

## Inventor Lecture #3

### Reading Assignment:

Read the following in Parametric Modeling with Autodesk Inventor 2009 by Randy Shih:  
Chapter 8

### Lecture Outline:

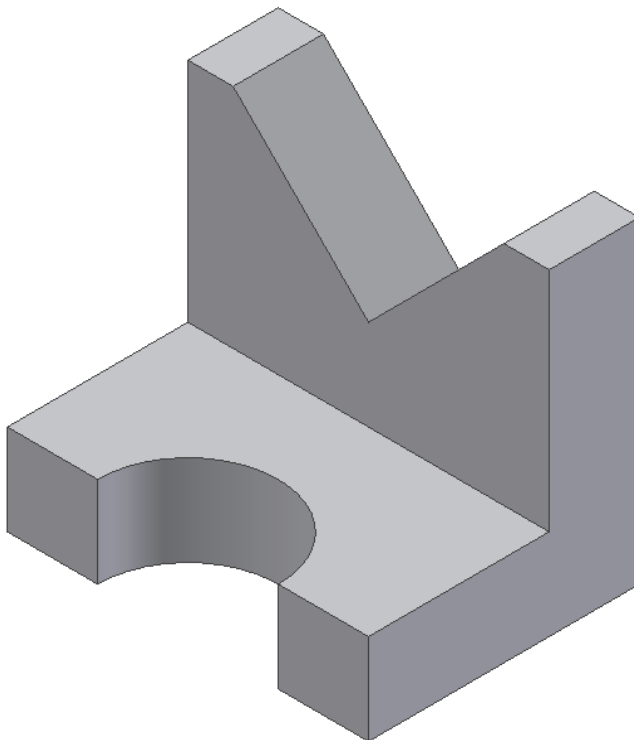
#### Drawing Files

So far we have printed directly from sketches or from the model view of a part. A better approach is to create a *drawing file* in Inventor. From the drawing file we can:

- Insert a part (a specified view)
- Create other views (top, front, side, isometric, auxiliary, section, ...)
- Annotate the drawing (dimensions, centerlines, notes, etc)
- Change dimension styles
- Scale the drawings (our models are always full size)
- Select/modify a title page

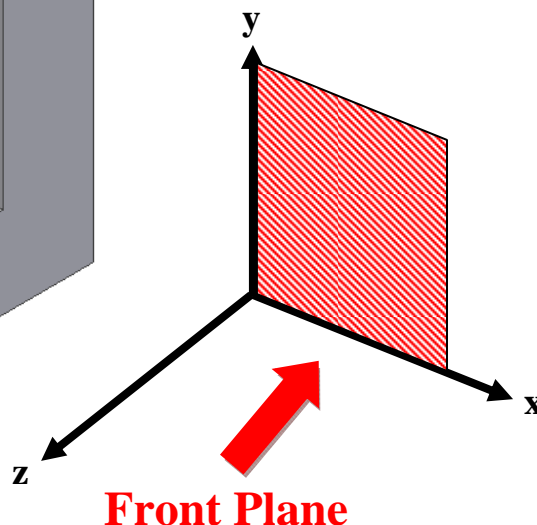
#### Example:

- 1) Create a part (such as Prob 4 on sheet A-2 (page D-23) in Engineering Graphics Text and Workbook, Series 1.2, by Craig & Craig). Use the scale 1 square = 0.25”.

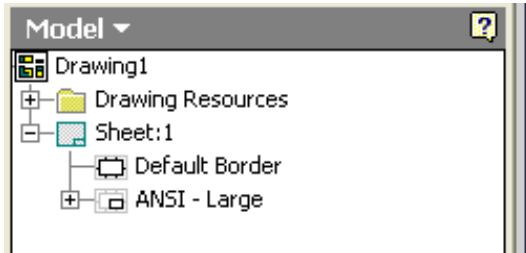


Note: It is often easiest if you draw an object so that what you think of as the front view agrees with what Inventor thinks of as the front view. Note that:

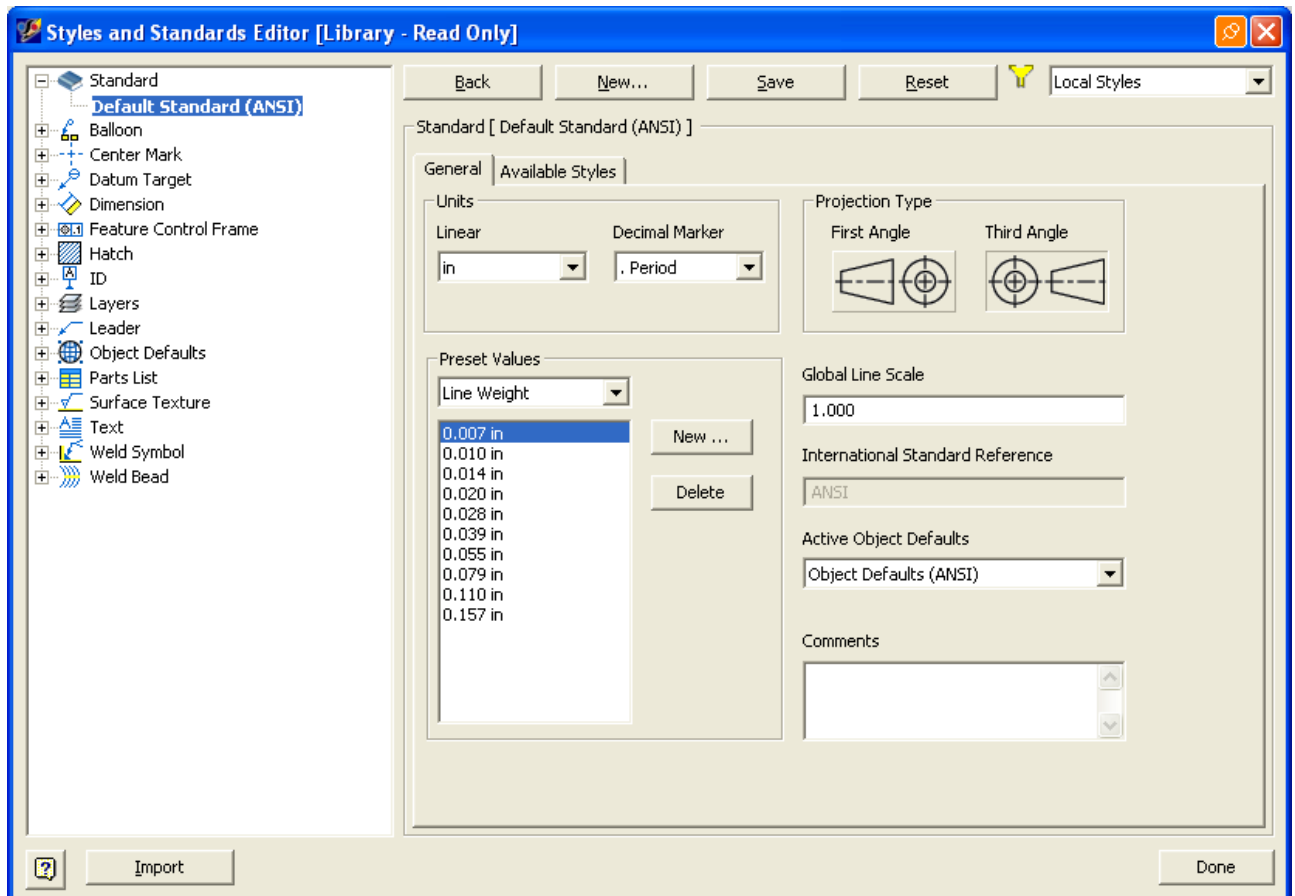
**The default sketch plane in Inventor is the front view.**



2. Create a drawing file. Use File – New – Standard.idw
3. Note that Inventor inserted a default drawing sheet (Sheet:1). It can be seen in the Browser.

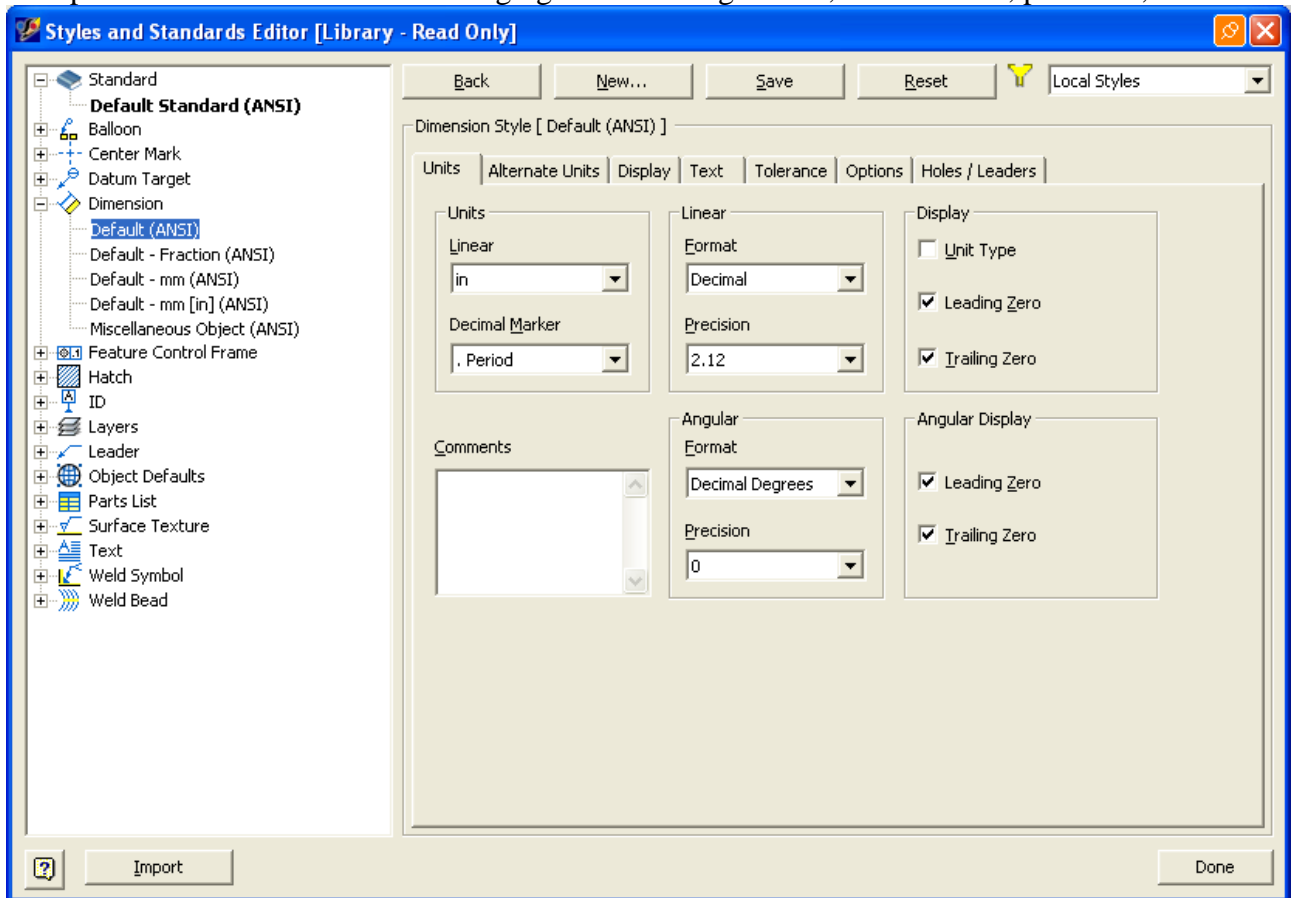


4. The default sheet size is C (17" x 22"). Right-click on Sheet:1 in the Browser and select **Edit Sheet**. Change it to an **A-size sheet** and select **Portrait** for the orientation as shown below.
  - a. Select Format – Active Standard from the main menu
  - b. Select Format – Active Standard from the main menu
5. Select **Format – Active Standard** from the main menu to modify the **Drawing Sheet Format**. Although we will use the default settings for now, look at various settings for future reference:
  - a. Be sure that Default Standard (ANSI) is selected.
6. Select **Format – Styles Editor** from the main menu to modify **Dimension Styles**. Although we will use the default settings for now, look at various settings for future reference:
  - a. Double-click on Default Standard (ANSI) in the Styles and Standards Editor dialog box. You should now see a screen like the one below.



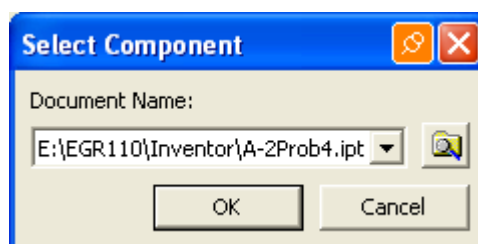
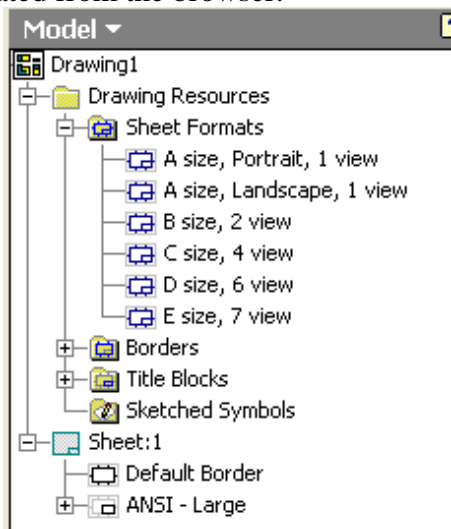
- b. Confirm that **Third Angle Projection** is selected in the screen above.

- c. Select **Dimension** on the left of the screen and select **Default (ANSI)**. Note that many options are available here for changing dimensioning format, such as units, precision, etc.

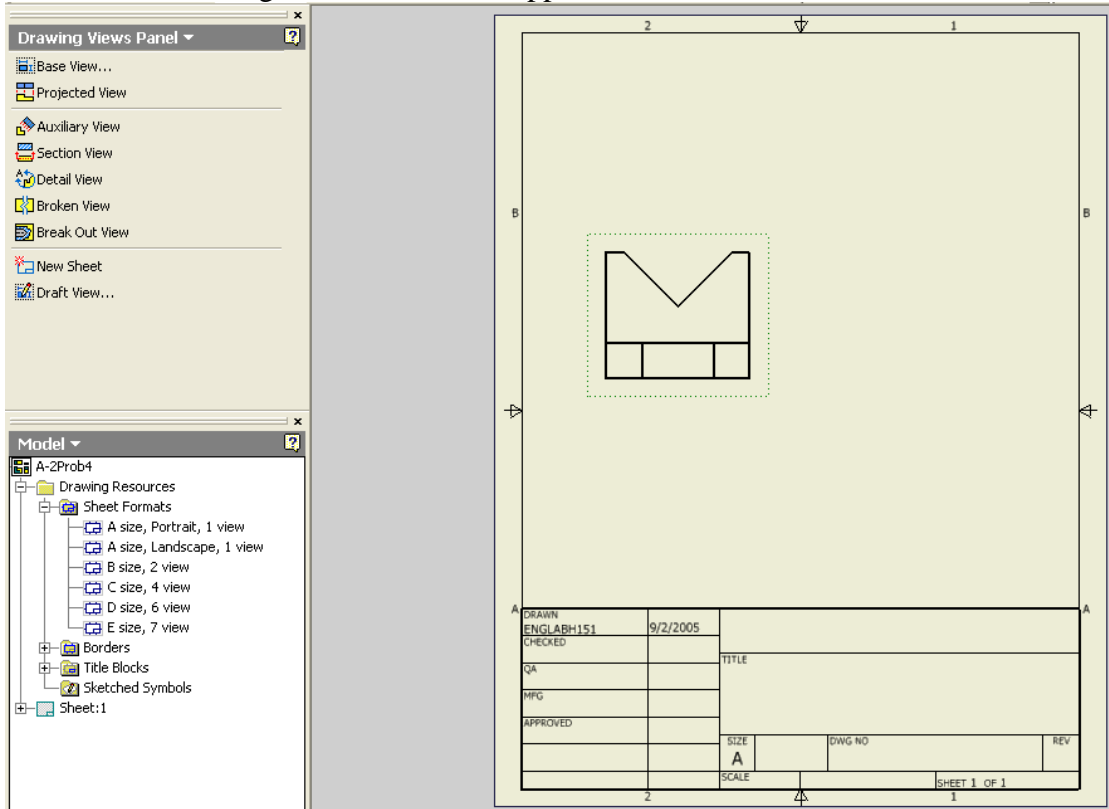


- d. Investigate some of the features available under the tabs above for Display, Text, Tolerance, etc.

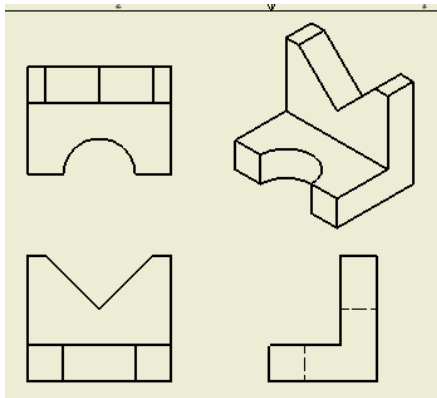
7. Inside the Drawing Browser, expand the **Drawing Resources** (click on +) and expand the **Sheet Format**. The browser should appear as shown below. Try different options (C size - 4 views, B size - 2 views). As you select each one, you will be prompted to select a component (part) as shown below. Finish with **A size - Portrait - 1 view**. You can delete any other sheets that you created from the browser.



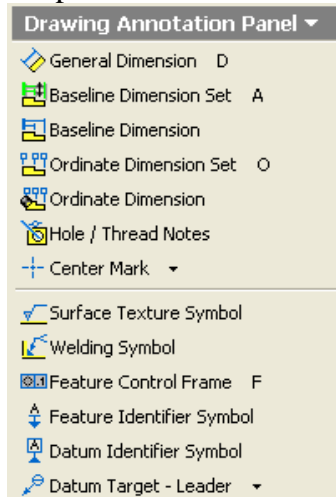
8. Your A-size drawing sheet should now appear with one view as shown below.



9. Right click on the view and select **Delete View** (sometimes the default view is not the view that you want so you may want to delete it and start with a new base view).
10. Select **Base View** from the Drawing Views Panel. Select the desired file and try different views for the desired base view (often the front view). Select OK to create the view.
11. Select **Projected View** from the Drawing Views Panel. This can be used to create orthogonal and isometric views. All views are projected from the base view, so it is important to carefully select your base view. Drag the mouse over the base view in the direction of the desired projected view. Right-click and select Create to create the view. You can also drag several views and then select Create once to create them all. Note that hidden and center lines are generated after you select Create. Produce a drawing with front, top, right side, and isometric views as shown below. Try repositioning views with the mouse.

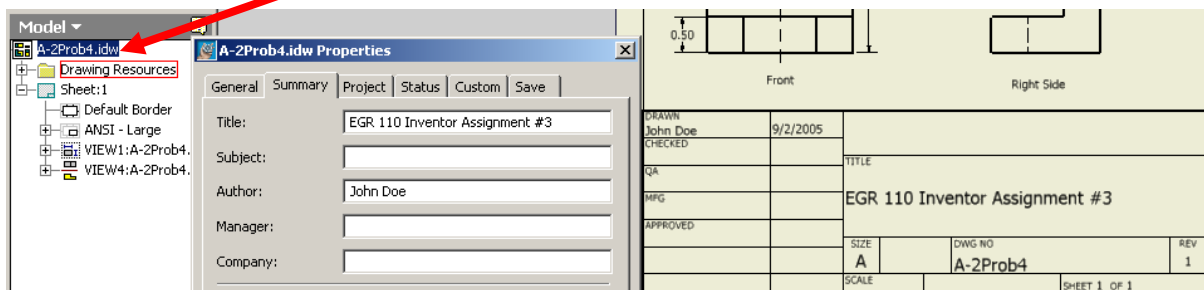


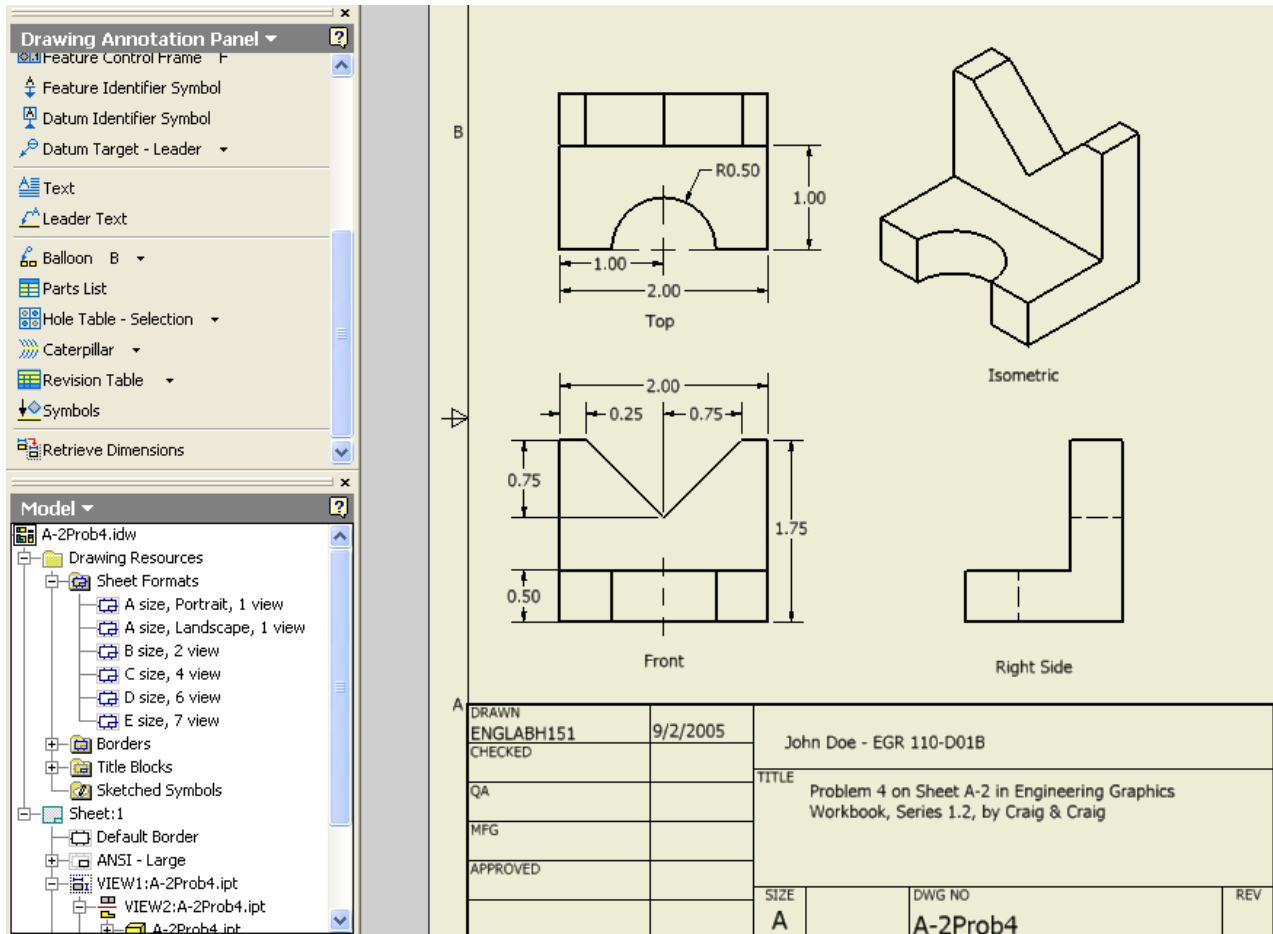
12. **Adding Dimensions.** By default, no dimensions are shown in the views created in the drawing file. To add dimensions:
- Click on the arrow next to the **Drawing Views Panel** and select the **Drawing Annotation Panel**. The panel should now look as follows:



- Right-click over one of the views and select **Retrieve Dimensions**.
- Choose **Select Parts** in the Retrieve Dimensions dialog box.
- Choose **Select Dimensions** in the dialog box and all available dimensions should now appear in the view. Pick the desired dimensions with the mouse.
- Select Apply to add the selected dimensions to the view.
- Use **Select View** to pick another view and use the same procedure to add dimensions to the remaining views. Note that some thought should be put into what dimensions to display.
- Select **Cancel** once all of the desired dimensions have been added to each view.
- Editing dimensions.** Try editing some of the dimensions. They can be easily dragged to new locations and the dimension format can be changed by right-clicking on the dimension.
- Adding center marks.** Select Center Mark from the Drawing Annotation window. Select the arc or circle to which a center mark is to be added.
- Adding center lines.** Select the arrow next to Center Mark from the Drawing Annotation window and select Center Line Bisector. Pick the two lines that the center line will run between.
- Title Block** – Use Zoom and Pan to zoom in on the title block area. Select Text from the Drawing Annotation Panel to add text to the title block and to label each view. Right click on any text box to edit it. Note that you can also add text to the boxes in the title block by right-clicking on the icon at the top of the browser and selecting **Properties** (see below).

Right-click on the icon next to the drawing filename and select Properties





### Re-opening a drawing file

When drawing files are re-opened, the page will often appear blank. To see the views, dimensions, etc., right-click on the Sheet (Sheet 1, for example) in the browser and select *Activate*.

### Dimensioning style

Good style in dimensioning will be covered in detail a little later in the course, but for now a few useful rules are presented below:

- Keep dimensions aligned as much as possible
- Neatly space the dimensions
- Avoid inside dimensions
- Avoid driven (redundant) dimensions
- Center the dimension text within the extension lines
- Specify diameter (using  $\phi$ ) for holes and radius (using R) for arcs
- Label views (neatly centered under each view)
- Always include the scale for the drawing
- Use centerlines where appropriate.