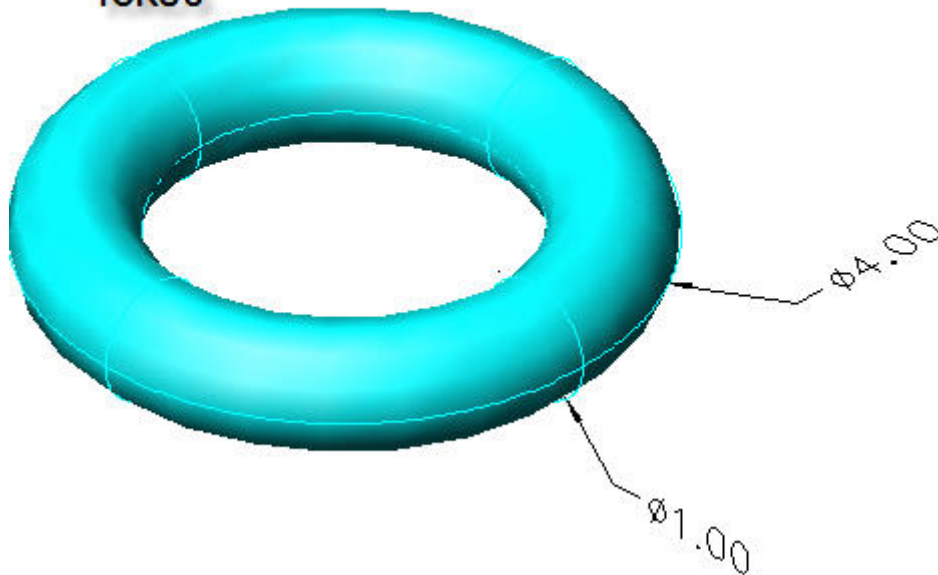
Pick the location of the center of the sphere as shown above.

Type **D** for Diameter > **3** (no dimensions yet).

Press the **Space bar** to end this command.



TORUS



TORUS

Pick the **Taurus** icon on the **Solids** toolbar.

Pick the location of the center of the tours as shown above.

Type **D** for Tours Diameter > **4**.

Type **D** for Tube Diameter > **1**.

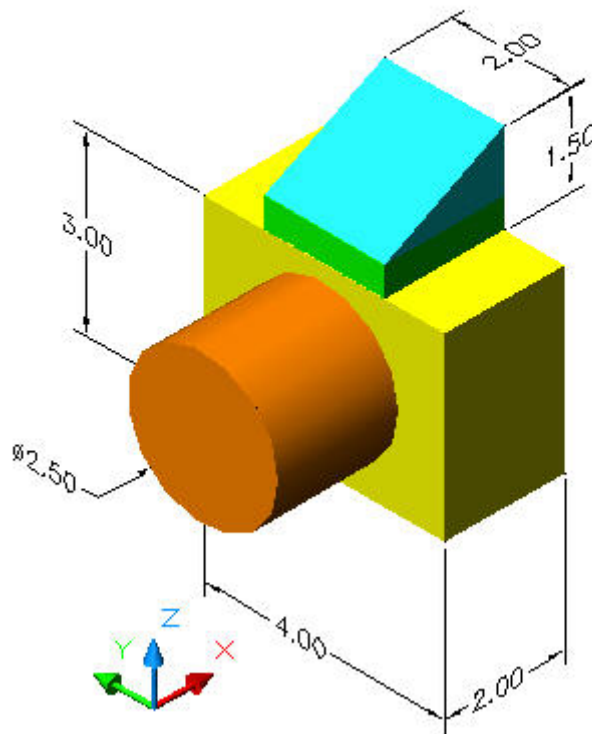
Dimensions

See, "ADD DIMENSIONS" page 28. Rev. 15Mar08

All dimensions are in the X-Y plane.

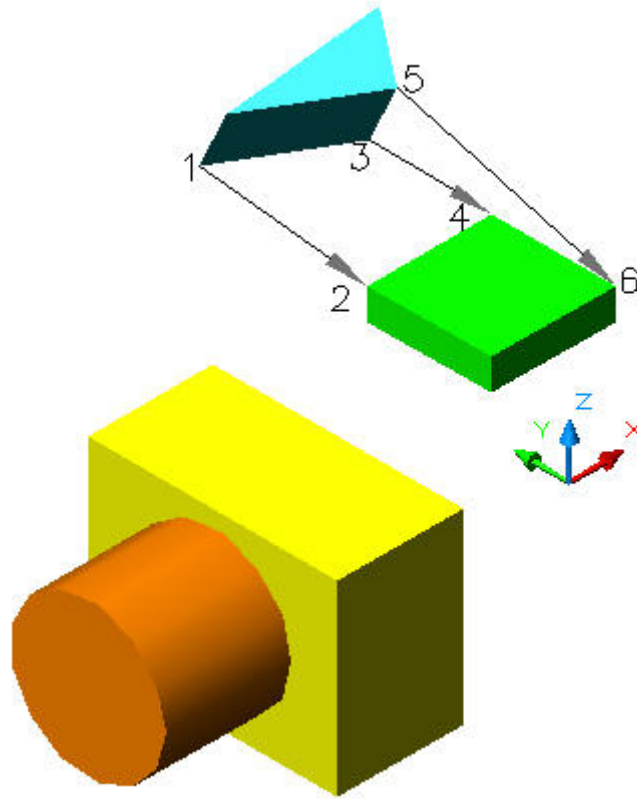
Pick the, "Wire-frame" icon when dimensioning.

Pick, "Hidden" icon on the Visual Styles toolbar and move dimensions until visible.



COMBINING SOLIDS

The Camera made from a cylinder, two boxes, and wedge have been **Aligned** and **Moved** as described below.



ALIGN

Type **AL** for Align on the command line.

Press the **Space bar**.

Select the objects: Wedge and Box shown above.

Pick first source point **1**.

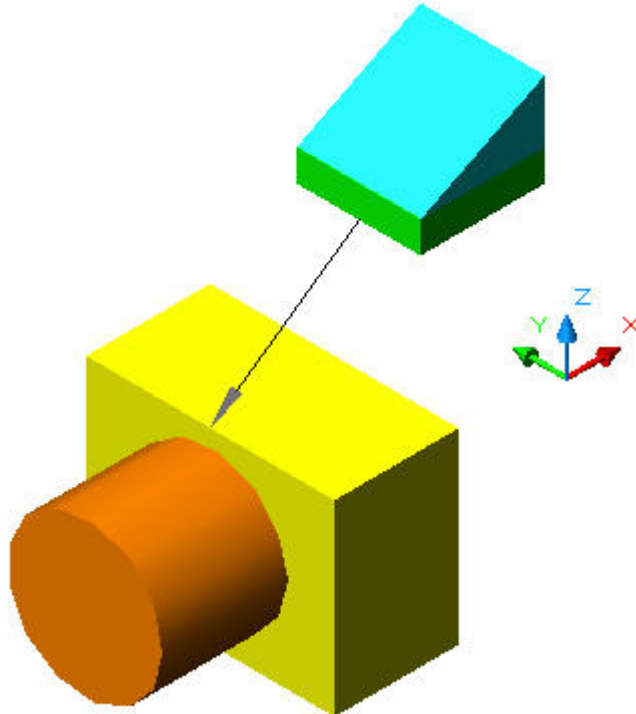
Pick first destination point **2**.

Pick second source point **3**.

Pick second destination point **4**.

Pick third source point **5**.

Pick third destination point **6**.



MOVE

The wedge and block are moved to the top center of the camera.

Type **M** for Move on the command line.

Press the **Space bar**.

Select the wedge and block.

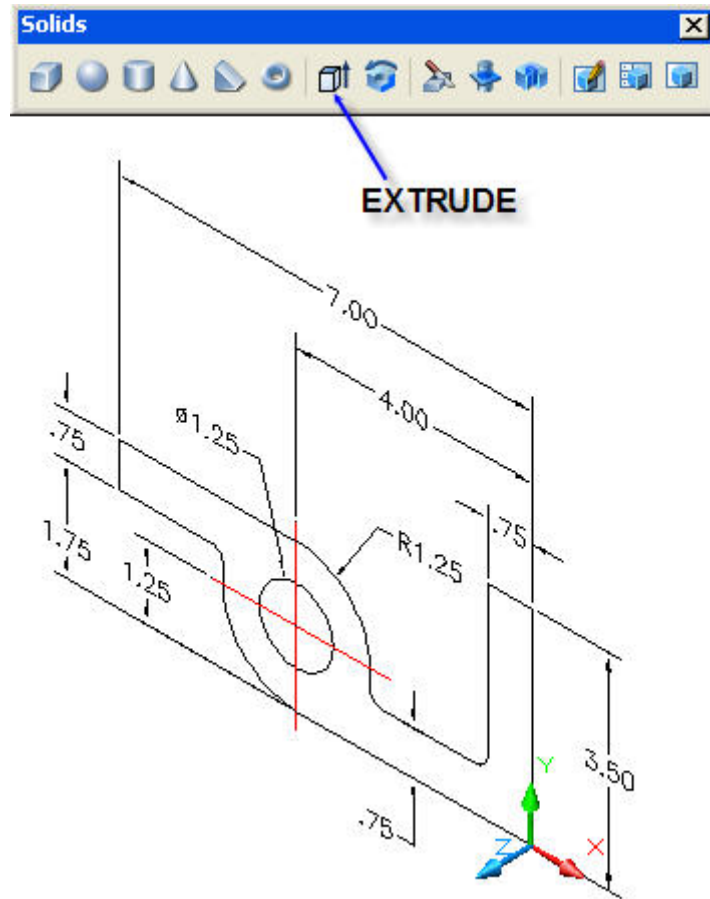
Press the **Space bar**.

Pick the **Mid-point** of the upper block as shown above.

Pick the **Mid-point** of the top center of the camera.

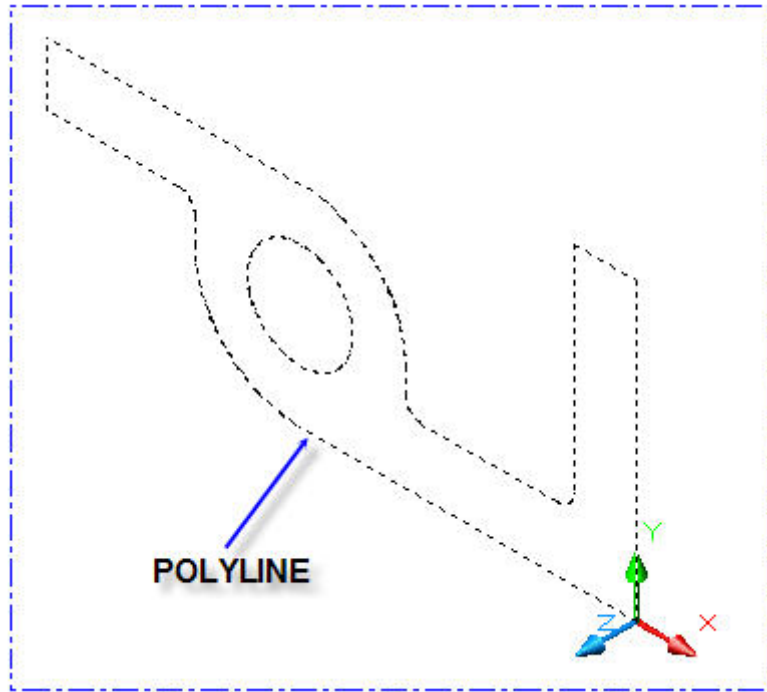
6. EXTRUSIONS

Commands for making 3D solid model extrusions are described below.



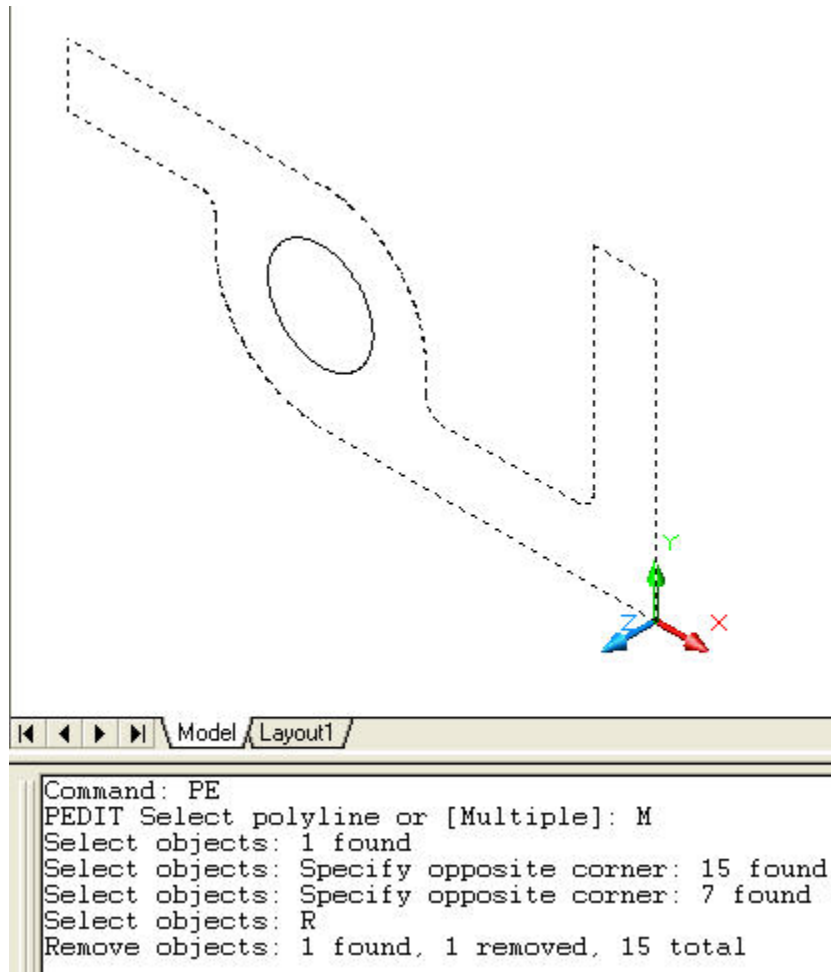
EXTRUDE EXAMPLE

Step-1 The part shape is drawn in the **X-Y** plane as illustrated above.



POLYLINE

Step-2 The outline is converted into one continuous line called a Polyline.
See below.



Step-3 Type **PE** for Polyline Edit as shown above.

Type **M** for multiple lines.

Select objects: 1 found. (Select one line at a time)

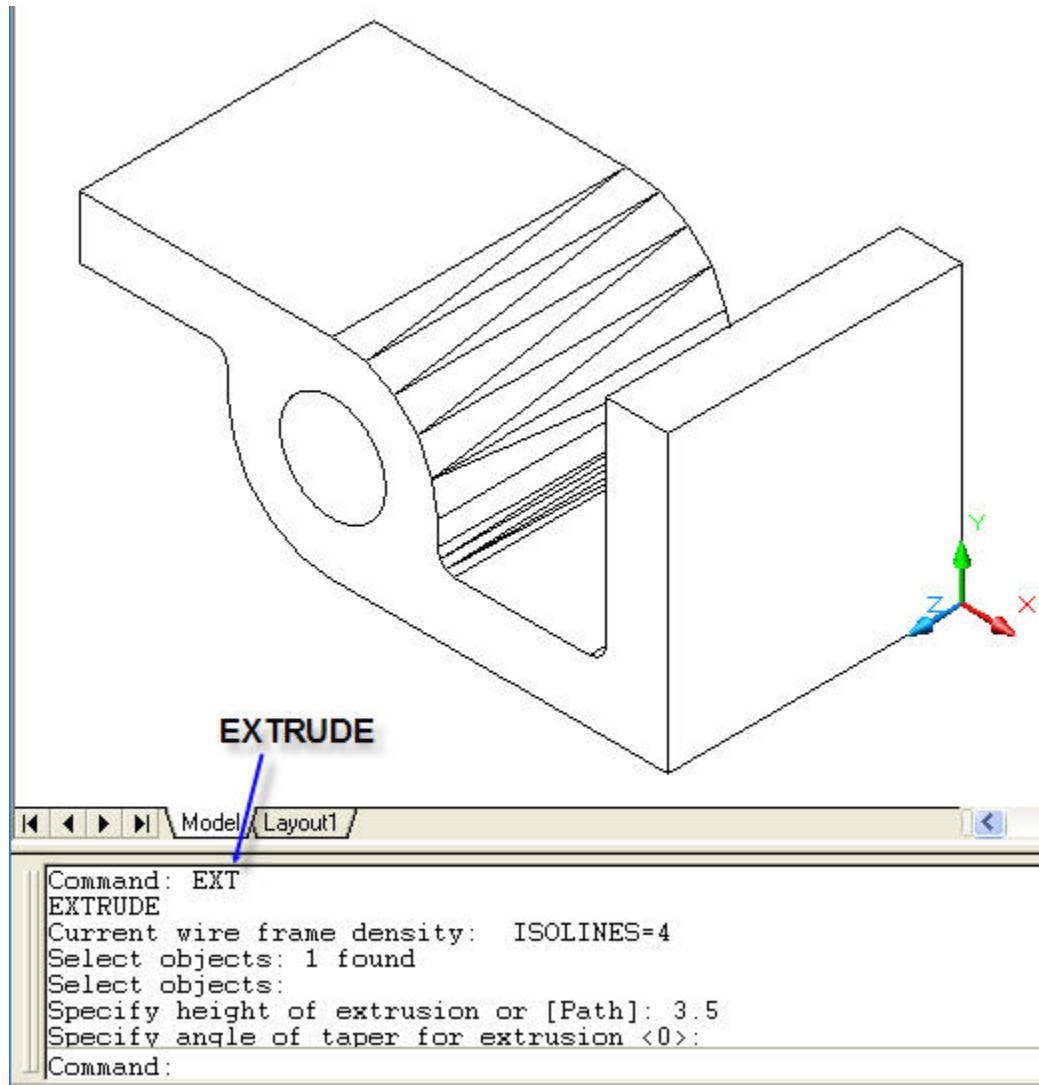
Select objects: Pick upper right corner. Specify opposite corner.

Select objects: Pick lower left corner. 15 found.

Select objects: Type **R** for Remove.

Pick the 1.25 diameter circle.

Remove objects: 1 found, 1 removed, 15 total.



EXTRUDE

Step-4 Type **EXT** for Extrude on the command line.

Press the **Space bar**.

Pick any point on the continuous outline of the part and the command line will read:

Select objects: 1 found, as shown above.

Press the **Space bar**.

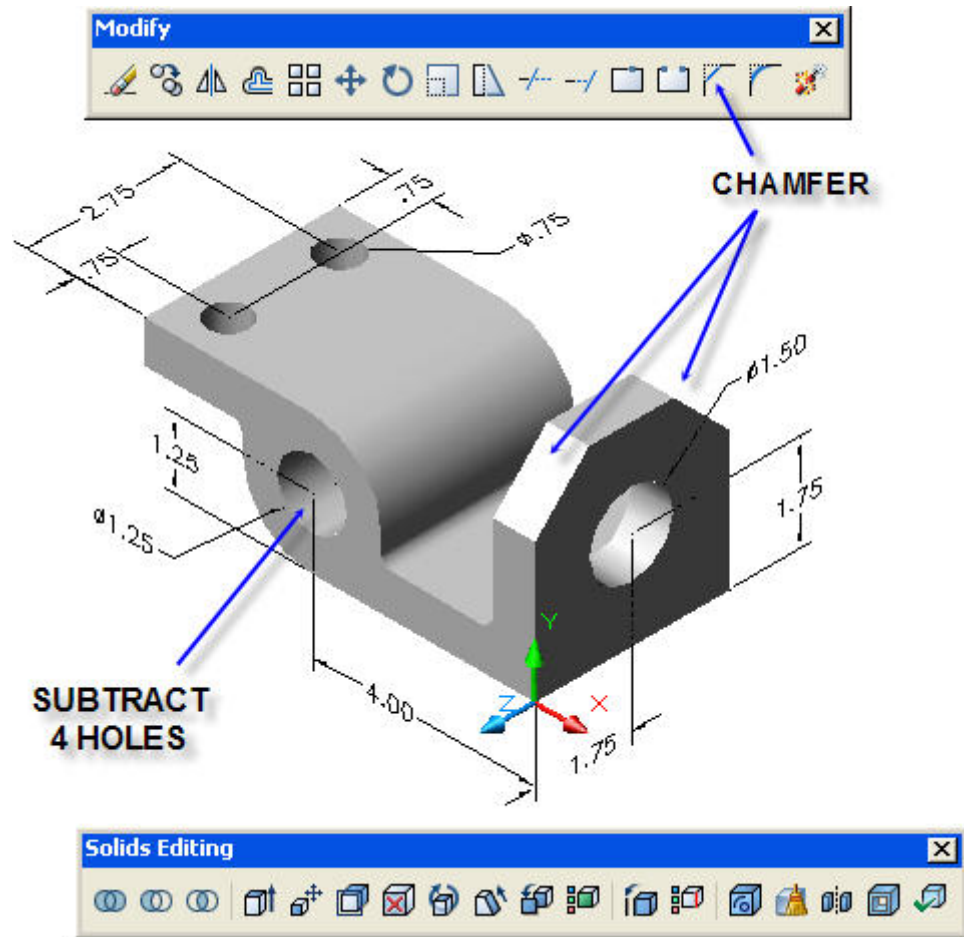
Specify height of extrusion: Type **3.5** in the **Z** direction.

Press the **Space bar**.

Specify angle of taper for extrusion <0>:

Press **Space bar** to accept the default value of **0** taper.

Press the **Space bar** to complete this command.



CHAMFER

Step-5 Type **CHA** for Chamfer on the command line or pick the **Chamfer** icon on the Modify toolbar shown above.

Select first line: Pick the top right corner edge.

Base surface selection(will appear on the command line)

Enter surface selection option <OK>

Press **Space bar** to accept the default **OK**.

Specify base surface chamfer distance: Type **.75**.

Press **Space bar**.

Specify other surface chamfer distance: Type **.75**.

Select the top right corner edge.

Press **Space bar**. The top right corner is now chamfered as shown above.

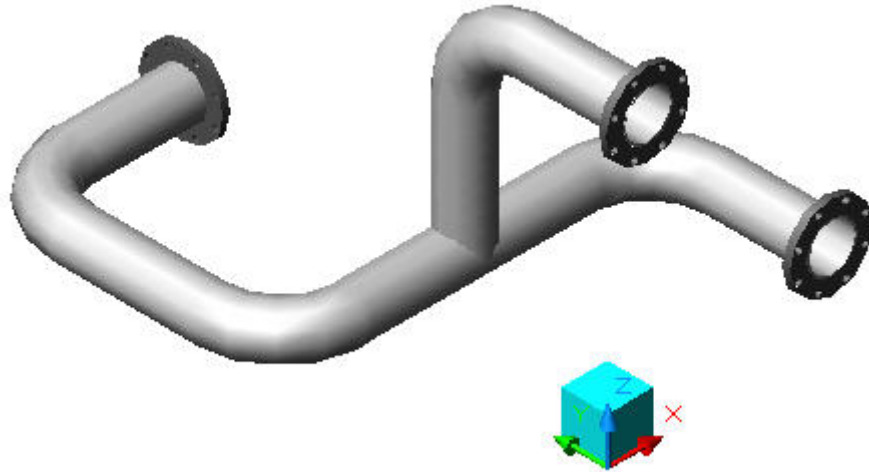
Repeat the **CHA** command for the top left corner.

SUBTRACT 4 HOLES

Step-6 Draw circles at the hole locations shown above.

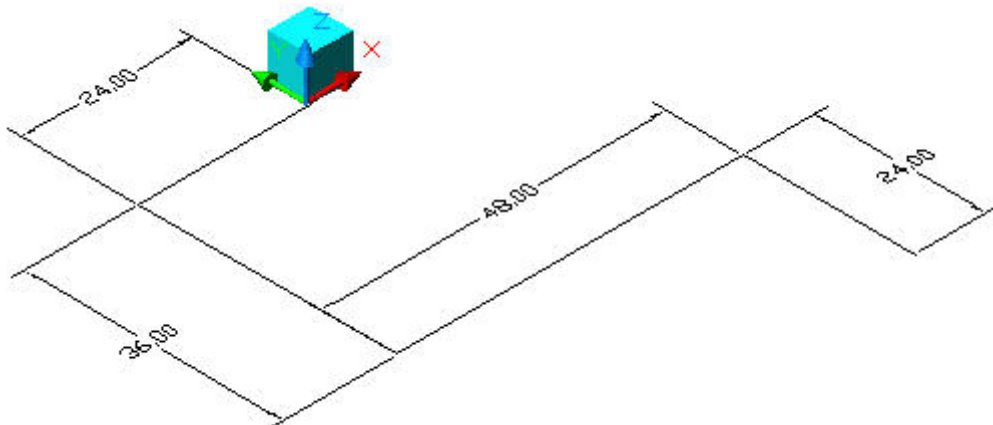
Extrude each hole circle thru the part.

Subtract all holes from the part as described in the above, **“Subtract Inner Cylinder From Outer”** note.



EXTRUDE PATH

The 3 dimensional solid model of the process pipe run above was created with the **Extrude Path** command group.



PATH

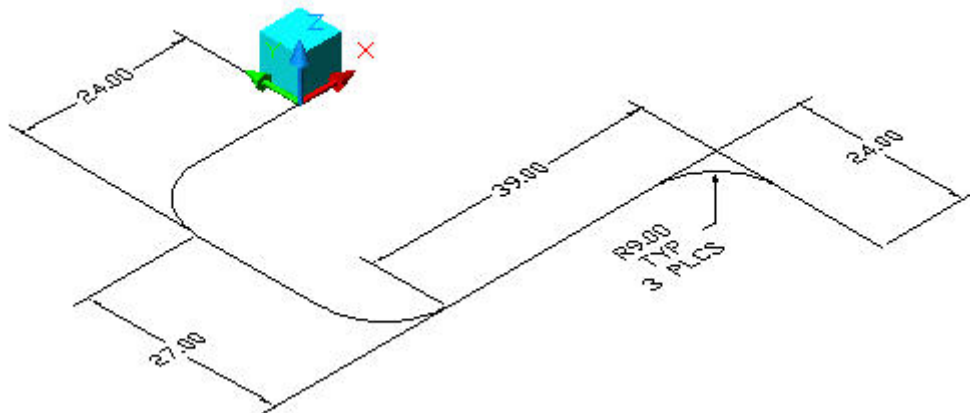
Step-1 Draw the pipe run path.

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

Press the **Space bar**.

Pick the bottom corner of the cube as the start of the 24.00 inch line shown above.

Toggle the **f8** key to turn **Ortho ON**.
Drag the mouse pointer in the negative **X** direction.
Type the length of the line, **24**.
Press the **Space bar**.
Drag the mouse pointer in the negative **Y** direction.
Type the length of the line, **36**.
Press the **Space bar**.
Drag the mouse pointer in the positive **X** direction.
Type the length of the line, **48**.
Press the **Space bar**.
Drag the mouse pointer in the negative **Y** direction.
Type the length of the line, **24**.
Press the **Space bar**.



FILLET

Step-2 Round the corners with fillets.

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

Press the **Space bar**.

Type **R** for Radius on the command line.

Press the **Space bar**.

Type **9**.

Press the **Space bar**.

Pick one intersecting line and then the next forming a fillet as shown above.

Add fillets at the remaining corners.

Step-3 Convert the pipe lines to one continuous Polyline

Type **PE** for Polyline Edit as shown above.

Press **Space bar**.

Type **M** for multiple lines.

Press **Space bar**.

Select objects: 1 found. (Select one line at a time)

Select objects: Pick upper right corner.

Specify opposite corner. (Select more than one line at a time)

Press **Space bar**.

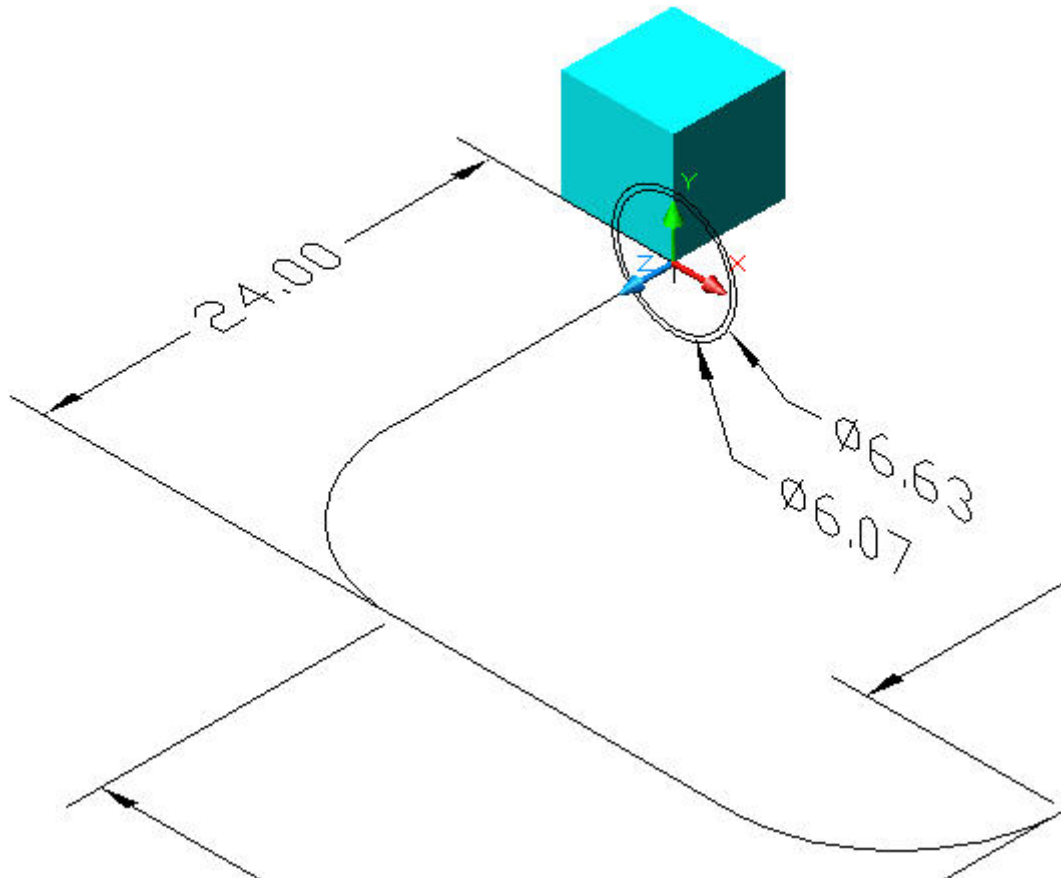
Press **Space bar** to accept default <Y> (Convert to polylines)

Type **J** for Join.

Press **Space bar**.

Press **Space bar** to accept default **Zero fuzz distance**.

Press **Space bar** to end this command.



CIRCLES

Extrusions are in the Z direction.

Step-4 Draw the pipe section.

Set the **X**, **Y**, and **Z** axes using the **UCS 3-Point** commands as shown above.

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

Press the **Space bar**.

Pick the location of the center point.

Press the **Space bar**.

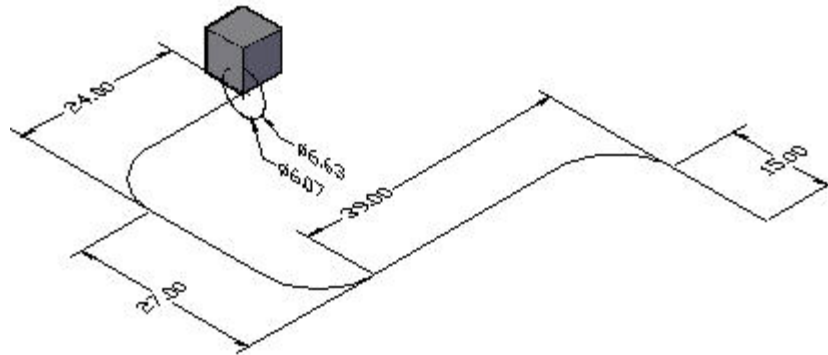
Type **D** for Diameter on the command line.

Press the **Space bar**.

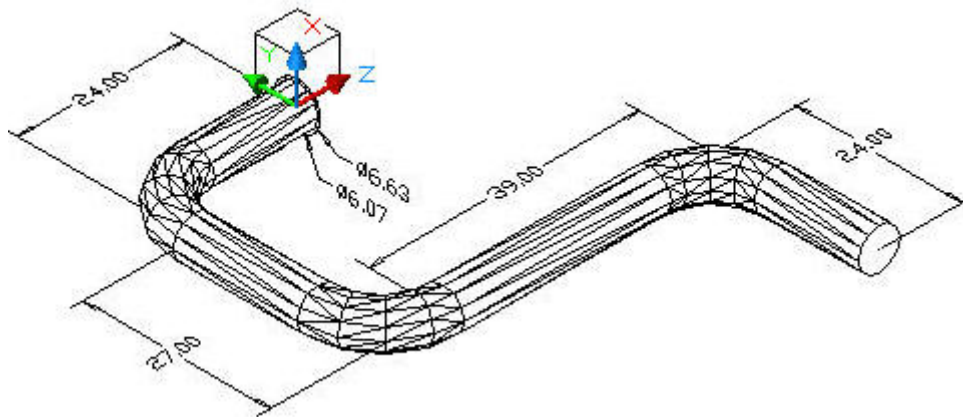
Type **6.07**.

Press the **Space bar** to repeat the Circle command.

Repeat the steps above to draw the **6.63** diameter circle.



EXTRUDE PATH CENTERLINE



EXTRUDE PATH – INSIDE PIPE

Step-5 Extrude the 6.07 inch diameter inside pipe.

Type **EXT** for Extrude on the command line.

Press the **Space bar**.

Pick any point on the continuous polyline pipe centerline.

Press the **Space bar**.

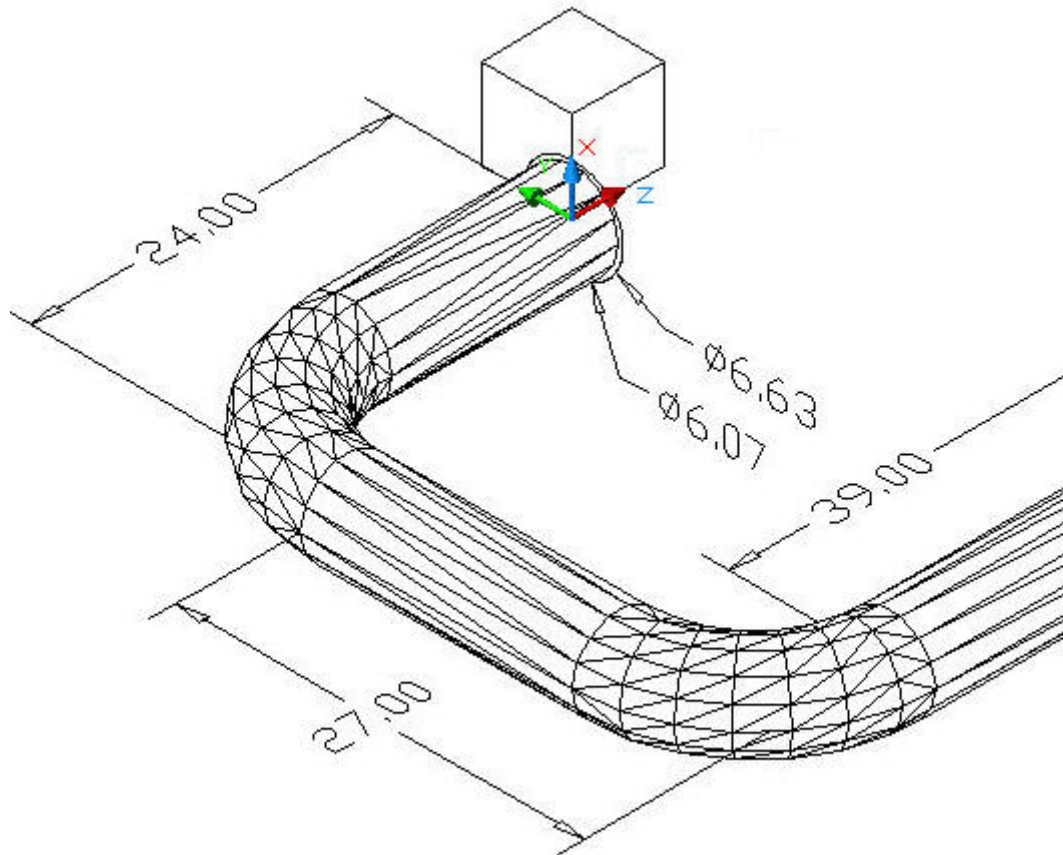
Type **P** for Path of the extrusion in the **Z** direction.

Press the **Space bar**.

Specify the Path Specify angle of taper for extrusion <0>:

Press **Space bar** to accept the default value of **0** taper.

Press the **Space bar** to end this command.



EXTRUDE PATH – OUTSIDE PIPE

Step-6 Extrude the 6.63 inch diameter outside pipe.

Type **EXT** for Extrude on the command line.

Press the **Space bar**.

Pick any point on the continuous polyline pipe centerline.

Press the **Space bar**.

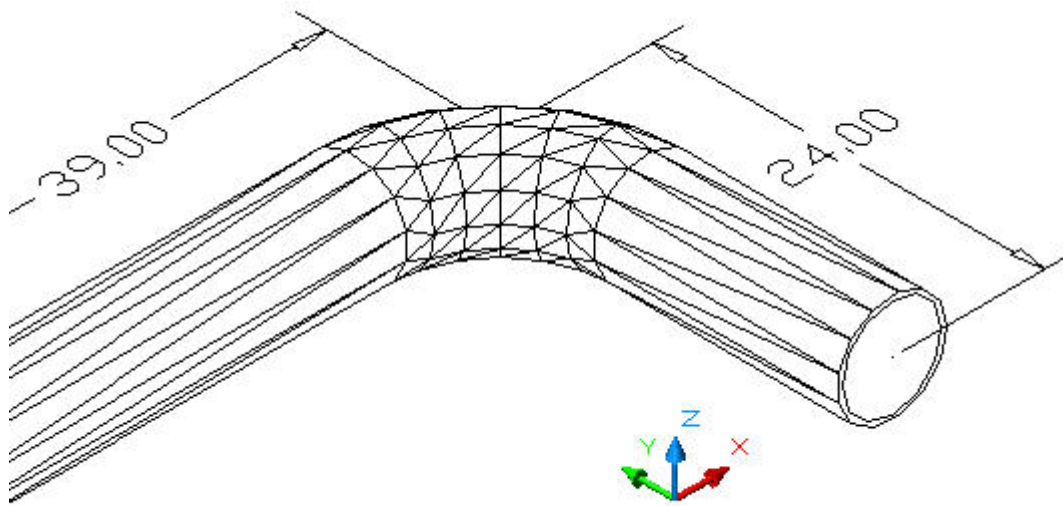
Type **P** for Path of the extrusion in the **Z** direction.

Press the **Space bar**.

Specify the Path Specify angle of taper for extrusion <0>:

Press **Space bar** to accept the default value of **0** taper.

Press the **Space bar** to end this command.



Step-7 Subtract the inner pipe from the outer pipe.

Type **SU** for Subtract on the command line.

Press **Space bar**.

Pick the outer pipe.

Press **Space bar**.

Pick the inner pipe.

Press **Space bar**.

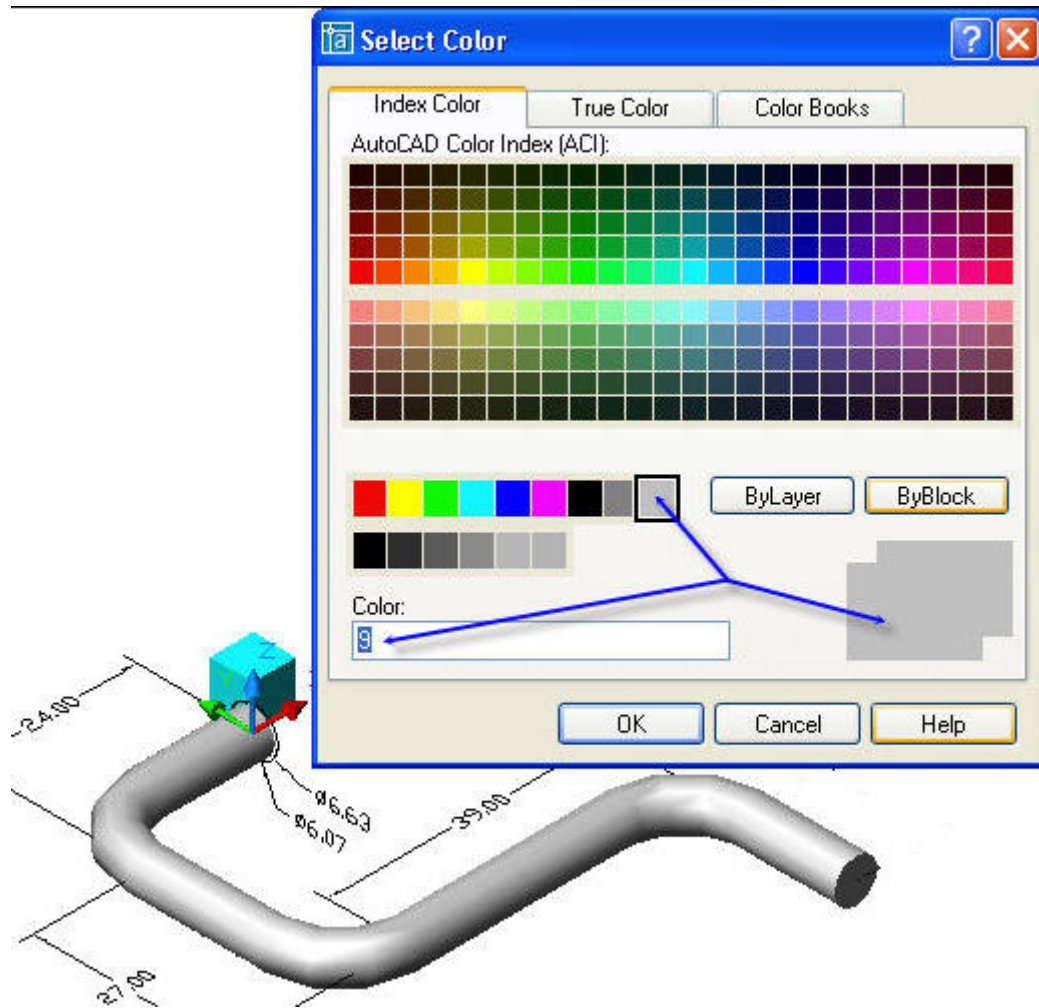


**COLOR
MENU**

Step-8 Select the color drop down menu.

Pick **Select Color ...** at the bottom of the color menu.

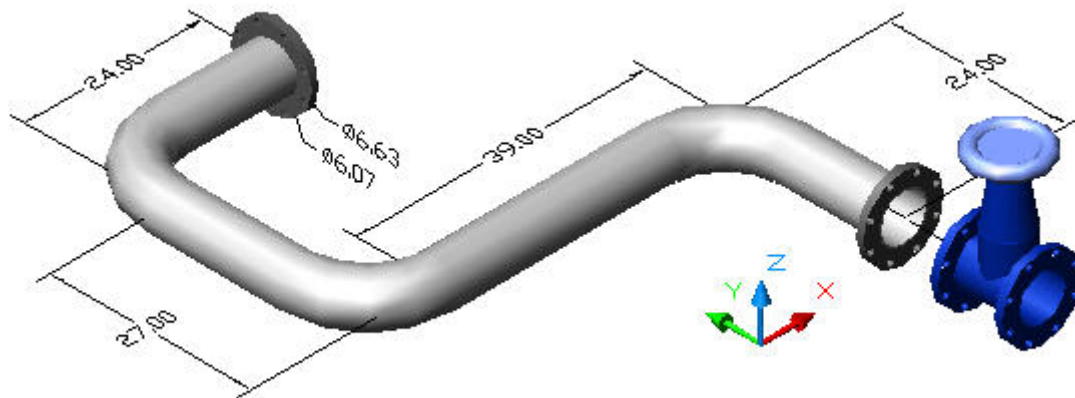
The **Select Color** dialog box will open as seen below.



Step-9 Apply color to the pipe.

Change the color of the pipe to Grey #9.

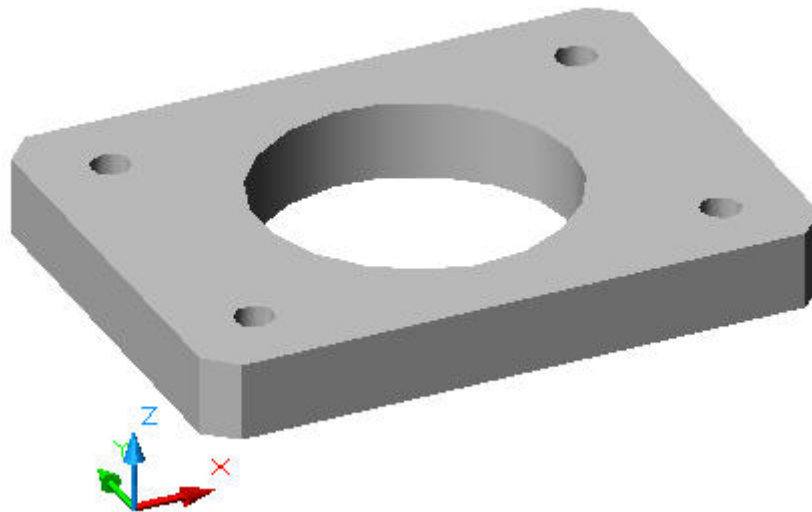
Press the **OK** button.



Step 10 Add flanges and a valve to the pipe run.

Create the flanges by following the steps below for creating the "Generic Hub".

Create the valve by combining: cylinders, a cone, and the Revolved handle formed when making the Generic Hub below.

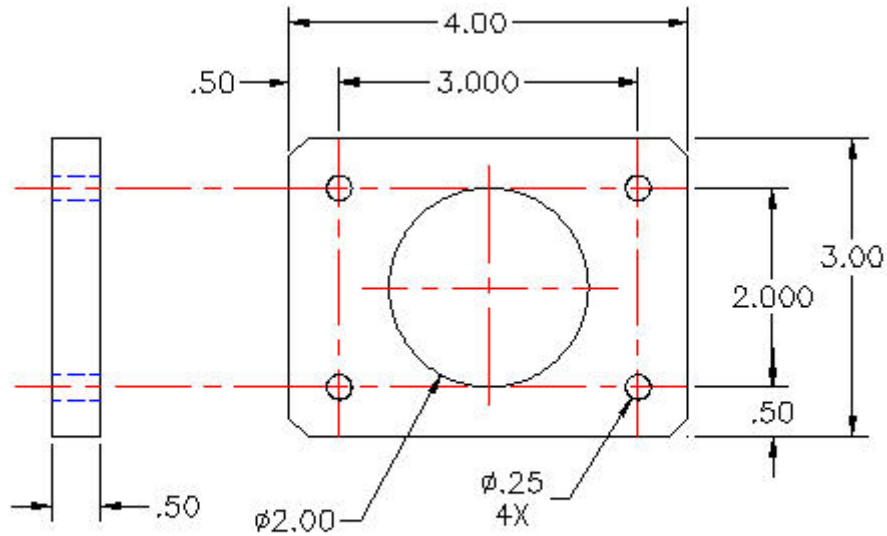


8. 2D DRAWING USED TO CREATE A 3D SOLID

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

EXAMPLE-1

This flat plate with holes has been chosen to demonstrate some of the most common AutoCAD commands used to make many types of 2D and 3D engineering drawings.



PART M2297 REV B

The 2D Drawing above will be used to create a 3D solid model

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



Step-1 Draw Line A-B

Open any version of AutoCAD from release 14 to 2007.

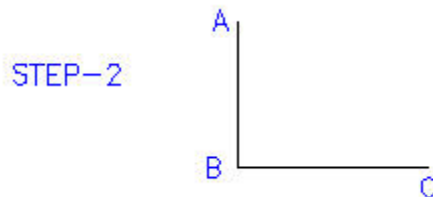
Type the command: **L** for Line.

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 **1.5**

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



Step-2 Draw Line B-C

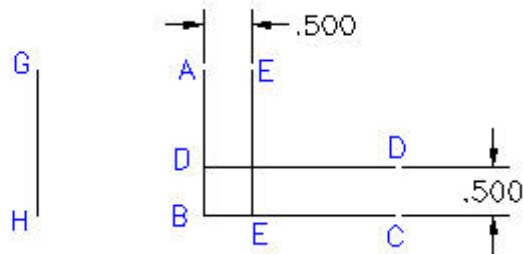
Press the **Spacebar** or **Enter Key** to repeat the last command.

Pick 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



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.

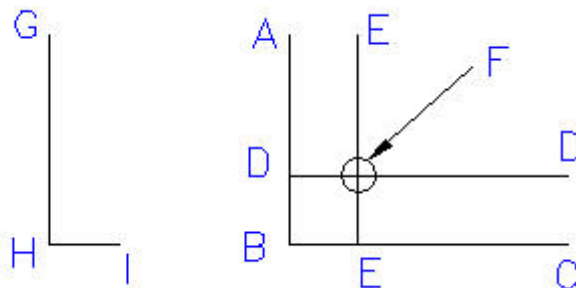
Pick a point above line B-C.

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.

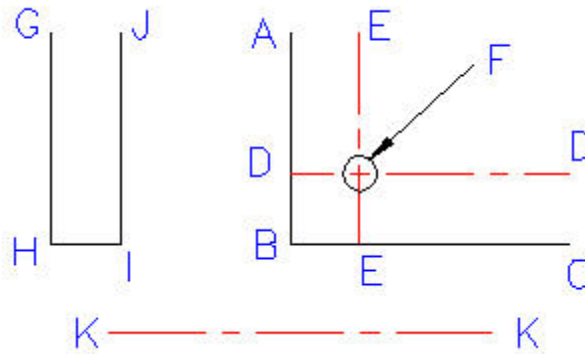
Pick the intersection of lines D-D and E-E.

Type **D** for Diameter.

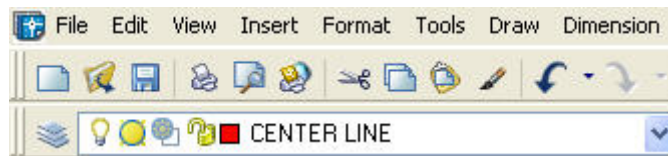
Type **.5**.

Press the **Spacebar**.

STEP-5



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



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

Type **L** on the command Line.

Draw center line K-K any length.

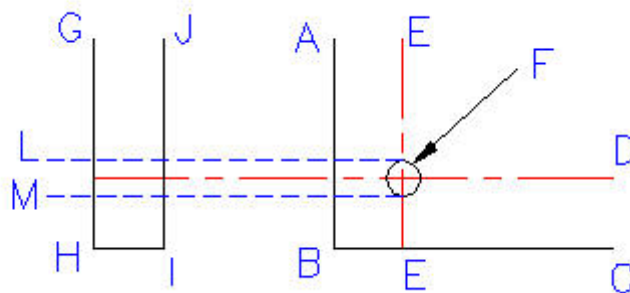
Pick the **“Paint Brush”** icon shown above.

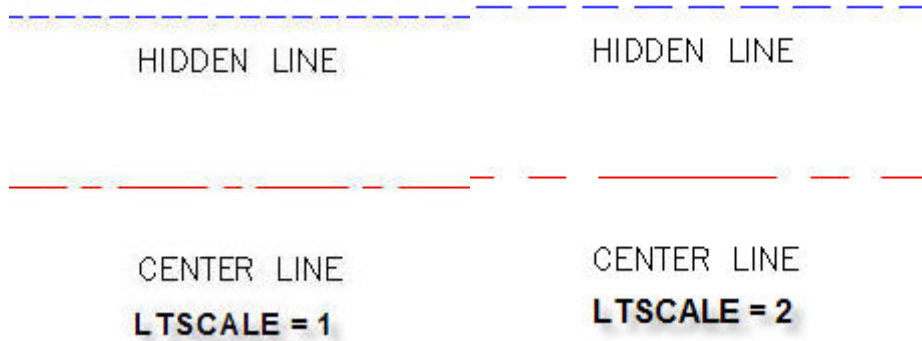
Pick any point on center line K-K.

Pick lines D-D & E-E.

Press the **Spacebar**.

STEP-6





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

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**.

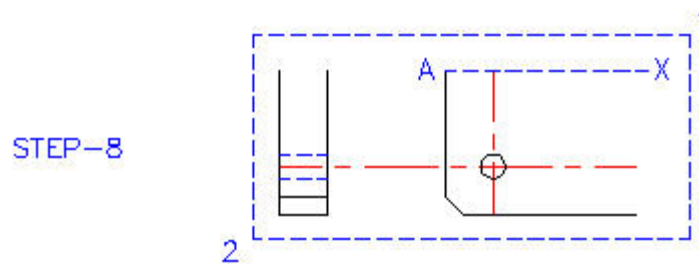
Type, **.188**.

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

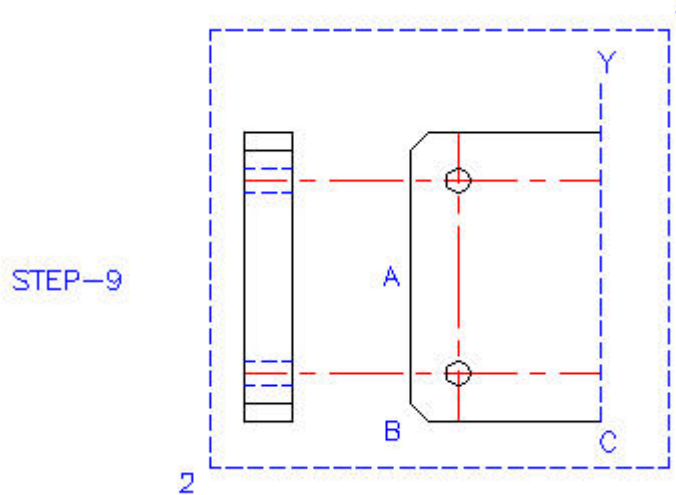
Toggle function key **f8** until you see, **Ortho on**.

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

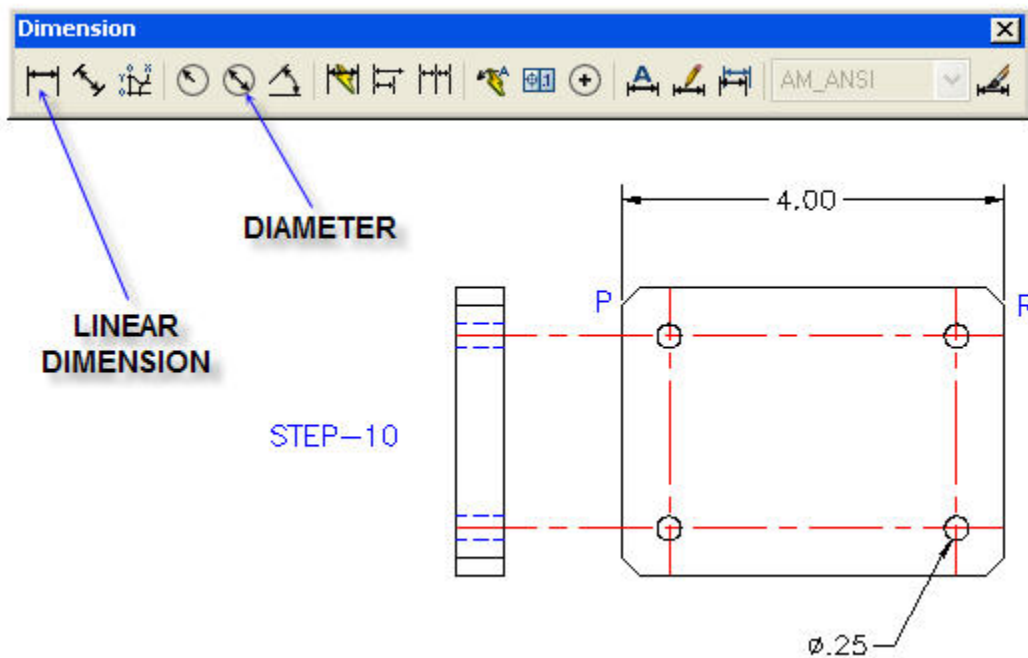
Toggle function key **f8** until you see, **Ortho on**.

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 the 4.00 and .25 diameter Dimensions

Pick the Linear Dimension icon.

Pick point **P**.

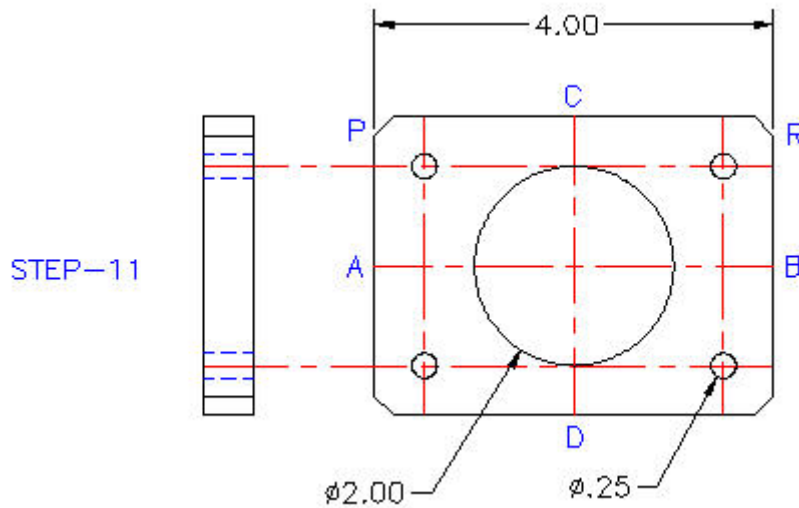
Pick point **R**.

Pick the location of the **4.00** dimension as shown above.

Pick the Diameter icon.

Pick point a point on the **.25** diameter circle.

Press the **Space bar** to end this command.



STEP 11 Add a 2.00 inch Diameter Hole

Type **L** for Line

Pick line mid points **A** and **B**.

Pick line mid points **C** and **D**.

Type **C** for Circle.

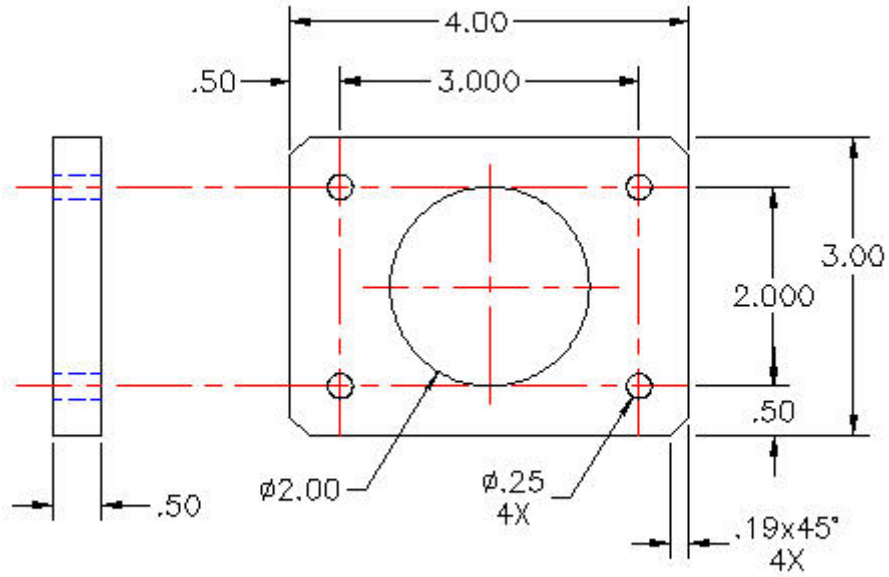
Pick the 2.00 circle center at the intersection of A-B & C-D.

Type **D** for Diameter.

Type **2** for diameter Dimension.

Pick the **Diameter** dimension icon.

Pick the dimension location at **Φ2.00**.



PART M2297 REV B

STEP 12 Add Dimensions to Complete the 2D Drawing

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

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

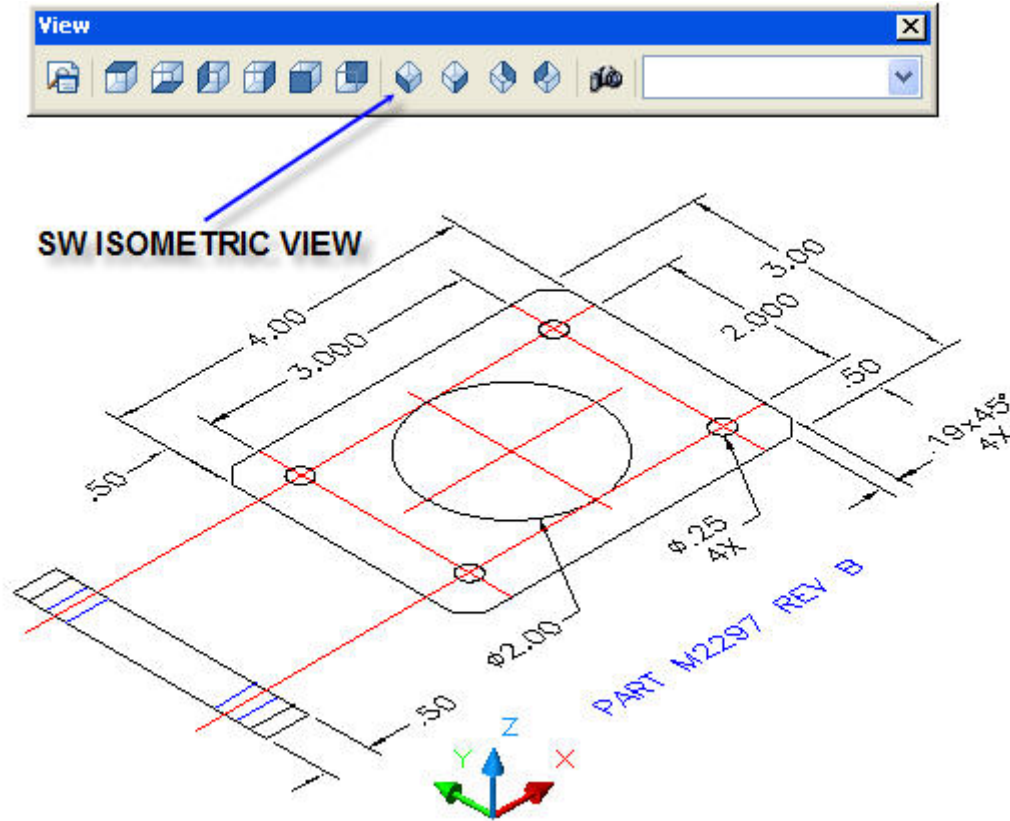
Pick the dimension location at **4.000**.

Pick the **Diameter** icon.

Pick a point **on** the circle.

Pick the dimension location at **φ.250**.

Press the **Spacebar**.



STEP 13 Move From 2D to 3D

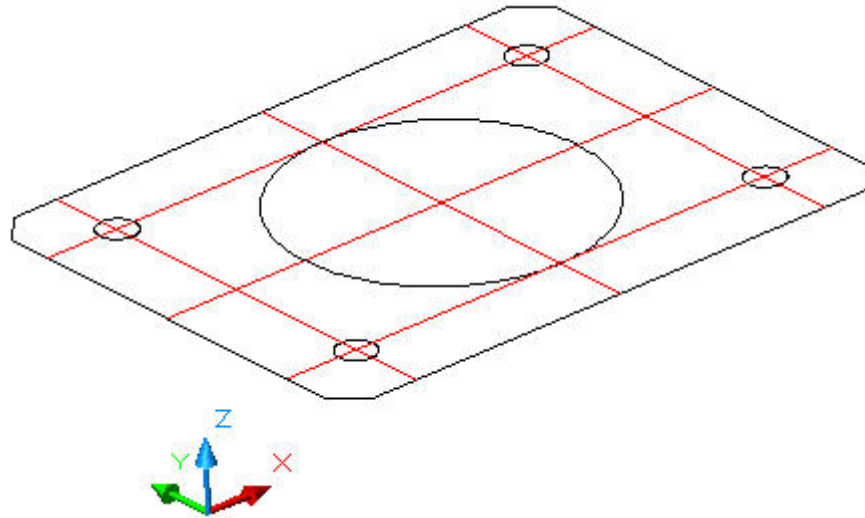
Pick the **SW Isometric View** icon on the **View Toolbar**.

Type, **E** for Erase.

Select each dimension and the side view.

Press the **Spacebar**.

All dimensions and the side view of the plate are erased as shown below.



STEP 14 Convert the Part Outline to a Polyline

Type, **PE** for Polyline Edit.

Type, **M** for Multiple Lines.

Select each line segment forming the part outline.

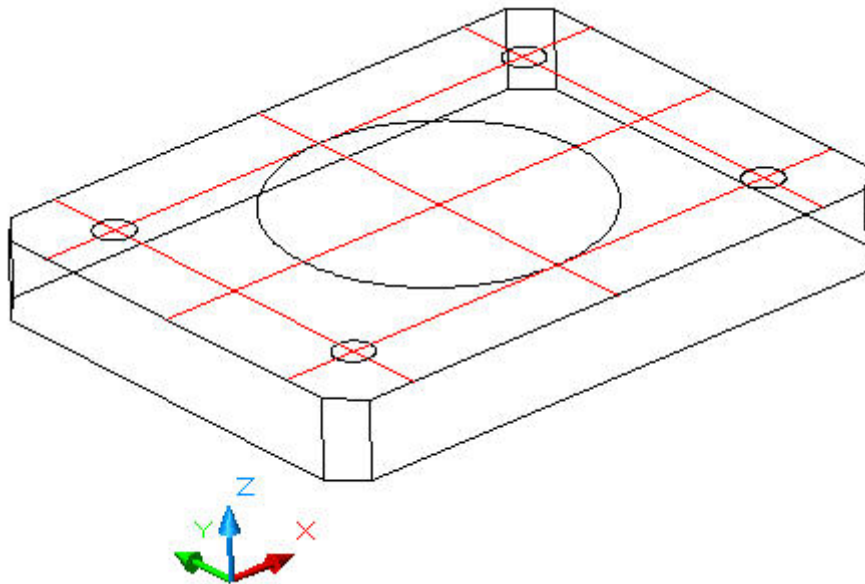
Press the **Spacebar** to accept, convert lines to Polylines.

Type, **J** to Join all of the Polylines.

Press the **Spacebar** again to say Fuzz Distance Zero, which is zero gap between line ends.

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

The part outline is now one continuous line.



STEP 15 Extrude the Part and the 5 Holes

Type, **EXT** for Extrude.

Select any point on the continuous outline of the part.

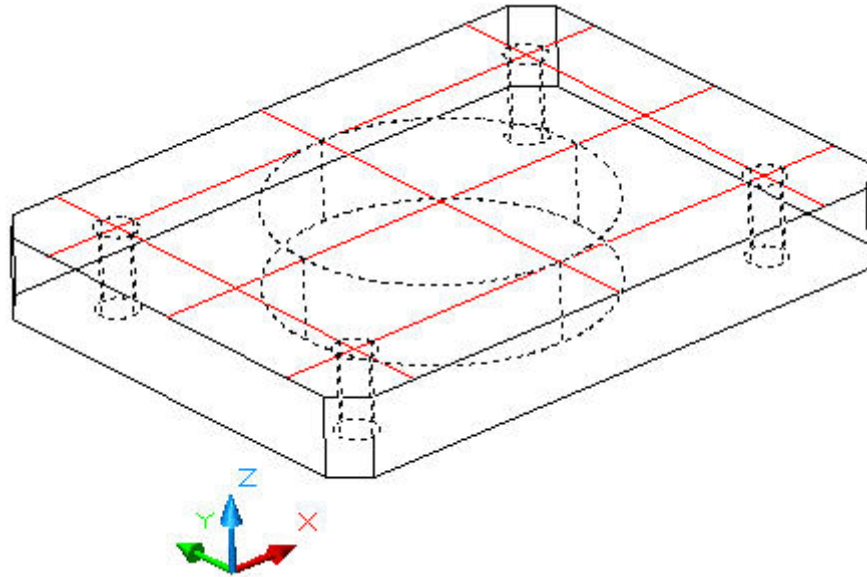
Press the **Spacebar**, indicating that all items have been selected.

Type height of extrusion in the **Z** direction, **-.5** (negative .5 inches).

Press the **Spacebar** to accept the default of zero taper.

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

Repeat the above commands to extrude each of the 5 holes.



STEP 16 Subtract the 5 Holes

Type, **SU** for Subtract.

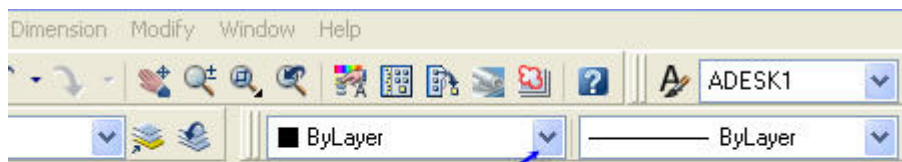
Select any point on the continuous outline of the part.

Press the **Spacebar**, indicating that all items have been selected.

Select any point on each of the 5 holes.

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

The five holes are now subtracted as illustrated below.



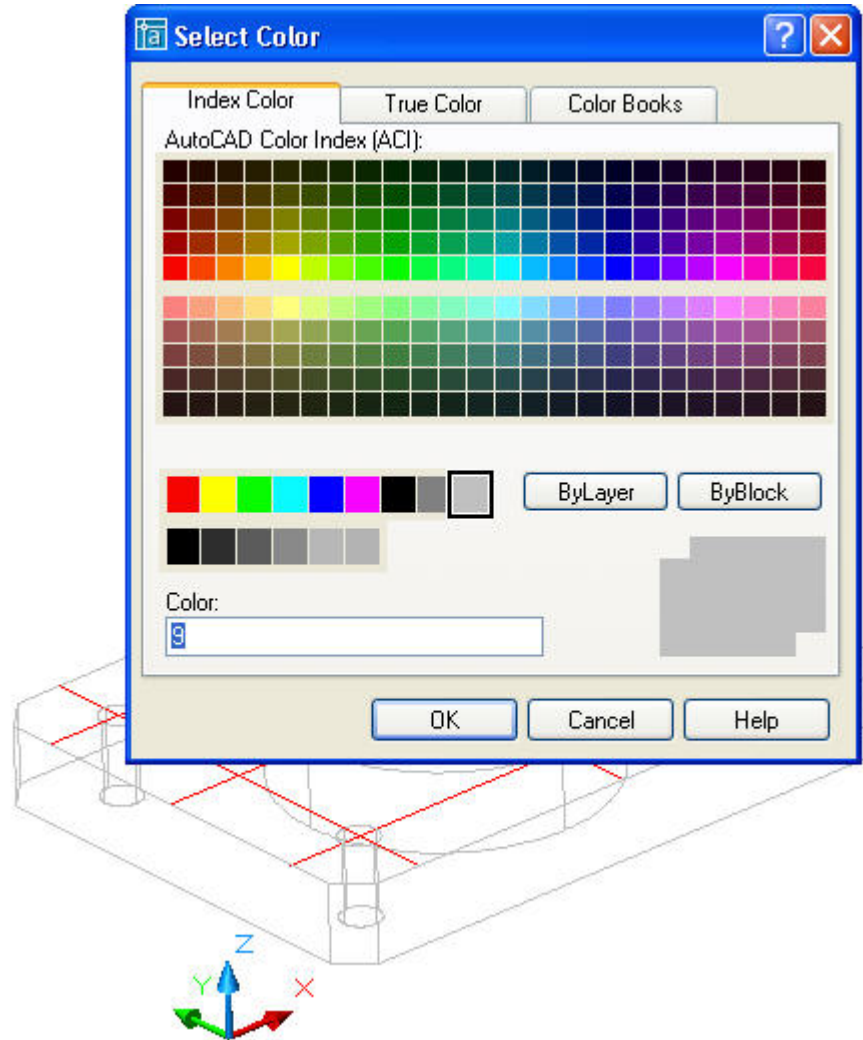
COLOR SELECTION MENU

STEP 17 Change the Color of the Part

Select the flat plate with the mouse pointer.

Select the **Color Selection** drop down menu shown above.

The open **Select Color** menu is pictured below.

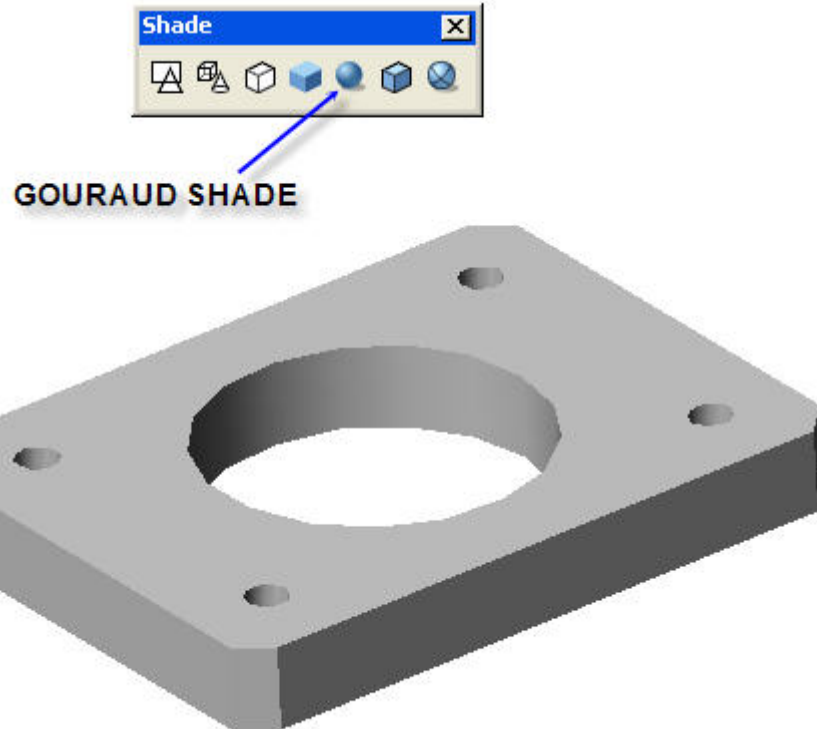


STEP 18 Change the Color to Light Grey

Pick color **9** for light grey.

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

The part is now light grey.

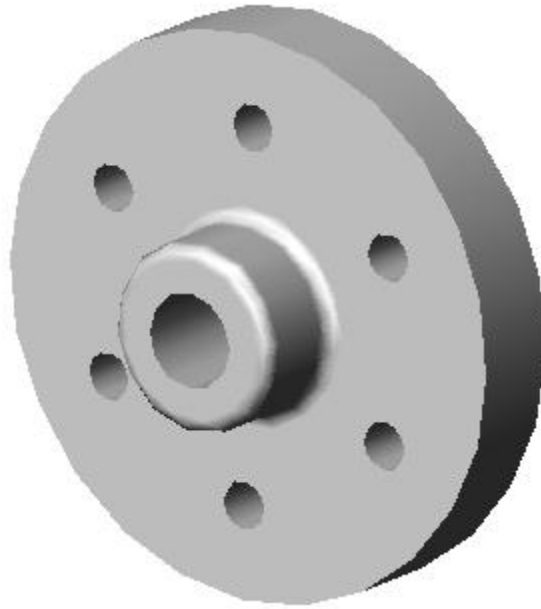


STEP 19 Change the Shade to Gouraud

Pick the Gouraud icon on the Shade Toolbar as illustrated above.
The part is now shaded as shown above.

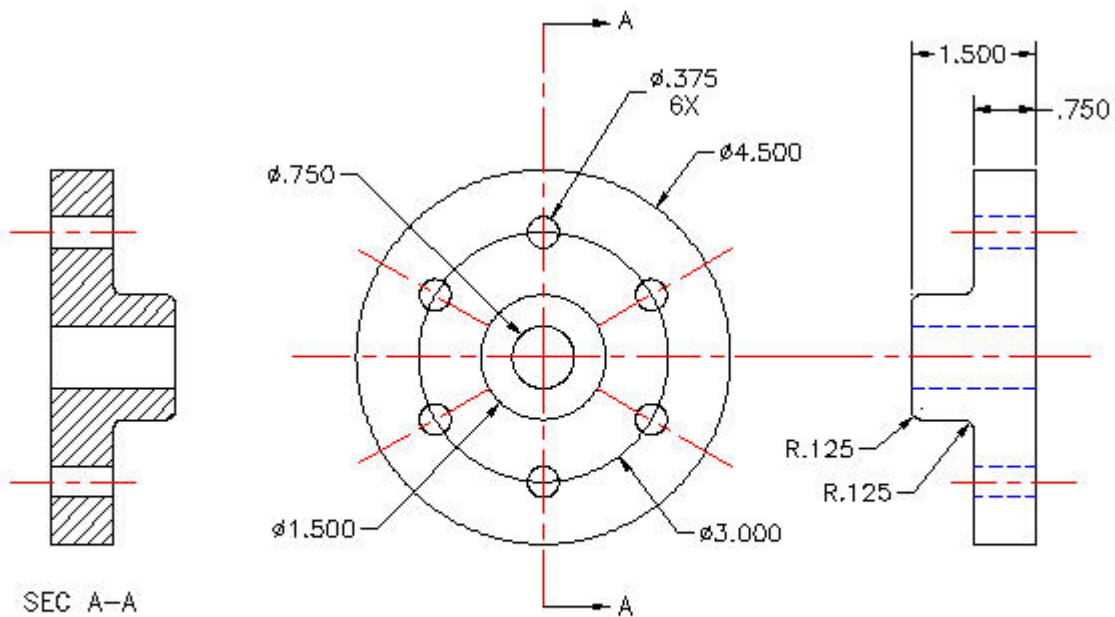
9. GENERIC HUB

The 3-dimensional solid model of the Generic Hub shown below could be drawn with all versions of AutoCAD from release 14 to 2007.



GENERIC HUB

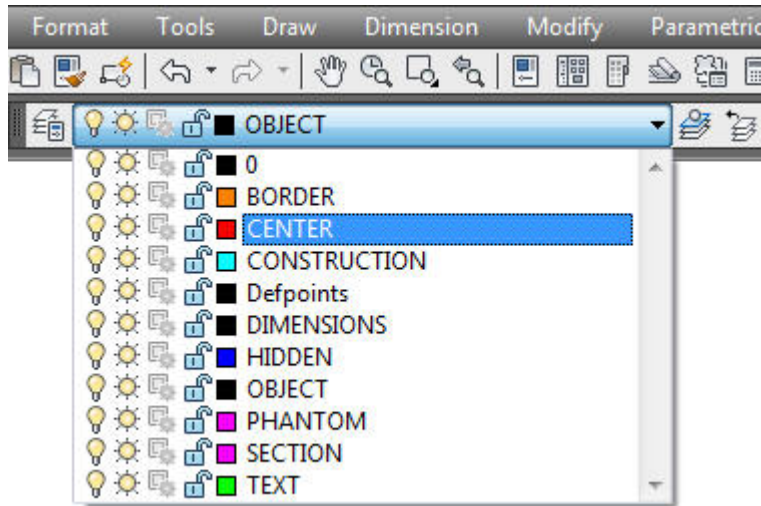
The AutoCAD commands used to make a 3D solid model of the generic hub pictured above will be used to demonstrate typical AutoCAD commands. A 2D drawing of the hub will be used to generate a 3D solid model.



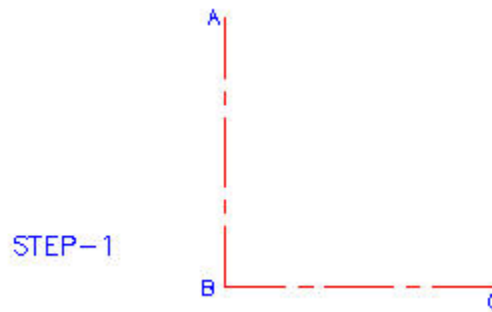
PART M3116 REV C

STEP-10

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



Pick drop-down “Layer Menu” above and select CENTER for centerline.



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

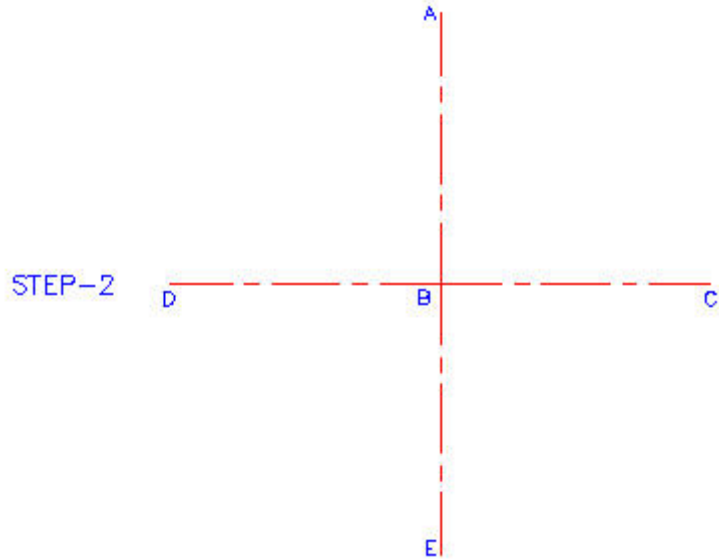
Type the command: **L** for Line.

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**.

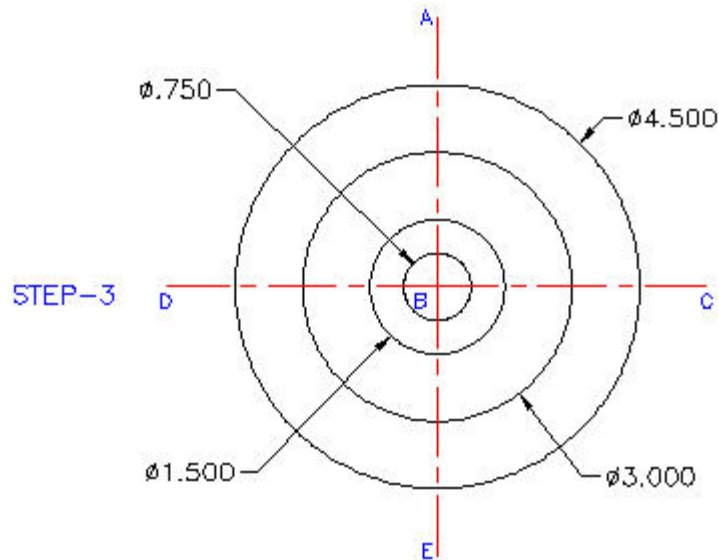
Type **MI** for Mirror on the command Line.

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.

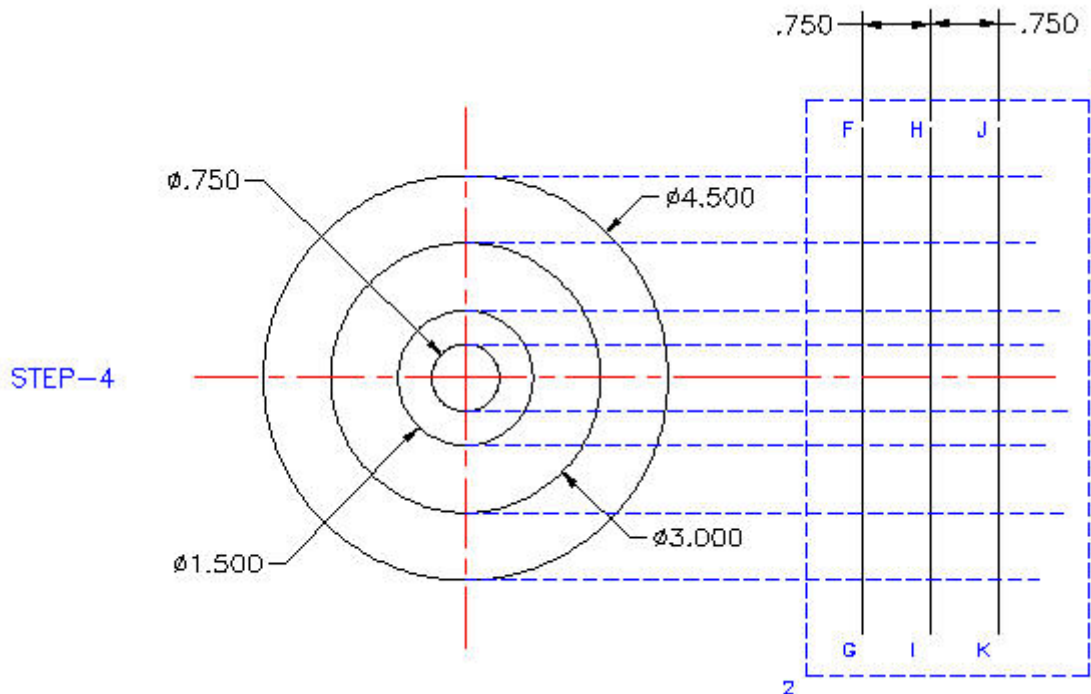
Press the **Spacebar** or **Enter Key** to repeat the command.

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**.

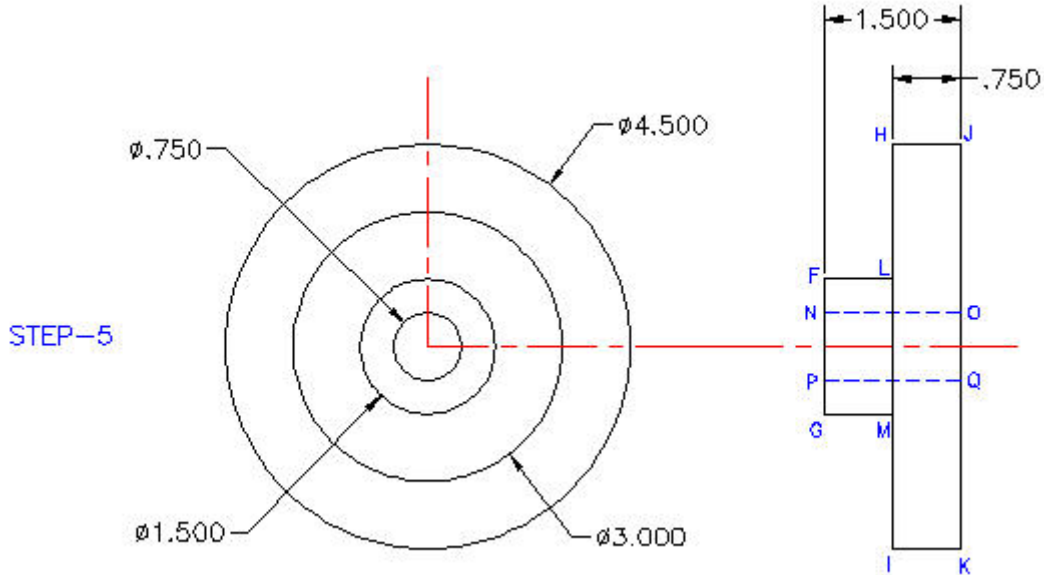
Press the **Spacebar**.

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

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

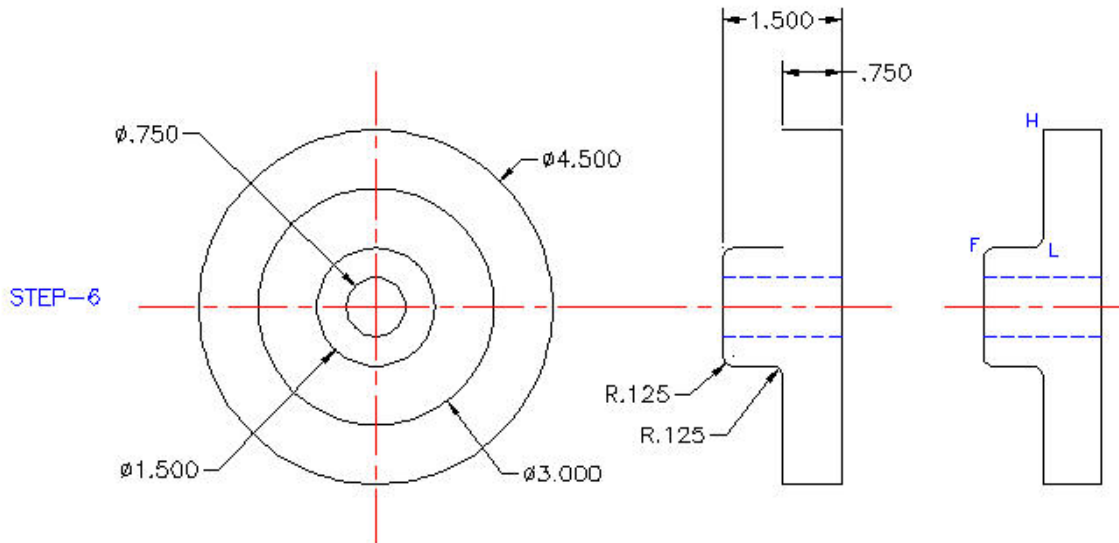
Pick line segments to be removed by trimming to obtain the side view below.

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.

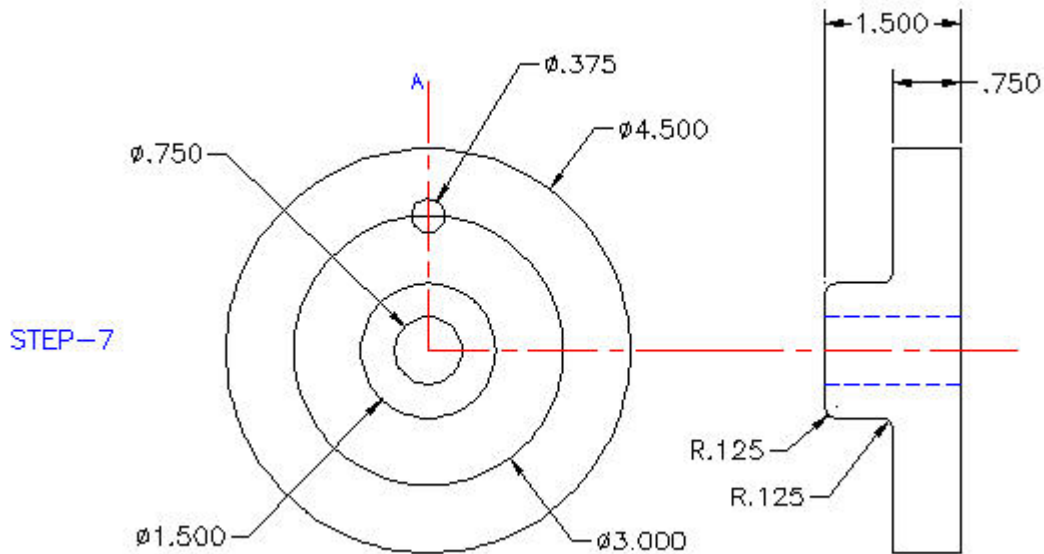
Type **L** for Line on the command Line.

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

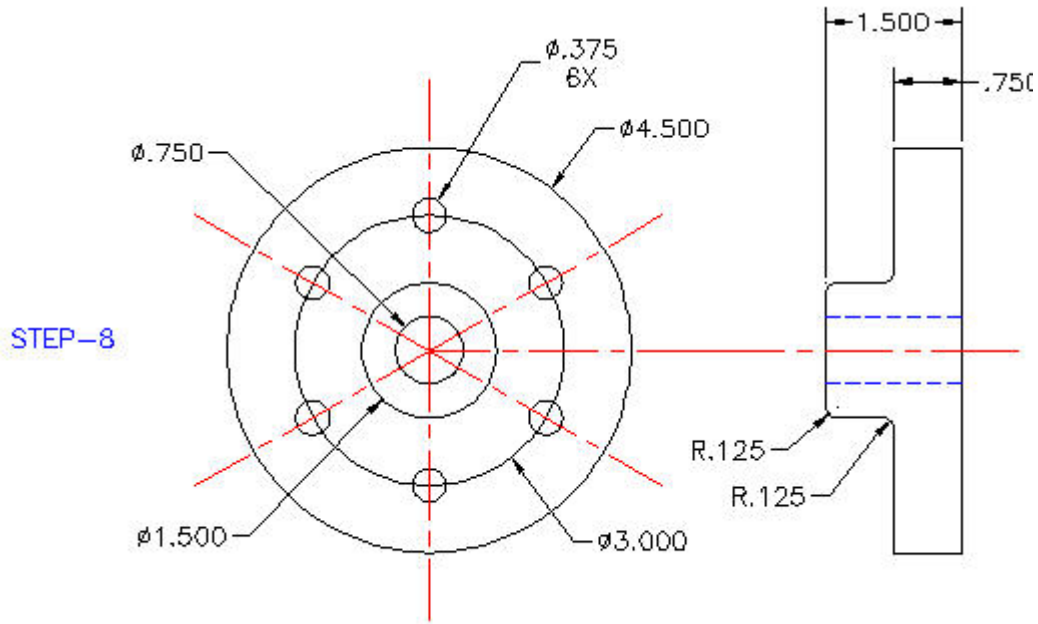
Type **C** for Circle on the command Line.

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

Type **D** for diameter.

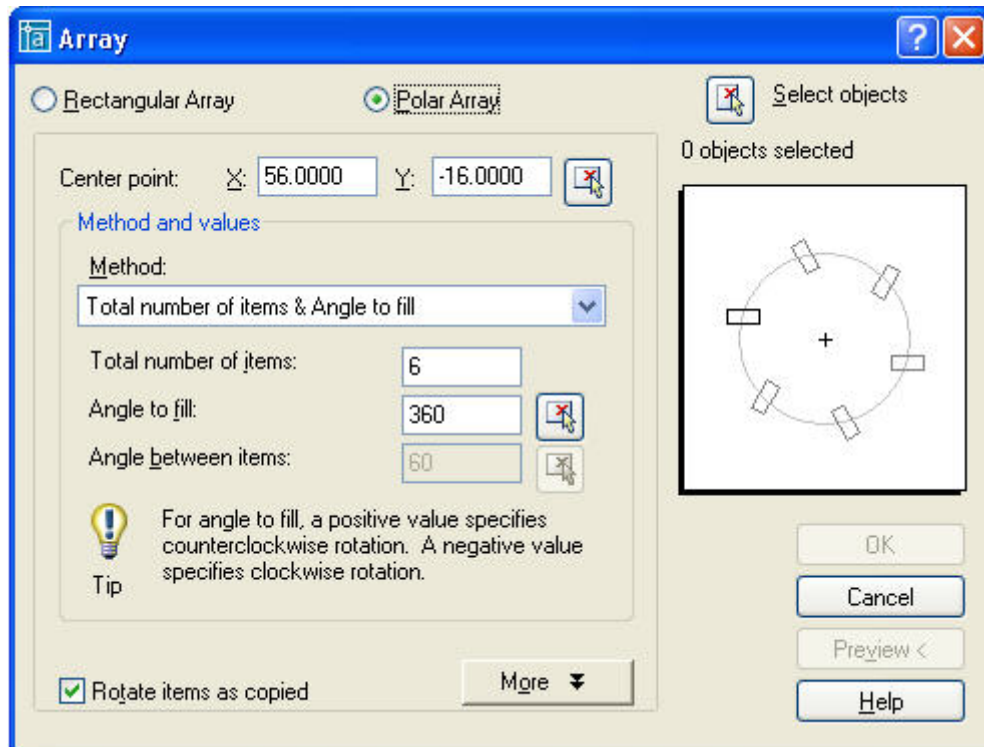
Type **.375** for the dimension.

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



Step-8 Add the Bolt Hole Circle to the Front View

Type **AR** for Array on the command line and the dialog box below will appear.



Pick **Polar Array**.

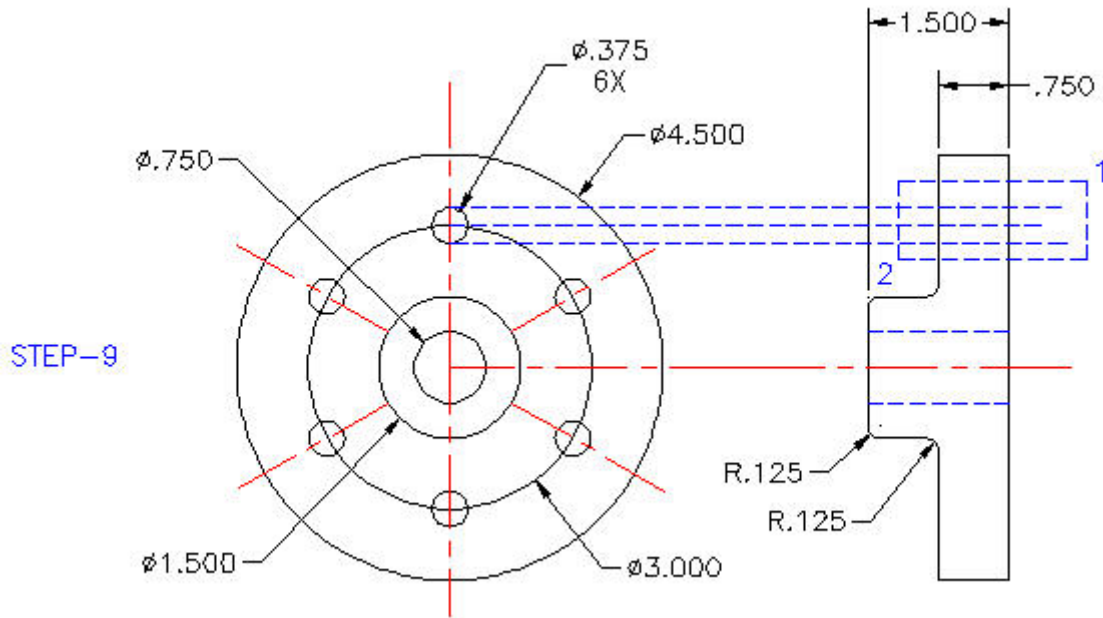
Pick **Select objects**.

Select the 0.375 diameter circle in the drawing.

Pick **Center Point**.

Select the center point of the bolt circle on the drawing.

Pick **OK**.



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.

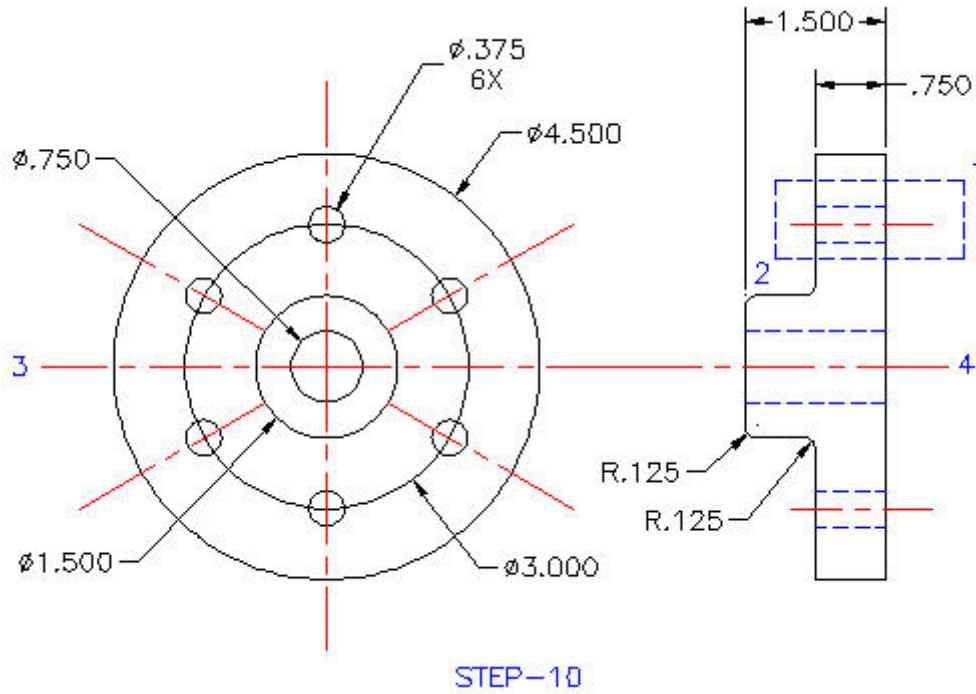
Press the **Spacebar**.

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

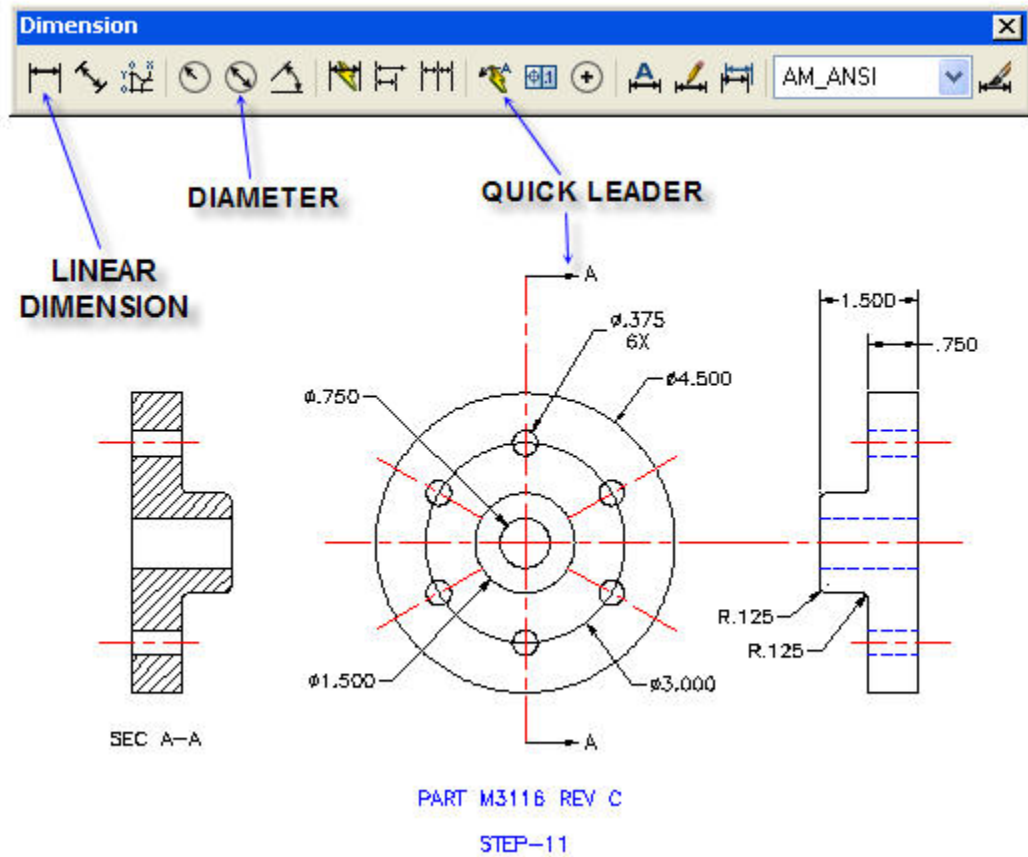
Toggle function key **f8** until you see, **Ortho on**.

Type **M** 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.

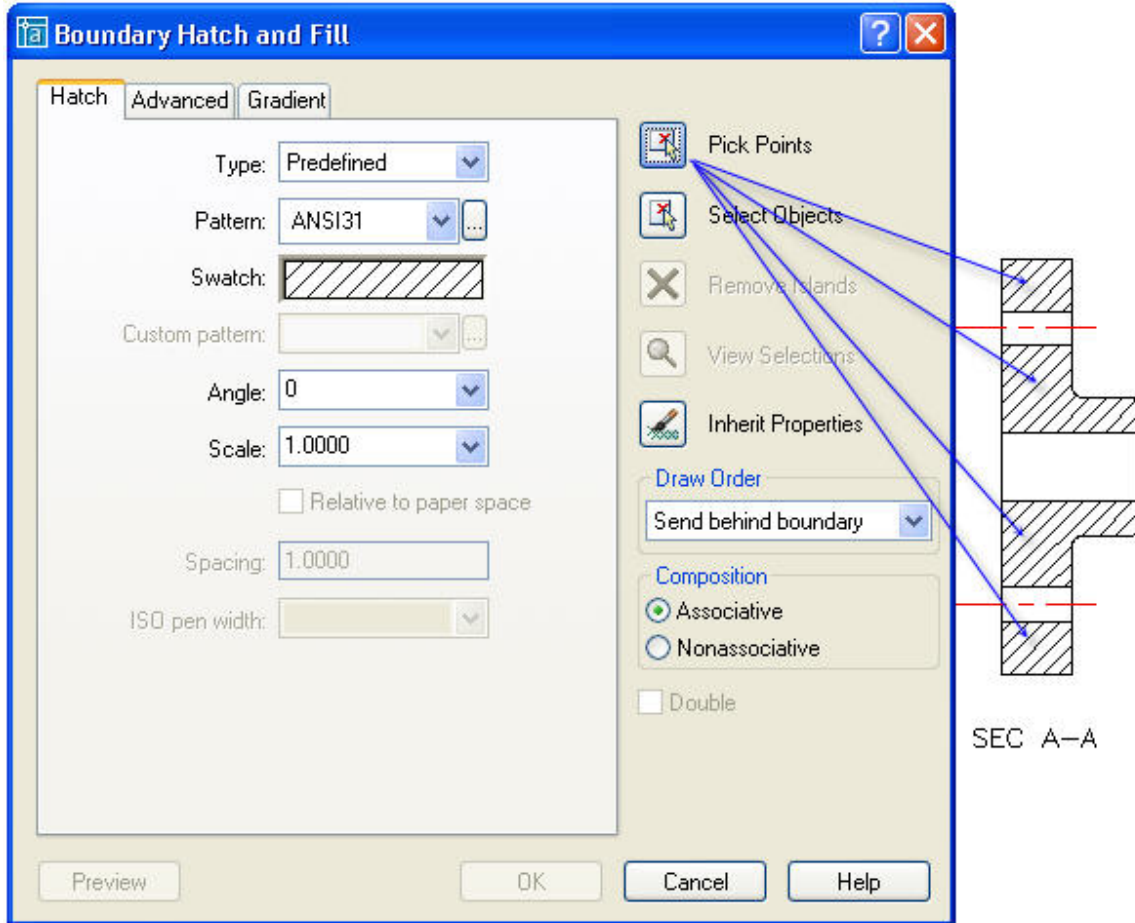
Pick the **Diameter** icon.

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**.



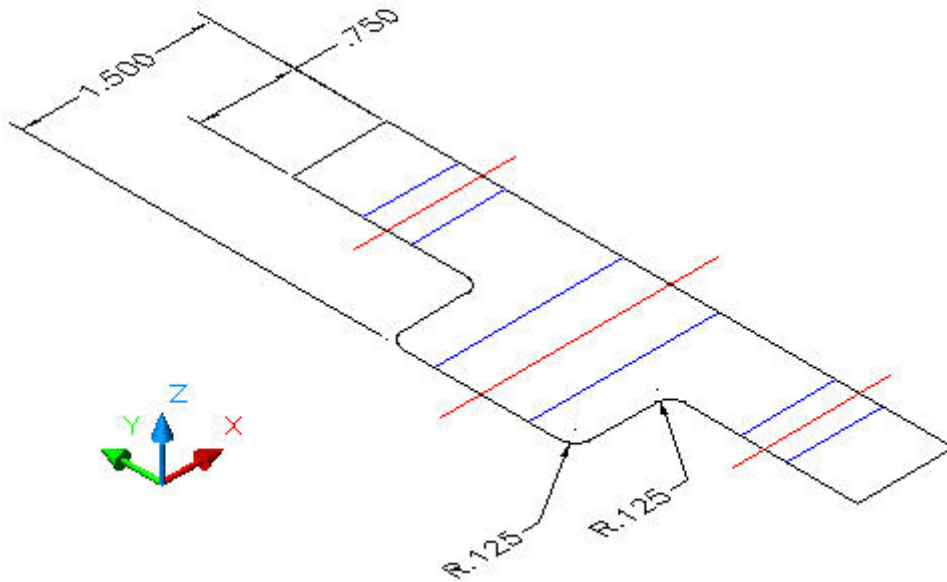
Step-12 Add Hatching to the Sectioned Side View

Type **BHATCH** for Boundary Hatch on the command line and the dialog box above will appear. Select **Pick Points** and pick the inside area points to be hatched. Press the **Spacebar**. Pick **OK**.



TOP VIEW

SW ISOMETRIC VIEW

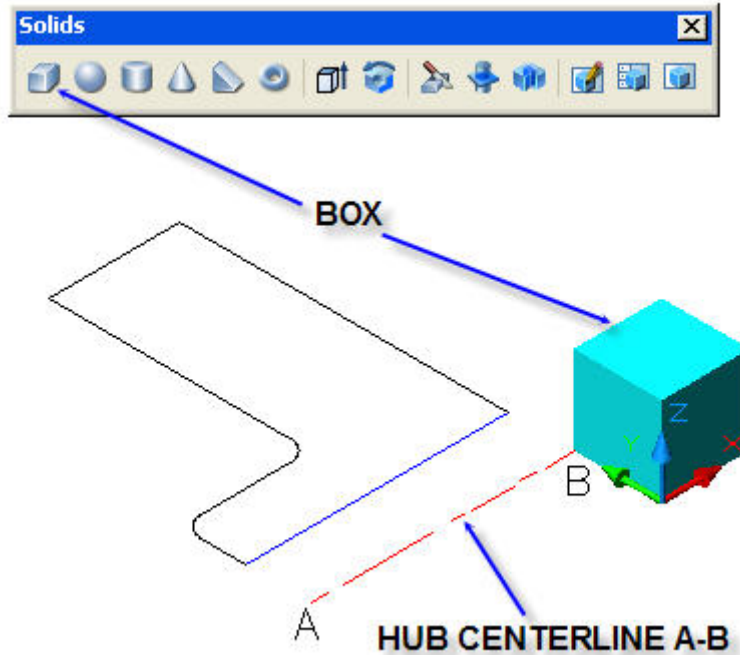


Step-13 Select the View Toolbar & Pick the SW Isometric View

The SW Isometric View of the Hub will be seen as shown above.

Select the Top View to return to the AutoCAD 2-dimensional zone.

Press the **Spacebar**.



Step-14 Erase One Half of the Hub and Place a Cube as Shown

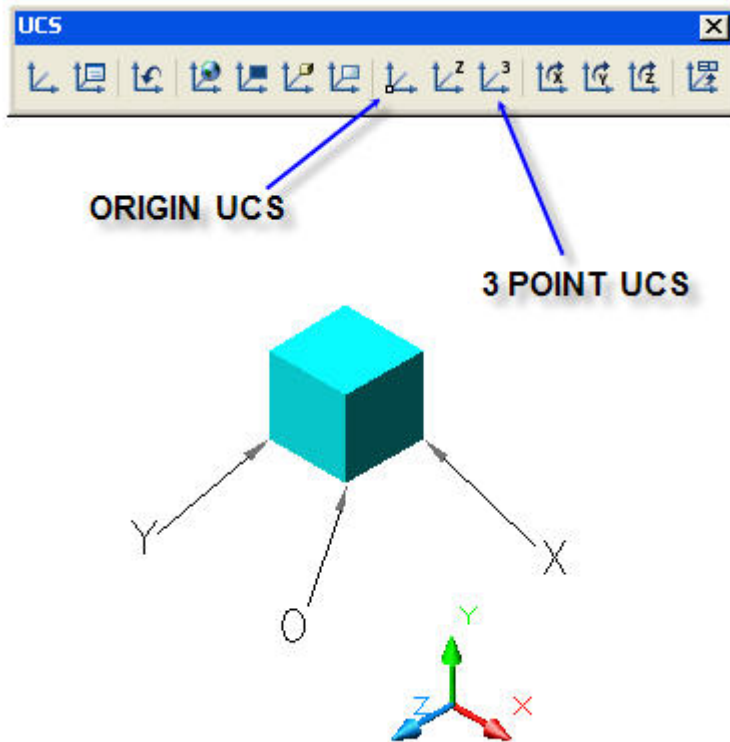
Type **E** for Erase and pick all lines to be erased, leaving one half of the Hub as seen above.

Select the Solids Toolbar and pick the Box icon.

Pick the location of the bottom left corner of the Box.

Type: **C** for Cube > **.5** for length.

Press the **Spacebar**.

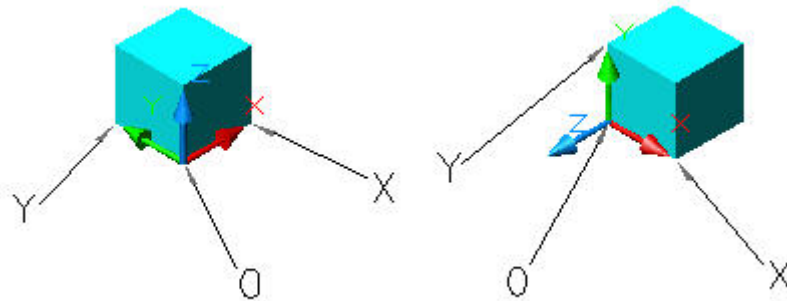


Step-15 Set the New User Coordinate System (UCS)

Select the UCS Toolbar.

Pick the 3 Point UCS icon.

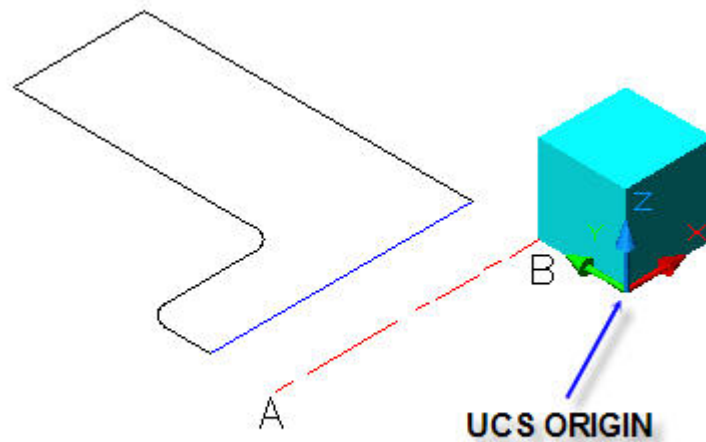
Pick the new: Origin **O**, **X** corner, **Y** corner, as illustrated above.



Step-16 The New UCS is Set and Reset.

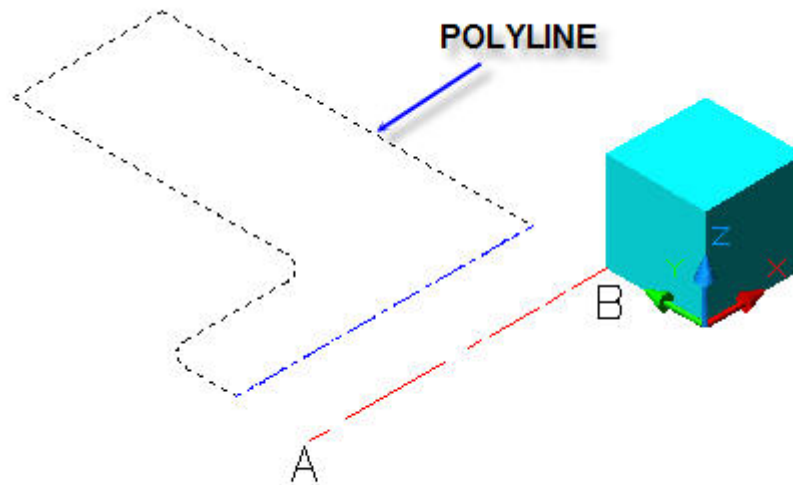
Left: The 3 Point UCS has been set with the X-Y plane on the Box bottom.

Right: The 3 Point UCS has been set with the X-Y plane on the Box left side.



Step-17 Set the New UCS Origin

The new User Coordinate Origin is shown above with the X-Y Plane on the box bottom.



Step-19 Convert Hub Half Section Line Segments to a Continuous Polyline

Type **PE** for Polyline Edit.

Type **M** for Multiple lines.

Select the Hub line segments shown dashed here.

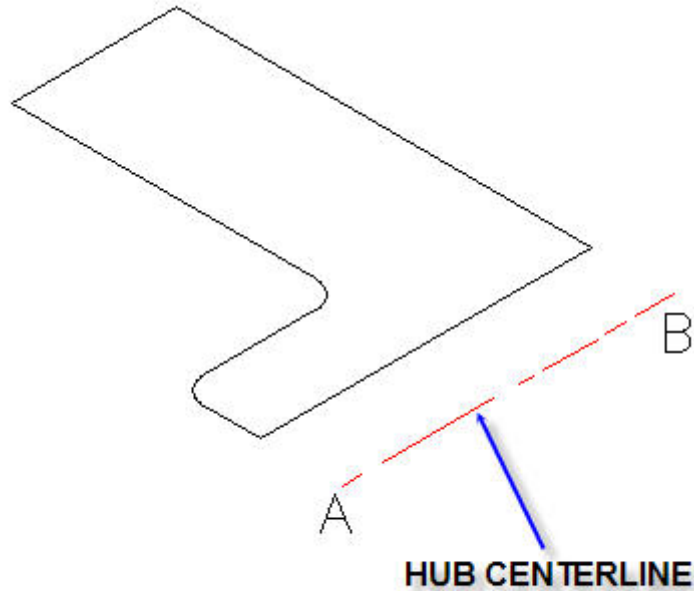
Press the **Spacebar** to indicate all lines have been selected.

Press the **Spacebar** to accept the default **Y** for Yes join line segments.

Type **J** for Join the line segments.

Press the **Spacebar** to accept default Fuzz distance of 0.000.

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



Step-20 Revolve Hub Half Section to Create the 3-D Solid Model

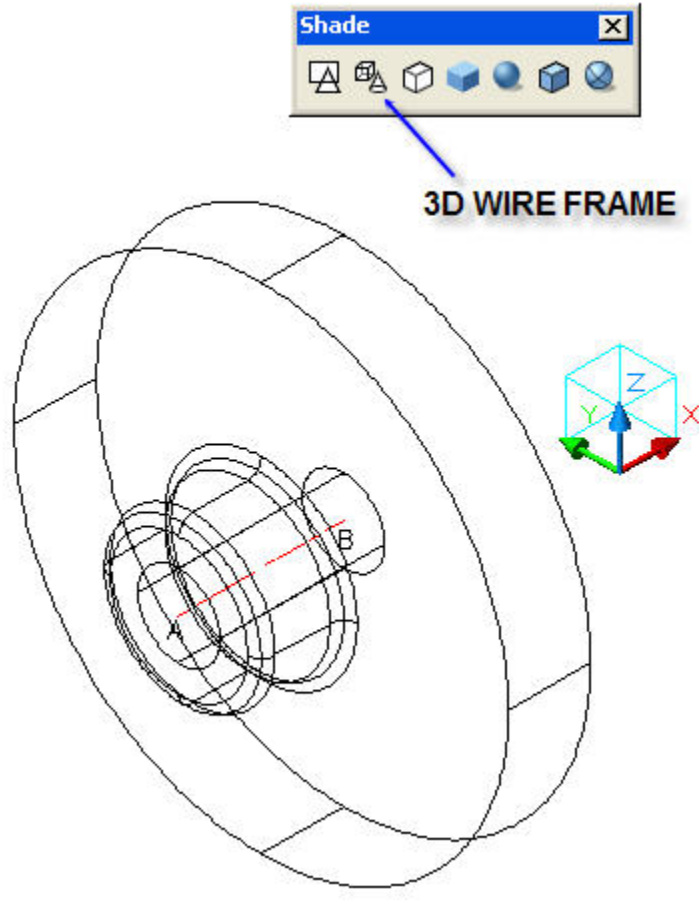
Type **REV** for Revolve.

Select the Hub Polyline shown dashed in Step-18.

Press the **Spacebar** to indicate the continuous Polyline has been selected.

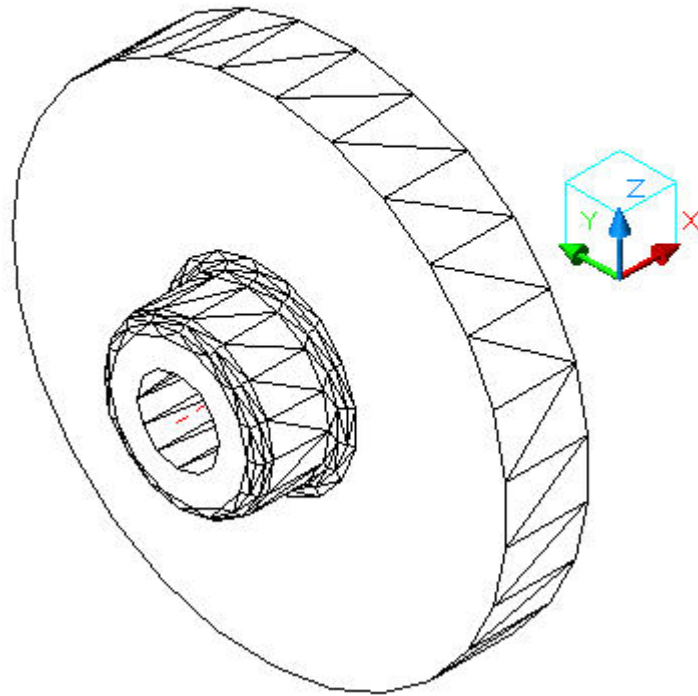
Define axis of revolution by picking **A** then **B**.

Press the **Spacebar** accept the default 360 degrees of rotation.



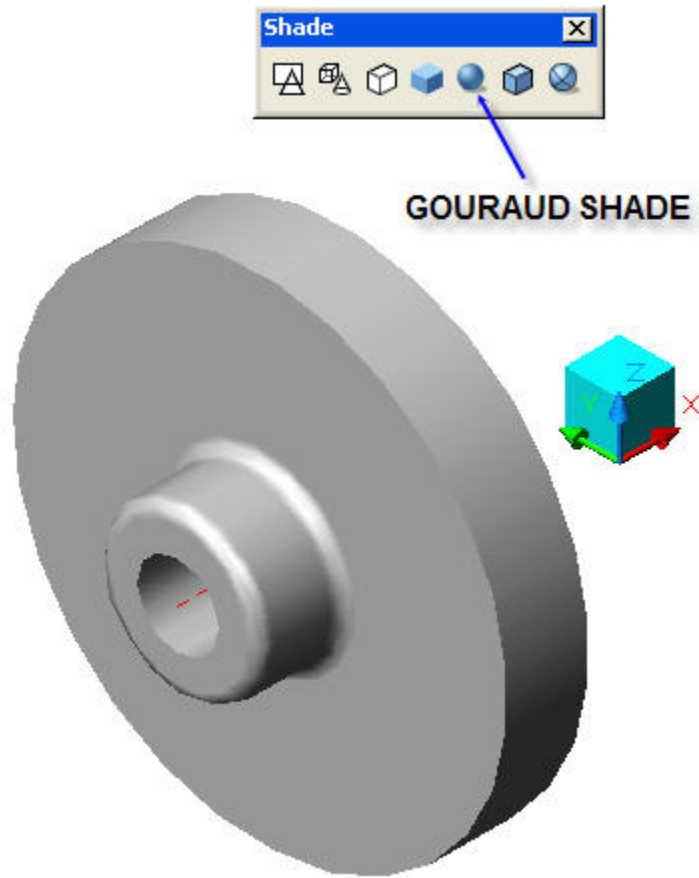
Step-21 3D Wire Frame

Select the Shade toolbar and pick the Wire Frame icon as illustrated.



Step-22 3D Hidden Lines

Select the Shade toolbar and pick the Hidden lines icon.



Step-23 3D Gouraud Shaded

Select the Shade toolbar and pick the Gouraud Shaded icon.

10. 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: _____

A DWG NO: _____

COMMENTS:

B DIMENSIONS:

- 1 Units: Inch or Metric
- 2 Overall Height, Width, Length
- 3 Zero Datum's
- 4 Hole Locations, Sizes & Shapes
- 5 All Features Dimensioned

- 6 Decimal Places are a Minimum
- 7 Dimensions are Not Crowded
- 8 Tolerances are Indicated

C VIEWS:

- 9 Orthographic Projection
- 10 Outline of Parts are 0.7 Line weight
- 11 All Other Lines are 0.3 Line weight
- 12 Section & Detail Views Required?

D NOTES:

- 13 Material?
- 14 Pre-finish Treatment?
- 14 Finish?
- 16 Hardware, Nearside or Far Side?
- 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

11. 3-D AutoCAD COMMAND LIST

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

CHK

Toolbars	Rt Key	Toolbars > Views > UCS > Solids
Object Snap F3	OS	OSnap On / Off

Iso Plane	F5	Top > Right > Left
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
Extrude	EXT	Select Objects > Height or Path
Isometric	SNAP	S > I > (.5) > OK, F5 > Top Plane > Right > Left
Drawing Limits	LIMITS	Specify Lower Left Corner > Upper Right Corner
Slice	SL	Pick 3 Points > Both
Subtract	SU	Pick large object > Pick small objects
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
User Coord Sys	UCS	Origin > 3 Point
Undo	U	
Union	UNI	Select Objects > OK
Zoom All	Z > A	
Zoom Window	Z > W	Pick Window Corner > Pick Opposite Corner
Pan	P	Pick Window Start > Pick Window End
Plot	ctrl + 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 > (Smoothes 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 Attribute Definition	XR	Xref-Manager > Attach > File Path to Drawing
Attribute Edit	ATT	Pick Insert Point > Tag > Prompt > Value > Wblock
	ATE	Pick Attribute Block > Edit

12. 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 3-D Basics course.

Please attempt the quiz.