Benha Faculty of Engineering
Mechanical Engineering Department

M1382 : **Computer Aided Design CAD**

First Semester 2018, Y3

Lecture No. 06

Presented by:
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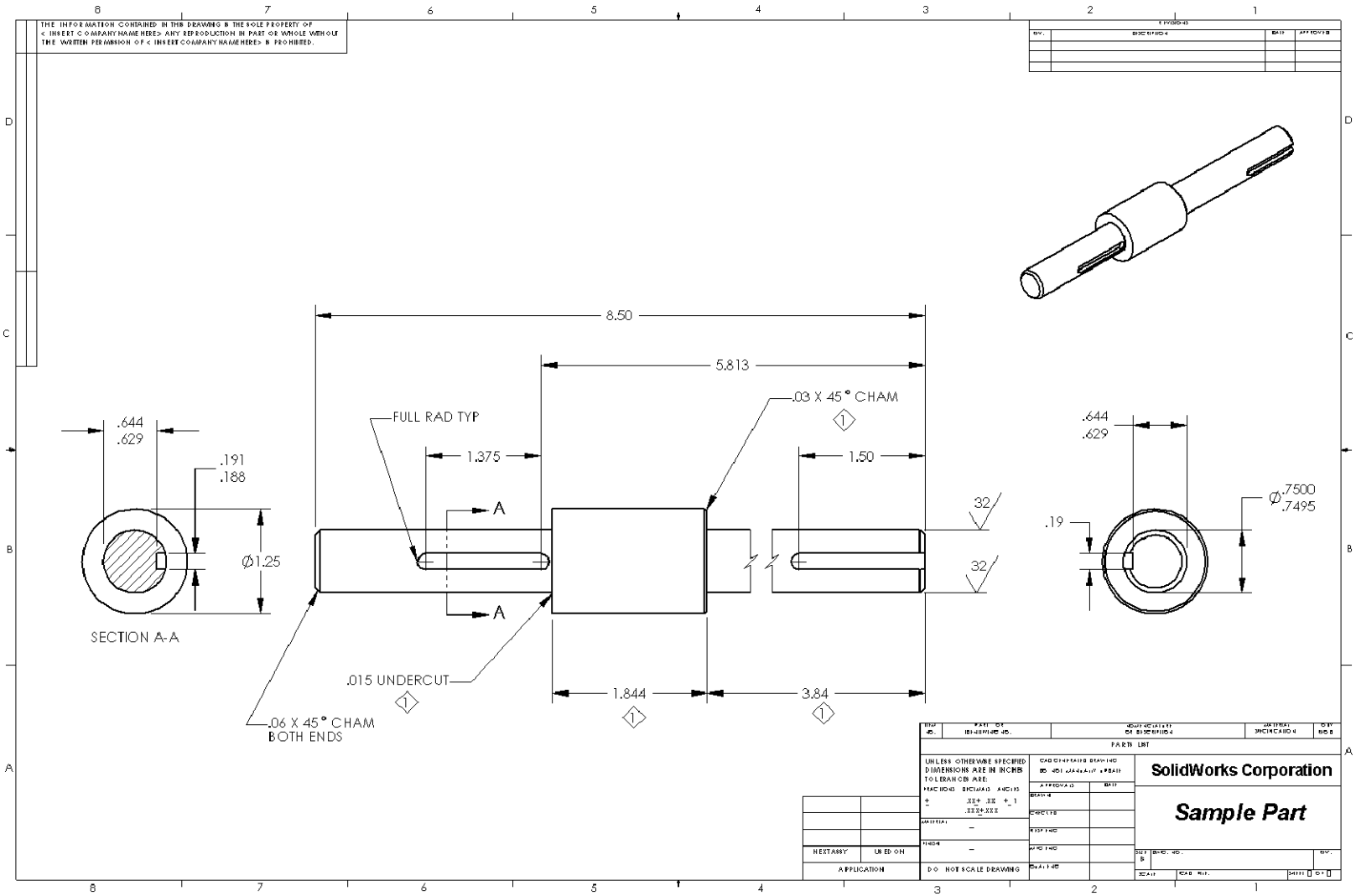
Week	Topics
1	Introduction
2	Introduction to CAD (Solid Modeling)
3	Part modeling
4	Finite element analysis (FEA)
5	Parts assembly using SolidWorks
6	**Basic concepts of engineering drafting**
7	Linear Static Analysis
8	Adaptive Analysis and Mesh Control
9	Modal Analysis
10	Design Optimization
11	Case study 1
12	Case study 2
13	Co-simulation SolidWorks and ADMS software
14	Project Discussion

Engineering Drawings

- **Drawings communicate three things about the objects they represent:**
  - **Shape – *Views* communicate the shape of an object.**
  - **Size – *Dimensions* communicate the size of an object.**
  - **Other information – *Notes* communicate non-graphic information about manufacturing processes such as drill, ream, bore, paint, plate, grind, heat treat, remove burrs, and so forth.**

Sample Engineering Drawing

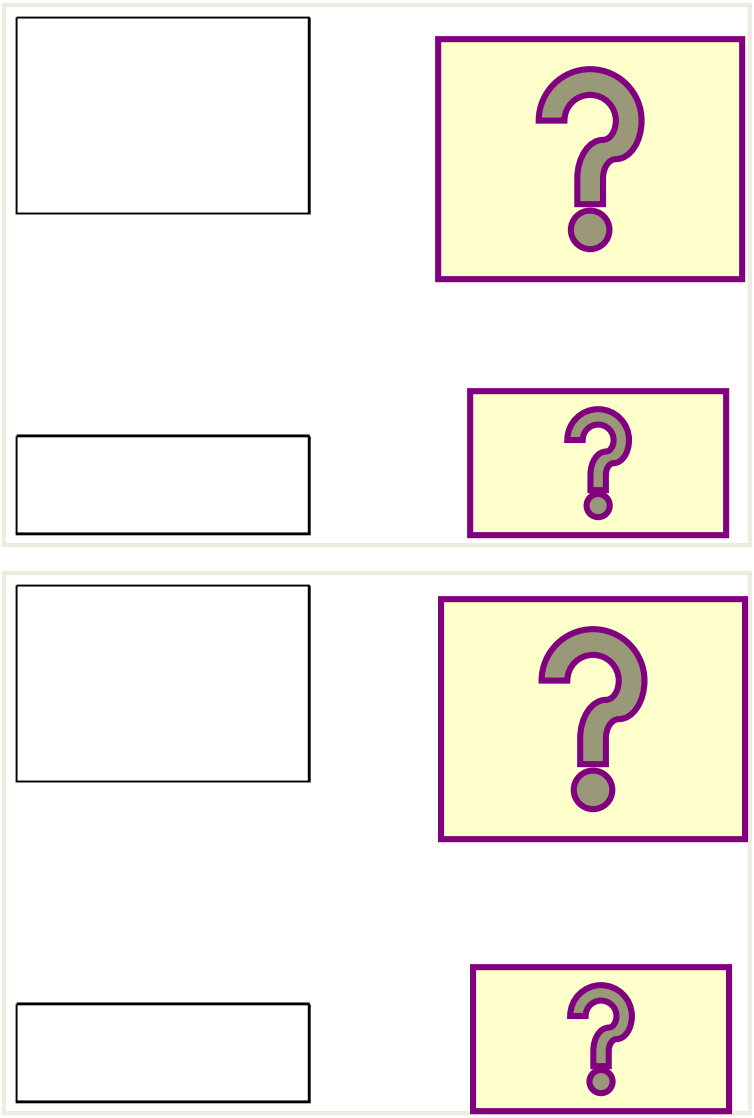


General Drawing Rules – Views

- **The general characteristics of an object will determine what views are required to describe its shape.**
- **Most objects can be described using three properly selected views.**
  - **Sometimes you can use fewer.**
  - **However, sometimes more are needed.**

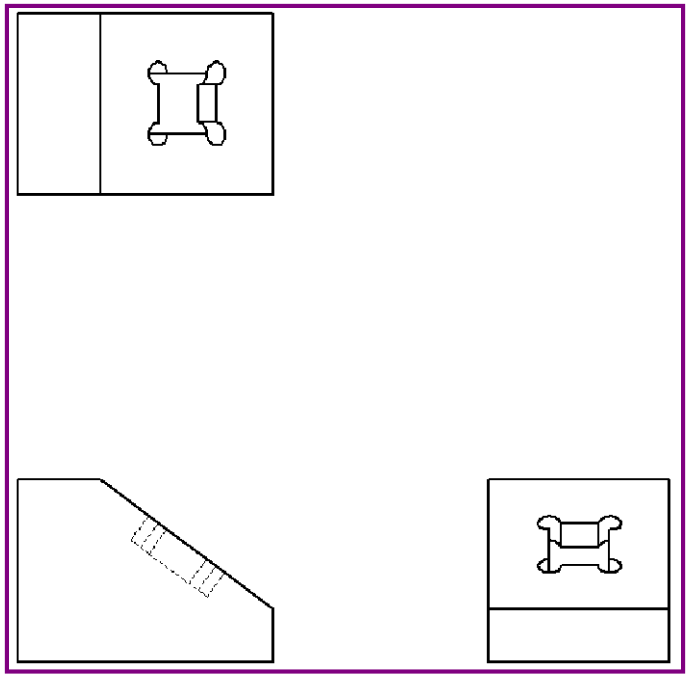
Drawing Views

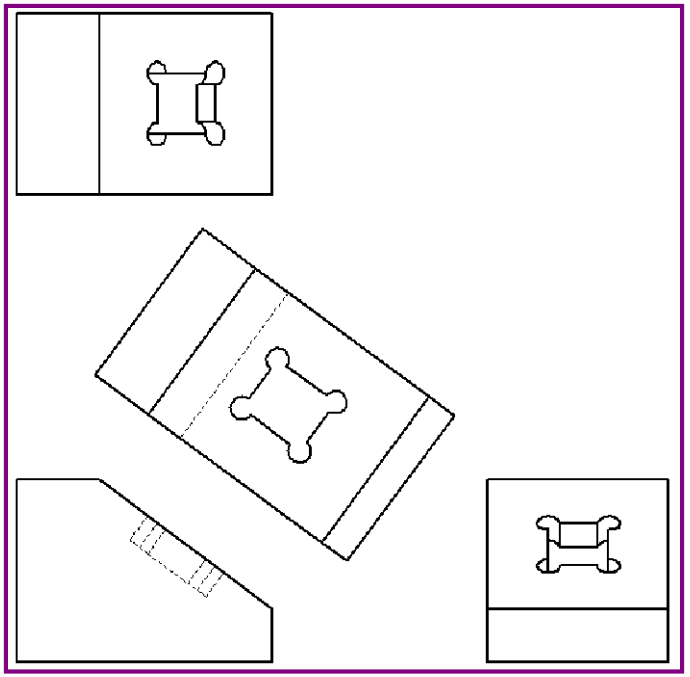
- **Why do we need three views?**
  - **The Front and Top views of both parts are identical.**
  - **The Right side view is necessary to show the characteristic shape.**



Drawing Views: When Three is not Enough

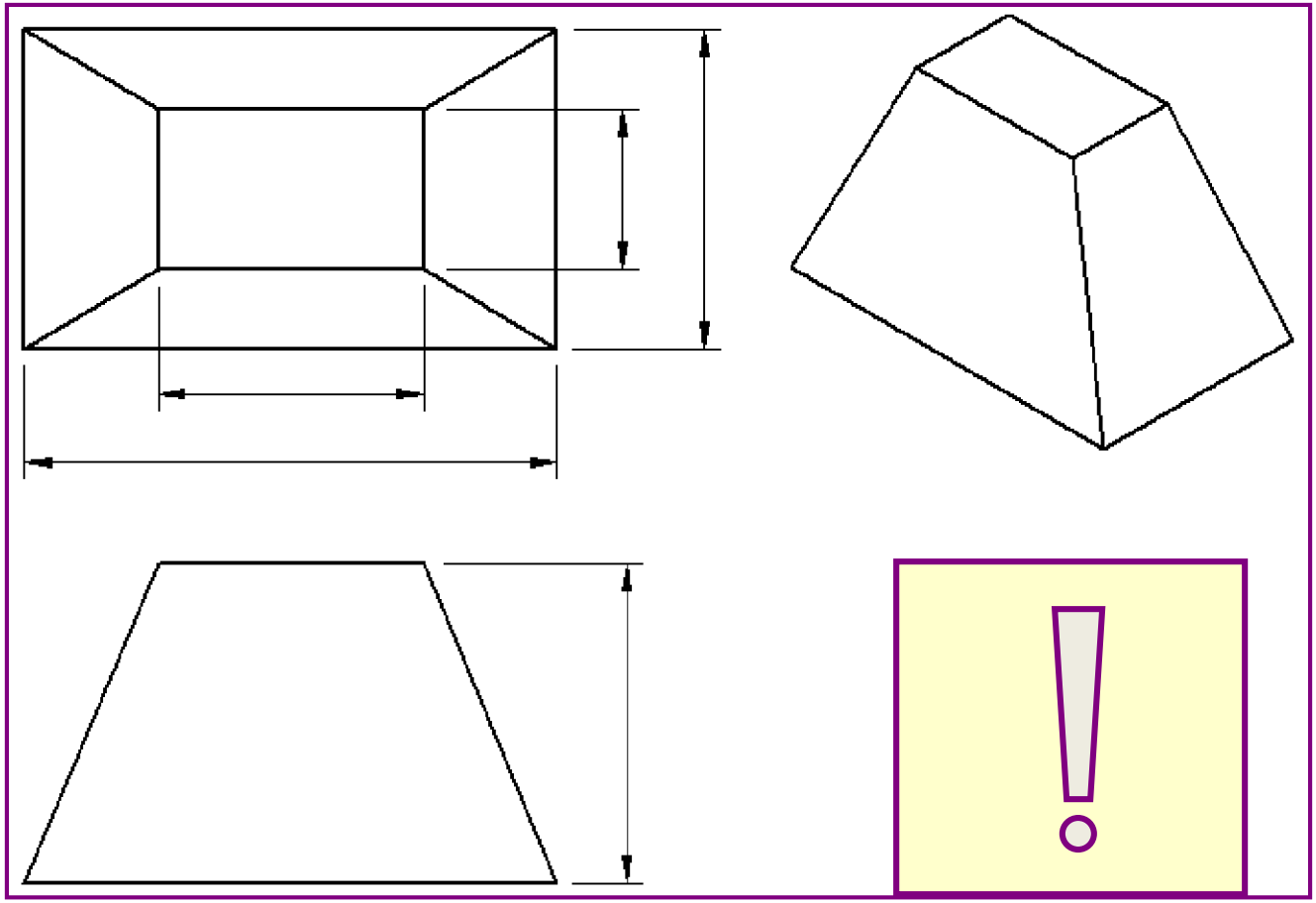
- **Three standard views do not fully describe the shape of the cut-out in the angled face.**





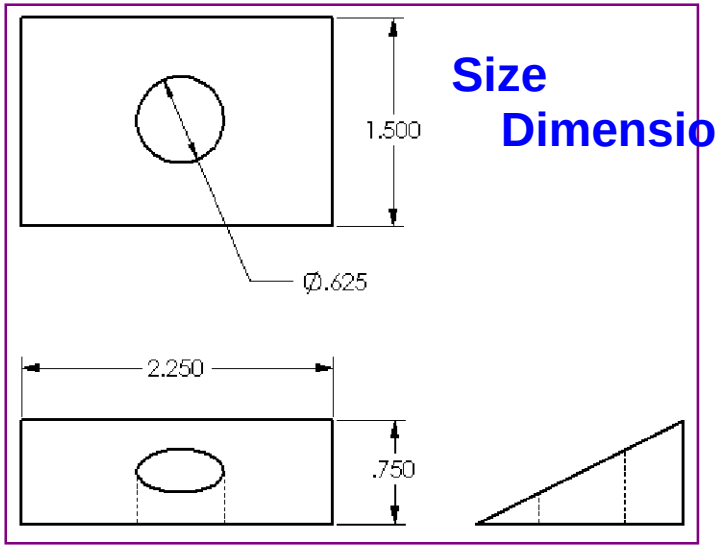
Drawing Views: When Three is too Many

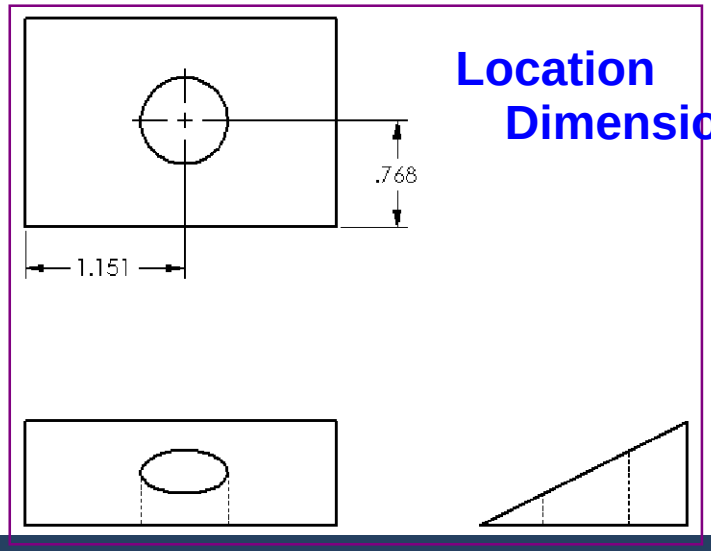
- **The Right side view is unnecessary.**



Dimensions

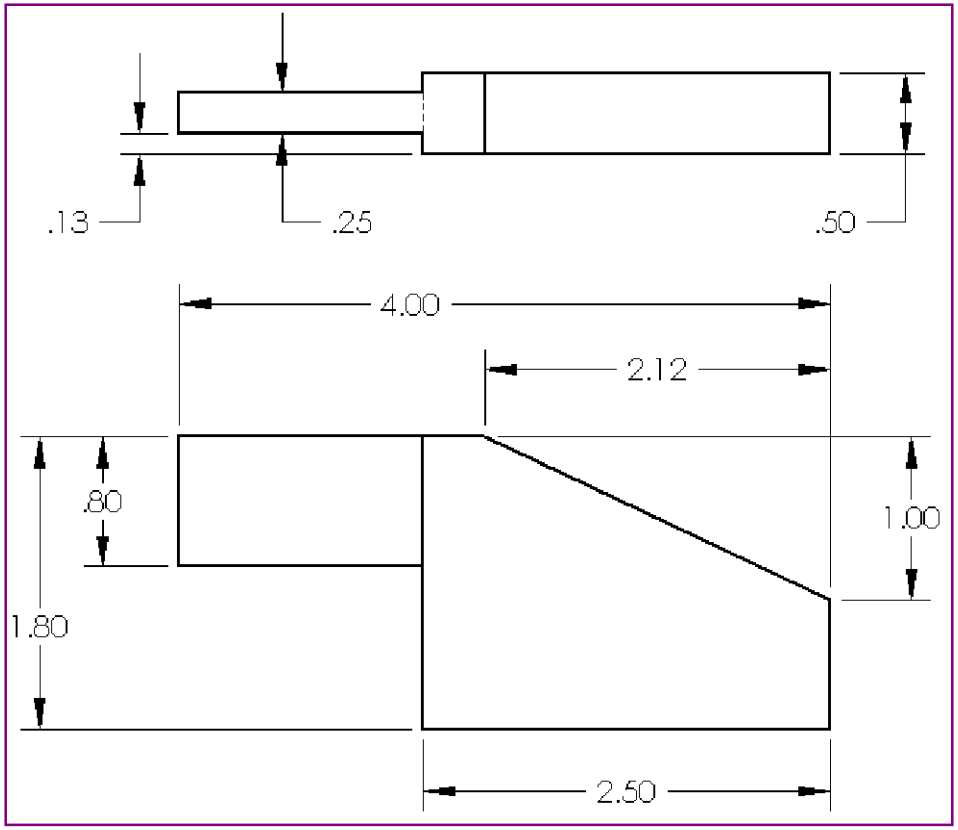
- **There are two kinds of dimensions:**
  - **Size dimensions – how big is the feature?**
  - **Location dimensions – where is the feature?**





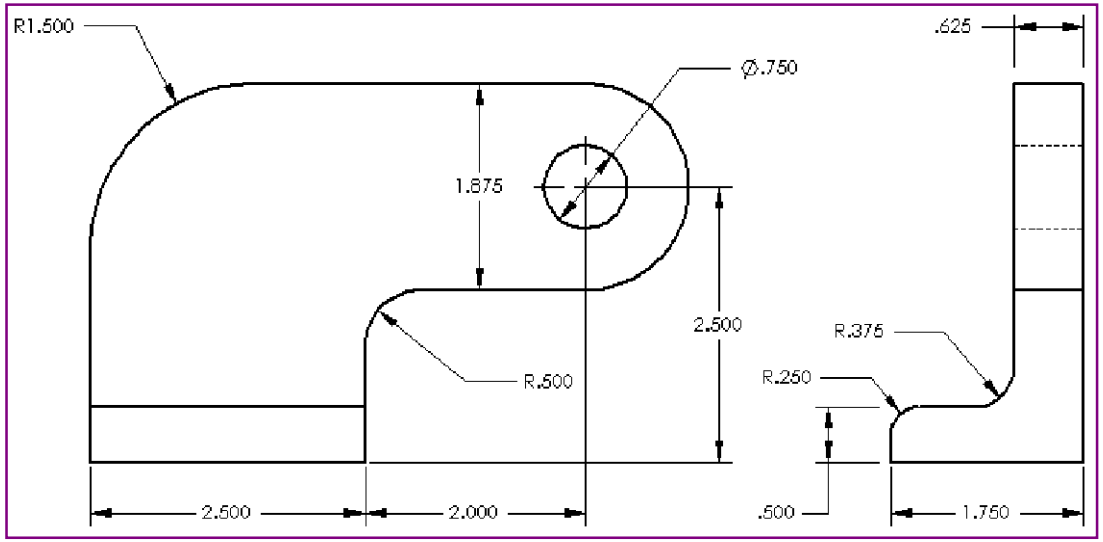
General Drawing Rules – Dimensions

- **For flat pieces, give the thickness dimensions in the edge view, and all other dimensions in the outline view.**



General Drawing Rules – Dimensions

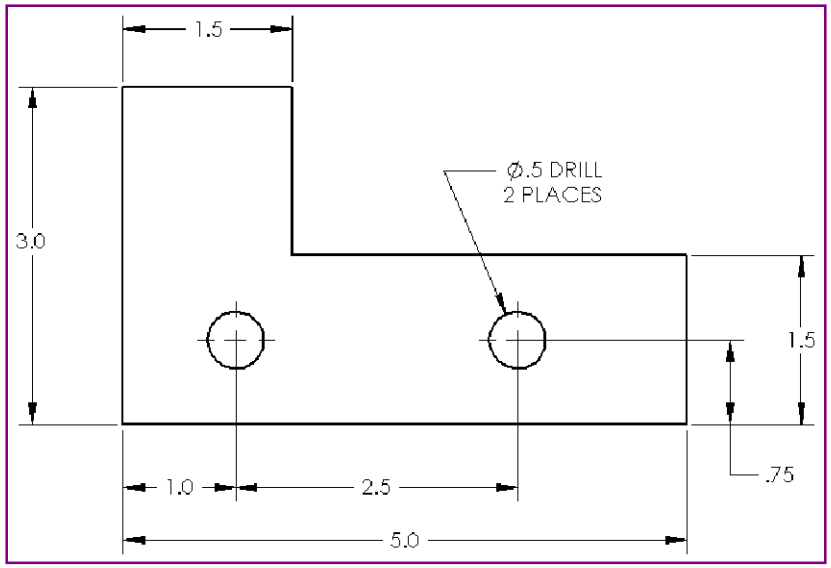
- **Dimension features in the view where they can be seen true size and shape.**



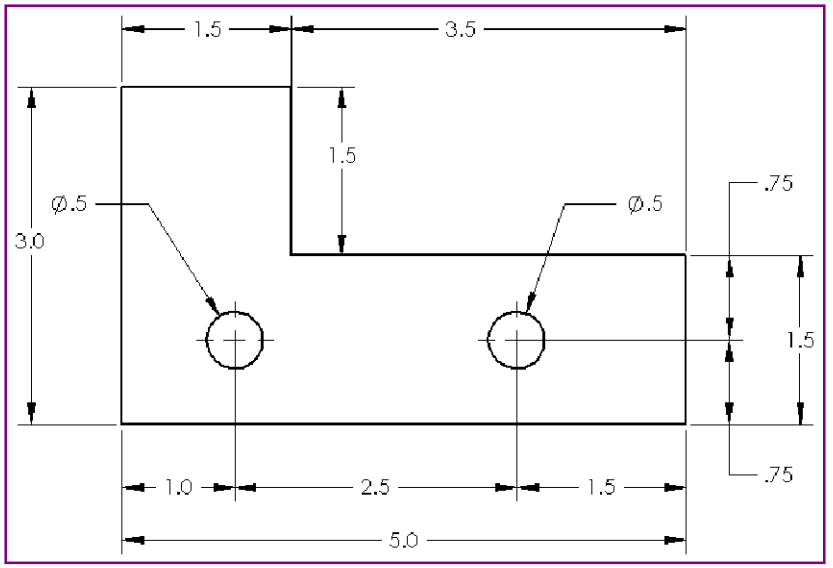
- **Use diameter dimensions for circles.**
- **Use radial dimensions for arcs.**

General Drawing Rules – Dimensions

- **Omit unnecessary dimensions.**



This

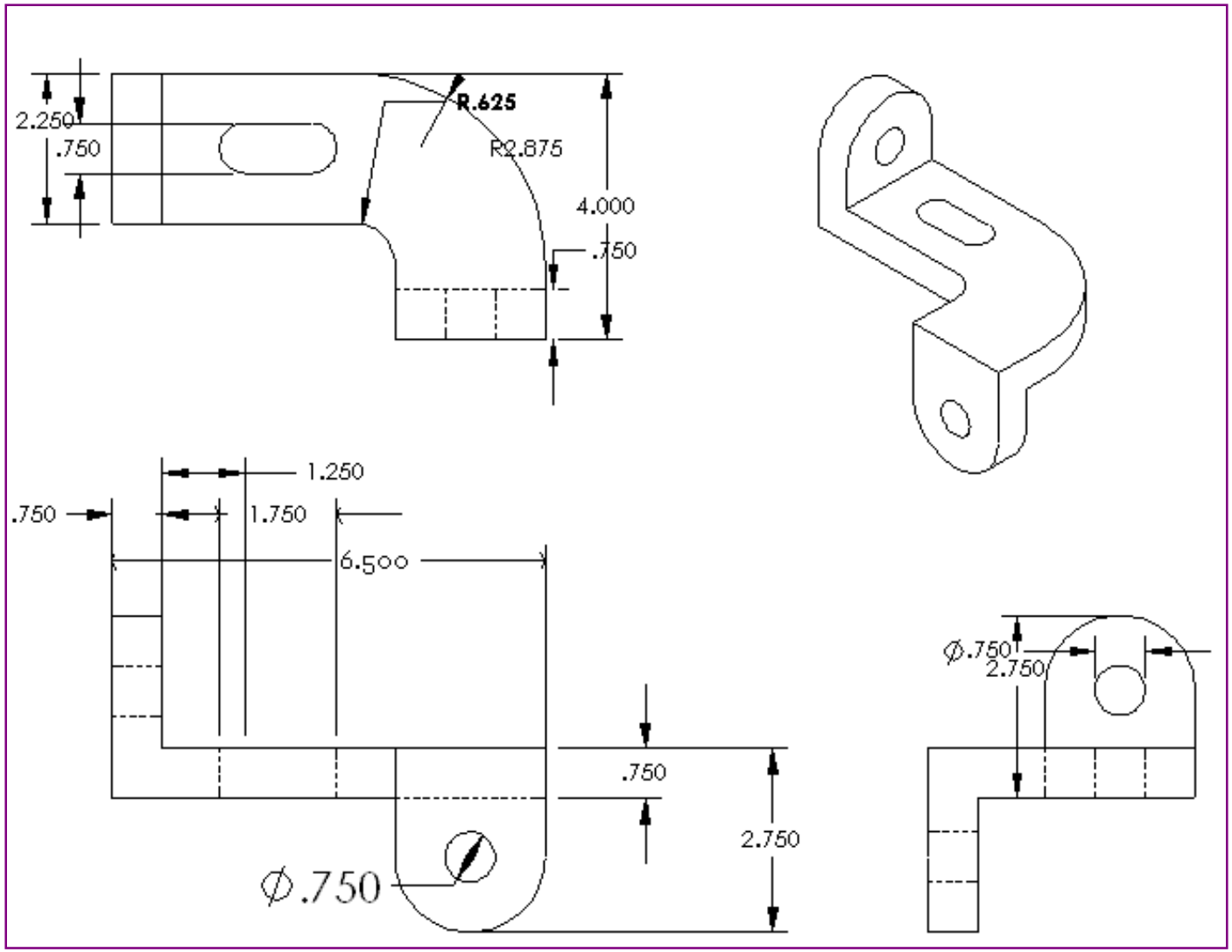


Not This

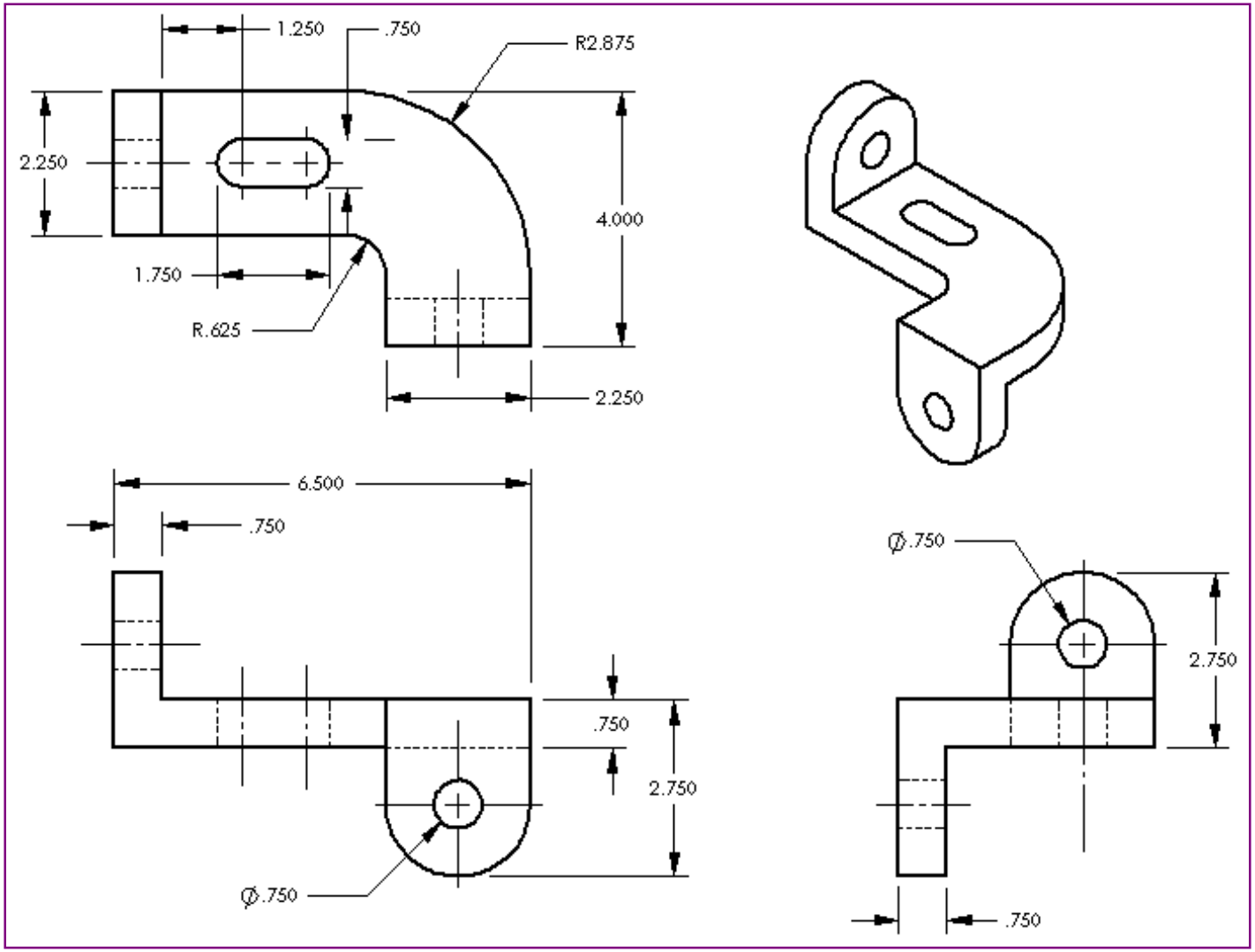
Dimension Guidelines – Appearance

- **Place dimensions away from the profile lines.**
- **Allow space between individual dimensions.**
- **A gap must exist between the profile lines and the extension lines.**
- **The size and style of leader line, text, and arrows should be consistent throughout the drawing.**
- **Display only the number of decimal places required for manufacturing precision.**

Drawing Appearance – Not Good



Drawing Appearance – Much Better



What is a Drawing Template?

- **A Drawing Template is the foundation for drawing information.**

A drawing template specifies:

- **Sheet (paper) size**
- **Orientation - Landscape or Portrait**
- **Sheet Format**
  - **Borders**
  - **Title block**
  - **Data forms and tables such as bill of materials or revision history**

Drawing Templates Choices in SolidWorks

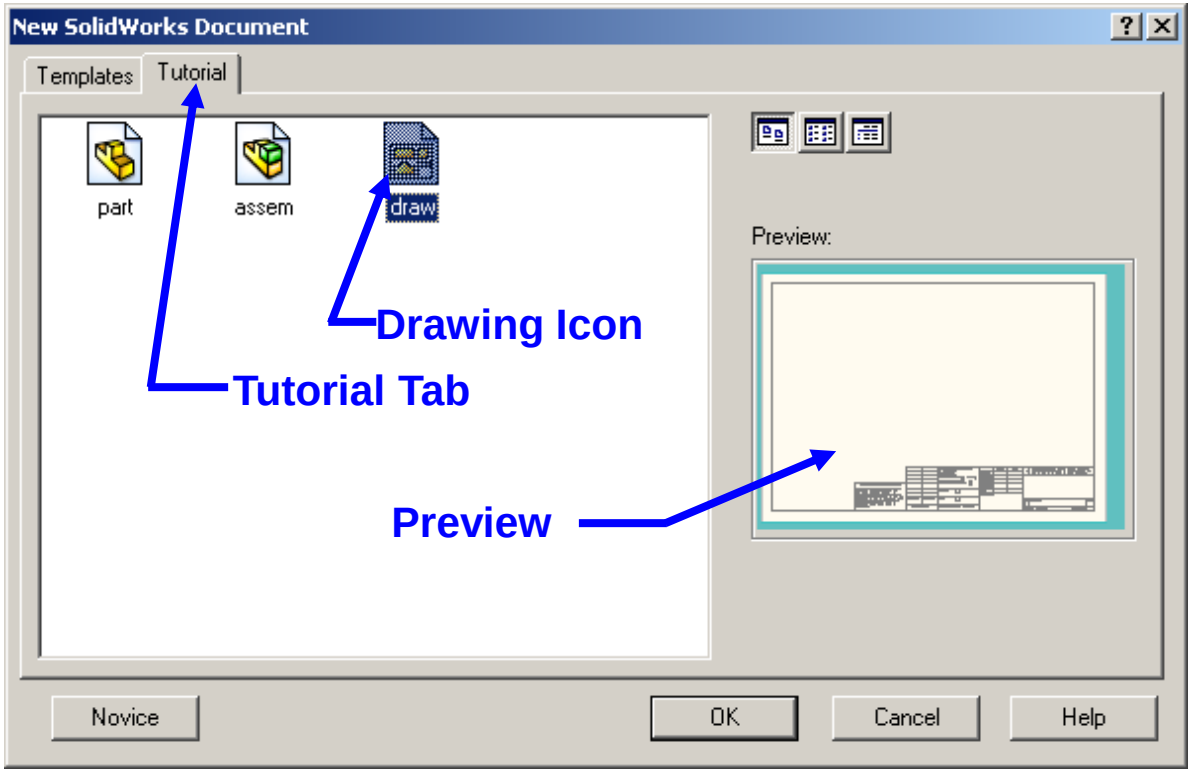
- **Standard SolidWorks drawing template**
- **Tutorial drawing template**
- **Custom template**
- **No template**

To Create a New Drawing Using a Document Template:

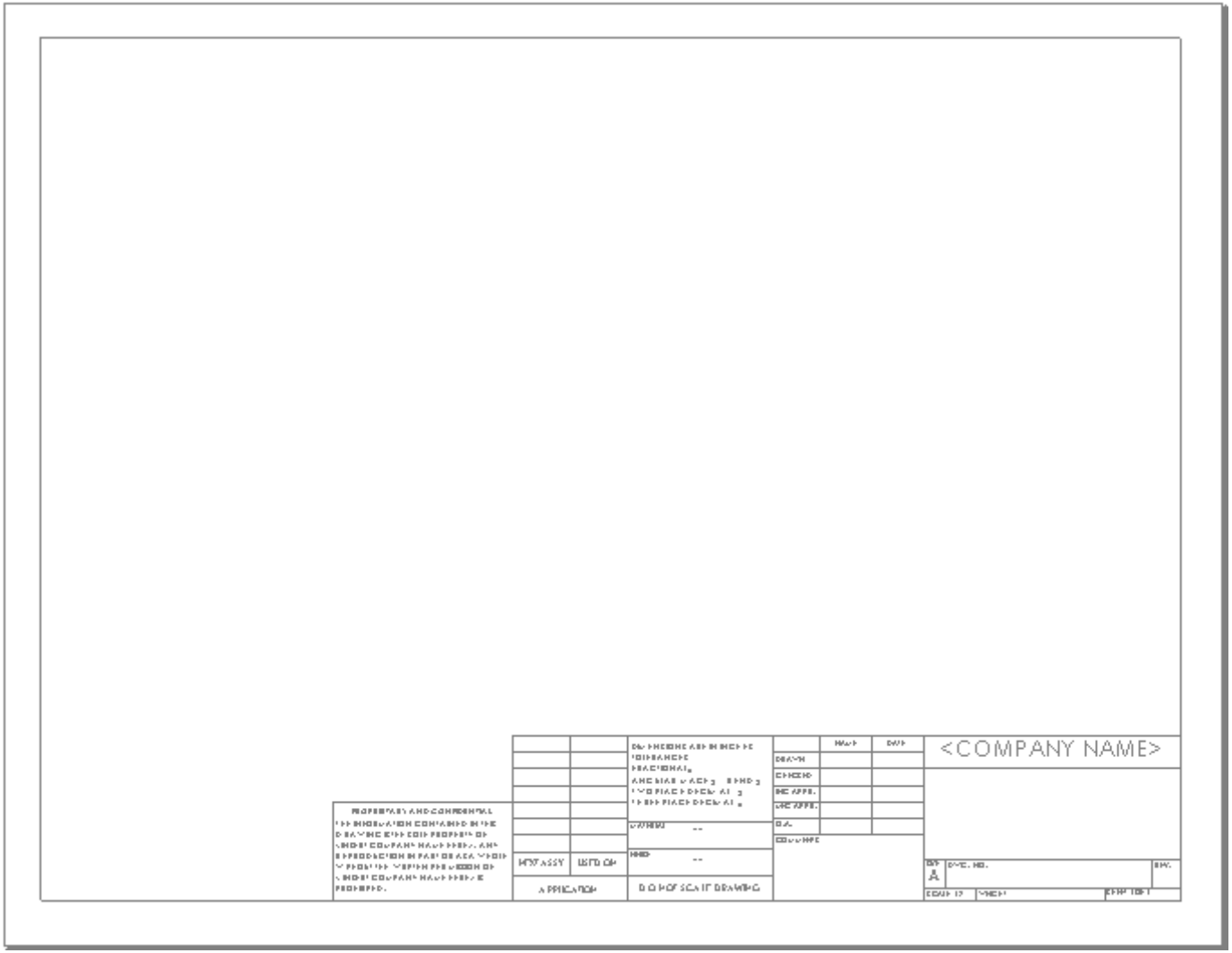
1. Click New  on the Standard toolbar.

2. Click the Tutorial tab.

3. Double-click the drawing icon.



Sample Drawing Template



Edit Sheet vs. Edit Sheet Format

There are two modes in the drawing:

- **Edit Sheet**
  - This is the mode you use to make detailed drawings
  - Used 99+% of the time
  - Add or modify views
  - Add or modify dimensions
  - Add or modify text notes
- **Edit Sheet Format**
  - Change the title block size and text headings
  - Change the border
  - Incorporate a company logo
  - Add standard text that appears on every drawing

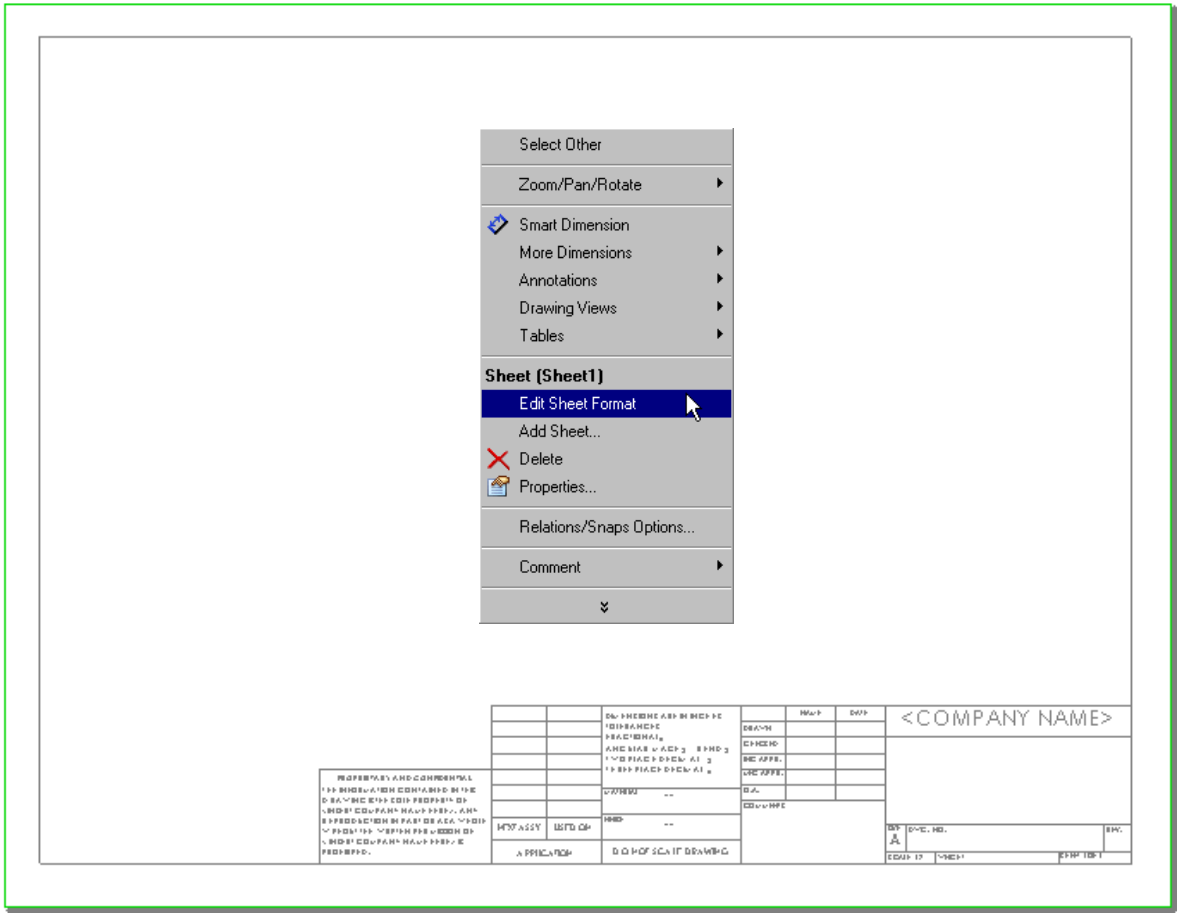
Title Block

- **Contains vital part and/or assembly information.**
- **Each company can have a unique version of a title block.**
- **Typical title block information includes:**

Company name	Material & Finish
Part number	Tolerance
Part name	Drawing scale
Drawing number	Sheet size
Revision number	Revision block
Sheet number	Drawn By/Checked By

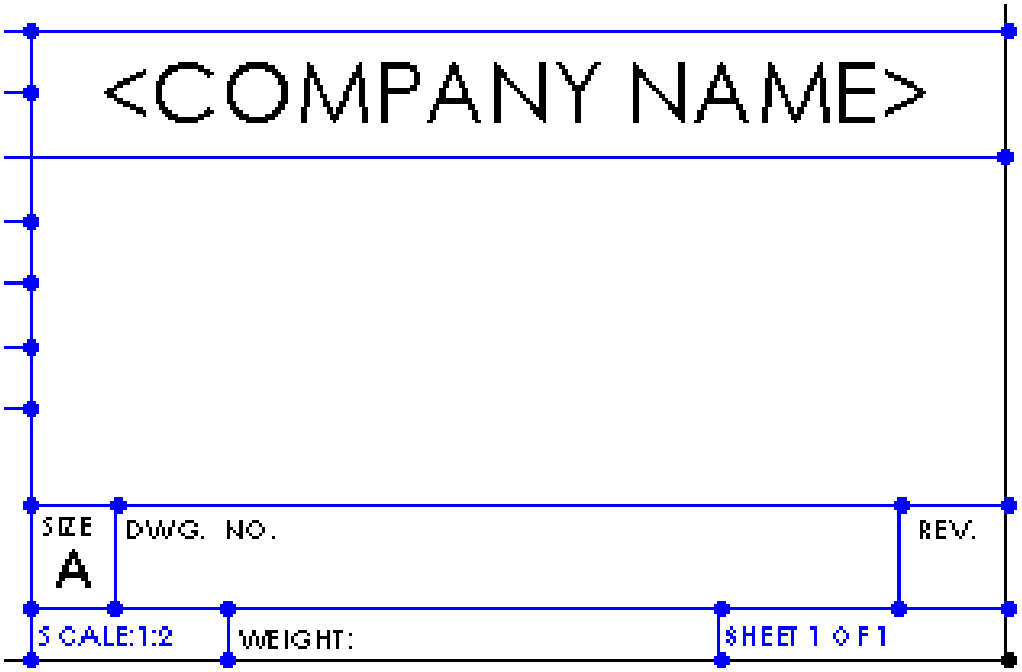
To Edit the Title Block:

1. **Right-click in the graphics area, and select <u>Edit Sheet Format</u> from the shortcut menu.**



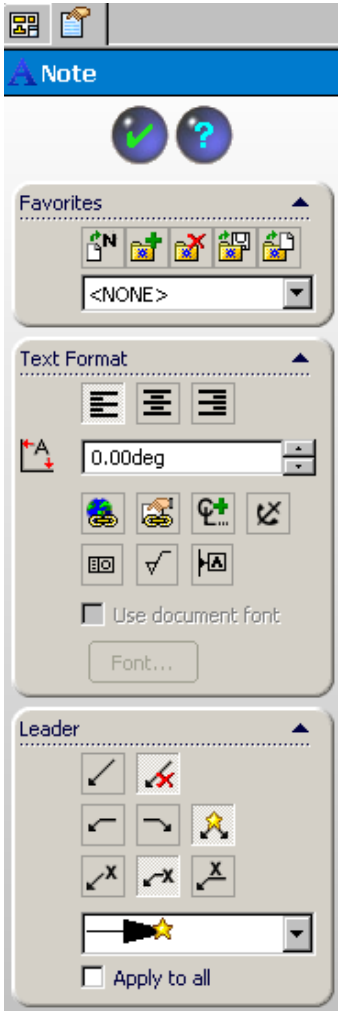
Editing the Title Block:

2. Zoom in on the title block.

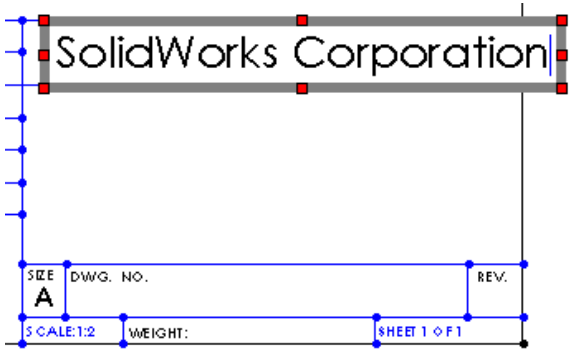


Editing the Title Block:

3. **Double-click the note that says <COMPANY NAME>.**
   The PropertyManager and the pop-up formatting toolbar appear.



4. **Enter your school name in the text insertion box.**



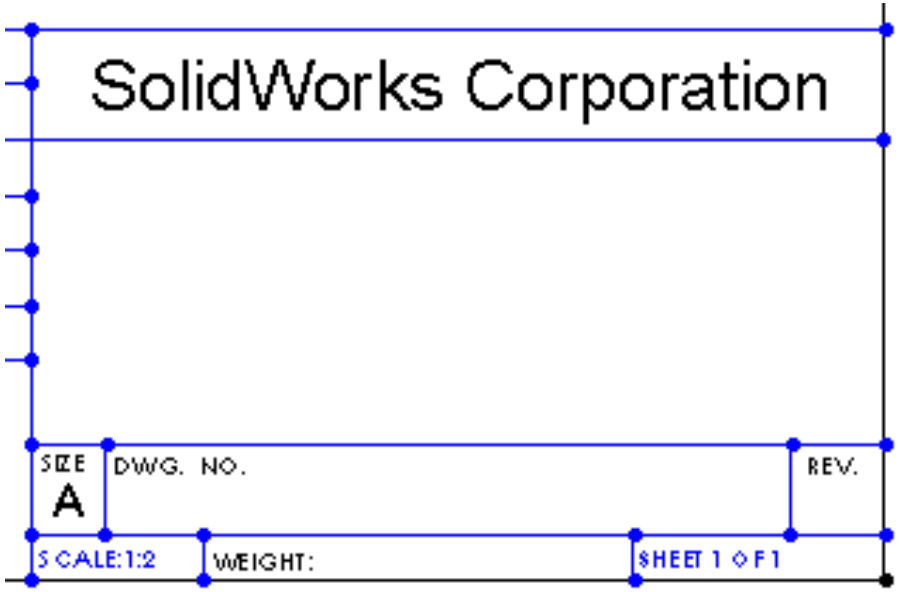
Editing the Title Block:

5. **Set the text justification to Align Left and change the size and style of the text font.**

6. **Click OK to apply the changes and close the PropertyManager.**

Editing the Title Block:

7. **Position the note so it is centered in the space.**



Customizing the Part Name

Advanced Topic

- **The name of the part or assembly shown on the drawing changes with every new drawing.**
- **It is not very efficient to have to edit the sheet format and the title block each time you make a new drawing.**
- **It would be nice if the title block would automatically be filled in with the name of the part or assembly that is shown on the drawing.**
- **This can be done.**

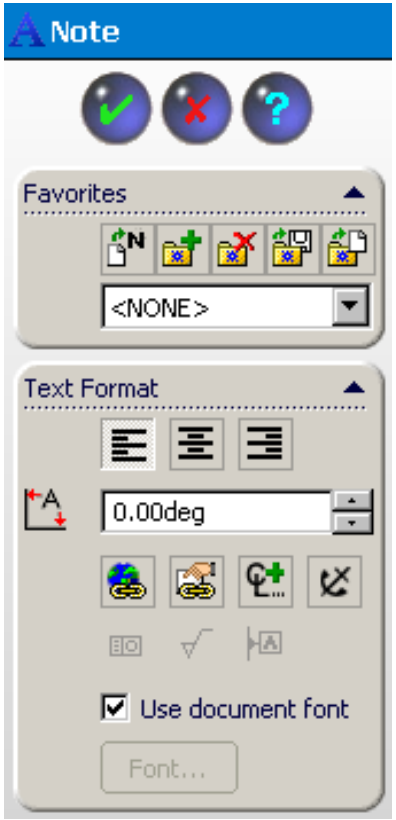
Editing the Part Name:

Advanced Topic

1. Click Note  on the Annotation toolbar, or click Insert, Annotations, Note.

   The PropertyManager appears.

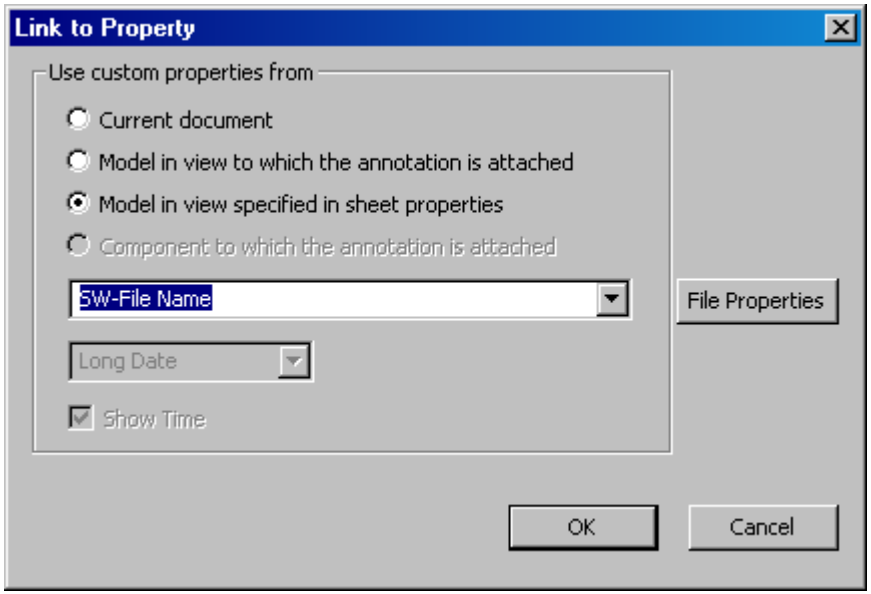
2. Click the  Link to Property button .



Editing the Part Name:

Advanced Topic

3. Click Model in view specified in sheet properties, and choose SW-File Name from the list of properties.

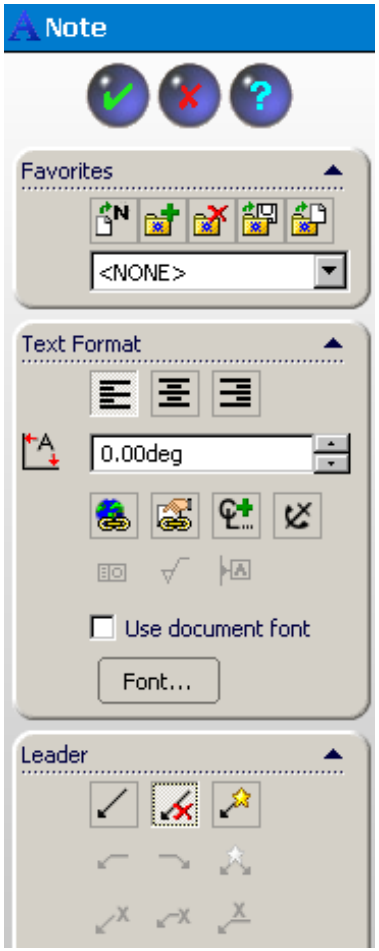


4. Click OK to add the property.

Editing the Part Name:

Advanced Topic

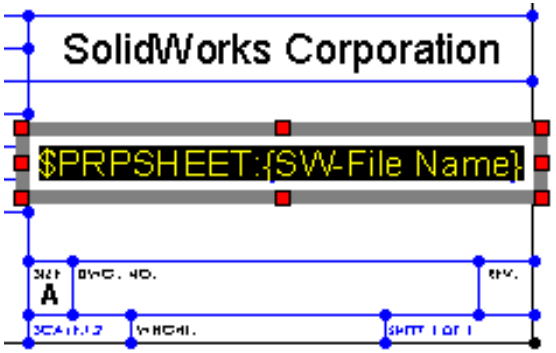
5. **In the PropertyManager, set any other text properties such as justification, or font.**



Editing the Part Name:

Advanced Topic

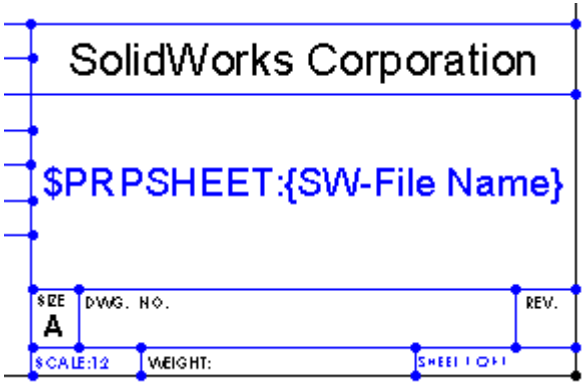
6. Click OK to apply the changes and close the PropertyManager.



Advanced Topic

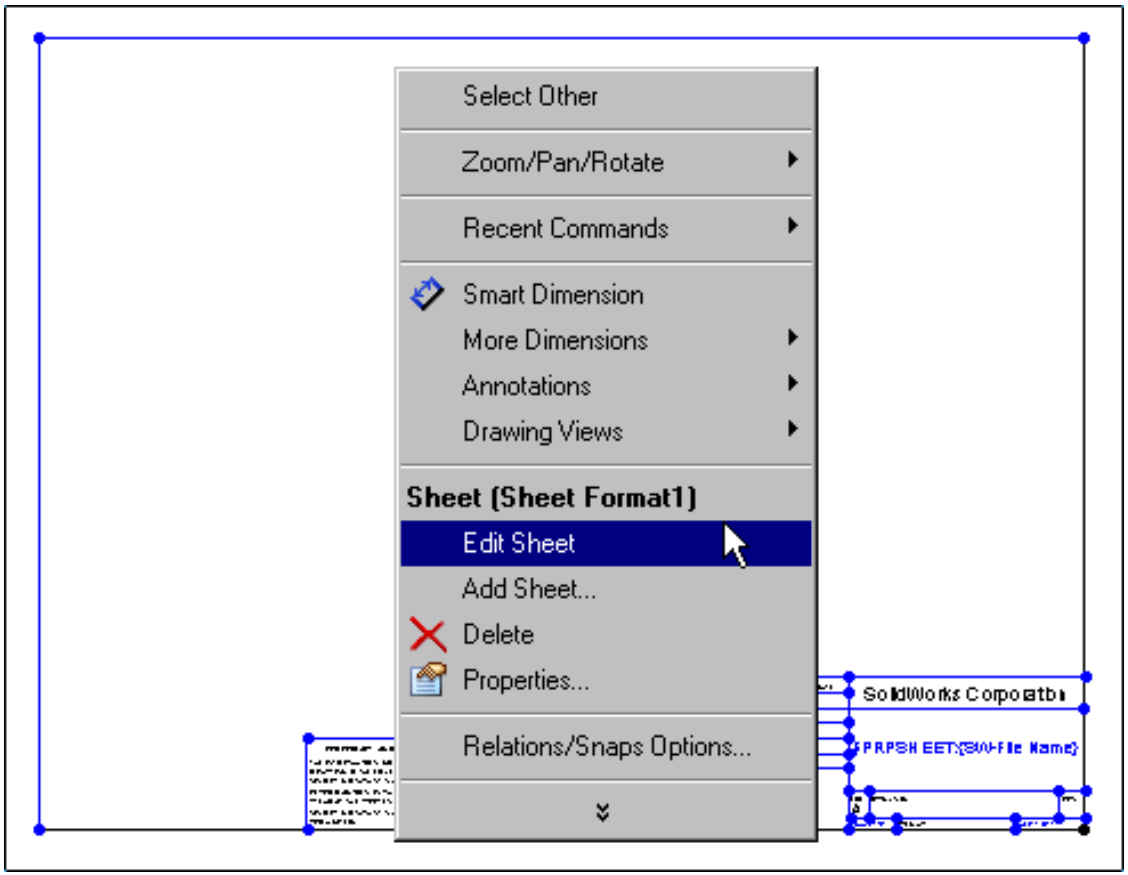
7. **Results.**

   Currently the title block shows the text of the property. However, when the first view is added to the drawing, that text will change to become the file name of the referenced part or assembly.



Switching to Edit Sheet Mode:

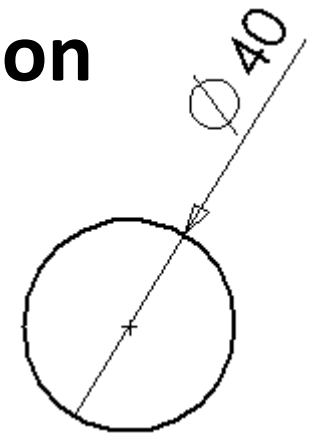
1. **Right-click in the graphics area, and select <u>Edit Sheet</u> from the shortcut menu.**
2. **This is the mode you must be in when you make drawings.**



Detailing Options

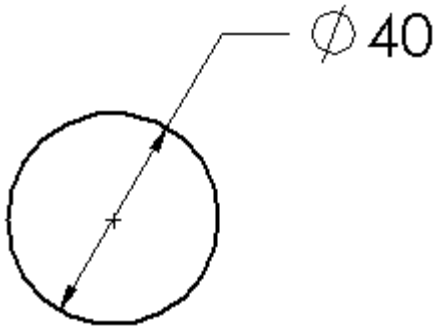
Dimensioning Standards

- **Dimensioning standards determine things such as arrowhead style and dimension text position.**
- **The Tutorial drawing template uses the ISO standard.**
- **ISO stands for International Organization for Standardization.**
- **ISO is widely used in European countries.**



Detailing Options

Dimensioning Standards

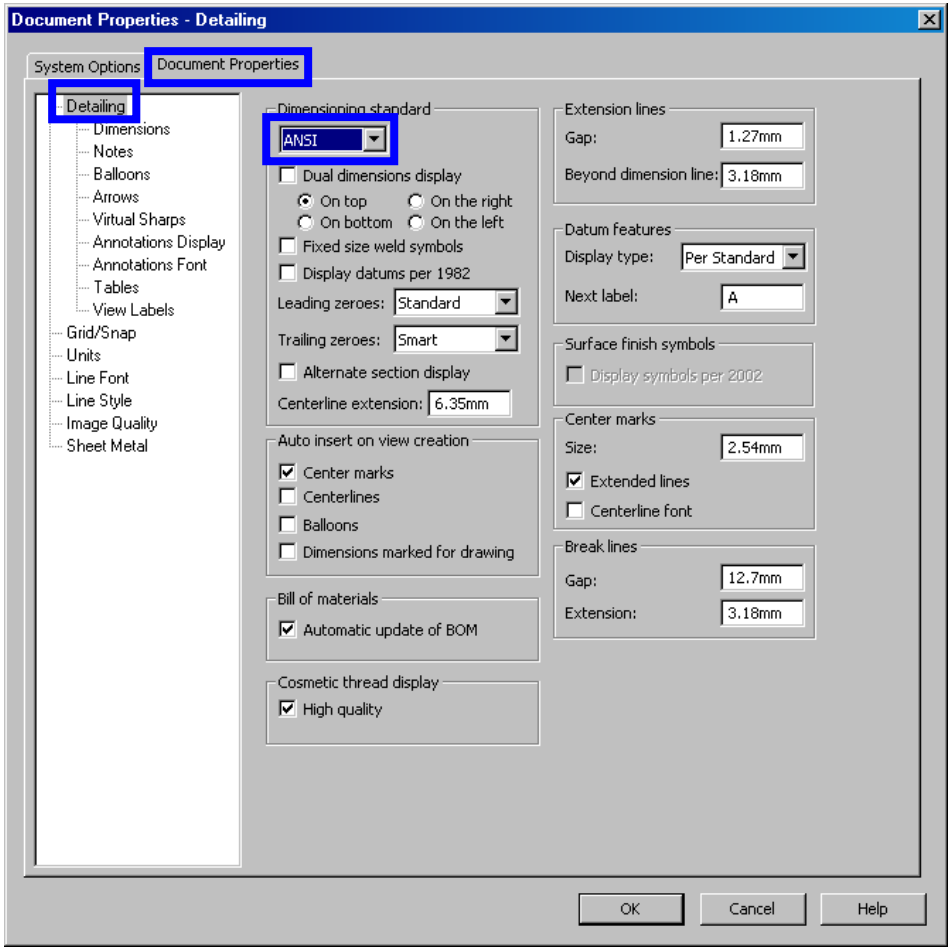


- ANSI is widely used in the United States.
- ANSI stands for American National Standards Institute.
- Other standards include BSI (British Standards Institution) and DIN (Deutsche Industries-Normen).
- Customize the drawing template to use the ANSI standard.

Detailing Options

Setting the dimensioning standard:

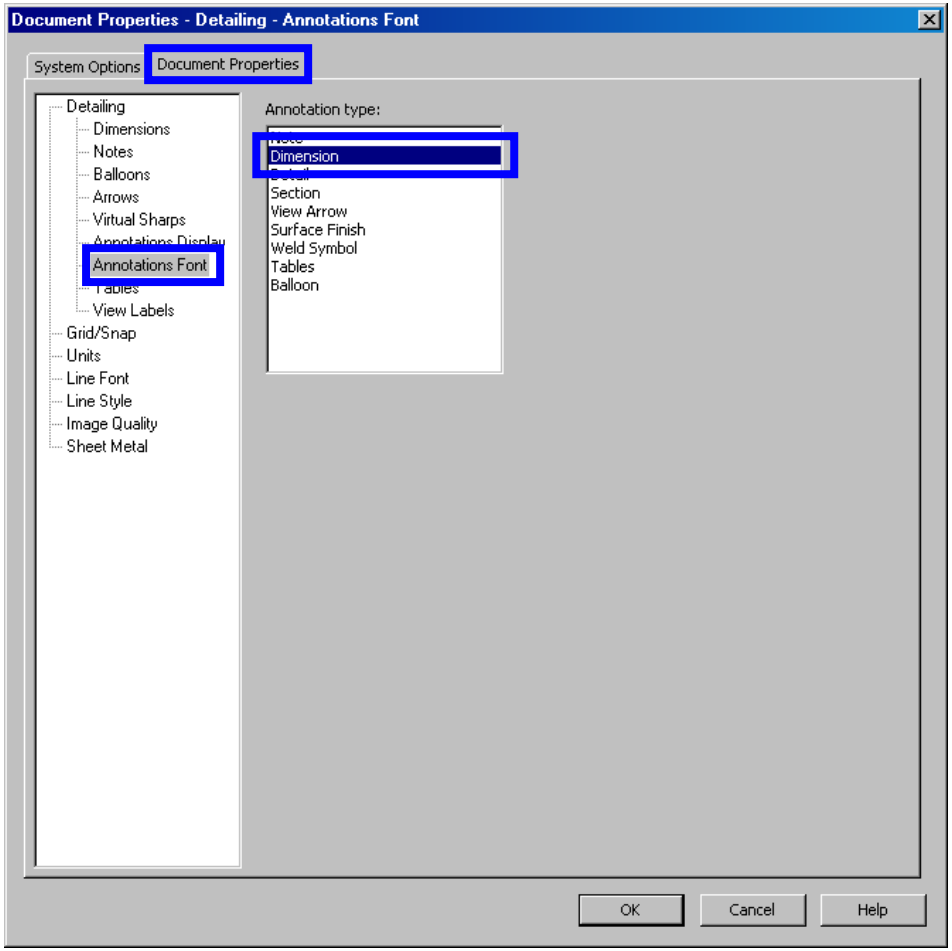
1. Click Tools, Options.
2. Click the Document Properties tab
3. Click Detailing.
4. Select ANSI from the Dimensioning standard list.
5. Click OK.



Detailing Options

Setting text fonts:

1. Click Tools, Options.
2. Click the Document Properties tab
3. Click Annotations Font.
4. Select the annotation type from the list.

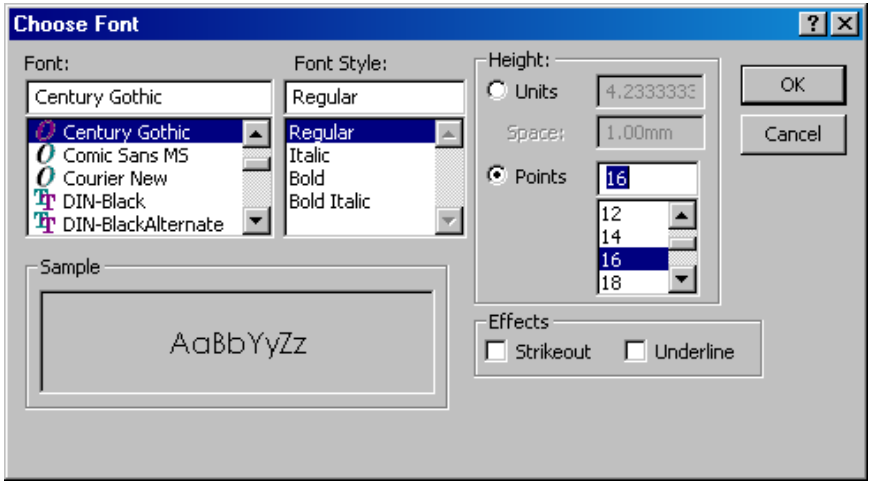


Detailing Options

Setting text fonts continued:

5. **The <u>Choose Font</u> dialog box opens.**

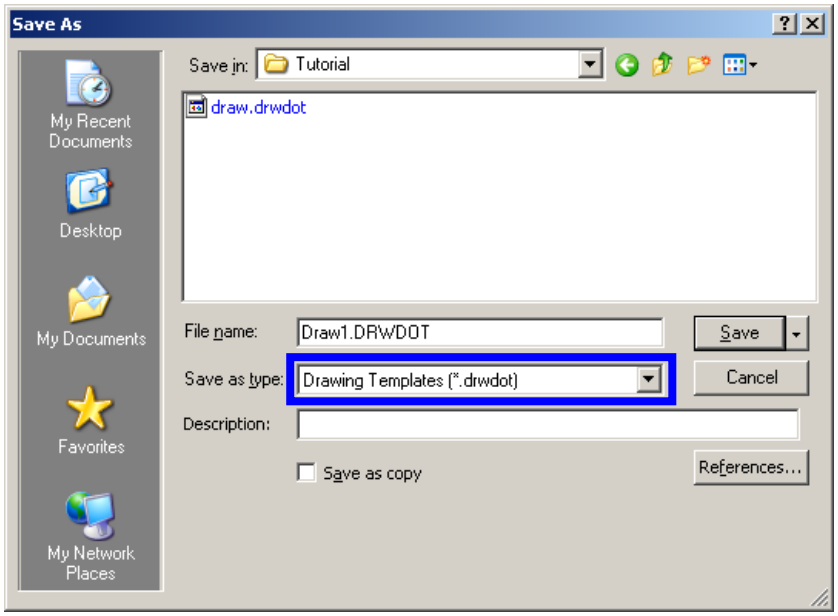
6. **Make the desired changes and click <u>OK</u>.**



Saving a Custom Drawing Template:

1. Click File, Save As...
2. From the Save as type: list, click Drawing Templates.

   The system automatically jumps to the directory where the templates are installed.

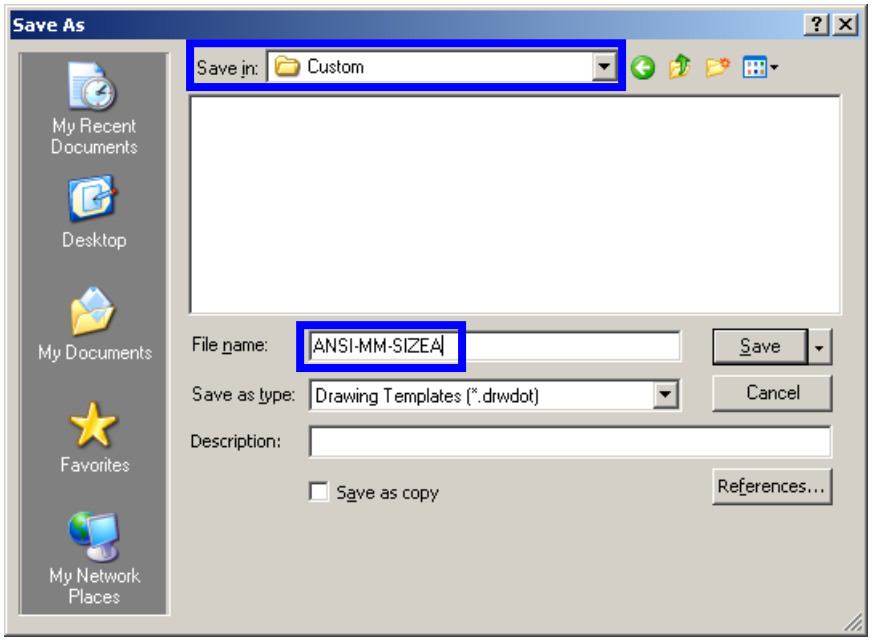


3. Click  to create a new folder.

Saving a Custom Drawing Template:

4. **Name the new folder *Custom*.**
5. **Browse to the *Custom* folder.**
6. **Enter *ANSI-MM-SIZEA* for the file name.**
7. **Click Save.**

Drawing templates have the suffix *.drwdot

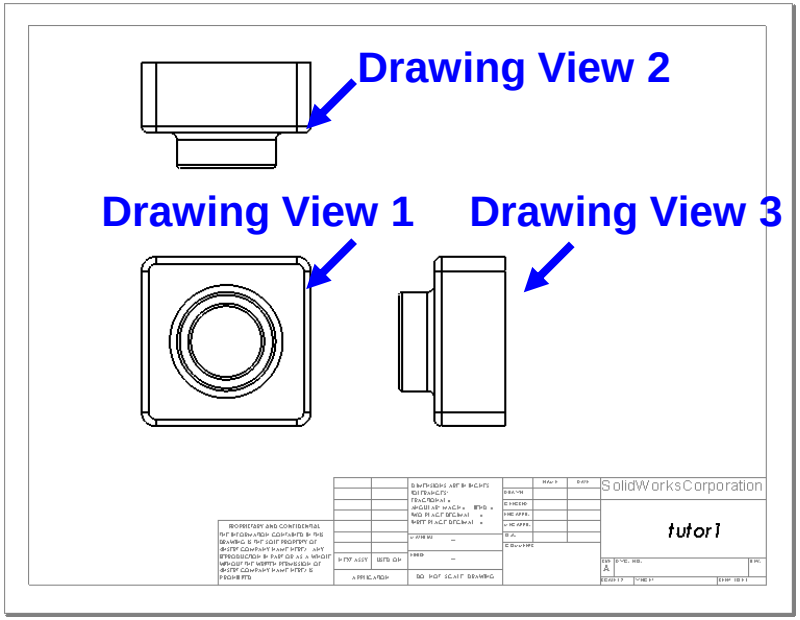


Creating a Drawing – General Procedure

1. **Open the part or assembly you wish to detail.**
2. **Open a new drawing of the desired size.**
3. **Add views: usually three standard views plus any specialized views such as detail, auxiliary, or section views.**
4. **Insert the dimensions and arrange the dimensions on the drawing.**
5. **Add additional sheets, views and/or notes if required.**

To Create Three Standard Views:

1. Click Standard 3 View  .
2. Select Tutor1 from the Window menu.
3. Click OK.



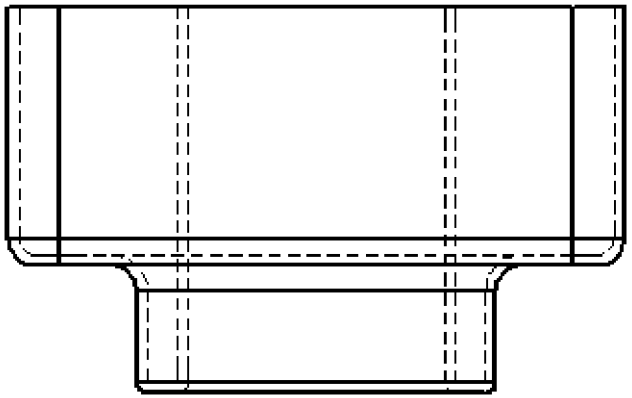
The drawing window reappears with the three views of the selected part.

Working with Drawing Views

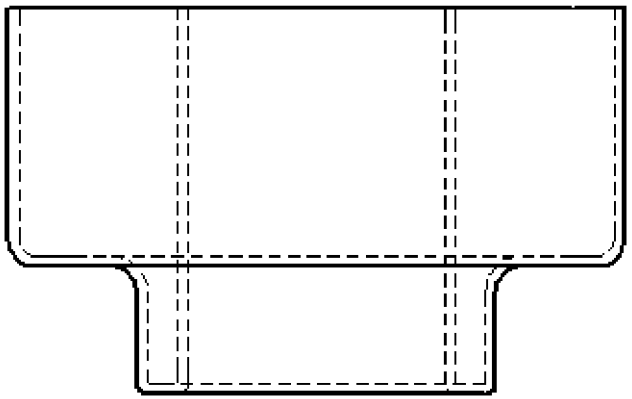
- **To select a view, click the view boundary. The view boundary is displayed in green.**
- **Drawing views 2 and 3 are aligned with view 1.**
- **Drag Drawing View1 (Front). Drawing View 2 (Top) and Drawing View 3 (Right) move, staying aligned to Drawing View1.**
- **Drawing View 3 can only be dragged left or right.**
- **Drawing View 2 can only be dragged up or down.**

Working with Drawing Views

- **Hidden line representation.**
  - **Hidden Lines Visible is usually used in orthographic views.**
  - **Hidden Lines Removed is usually used in isometric views.**



- **Tangent edge display.**
  - **Right-click inside the view border.**
  - **Select Tangent Edge, Tangent Edges Removed from the shortcut menu.**



Dimensioning Drawings

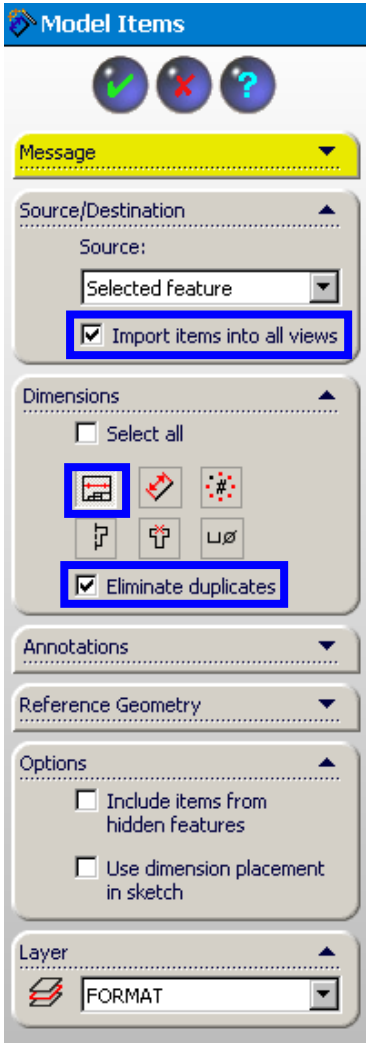
- **The dimensions used to create the part can be imported into the drawing.**
- **Dimensions can be added manually using the Dimension tool  .**

Associativity

- **Changing the values of imported dimensions will change the part.**
- **You cannot change the values of manually inserted dimensions.**

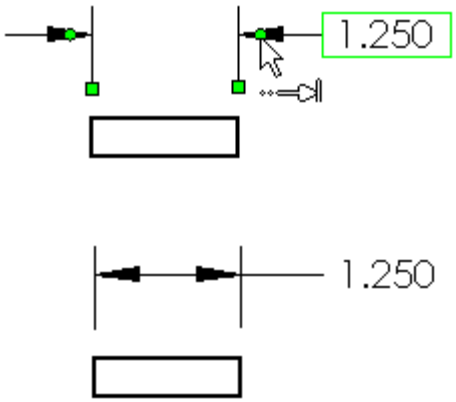
To Import Dimensions into the Drawing:

1. Click Model Items on the Annotation toolbar, or click Insert, Model Items.
2. Click the Import items into all views check box.
3. Click the ption for Marked for drawing and Eliminate duplicates check box.
4. Click OK.



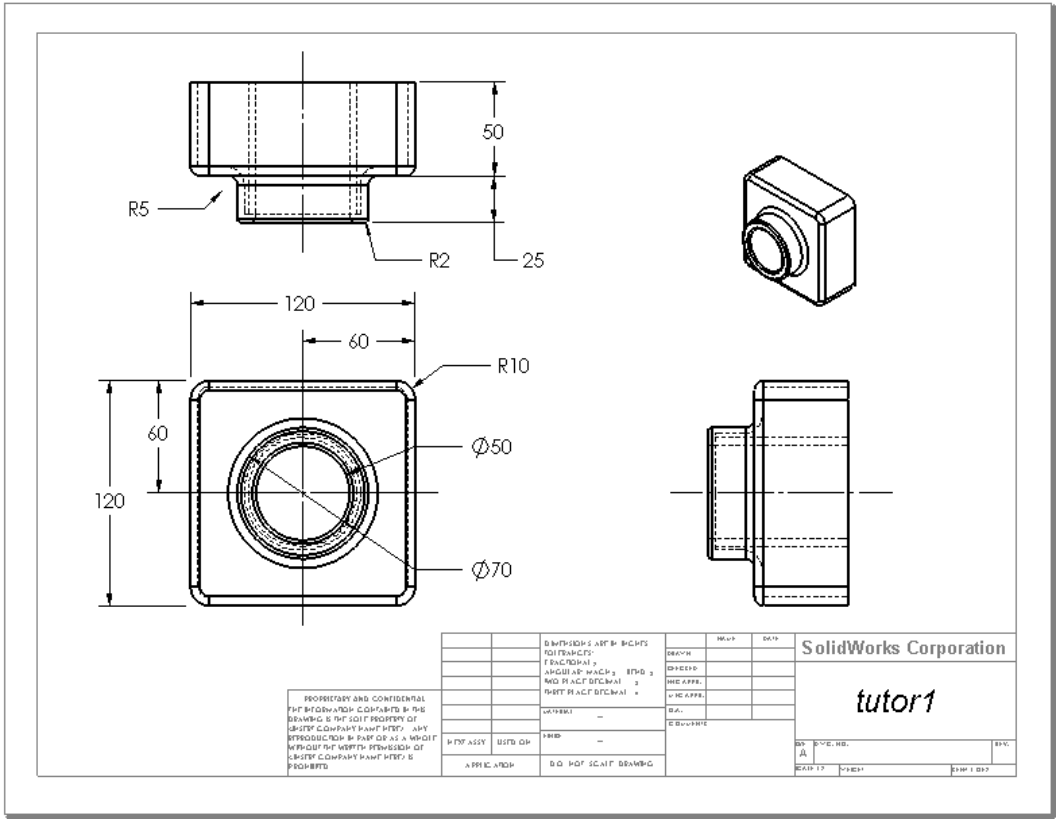
Manipulating Dimensions

- **Moving dimensions:**
  - **Click the dimension text.**
  - **Drag the dimension to the desired location.**
  - **To move a dimension into a different view, press and hold the Shift key while you drag it.**
- **Deleting dimensions:**
  - **Click the dimension text, and then press the Delete key.**
- **Flipping the arrows:**
  - **Click the dimension text.**
  - **A green dot appears on the dimension arrows.**
  - **Click the dot to flip the arrows in or out.**



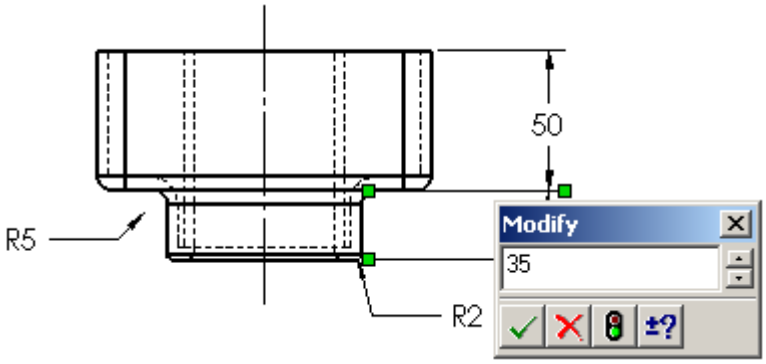
Finish the Drawing

- **Position the views.**
- **Arrange the dimensions by dragging them.**
- **Set hidden line removal and tangent edge display.**

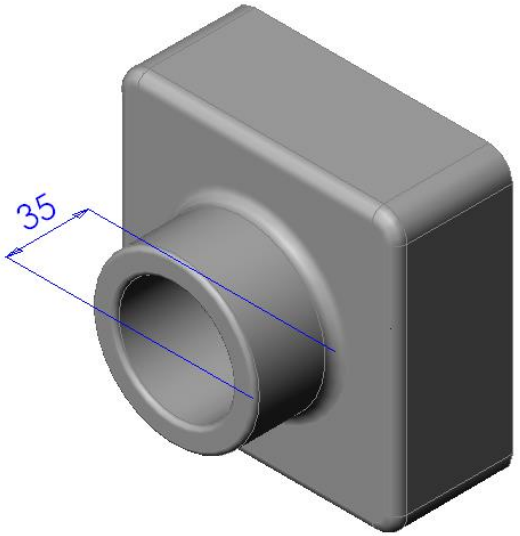


Associativity

- **Changing a dimension on the drawing changes the model.**
  - **Double-click the dimension text.**
  - **Enter a new value.**
  - **Rebuild.**



- **Open the part. The part reflects the new value.**
- **Open the assembly. The assembly also reflects the new value.**

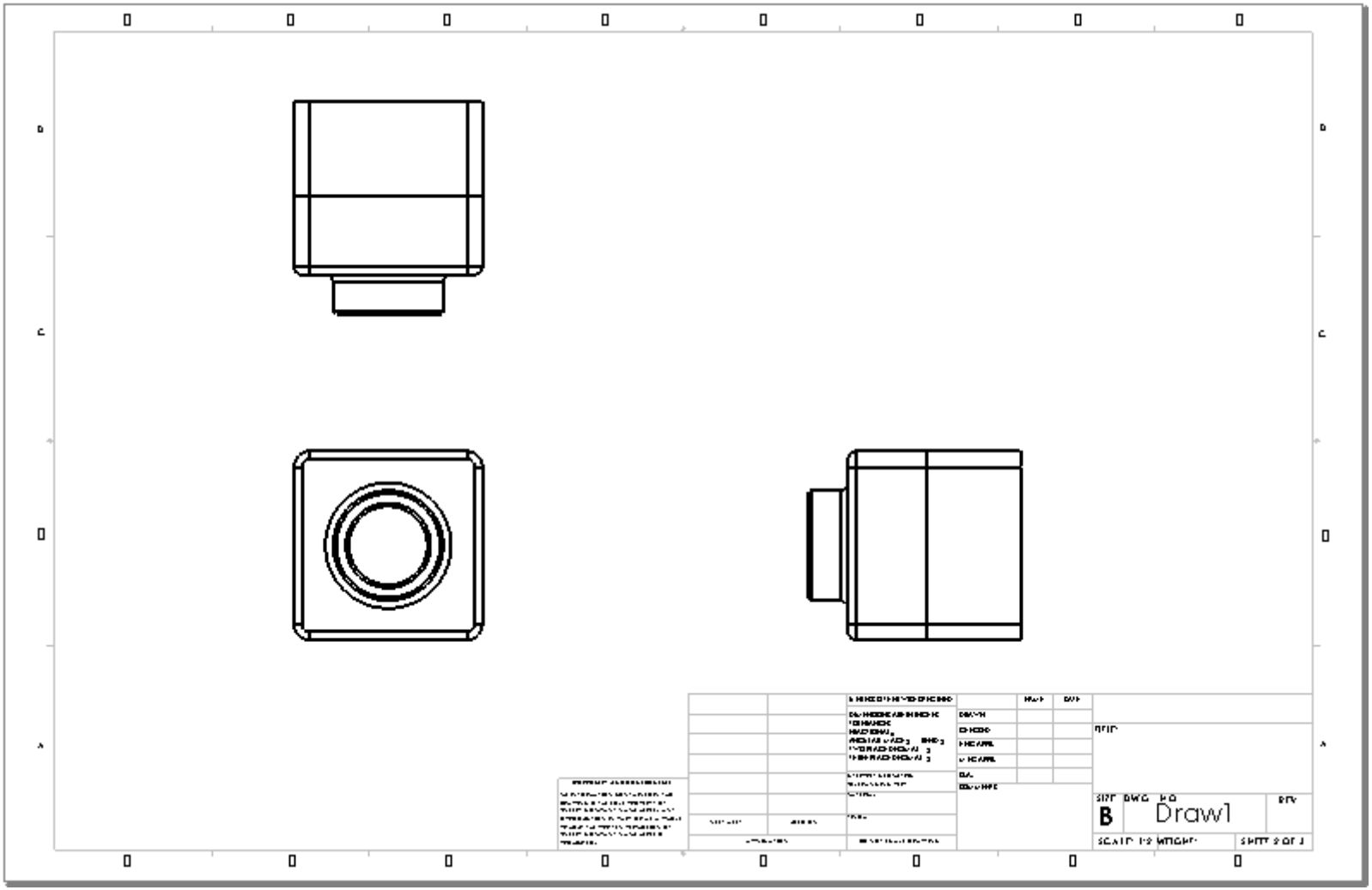


Multi-sheet Drawings

Drawings can contain more than one sheet.

- **The first drawing sheet contains Tutor1.**
- **The second drawing sheet contains the Tutor assembly.**
- **Use the B-size landscape (11” x 17”) drawing Sheet Format.**
- **Add 3 standard views.**
- **Add an Isometric view of the assembly. The Isometric view is a model view.**

Three View Drawing of Assembly

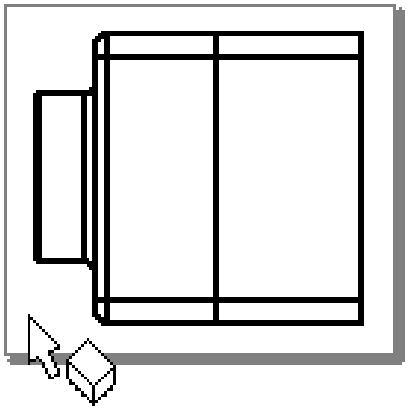


Model Views

- **A model view shows the part or assembly in a specific orientation.**
- **Examples of model views are:**
  - **Standard Views such as Front, Top or Isometric view.**
  - **User-defined view orientations that were created in the part or assembly.**
  - **The current view in a part or assembly.**

To Insert a model View:

1. Click model View  , or click Insert, Drawing view, Model.
2. Click inside the border of an existing view.



Important: Do not click directly on one of the parts in the assembly. Doing so will create a named view of that specific part.

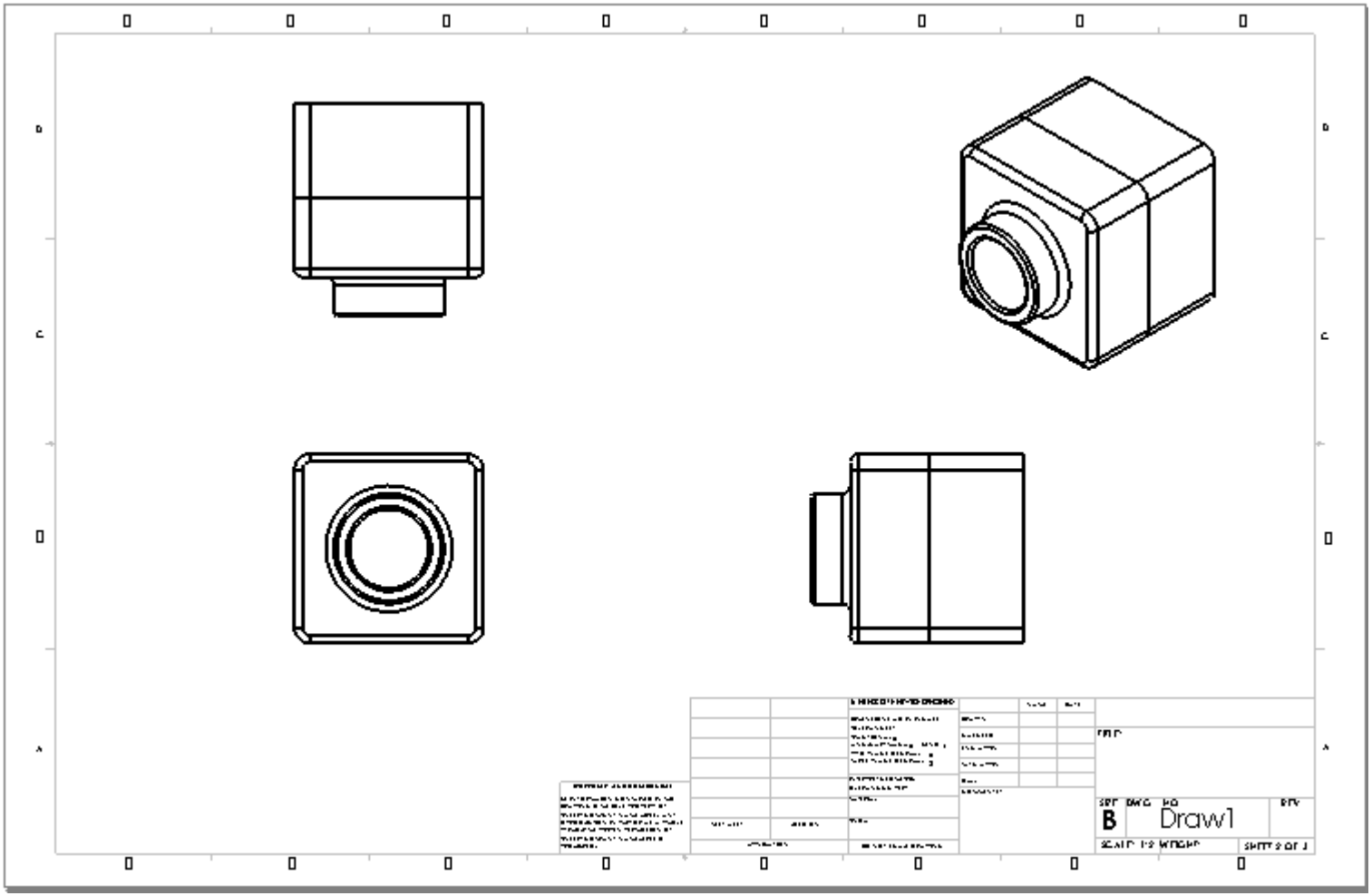
Inserting a Model View:

3. **A selection of model view icons appears in the PropertyManager.**

   Select the desired view, in this case, Isometric, from the selection.

4. **Place the view in the desired location on the drawing.**

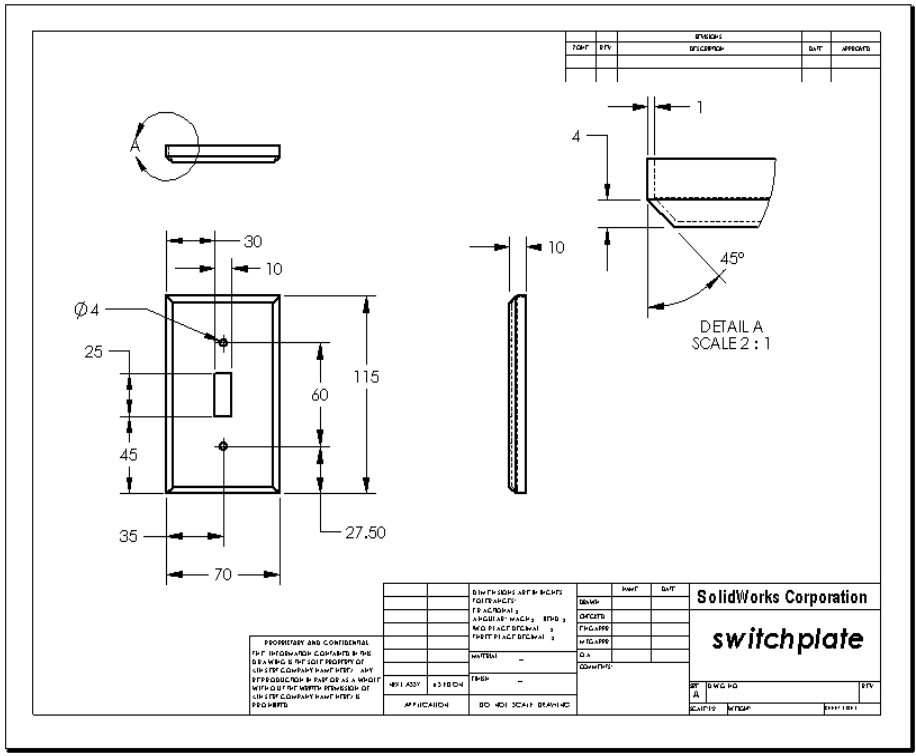
Isometric View Added to Drawing



Specialized Views

Detail View – used to show enlarged view of something.

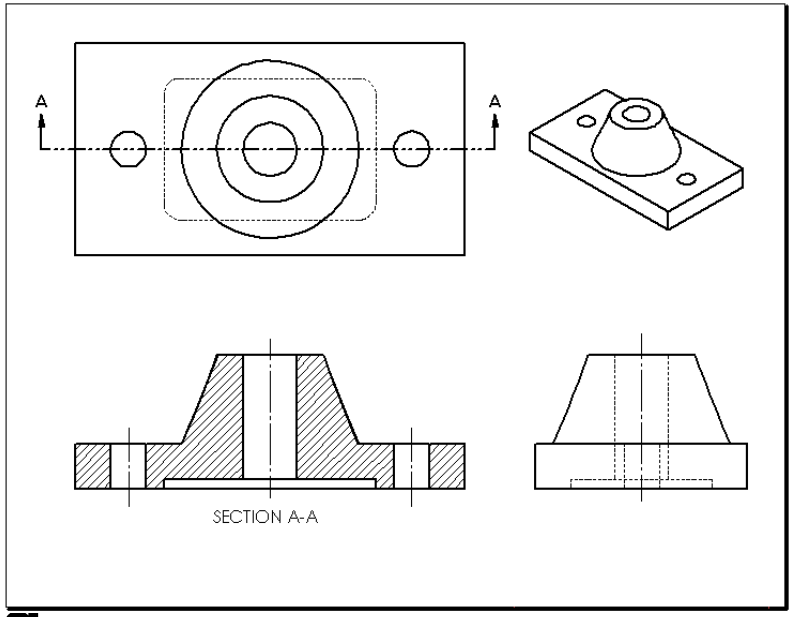
1. **Click  or click Insert, Drawing View, Detail.**
2. **Sketch a circle in the "source" view.**
3. **Position the view on drawing.**
4. **Edit the label to change scale.**
5. **Import dimensions or drag them into view.**



Specialized Views

Section View – used to show internal aspects of object.

1. **Click or click Insert Drawing View, Section.**
2. **Sketch line in the "source" view.**
3. **Position the view on drawing.**
4. **Section view is automatically crosshatched.**
5. **Double-click section line to reverse arrows.**



Ref: SolidWorks Teacher Guide Lesson

Thank You for Attention !!

Any Questions