Process For Creating A Multi-View Drawing

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Basic Rules of Projection

Direction of projection:

* The top view and the front view (as well as the bottom view) share the same length dimensions, and the length dimensions are projected vertically;

* The front view and the right side view (as well as the other "elevation" views, such as left side and rear views) share the same height dimensions, and the height dimensions are projected horizontally;

* The top view and the side view share the same width (depth) dimensions, and the width (depth) dimensions can be projected through a miter line (drawn at 45° from horizontal, from a corner where the three planes, i. e., the horizontal plane where the top view is drawn, the frontal plane where the front view is drawn, and the profile plane where the side view is drawn, meet). the width (depth) from the top view can be first projected horizontally to the miter line; then, from the points of intersection, they can be projected vertically to the right side view. Similarly, the width (depth) dimensions can be projected from the side view vertically up to the miter line and then horizontally to the top view.
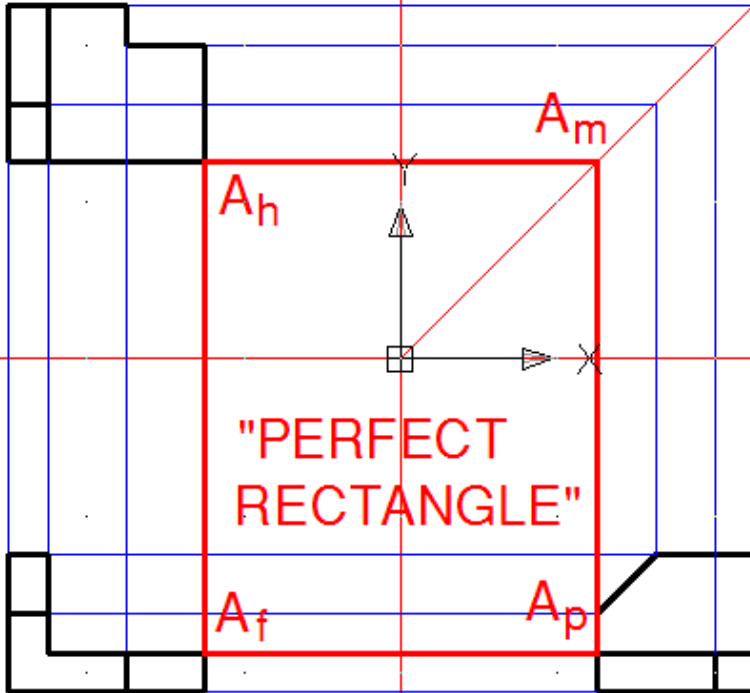
Perfect Rectangle Rule:

* The locations of any point on the object in the top view, front view, miter line and side view form a perfect rectangle
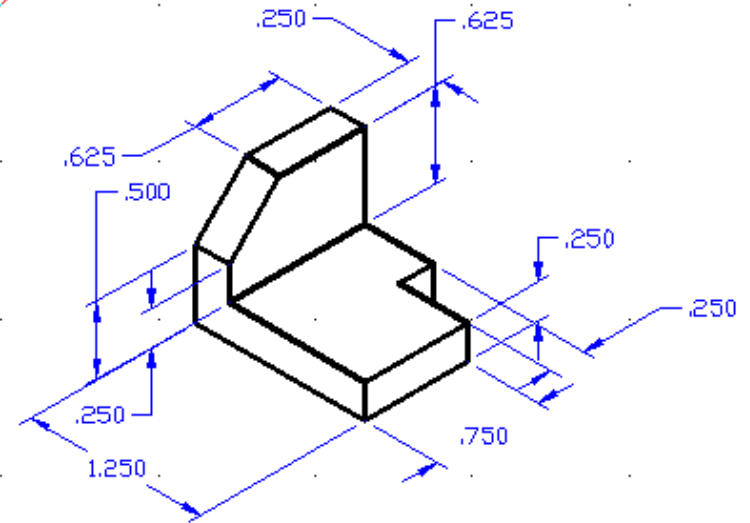
TOP VIEW

MITER LINE

45°

A_m

A_h

"PERFECT
RECTANGLE"

A_f A_p

TOP VIEW   TOP VIEW

.250   .625
.625
.500
.250
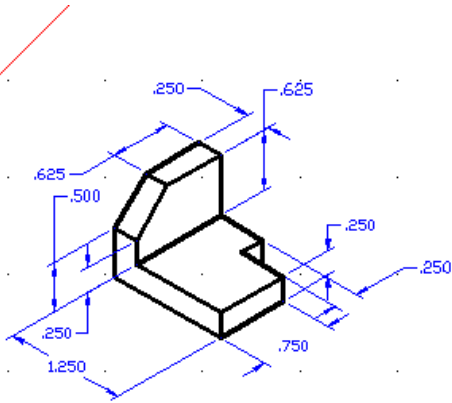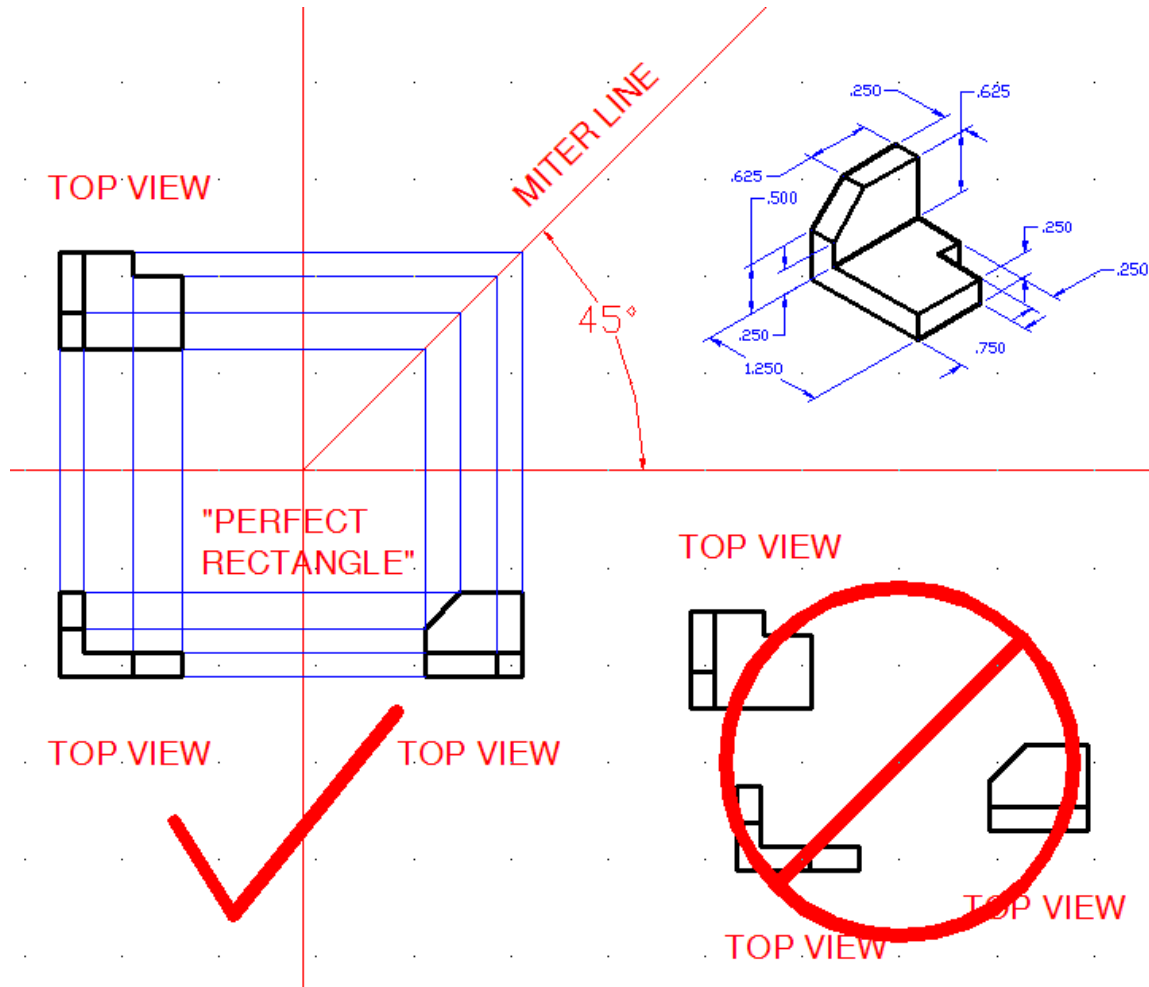.250
.250   .750
1.250

Direction of projection:
Top view and front view →
    vertical projection;
Front view and side view
    → horizontal
    projection;
Top view and side view →
    projection through the
    miter line.

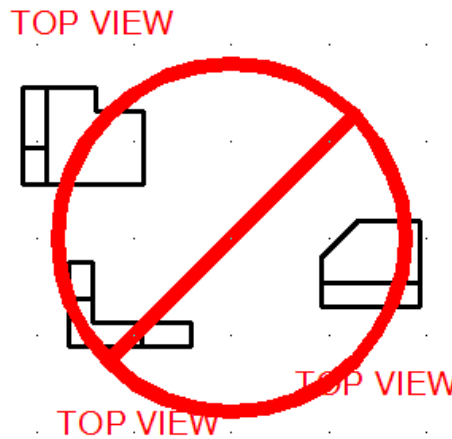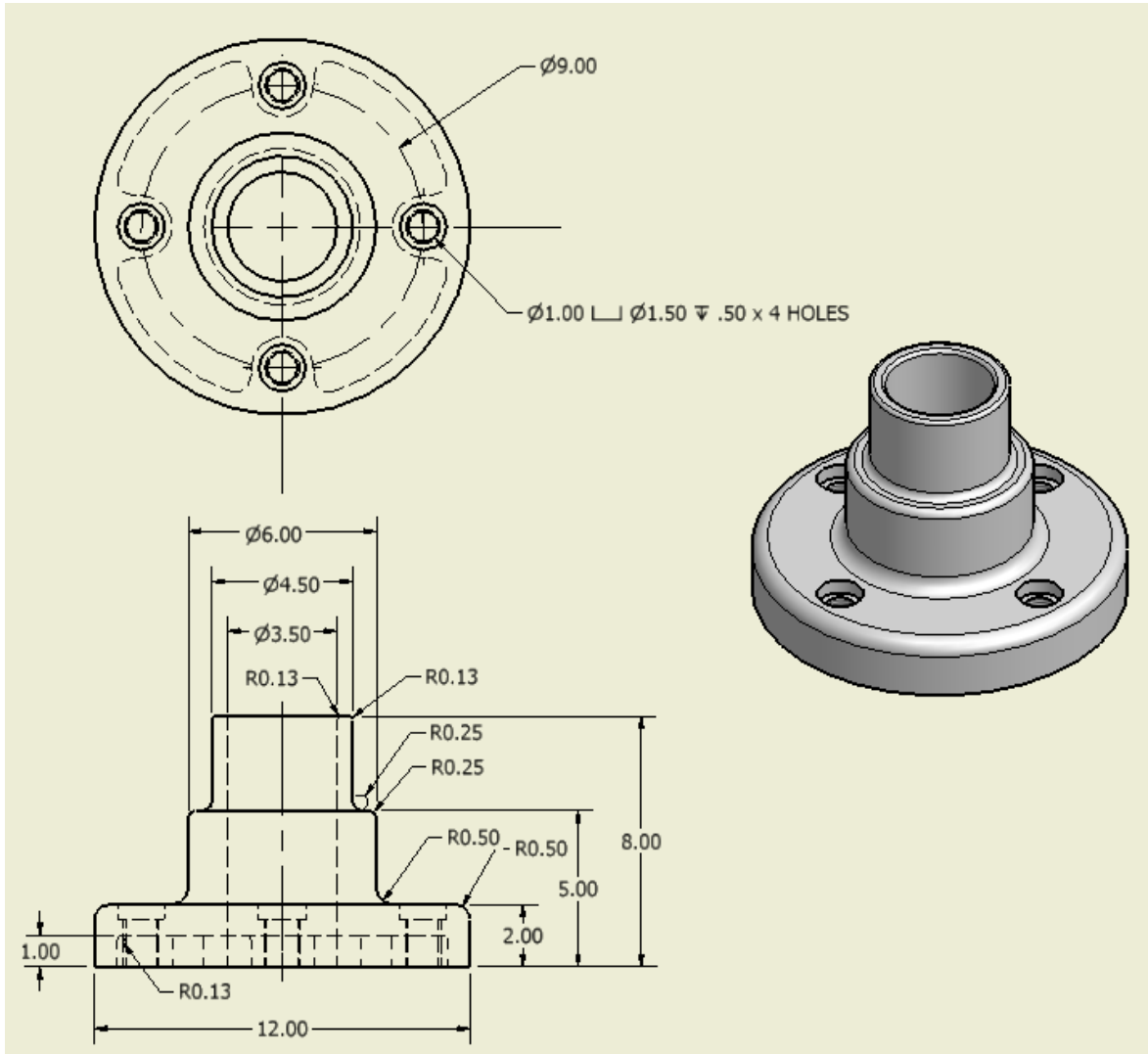# Alignment and Spacing Between
The Three Principal Views

Alignment:

- The top view and the front view must align vertically;

- The front view and the right side view must align horizontally.
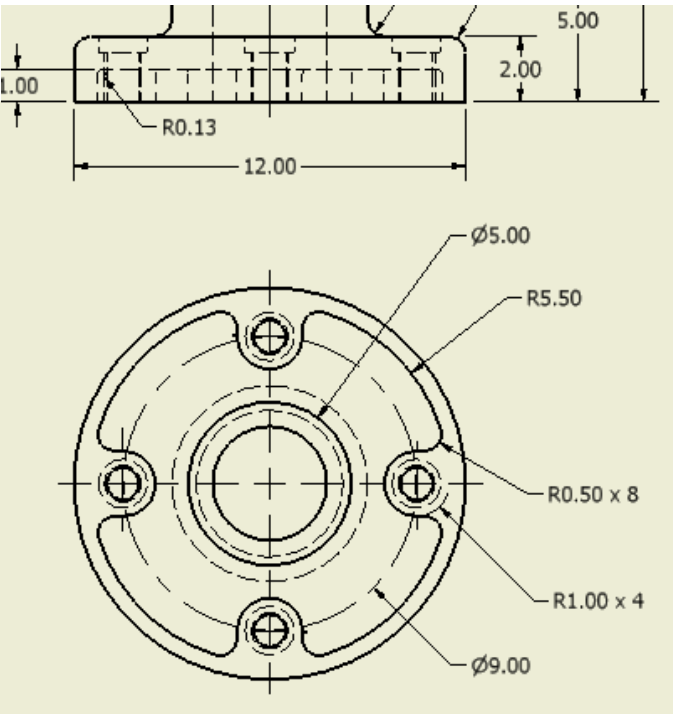
Spacing:

- About 3 inches or more between the top and front view, and between the front and side views (to make room for the placement of dimensions, notes, etc., and to avoid crowdedness).

The Assignment

Ø9.00

Ø1.00 ⌴ Ø1.50 ▽ .50 x 4 HOLES

Ø6.00
Ø4.50
Ø3.50
R0.13 — R0.13
R0.25
R0.25
R0.50 — R0.50
8.00
5.00
2.00
1.00
R0.13
12.00

Top view and front view
(not sectioned)

1.00
5.00
2.00
R0.13
12.00

Ø5.00
R5.50
R0.50 x 8
R1.00 x 4
Ø9.00

Bottom view

Assignment:

- Draw the top view, the front full section view and the right side view of the flag pole holder.

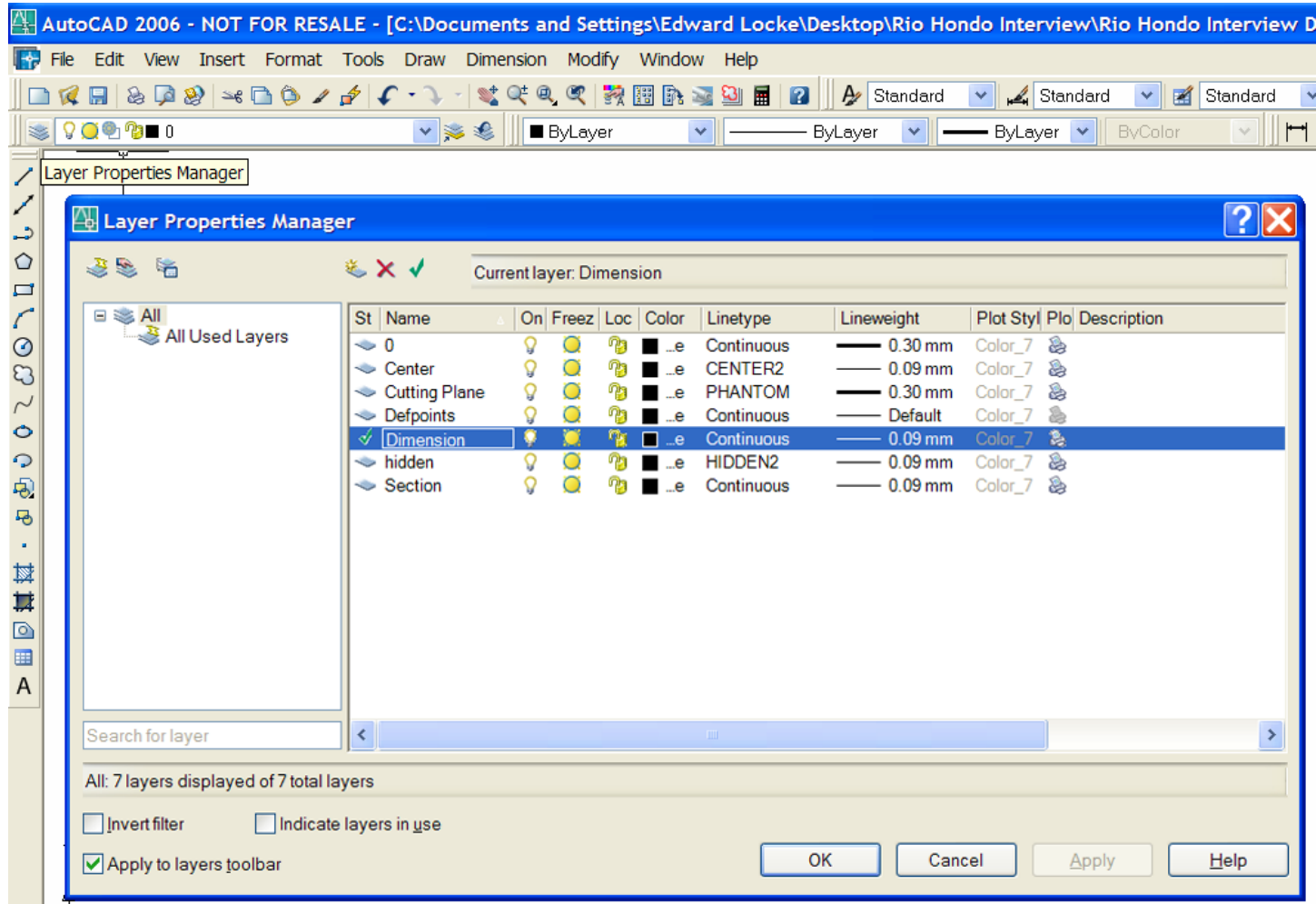- Extra credit: draw the bottom view.

Starting An "Imperial" File
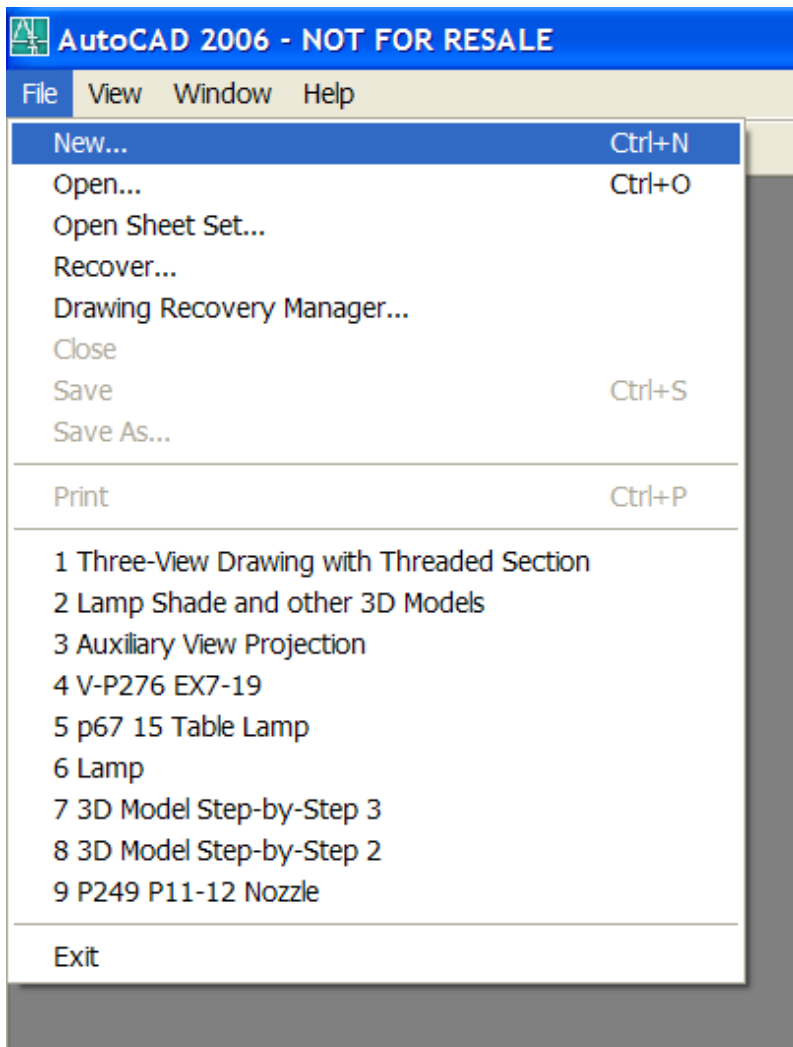
Laying out the three views:

- File → New…

- Imperial (feet and inches) → OK
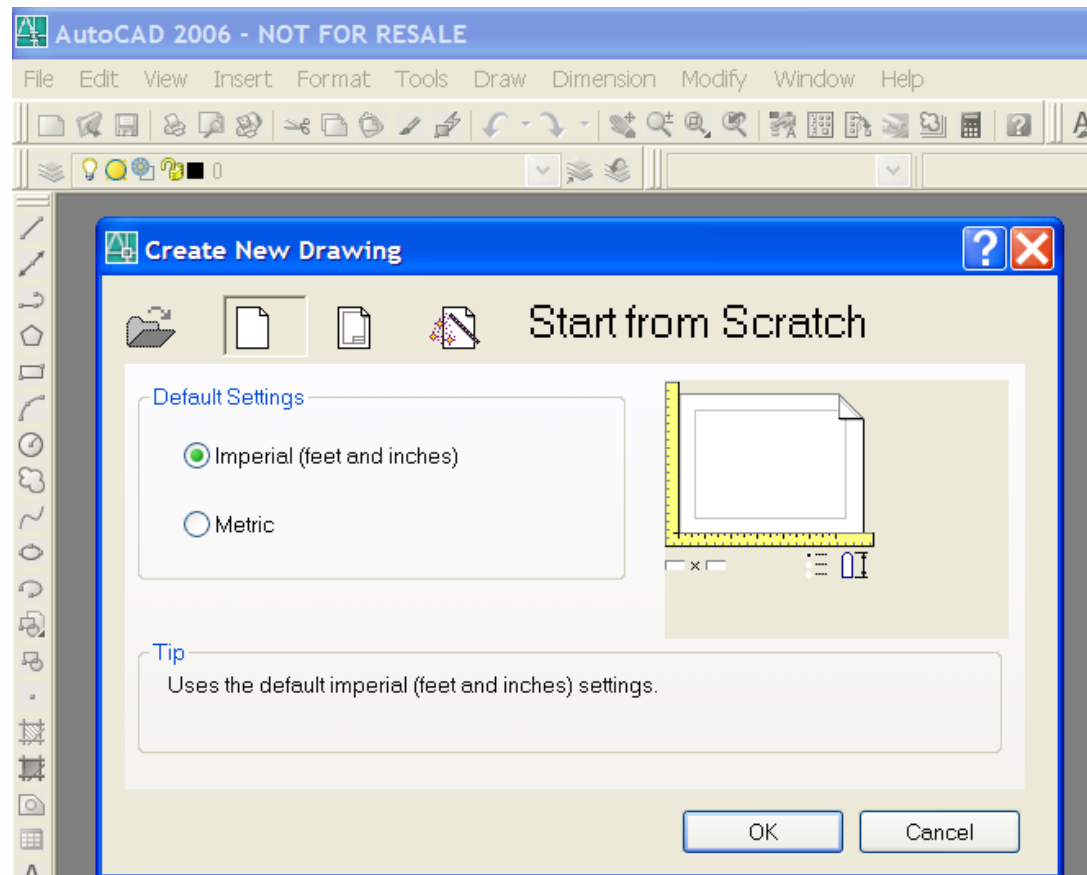
Setup The Layers

Creating different Layers

Layer function allows collective change of the properties of objects drawn on particular layers.
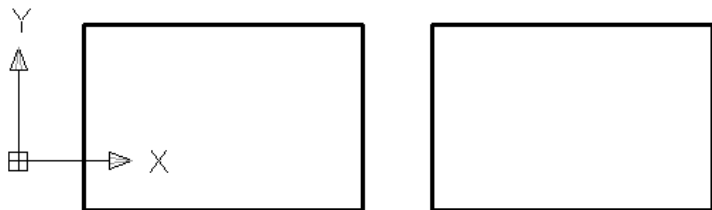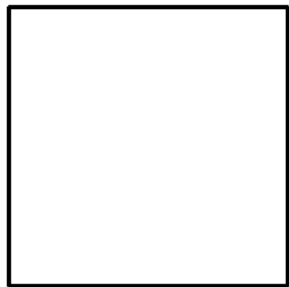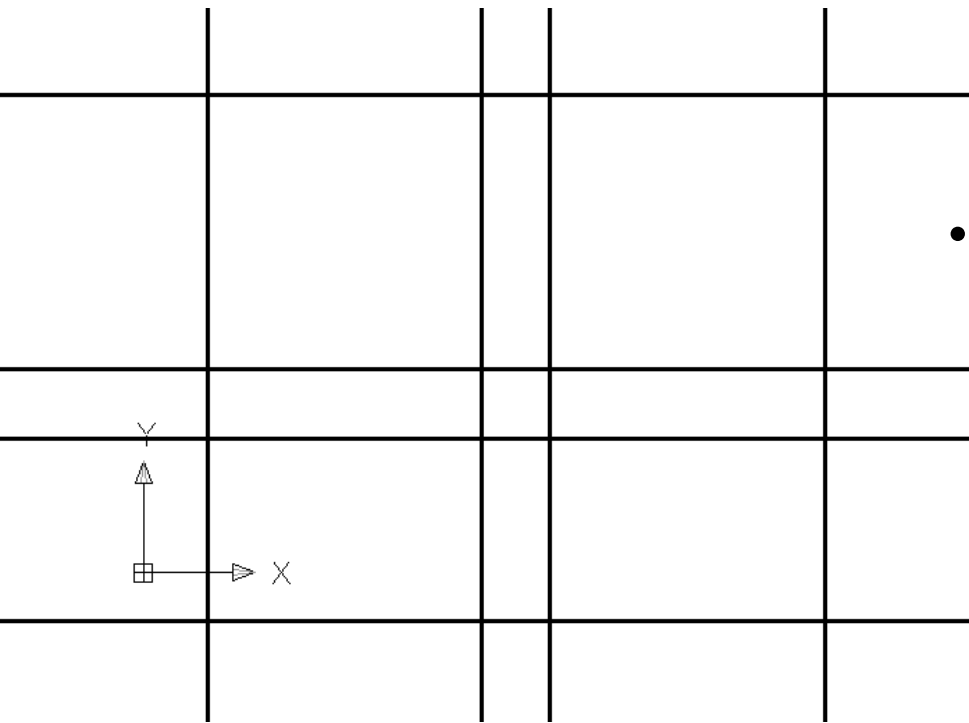
# Step-by-Step Creating A Three-View Drawing in AutoCAD
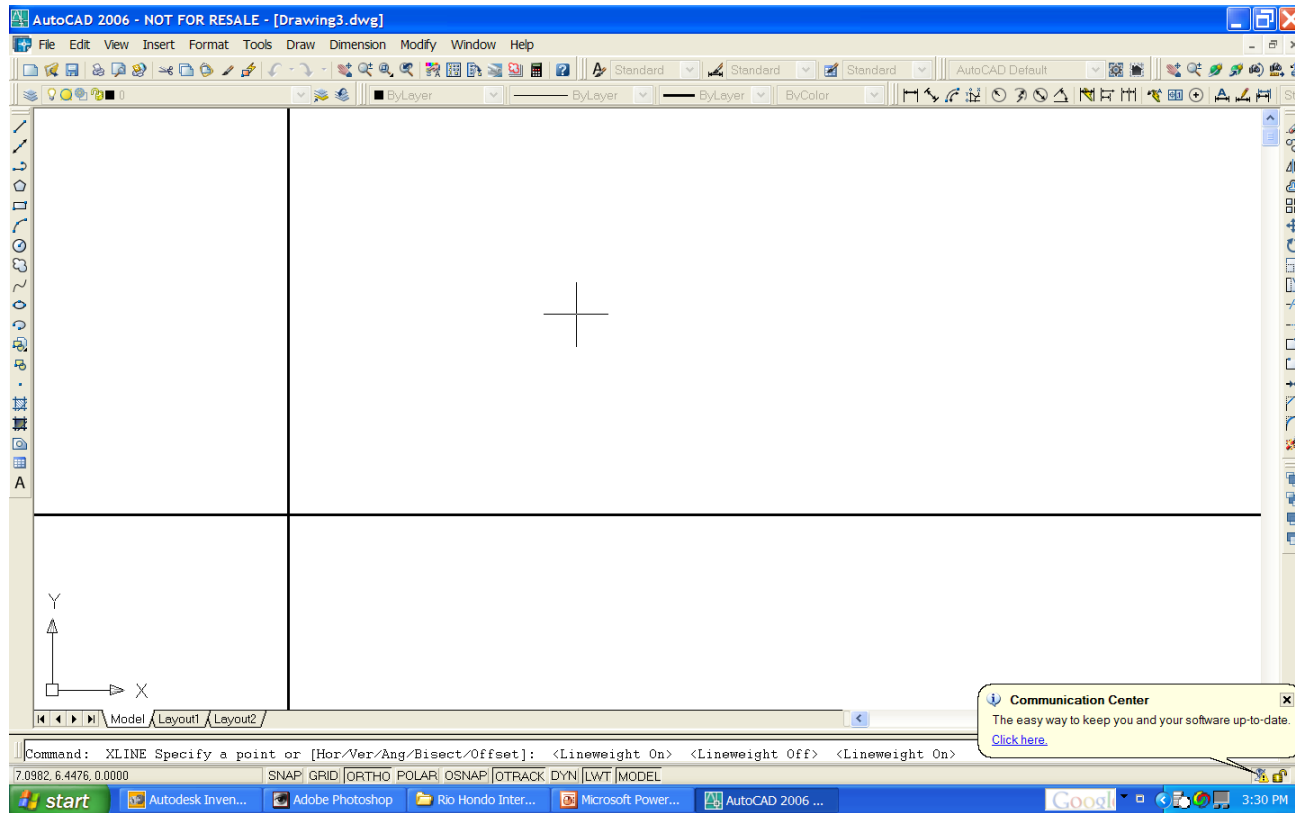
Laying out the three views:

- Turn ORTH on (F8);
- Use the Construction Line (xl) tool to draw a vertical line and a horizontal line.

Laying out the overall dimensions across all three views first helps to avoid the problem of misalignment between the views.
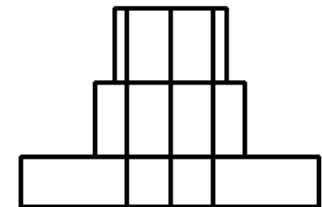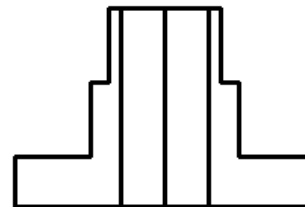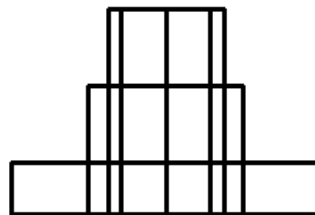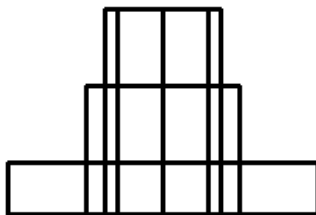
- Use the Offset (o) tool to set the overall length of the object across the top and front views, the spacing between the front and side views, and the width on the side view, by offsetting the vertical construction line. Next, use the Offset (o) tool to set the overall height of the object across the front and side views, the spacing between the front and top views, and the width on the top view, by offsetting the horizontal construction line;

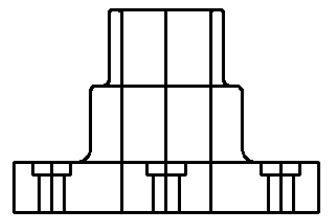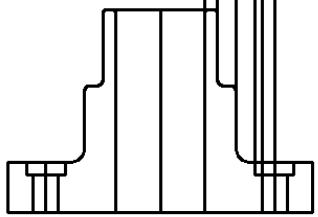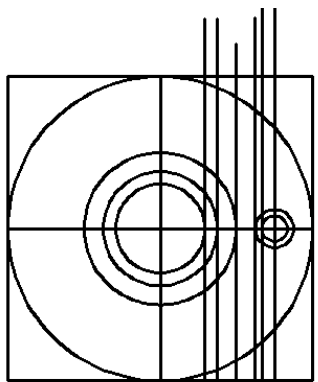- Use the Trim (tr) tool to trim off the unneeded line segments, and to complete the basic layout of the three views.
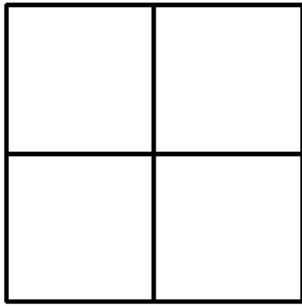
Adding detailed dimensions across the three views:

- Use the Offset (o) tool to set the basic length, height and width (depth) dimensions in two adjacent views as pairs;

- Use the Trim (tr) tool and the Delete key to eliminate unneeded line segments;

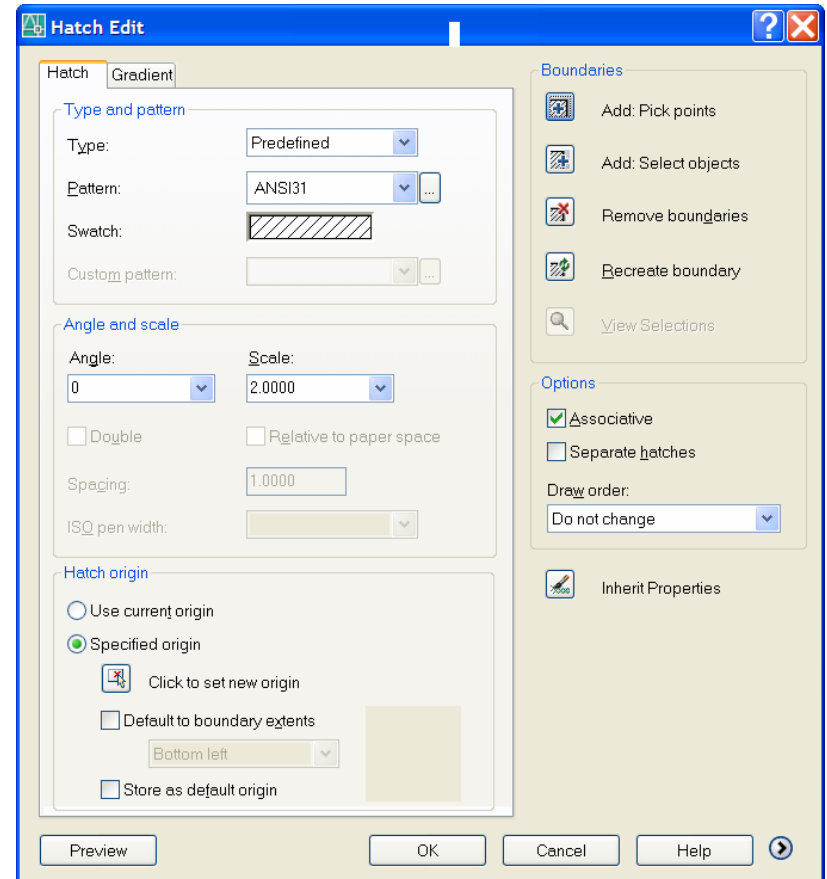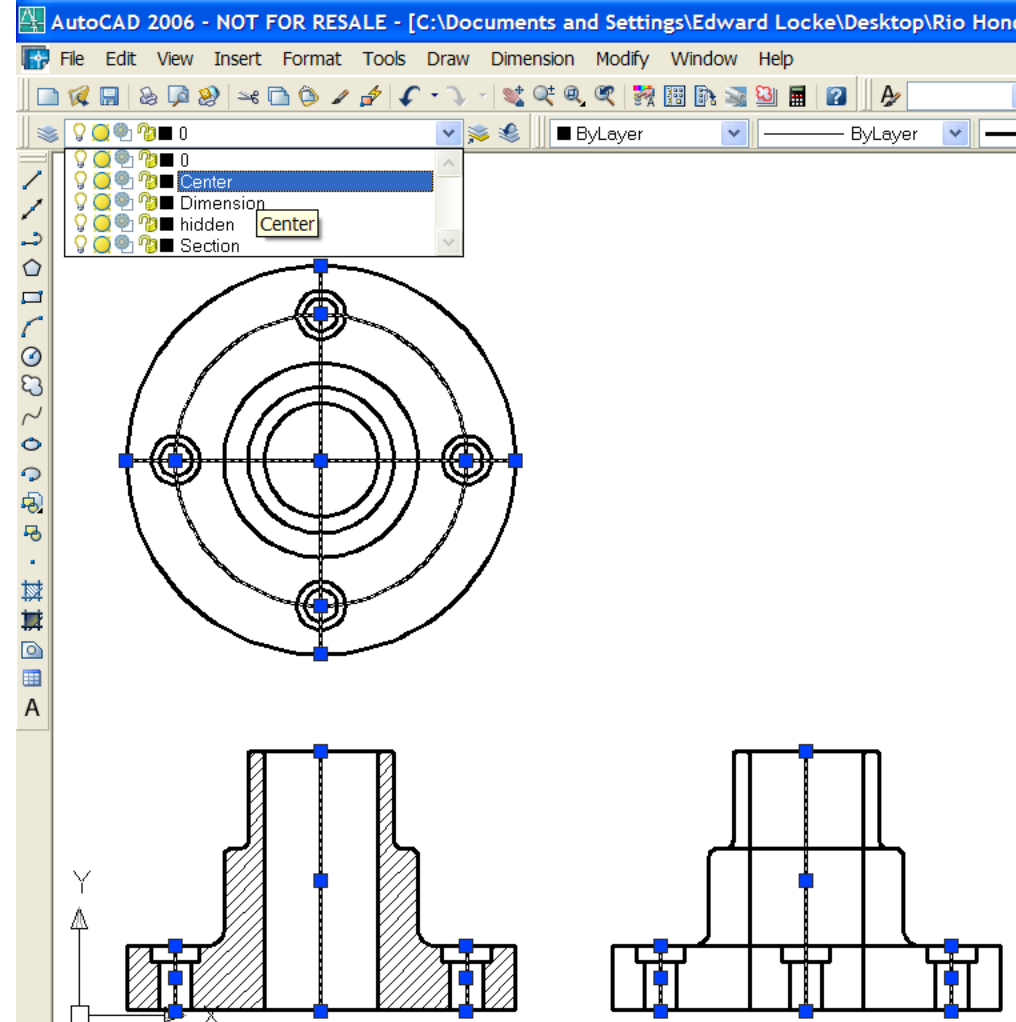File   Edit   View   Insert   Format   Tools   Draw   Di

0

0
Center
Dimension
hidden
Section

- Use a variety of tools to add some details to each view (I omit the cavity on the base plate in this demonstration);

- Select the Section layer and use the Hatch (h) tool with "Add Pick points" option to add section lines to the front view;
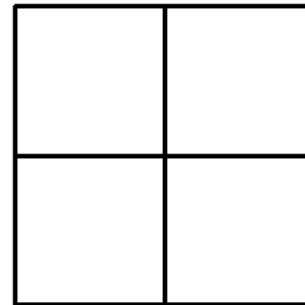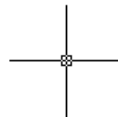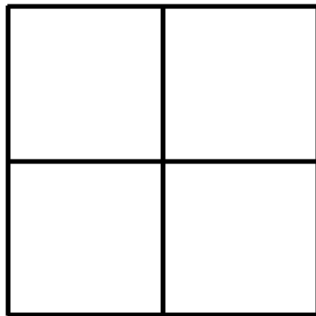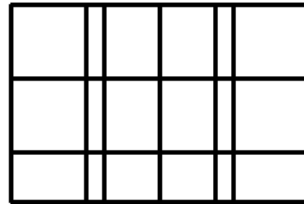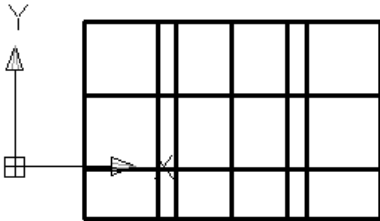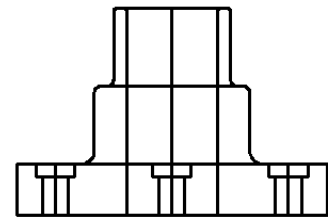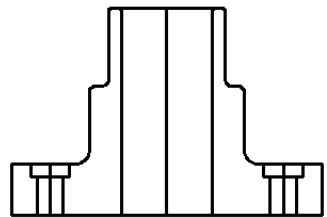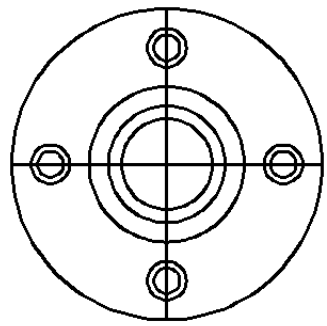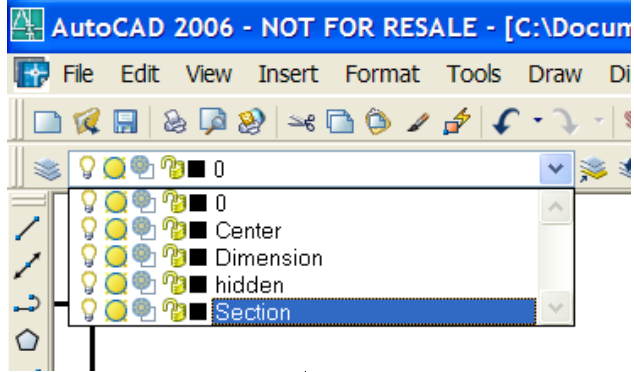
- Notice that hidden lines usually are not drawn on the section view

Hatch Edit

Hatch | Gradient

Type and pattern

Type: Predefined
Pattern: ANSI31
Swatch:
Custom pattern:

Angle and scale

Angle: 0   Scale: 2.0000
Double   Relative to paper space
Spacing: 1.0000
ISO pen width:

Hatch origin

○ Use current origin
● Specified origin
   Click to set new origin
   □ Default to boundary extents
      Bottom left
   □ Store as default origin

Boundaries

Add: Pick points
Add: Select objects
Remove boundaries
Recreate boundary
View Selections

Options

☑ Associative
□ Separate hatches

Draw order:
Do not change

Inherit Properties

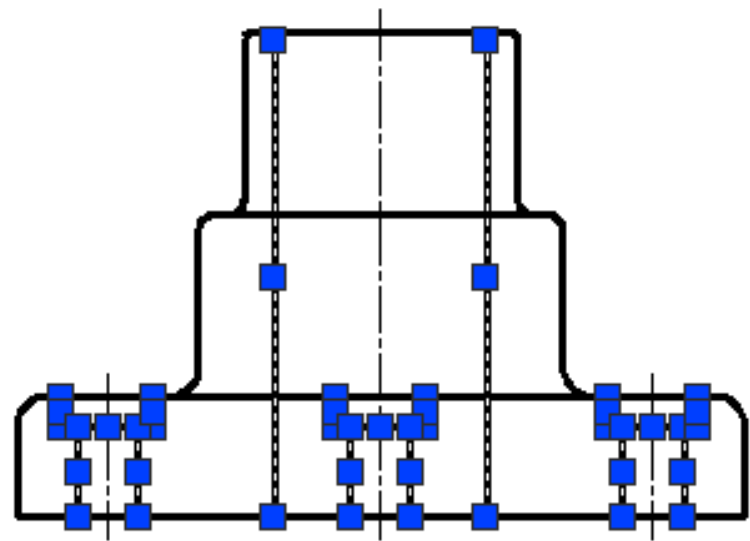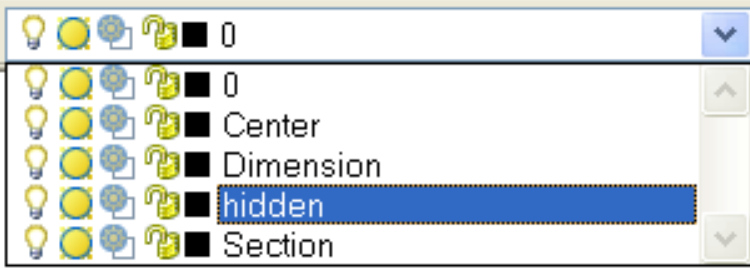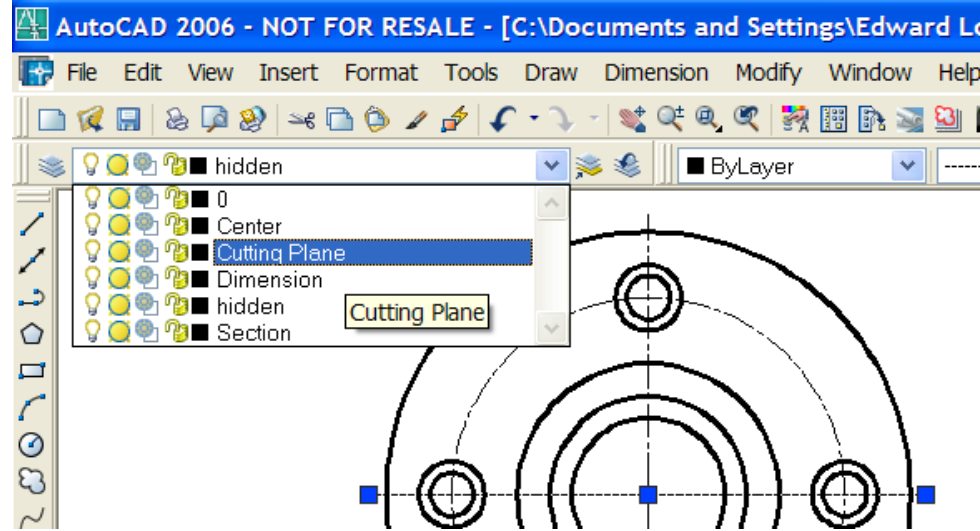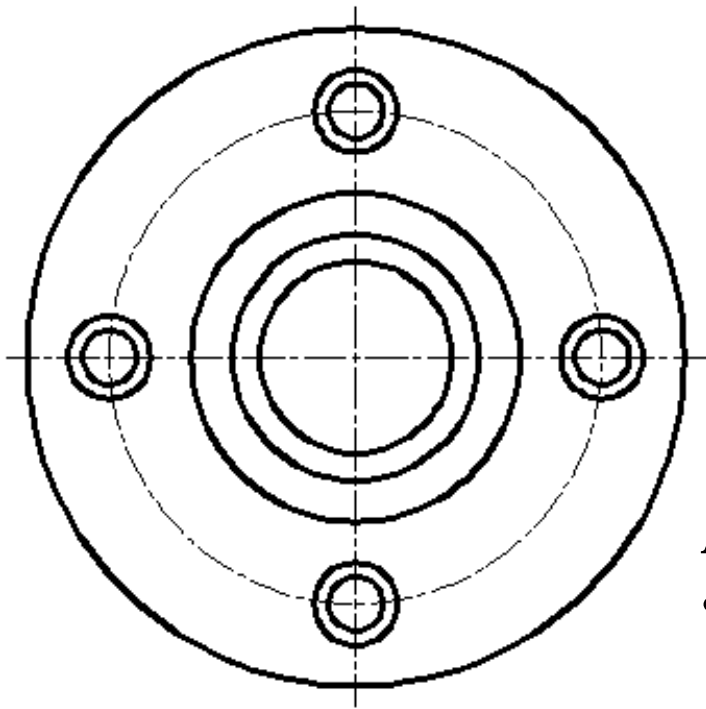Preview   OK   Cancel   Help

- Select the center lines and go to the Layer Manager field to choose the Center layer. The line type of all center lines changes to CENTER2;
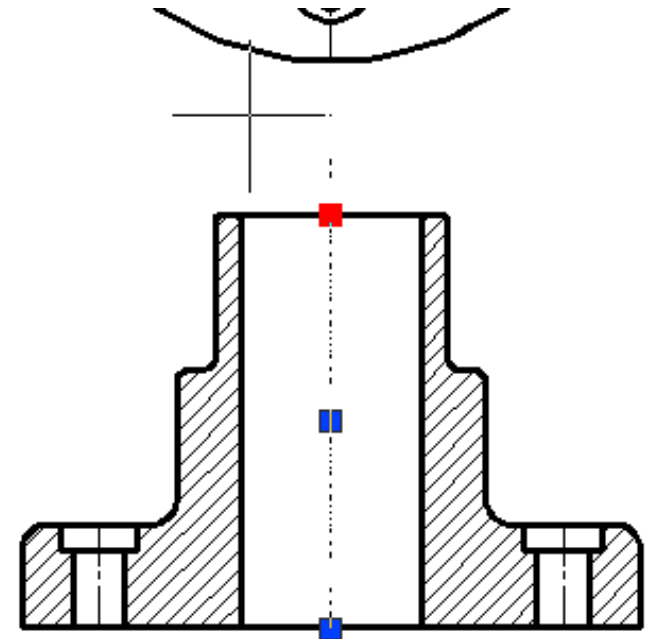
- Centerlines should extend out of the edges. To extend them by about 3/8", first turn ORTHO On and click-select a centerline; then click the endpoint (which turns red), and drag the mouse outward from the edge; then type .375 and press Enter; repeat the same step for all centerline and press Enter to finish.
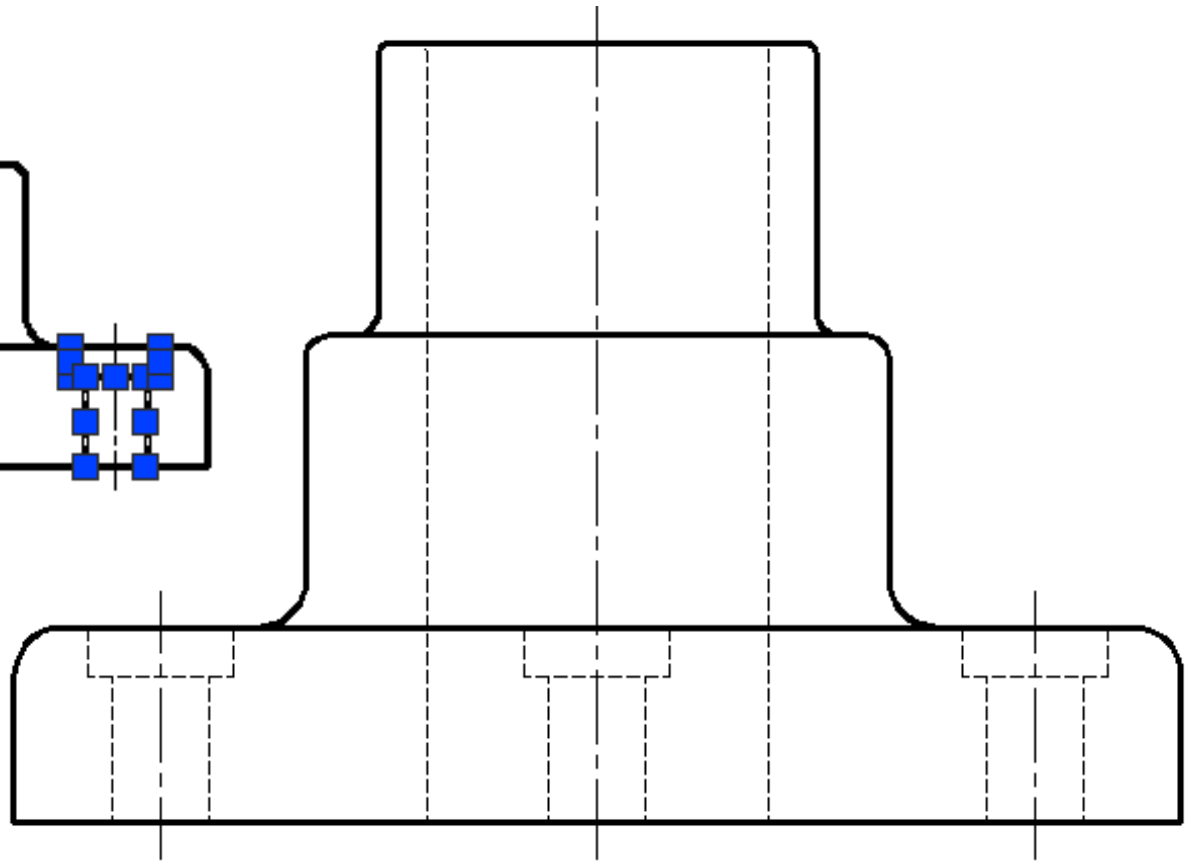
- Select the hidden lines on the right side view and go to the Layer Manager field to choose the Hidden layer. The line type of all center lines changes to HIDDEN2;

File   Edit   View   Insert   Format   Tools   Draw   Dimension   Modify   Window   Help

hidden

ByLayer

0
Center
Cutting Plane
Dimension
hidden
Section

Cutting Plane

Add the cutting plane line on the top view

- Select the horizontal center line on the top view; go to the Layer Manager and choose Cutting Plane layer, then press the Esc key. The linetype changes to PHANTOM.
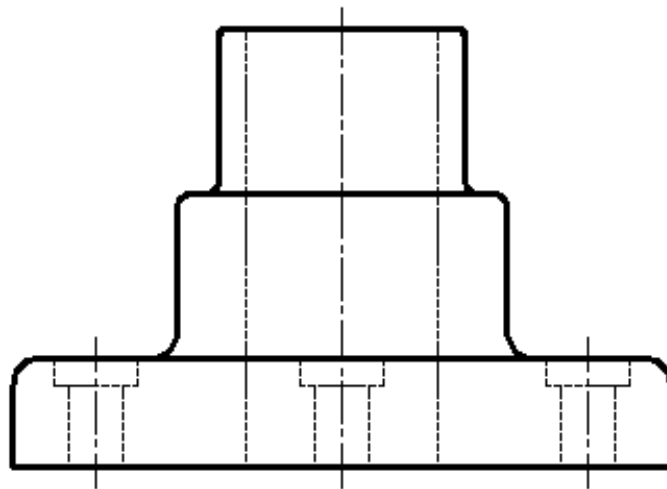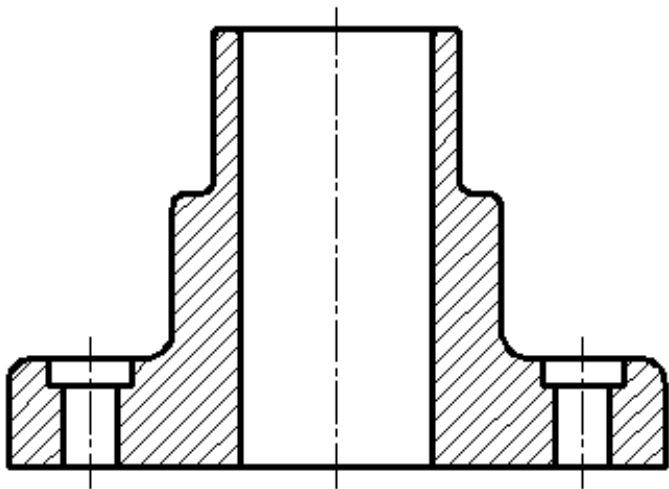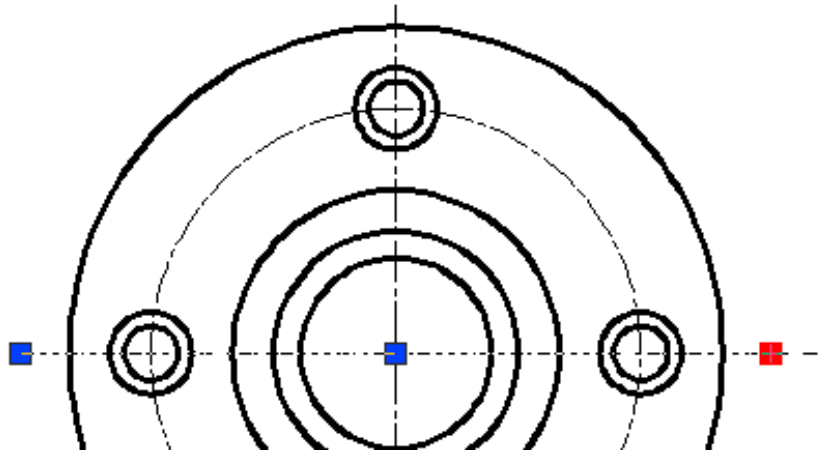
- Select the cutting plane line and extend the line on both side by 0.5";
- Turn the ORTHO On and Osnap (Endpoint) On; use the Poliline (pl) tool with the Width option, and Direct Distance Entry method to create one arrow with a stem length of 0.5", a width of 0.5" and an arrow length of 0.25"; then use the Copy (co) tool to copy it to the other side;
- Use the Text (dt) tool to draw the section label A – A above the arrows.
- Draw the SECTION A-A label below the front view.

A

A

0.25

0.50

0.50
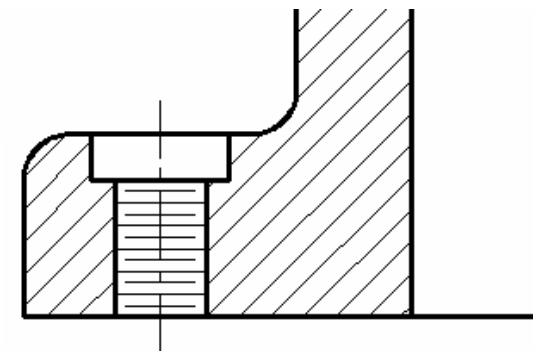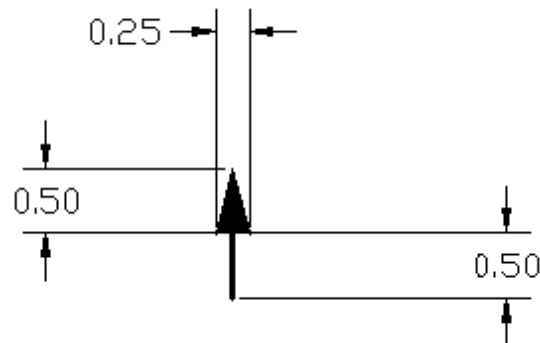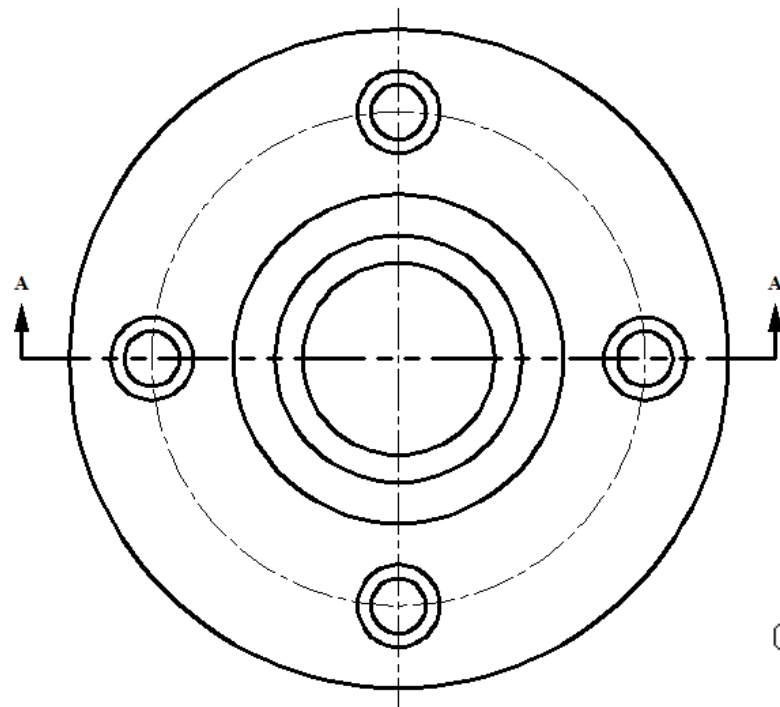
SECTION A-A
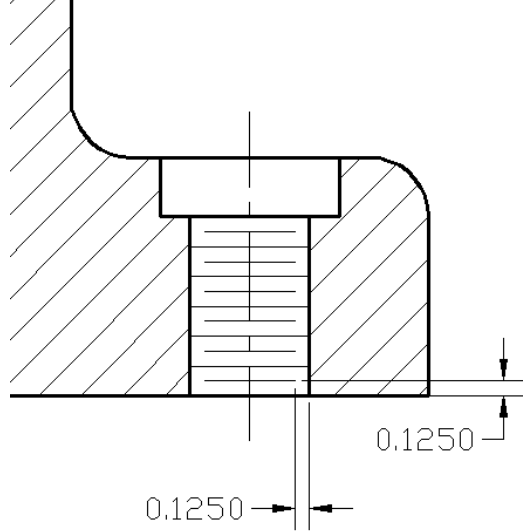
SCHEMATIC THREAD SYMBOL

0.1250

0.1250
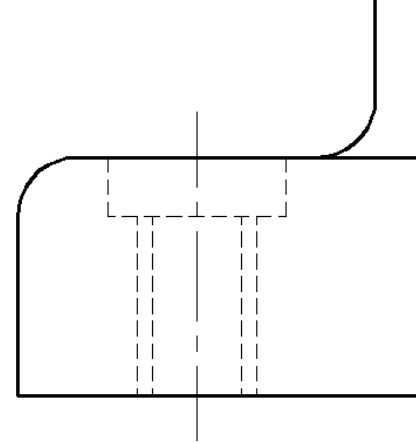
**SIMPLIFIED THREAD SYMBOL**

Creating th

- Threads can be drawn in three ways (schematic, simplified and detailed). Schematic and simplified thread symbols will be used here. Threads are among the few objects that are not drawn "what-you-see", but "symbolically," in mechanical engineering drawings. Threads are specified by "call-outs" or notations. Threads are not modeled in 3D digital models either but are indicated with pictures. The minor diameter for holes and the major diameter for shafts are what machinists care about; the threads are cut with special machines. The "accurate" drawing of threads are not important.

- Use a variety of tools such as Offset (o), Trim (tr), and/or Array (ar, with Rectangular option) to draw the thread symbols.

Ready for dimensioning…

- The three-view drawing is completed. The next step is to place dimensions on the views.
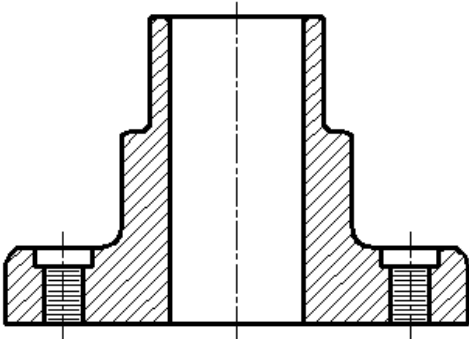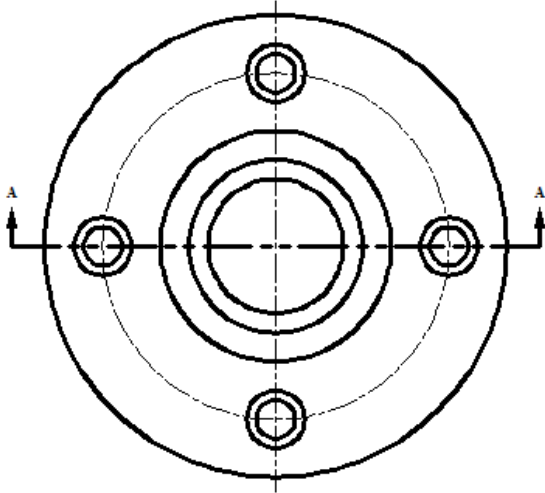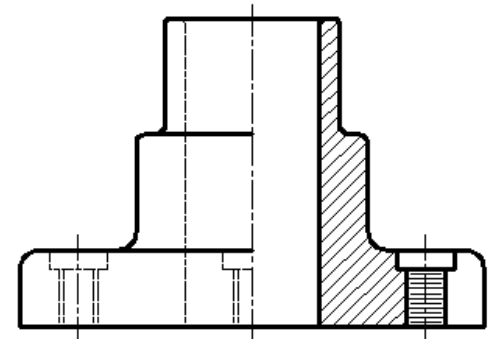
SECTION A-A
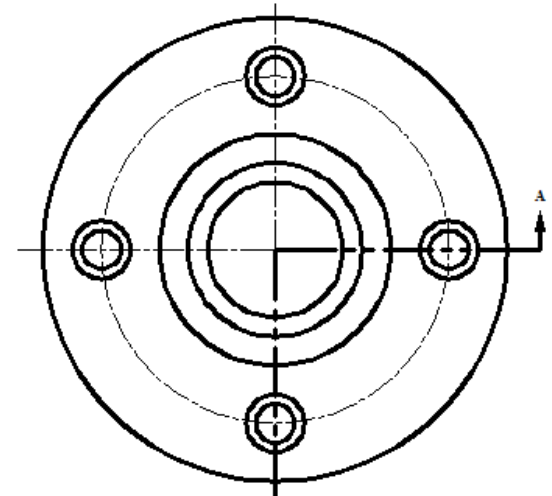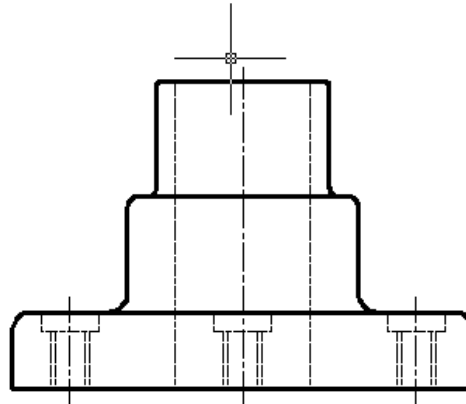
SECTION A-A

Some drafting conventions…

- For a symmetrical object like this one, only two views are needed: the top view (with half-section cutting plane lines), and the front view with a half-section (right) and a half regular view (left).

SECTION A-A

The End