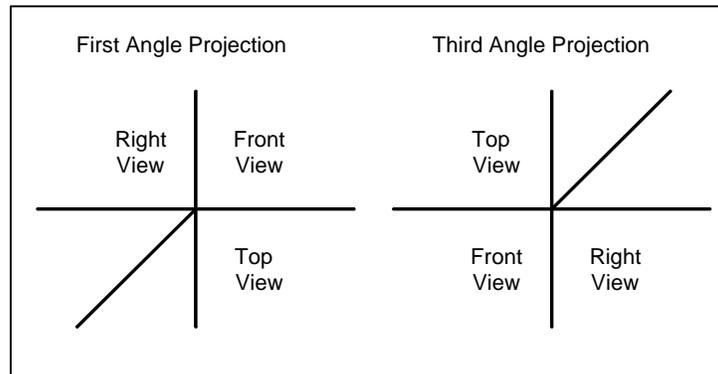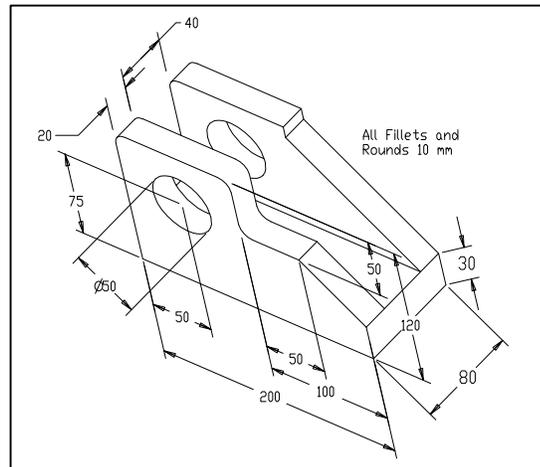# 3 – More Orthographic Drawings

In Chapter 2 we looked at drawing third angle orthographic drawings. We used projection lines to aid in the drawing the TOP and RIGHT views. In this chapter, we will draw first angle drawings using projection lines to assist in drawing all of the views. Our intent is to use the projection lines as fully as possible in order to eliminate the use of coordinates.

We are illustrating the techniques used here with first angle projections. The same techniques can be used just as well with third angle projections.

First and third angle projections only differ in the placement of the views. This difference is shown in the diagram below.

Creating Layers

Before we start, you must create layers for the drawing. We are going to use the same layers that you created last time. They are:

Name	Linetype	Weight
OBJ	Continuous	0.3
HIDE	HIDDEN2	Default
PROJ	Continuous	Default
CENTER	CENTER2	Default
FOLD	Continuous	Default

You may choose any color you want for the lines. Remember that it is frequently better to have a different color for the projection lines than for the other lines in the drawing. This makes it much easier to distinguish projection lines from other types of lines.

Starting the Drawing

Now that you have created all of the layers, switch to the OBJ layer. We will start by drawing the bottom edge of the object on the left side of the pictorial shown above. This first line is horizontal and 200 mm long. Type:

```
Line
Specify first point: 10,10        {Actually we could start the line anywhere and you
                                   could use the line icon and select a starting
                                   location with the mouse}
Specify next point: @200,0        {The line is horizontal and 200 units long}
Specify next point:               {Press enter to end the command}
```

More than likely, you are not seeing all of the line. Some of it may have disappeared off the screen or the line may not be visible at all. This is not a problem. You can always bring the line back onto the screen with the command:

```
zoom
Specify corner of window: all     {We tell AutoCAD we want to see everything
                                   we have drawn. It will rescale the window so
                                   that everything appears}
Regenerating model                {This message indicates that AutoCAD had
                                   to create a new viewing range to display the
                                   picture.}
```
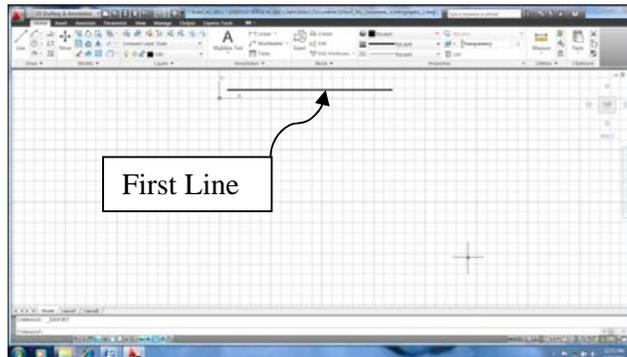
At this point, the line should appear on the screen. It may extend across the entire screen so that you cannot see the end points. If this is the case, place the mouse pointer on the line and roll the wheel towards you to make the line smaller. Continue rolling the wheel towards you until the line is a small as it can be. This is the limit of the AutoCAD regeneration range. To see the line even smaller, you must regenerate the drawing again. Enter the command:

```
regen
Regenerating model                        {AutoCAD creates a new viewing range}
```

Now you can roll the wheel and make the line appear even shorter. You are not actually changing the length of the line. The line is still 200 mm long. The only thing you are changing is how AutoCAD displays the line. You can always regenerate a drawing if you run into a limit while trying to make it larger or smaller on the screen.

Projection Lines

We are going to start drawing projection lines. These lines extend from the line we have drawn in the FRONT view down to the TOP view of the object and over to the RIGHT view. Press down
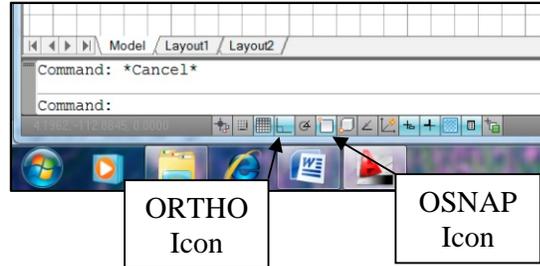

First Line

on the mouse wheel and drag the line to the top of the screen. When you have finished drawing the line and moving it to the top of the screen, your drawing should resemble the one shown above.
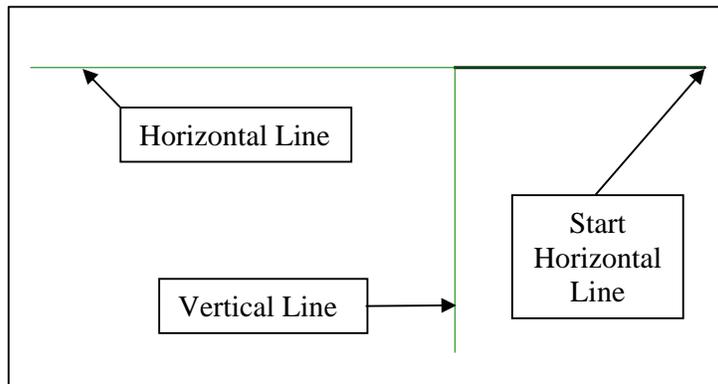
Make sure that ORTHO is on. You can do that with the **ORTHO** command.

```
ortho
Enter mode [ON/OFF] <OFF>: on
```

You can turn ORTHO on by with the command illustrated above or by clicking on the ORTHO icon at the bottom of the AutoCAD window. We want ORTHO on because we are going to draw vertical and horizontal projection lines.

Make sure that object snap is turned on. We want to draw lines with the mouse by snapping to endpoints, intersections, centers, and other features of lines that have been drawn. OSNAP can be turned on by clicking on the icon in the bottom left of the AutoCAD window. OSNAP is on when the button is highlighted. The icon is shown in the figure on the right.
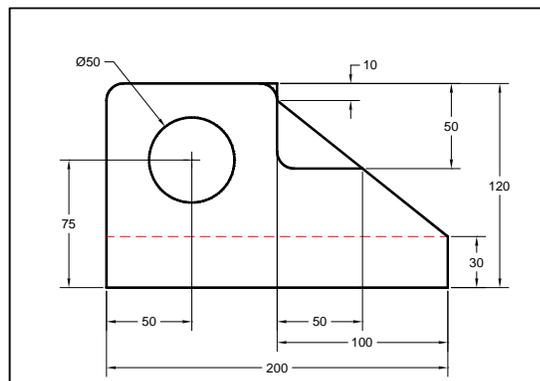
Switch to the PROJ layer and draw a vertical line extending down from the left end of the line you have already drawn. Draw a horizontal line extending from the right end of the line as shown in the drawing above.

The horizontal projection lines should be long enough to extend through the RIGHT view and the vertical lines should be long enough to extend through the TOP view.

The FRONT view of the object we are drawing is shown below. Notice that it has several features that will project to the TOP and RIGHT views. We can quickly create projection lines for these features by copying the projection lines we have already created. We will copy the lines with the **OFFSET** command.

The **OFFSET** command copies a projection line and at the same time

offsets the new line a specified distance from the original. You can use this command to quickly draw both horizontal and vertical projection lines.

We start with the horizontal lines. These lines project to the RIGHT view. The first projection line will be at the top of the object. It is offset 120 mm from the bottom line that we have already drawn.
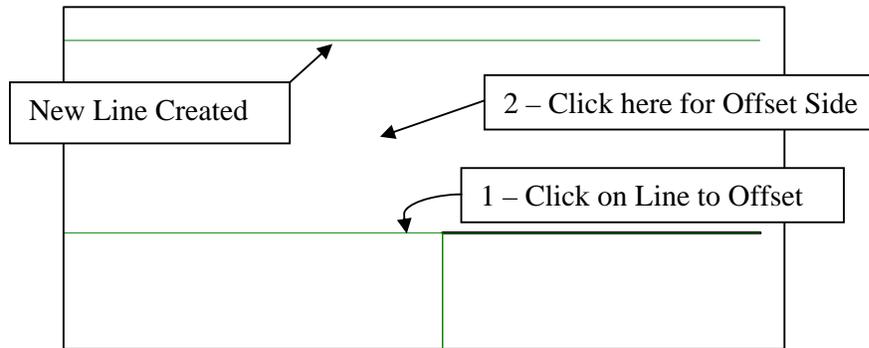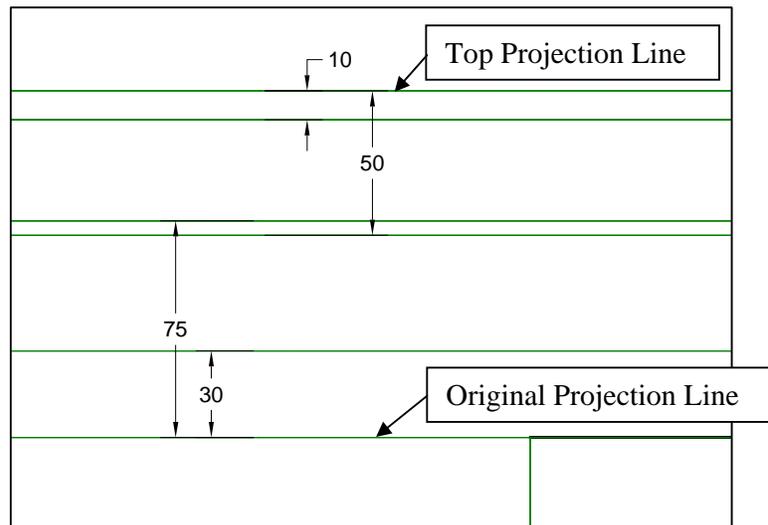
Enter:

```
offset
Specify offset distance: 120          {Offset distance to top of the object}
Select object to offset:              {1 - Click on the horizontal projection line}
Specify point on side to offset:      {2 - Click somewhere above the horizontal
                                      projection line.  This will create the new line
                                      above the existing line}
Select object to offset:              {Press Enter to end the command}
```

The results are shown below.

The other horizontal projection lines can be easily created using the OFFSET command. The diagram below shows the offset distances for new lines. Use this diagram to create the remaining horizontal projection lines. The dimensions show the offset distance between the existing projection line and the new projection line you are creating.
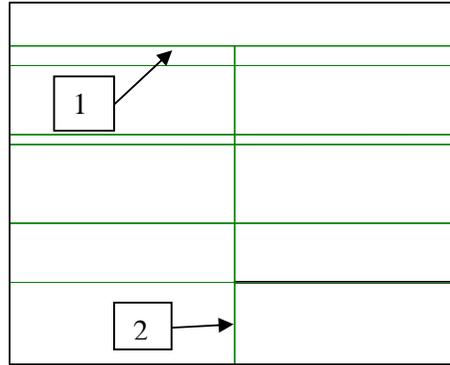
Next, we create the vertical projection lines. We start by extending the vertical projection line that we have already drawn to top horizontal projection line. This extends the projection line through the FRONT view so that it can be used to draw lines and other features in the FRONT view. Enter the following command.
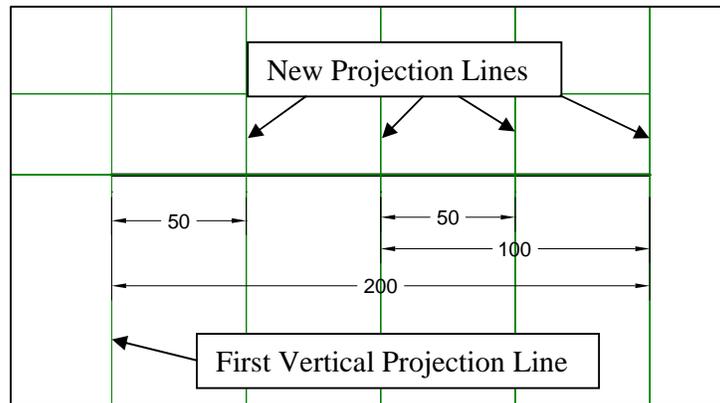
```
extend
Select boundary edges ...

Select objects: 1 found        {1 – Click on the top horizontal projection line}
Select objects:                {Press Enter to end the selection}
Select object to extend:       {2 – Click on the vertical projection line}
Select object to extend:       {Press Enter to end the command}
```

Now you can offset the vertical projection line to create all of the other vertical projection lines. Use the diagram below to assist in creating these lines.
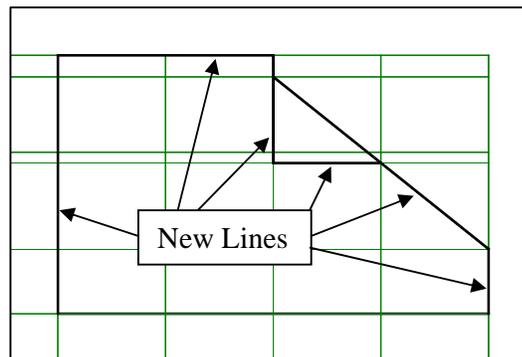
Creating the FRONT View

The FRONT can be created now. Switch to the OBJ layer and use the **LINE** command to draw around the FRONT view. Each corner of the object is at an intersection of a horizontal and vertical projection line so the entire outline of the object can be created by clicking.

The object contains rounds and fillets but do not worry about those at this time. We will create them later. For now, just draw the view with sharp corners. When you have finished, you should have a drawing similar to the one shown on the right.
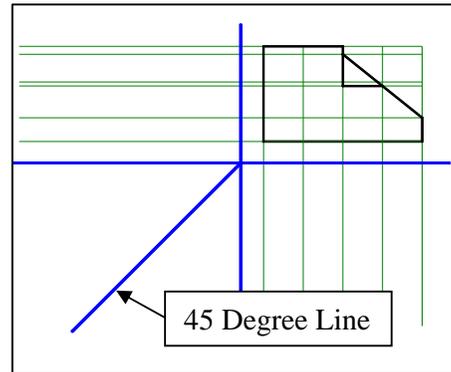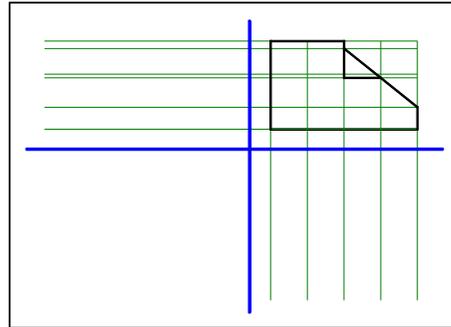
Creating Folding Lines

We can now create the folding lines which separate the FRONT view

from the TOP and RIGHT side views. Switch to the FOLD layer and draw the lines with the mouse. They should extend a little farther than the horizontal and vertical projection lines. Your drawing should look like the one shown at the right.

The folding lines in the drawing are drawn with a heavy line weight. Your lines might be thinner depending upon how you defined them in the FOLD layer. The thickness of the lines is not important because we will not display them in the final drawing.

45 Degree Line

The 45 degree line separating the TOP and RIGHT views can be created with the following commands.
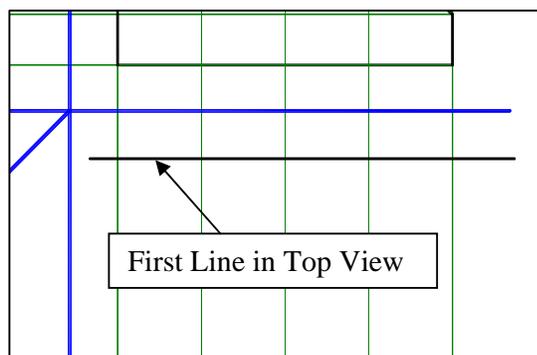
```
line
Specify first point:

Specify next point: @300<-135



Specify next point:
```

{1 - Click at the intersection of the two folding lines}
{Angles are measured from the horizontal axis with positive values in an anticlockwise direction and negative values in a clockwise direction.}

{Press Enter to end the command}

The folding lines with the 45 degree line are shown in the drawing on the right.

First Line in Top View

Creating the TOP View

Start theTOP view by switching to the OBJ layer then drawing a horizontal line below the horizontal folding line. This line represents the back of the object.

Draw the line with the mouse and extend it beyond the left and right vertical projection lines as shown in the drawing on the right.
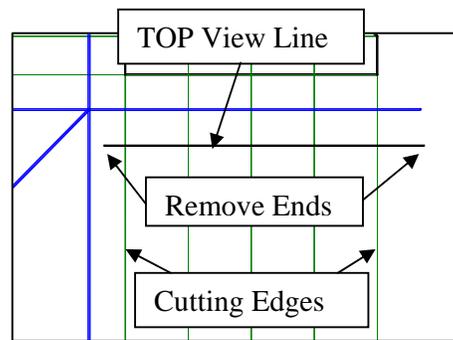
Use the **TRIM** command to trim off the ends of the line that extend beyond the right and left projection lines. The projection lines can be used as the cutting edges for the **TRIM** command.

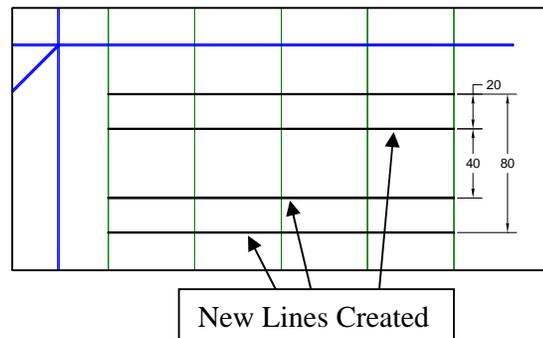The commands for trimming the line are shown on the next page.

```
trim
Select cutting edges ...
Select objects: 1 found          {Click on the right most projection line}
Select objects: 1 found          {Click on the left most projection line}
Select objects:                  {No more cutting edges – press Enter}
Select object to trim:           {Click on the right end of the TOP view line}
Select object to trim:           {Click on the left end of the TOP view line}
Select object to trim:           {Press Enter to end the command}
```

The OFFSET command can be used to create the other horizontal lines in the TOP view. The offset distances are shown in the figure on the right.
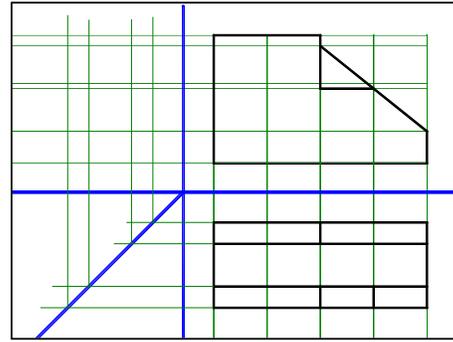
The vertical lines in the view can be drawn with the mouse by clicking on the intersection of the projection lines and the horizontal lines we have just created. The completed view is shown on the right.
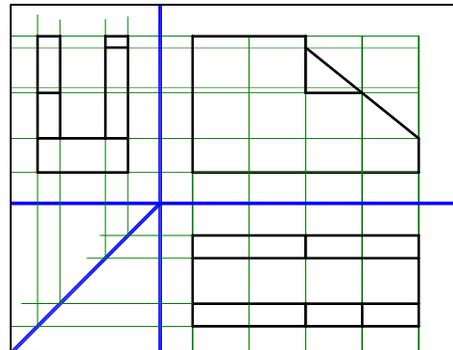
Creating the RIGHT View

The RIGHT view is easily created by projecting horizontal lines from the TOP view to 45 degree folding line. From here the lines are projected up to the TOP view. This is very similar to the work you did in Chapter 2 of this tutorial. After you have completed these projections, your drawing should look like the one on the right.

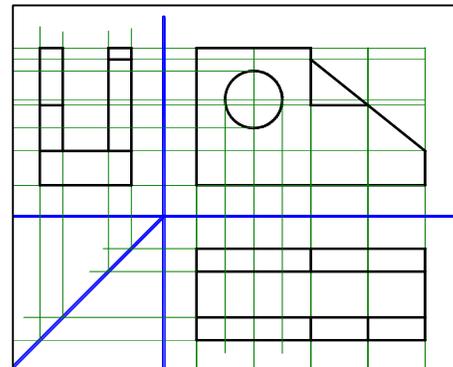Remember to switch to the PROJ layer before drawing more projection lines.

You can now create the lines in the TOP view. Switch back to the OBJ layer and draw the lines with the mouse. The completed TOP view is shown on the right.
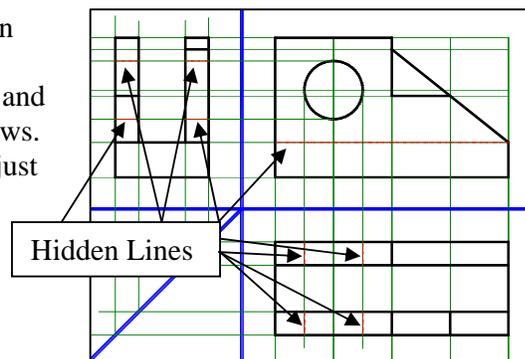
Hole in the FRONT View

The hole in the FRONT view can be drawn by adding a circle in the view. It has a diameter of 50 mm. There is a projection line crossing at the center of the circle so it is very easy locate the center with the mouse.

After the circle is drawn, extend projection lines from the top and bottom to the RIGHT view and from the right and left to the TOP view. The drawing with these projection lines is shown on the right.
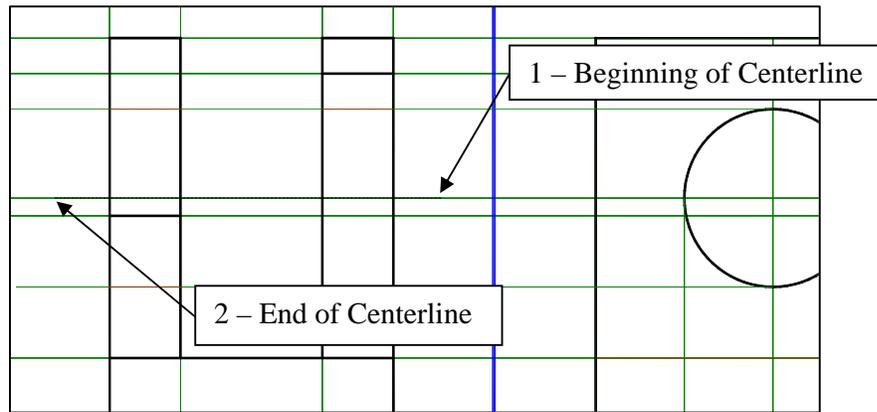
The next step is to draw the hidden lines produced by the hole (circle) in the FRONT view. Switch to the HIDE layer and draw the hidden lines in each of these views. While you are at it, draw the hidden line just above the base in the FRONT view also.

Hidden Lines

The holes in the RIGHT and TOP views should have centerlines. These centerlines should extend beyond the outside edge of the views. We will use the projection lines going through the centers of the holes and the **near** attribute to place the centerlines on these projection lines and to start and end the centerlines beyond the edges of the object.

You must switch the current layer to CENTER before drawing the centerlines.

1 – Beginning of Centerline

2 – End of Centerline

```
line
Specify first point: near        {Use near to put the point on the projection line}
To                               {1 - Click on the projection line where you want the
                                 centerline to begin}
Specify next point: near         {Use near for the end point of the centerline}
To                               {2 - Click on the projection line where you want the
                                 centerline to end}
Specify next point:              {Press Enter to end the command}
```
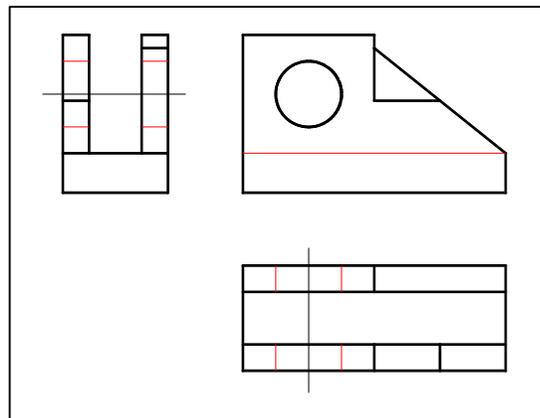
When you have finished the centerline in the RIGHT view, continue on and draw the centerline in the TOP view using the same technique.

Adjusting the Linetype

We have completed most of the drawing, it is time for us to turn off the projection and folding lines. Click on the layer control box and then click on the light bulb icons for the PROJ and FOLD layers to turn them off. Your picture should look something like the one shown on the right.
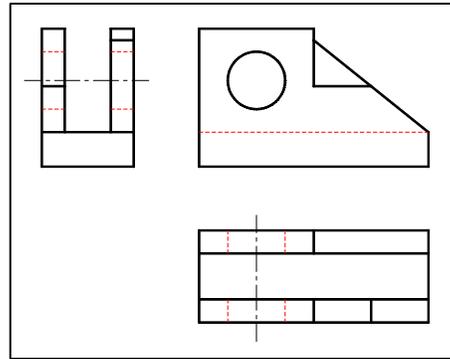
Frequently when we use different linetypes, the dashes do not show. The reason may be that the scale of the drawing is such that the entire line is only one dash long or it may be that the dashes are so small they run together. The picture shows this latter case. The dashes are too small and they run together. We can correct that with the **LTSCALE** command.
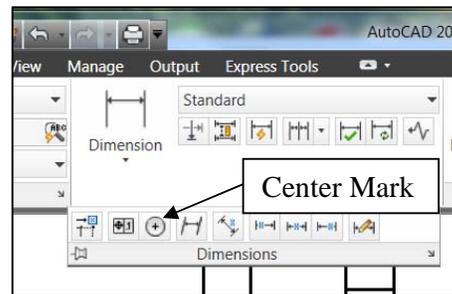
```
ltscale
Enter new linetype scale factor <1.0000>: 30
```

In this case, we are entering a scale factor of 30. This factor makes the dashes 30 times longer than they were originally. That will work for this drawing but it will not work for all drawings. If the dashes are not showing, or if they are too small, try entering a scaling larger than 1. You might have to try several different scaling factors to get the linetypes to appear correctly. If the dashes are too large, enter a scaling factor smaller than 1. The corrected picture is shown on the right.

At this point, we do not have centerlines for the hole in the FRONT view. These should be drawn by placing a small + in the center of the circle. In the previous lesson, we used the "center mark" icon to create this +. The menu showing this icon is shown in the picture on the right.
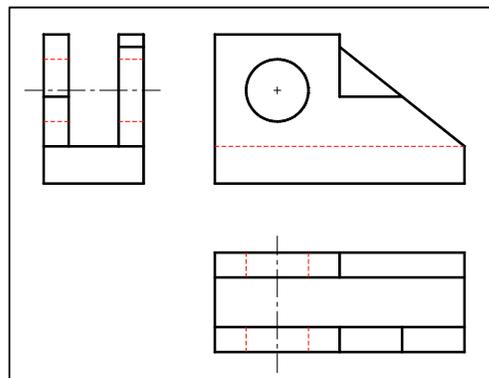
Click on the "center mark" icon and then click on the circle in the FRONT view. You may notice that the center mark is far too small or that nothing happened at all. We need to change the size of the center mark and this can be done with the **DIMSCALE** command. The **DIMSCALE** command changes the scale factor for dimensions. Enter the command:

```
dimscale
Enter new value for DIMSCALE <1.0000>: 30
```
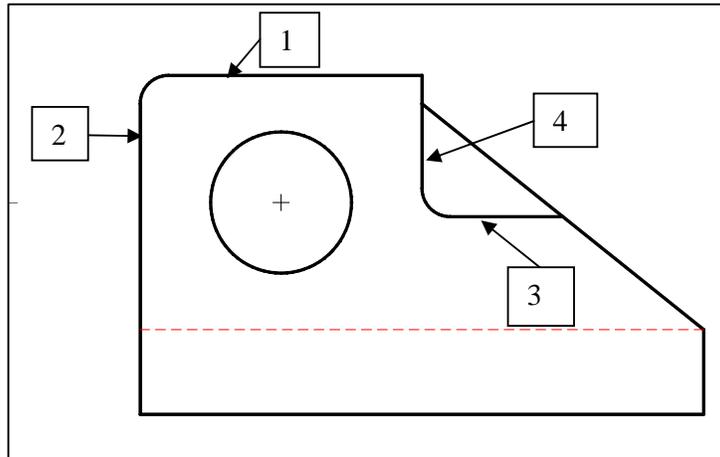
In this case we are making the dimensions 30 times larger than they currently are. You will have to redraw the center mark. It is not dynamic so once it is drawn; its size will not change even though you change the dimension scaling factor.

The center mark is shown in the drawing on the right.

Adding Fillets

The drawing is complete except for fillets and rounds in some of the corners. We could add those with the arc command but there is a much easier method. AutoCAD has a **FILLET** command that makes creating fillets and rounds a snap.

Enter the command:

```
fillet
Current settings: Mode=TRIM,Radius=0.0        {The radius is zero which is too
                                              small}
Select first object or
      [Undo/Radius/Trim/Multiple]: r          {Enter "r" to change the radius
                                              of the fillet}
Specify fillet radius <0.0000>: 10            {The new radius is 10}
Select first object:                          {1 – Click on line}
Select second object:                         {2 – Click on line}

Command:                              {Press Enter to start the fillet command again}

FILLET
Current settings: Mode=TRIM,Radius=10.0
Select first object:                          {3 – Click on line}
Select second object:                         {4 – Click on line}
```
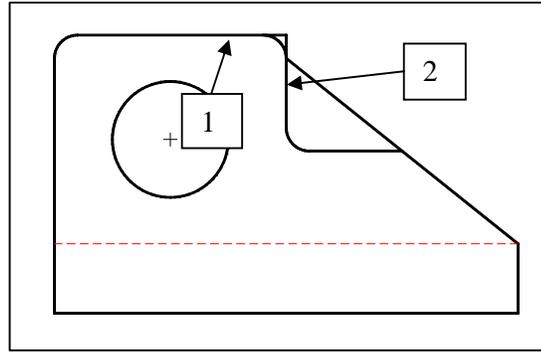
The FRONT view with the two new fillets is shown above.

The right side of the FRONT view also has a fillet on the front surface but not on the back. We can again use the FILLET command to create this fillet but this time we will not have the program trim the lines. Enter:


```
fillet
Current settings: Mode=TRIM,Radius=10.0
Select first object or
    [Undo/Radius/Trim/Multiple]: t      {We want to change how the
                                         lines are trimmed after inserting
                                         the fillet}

Enter Trim mode option
    [Trim/No trim] <Trim>: n            {Turn off trimming}
Select first object:                    {1 – Click on line}
Select second object:                   {2 – Click on Line}
```

The figure above shows the locations for references to line 1 and 2.

This completes our work. The finished drawing is shown below.