

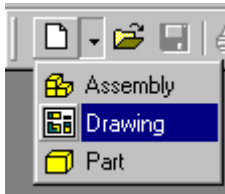
## Lesson 20

### Orthographic Views

#### Learning Objectives

At the conclusion of this lesson, the user will have gained mastery of the Drawing Management tools.

Open a new drawing file.

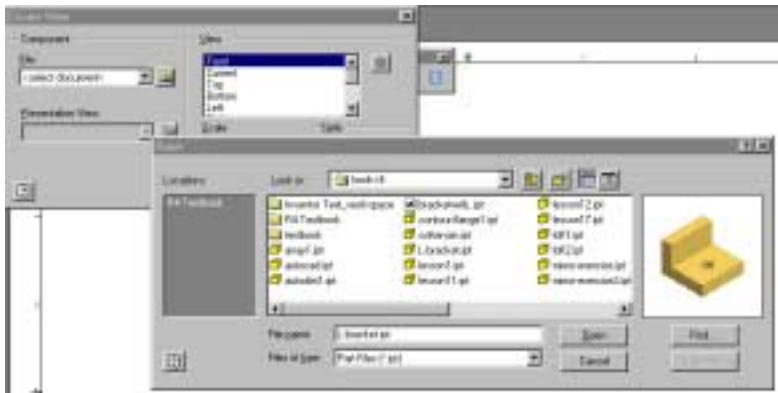


#### Create a Base View

Inventor automatically assumes that the first view placed on a drawing sheet is the Base or primary view. To create a base view, issue the 'Create View' command.



Right click anywhere inside the drawing sheet and select 'Create View'.



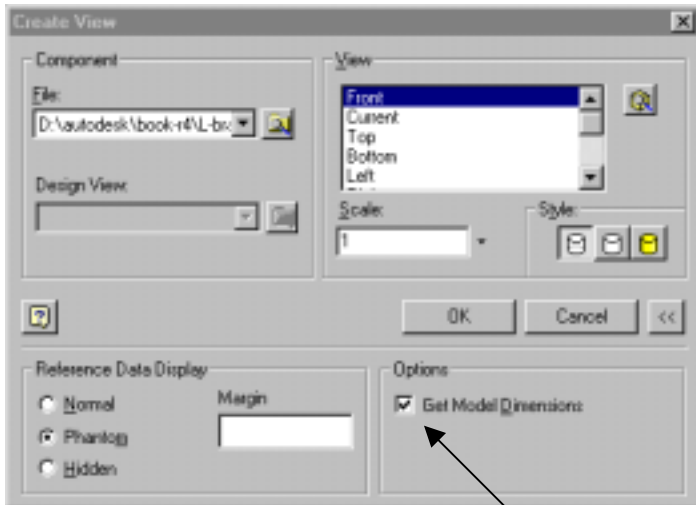
Verify that the 'Front' view is highlighted in the dialog box.

Select 'Hidden' as the style as shown.

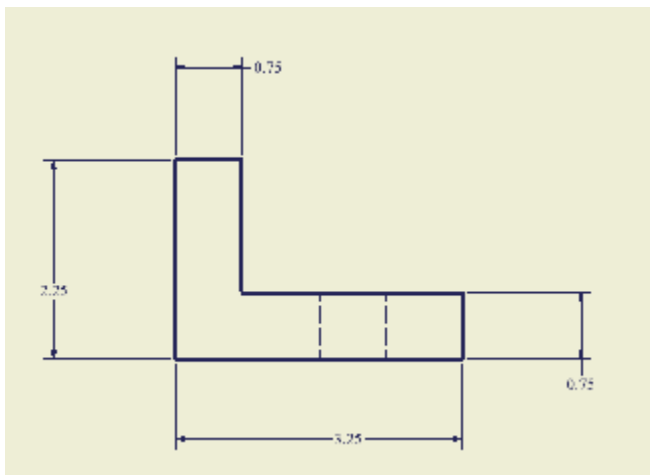
Set the Scale to 1 as shown.

Enable Dimensions by placing a check in the box next to Dimensions in the lower right corner.

Click on the file browser button. Locate the file called 'L-bracket.ipt' and press 'Open'.

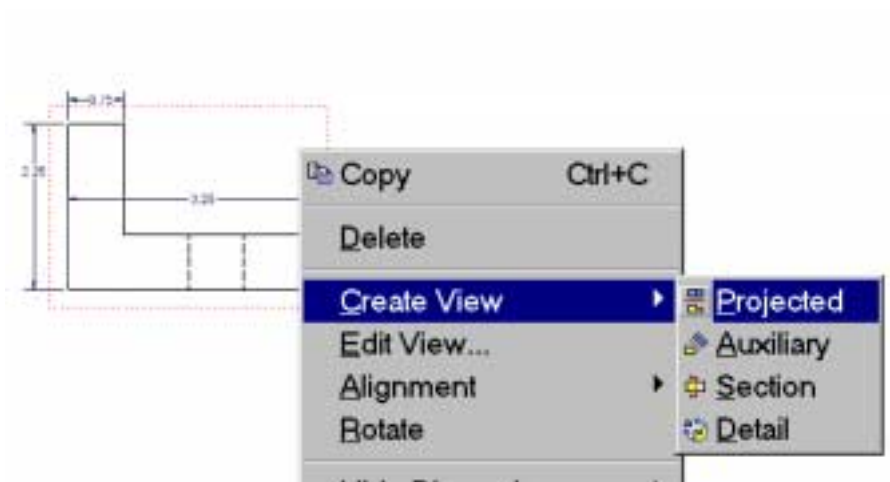


Make sure the 'Get Model Dimensions' is enabled.

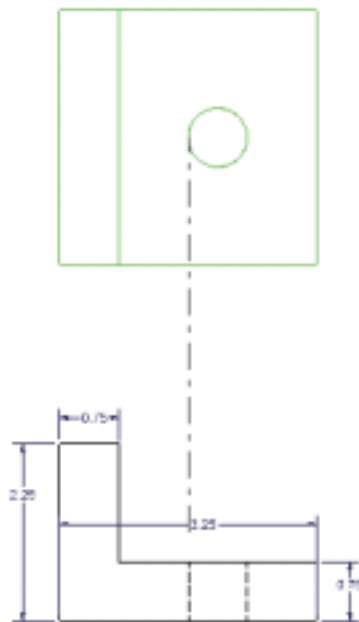


Place the view by picking in the lower left side of the drawing sheet.  
Note that the dimensions appear with the view.

### Create a Projected View



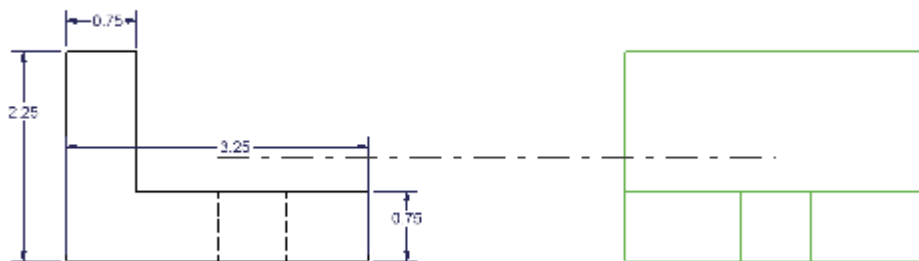
Highlight the base view by picking on it. (A red dotted rectangle should appear as shown)  
Right click the mouse to bring up the menu. Select Create View->Projected.



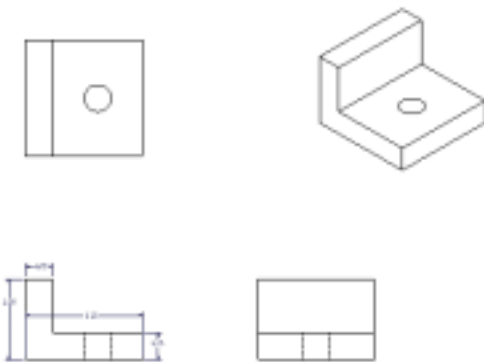
Pick the location for the top view.

Create the right side view

Move the mouse to the right of the base view.



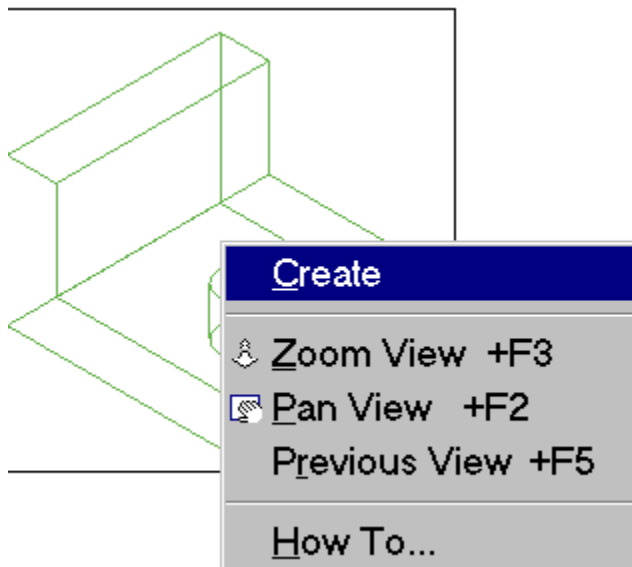
Pick to place the right side view.



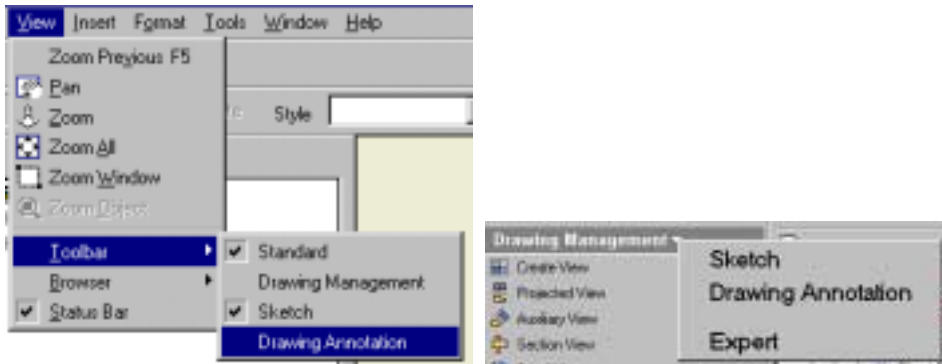
### Place isometric view

Move the mouse above the right side view and pick to place the isometric view.

Right click the mouse and select 'Create' to set the views in place and finish.



Bring up the Drawing Annotation toolbar.



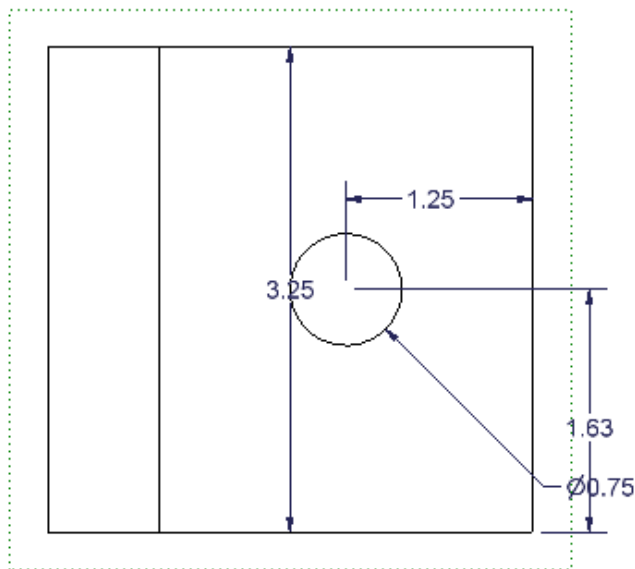
To activate the Drawing Annotation Toolbar:

- ◆ Go to View->Toolbar->Drawing Annotation and select.
- ◆ Left pick on the top bar of the panel bar toolbar and select Drawing Annotation

## Add dimensions to a View

Use 'Zoom Window' to zoom into the top view.

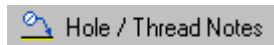
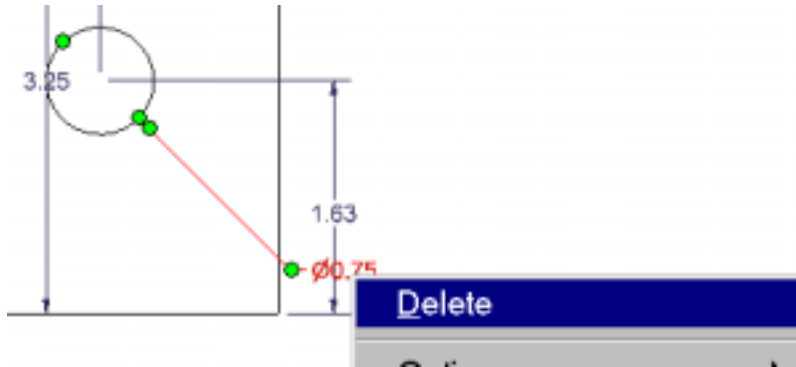
Pick on the view so that it highlights. Right click the mouse and select 'Get Model Dimensions'.



The dimensions will appear as shown.

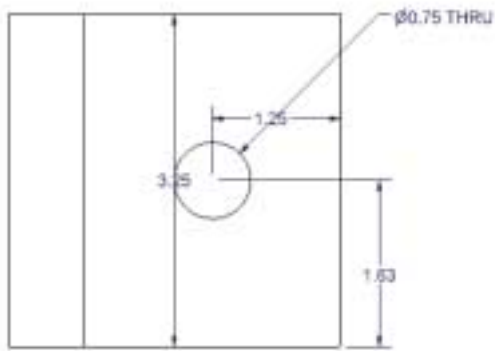
## Deleting Dimensions

Select the hole diameter dimension. Right click and select delete.



## Create Hole/Thread Notes

Select the Hole/Thread Notes tool.



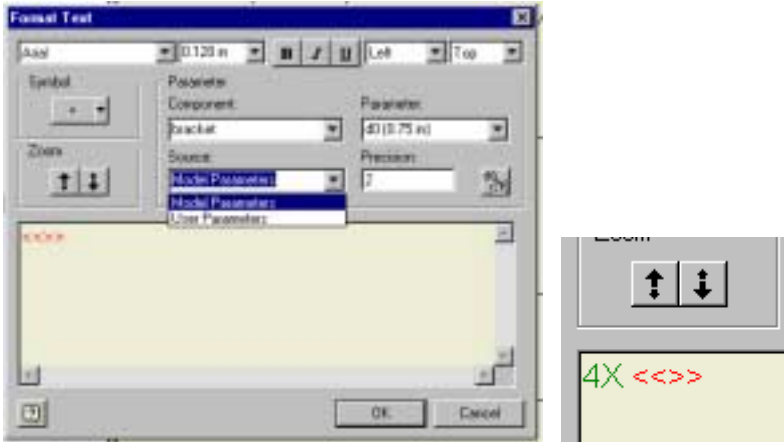
Pick the hole where you want to add the annotation. Then drag out to place the leader shoulder. The hole note is automatically created using the hole definition. Right click and select 'Done' when the text is located correctly.

## Editing A Hole Note

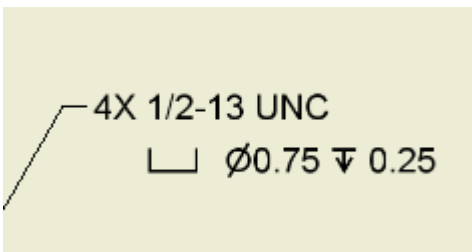
If for some reason, you don't like the appearance of a hole note, it can be modified.



Make sure the Select button is active. Then pick the hole note and right click. Select 'Text'.



The <<> in the edit box indicates the default value for the text. Simply delete the default and type in the desired note or add text.

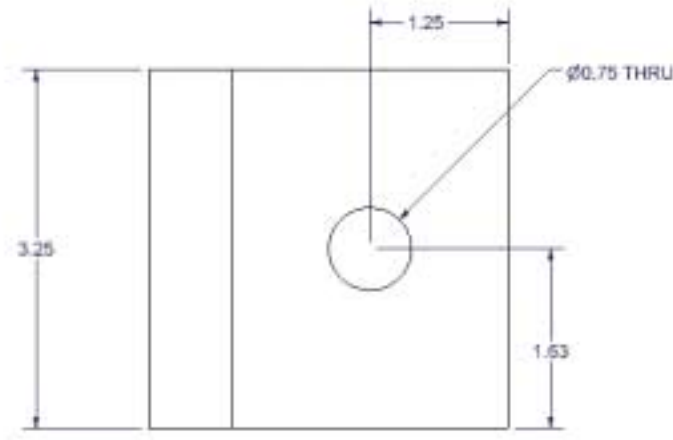


**TIPS:** If you accidentally delete the default value, simply type <<> in the edit box and the default value will be restored.



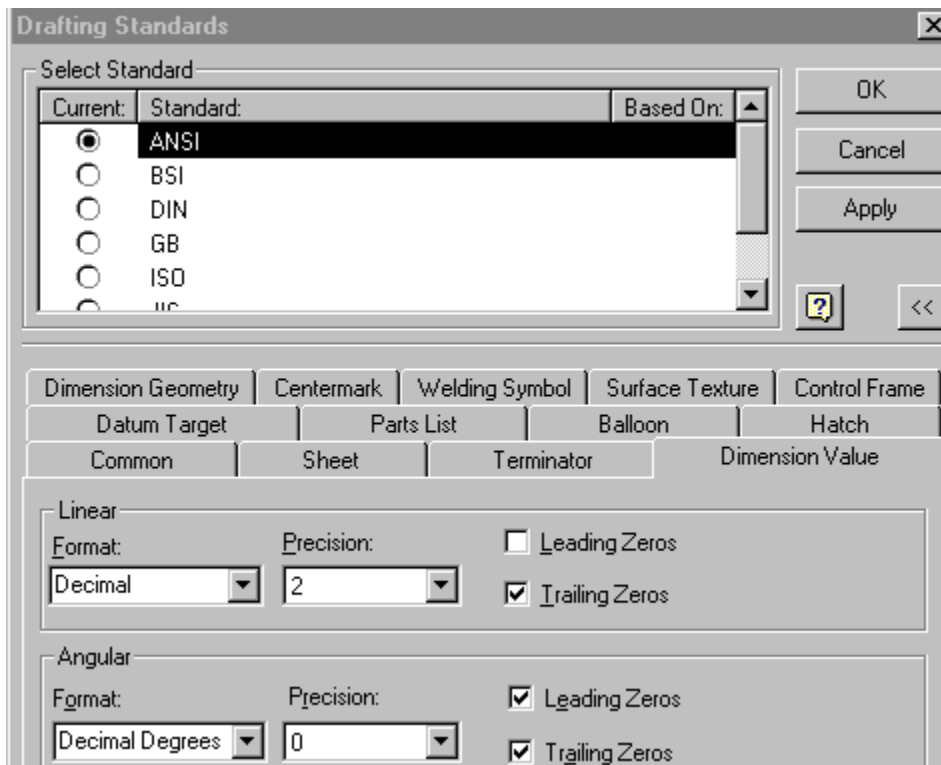
## Move Dimensions

To relocate a dimension, select it and grab one of the green dots. These act like grips. Drag the dimension to the desired location.



## Dimension Standards

The format of dimensions, such as precision and font, is set in the Drafting Standards. To access, select Format->Standards from the menu.



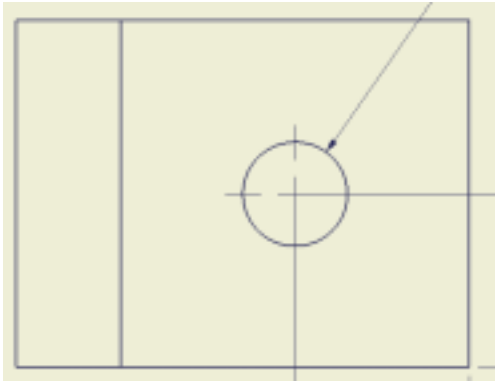
The Common tab controls the font of dimensions.

## Create Center Mark

Add a center mark to the circle.



Select the Center Mark tool from the Drawing Annotation toolbar.  
Then pick the circle.



Center mark has now been added.

## Create Centerline

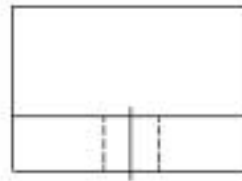
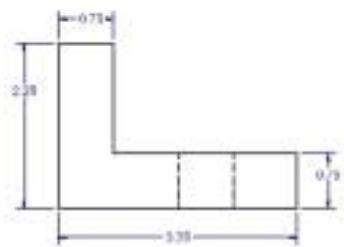
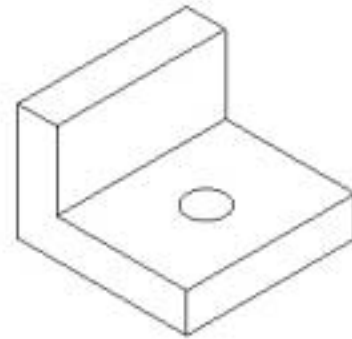
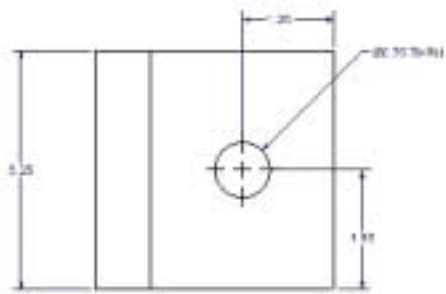


Let's create a centerline for the hole shown in the right side view.

The Centerline tool is located on the Drawing Annotation toolbar under the Center mark drop down.

Select the Centerline tool. Select the top horizontal line and then the bottom horizontal line of the bracket. Right click the mouse and select 'Create'.





Our completed drawing.

Save the drawing as 'ortho1.idw'.

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