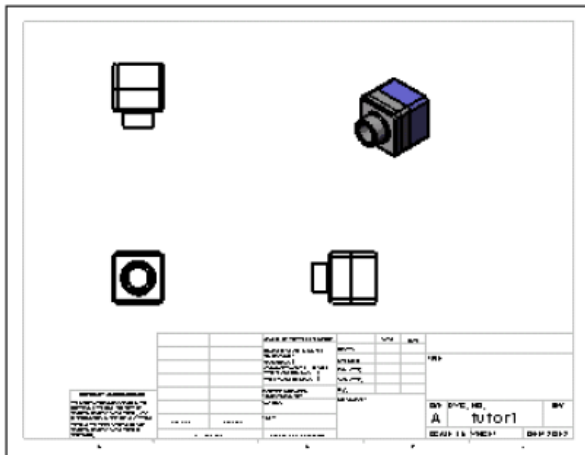In this lesson, you create a multi-sheet drawing of the parts and assembly from Lessons 1 and 2. This lesson includes:

- Opening a drawing template and editing a sheet format

- Inserting standard views of a part model

- Adding model and reference annotations

- Adding another drawing sheet

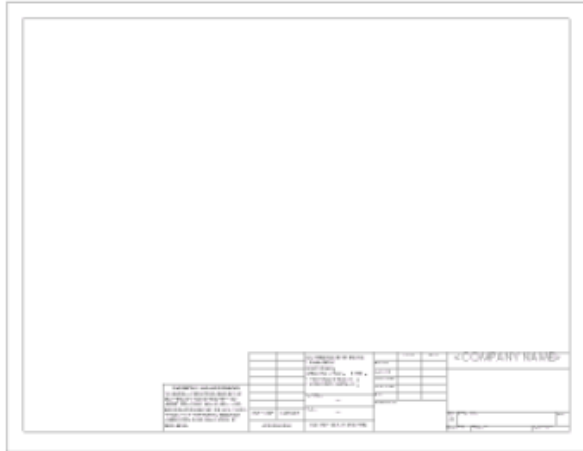- Inserting a named view

- Printing the drawing

Opening a Drawing Template

First you open the drawing template.

1. Click **New** on the Standard toolbar.

2. Click **Drawing**, then click **OK**.

   A new drawing appears in the graphics area, and the **Model View** PropertyManager appears.

Opening a Drawing Template (continued)

Next you edit the sheet format by changing some text properties. Since you are working on the sheet format, and not inserting a model in the drawing yet, cancel the PropertyManager.

1. Click **Cancel** ✖ in the PropertyManager.

   💡 The **Model View** PropertyManager appears only if **Start command when creating new drawing** is selected in the PropertyManager.

2. Right-click anywhere in the drawing sheet, and select **Edit Sheet Format**.

3. In the title block, double-click the variable text that appears.

   💡 You can use the zoom tool to make selection easier. Click **Zoom to Area** 🔍 on the View toolbar, and drag-select to the title block at the lower right. Click **Zoom to Area** 🔍 again to turn off the tool.

   The text appears in an edit box.

4. Change the text to the name of your company.

5. Change the font, size, or style, in the Formatting toolbar.

6. Click outside of the text area to save your changes.

7. Click **Zoom to Fit** on the View toolbar.

8. Right-click anywhere in the drawing sheet, and select **Edit Sheet** to exit the edit sheet format mode.

Saving the Drawing Sheet Format

Next you save the updated sheet format. This is different from saving the drawing itself.

1. To replace this format as the standard **A-Landscape** format, click **File**, **Save Sheet Format**.

2. In **Save in**, navigate to the file location where you save templates. The file location is set in **Tools > Options > System Options > File Locations** for **Document Templates**.

3. Click **a-landscape.slddrt**, then click **Save**.

4. Click **Yes** to confirm that you want to overwrite the existing sheet format. When you choose this sheet format for your own drawings, you do not need to perform these edits again.

Setting the Detailing Options

Next, set the default dimension font, and set the style of dimensions, arrows, and other detailing options. For this lesson, use the settings described below. Later, you can set the detailing options to match your company's standards.

1. Click **Options** on the Standard toolbar.

2. On the **Document Properties** tab, under **Drafting Standard**, click **Annotations**.

3. Under **Text**, click **Font**.

   The **Choose Font** dialog box appears.

4. Under **Height**, set **Points** to **12**, then click **OK**.

5. Under **Drafting Standard**, click **Dimensions**.

6. Select **Remove** in **Trailing zeroes** to remove all trailing zeroes from the dimensions displayed.

7. Click **OK**.

Creating a Drawing of a Part

1. Open **Tutor1.sldprt** if it is not open. Then return to the drawing window.

2. Click **Model View** on the Drawing toolbar.

   The pointer changes to .

3. In the PropertyManager, do the following:

   a. Under **Part/Assembly to Insert**, select **Tutor1**.

   b. Click **Next** .

   c. Under **Orientation**:

      - Click ***Front** under **Standard views**.

      - Select **Preview** to display a preview in the graphics area.

   d. Under **Options**, select **Auto-start projected view** to automatically display the **Projected View** PropertyManager when you place an orthogonal model view.

   e. Under **Display Style**, click **Hidden Lines Removed** .

   f. Under **Scale**, select **Use custom scale, User Defined,** and set to **1:4**.

4. Move the pointer into the graphics area.

   The pointer changes to with a preview of the front view of **Tutor1.sldprt**.

5. Click to place the front view as **Drawing View1**, as shown below.

6. Move the pointer up, and click to place **Drawing View2**, then move to the side and click to place **Drawing View3**.
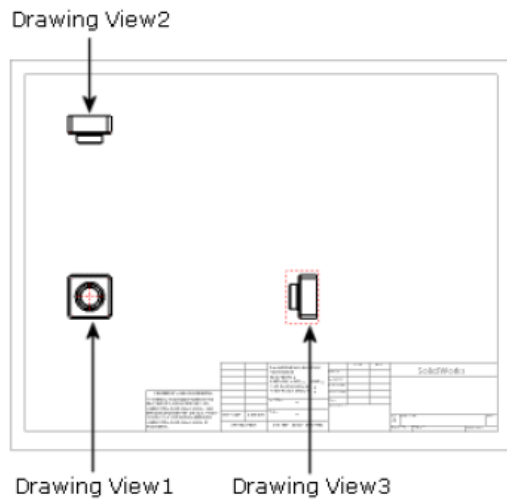
7. Click **OK** ✔.

This tutorial uses **Third angle** projection, so **Drawing View2** is the **Top** view, and **Drawing View3** is the **Right** view.

In first angle projection, **Drawing View2** is the **Bottom** view and **Drawing View3** is the **Left** view.

Moving Drawing Views

You move a view by clicking and dragging when the pointer changes to include ✛. This pointer appears when you are over the view border, a model edge, and so on. You can drag the view in its allowed directions.

1. Click **Drawing View2** (the upper left view on the sheet), then drag it up and down.

2. Click **Drawing View3** (the lower right view), then drag it left and right.

   Drawing View2 and **Drawing View3** are aligned to **Drawing View1**, and move in only one direction to preserve the alignment.

3. Click **Drawing View1** and drag it in any direction. The other two views move to maintain

alignment with **Drawing View1.**

4. Move the views on the drawing sheet to the approximate positions shown.
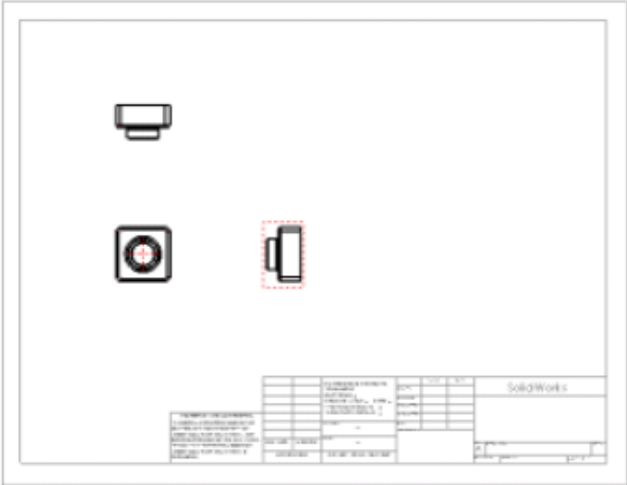
Adding Dimensions to a Drawing

Drawings contain 2D views of models. You can choose to display dimensions specified in the model in all of the drawing views.

1. Click **Model Items** on the Annotations toolbar.

    The **Model Items** PropertyManager appears. You can select which types of dimensions, annotations, and reference geometry to import from the model.

2. Under **Source/Destination**:

    - Under **Source**, select **Entire model** to import all the model dimensions.

    - Select **Import items into all views**.

3. Under **Dimensions**:

    - Click **Marked for drawing** to insert only those dimensions that are marked in parts for drawings.

    - Select **Eliminate duplicates** to insert unique model items only.

4. Click **OK** .

    Dimensions are imported into the view where the feature they describe is most visible.
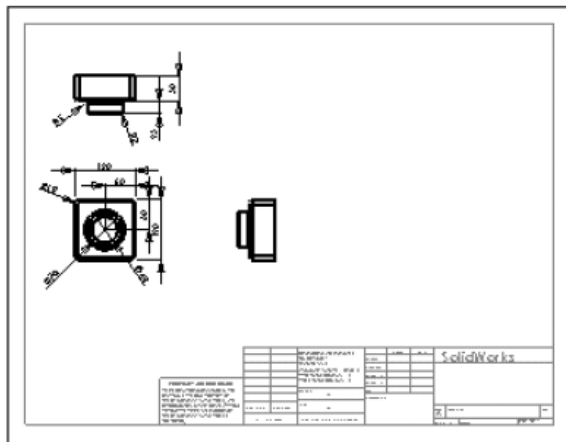

5. Drag the dimensions to position them as shown.

6. Click **Save** on the Standard toolbar and save the drawing document as **Tutor1**. The default extension is **.slddrw**.

Modifying Dimensions

When you change a model dimension in the drawing view, the model is automatically updated to reflect the change, and vice versa.
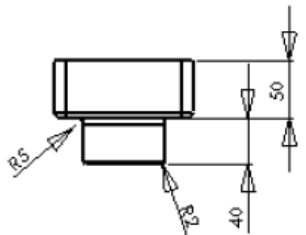
1. In **Drawing View2**, double-click the dimension for the depth (**25**) of the boss extrusion.

    The **Modify** dialog box appears.
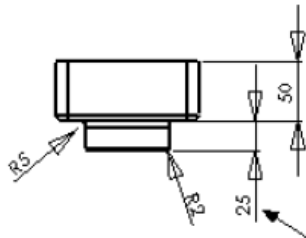
2. Change the value from **25** to **40**, and click **Rebuild** .

    The part rebuilds using the modified dimension. Both the drawing and the part are updated.

3. Click .

4. Save the drawing.

    The system notifies you that the model referenced in the drawing has been modified, and asks if you want to save it.

5. Click **Save All** to save both the drawing and the updated model.

Modifying Dimensions (continued)

Now check the part.

1. Click **Window**, and select the **Tutor1.sldprt** window.

2. Double-click **Boss-Extrude2** in the FeatureManager design tree to display the dimensions of the feature.

   Notice that the depth dimension is 40mm.

3. Click anywhere in the graphics area to turn off the dimensions.

Now rebuild the assembly that contains the modified part.

1. Open **Tutor.sldasm** if it is not still open.

   If a message appears asking you to rebuild the assembly, click **Yes**.

   ☀ If the message does not appear, click **Rebuild** ⬛ on the Standard toolbar.

   The assembly rebuilds with the new dimensions.

2. Save **Tutor.sldasm**, then return to the drawing window.

Adding Another Drawing Sheet

Now you create an additional drawing sheet for the assembly.

1. If the PropertyManager is still open, click **OK** ✔ to close it.

2. Right-click on any open area of the drawing sheet and select **Add Sheet**.

   Another sheet of the same size as Sheet1 is added to the drawing.

3. Click **Standard 3 View** 🖼 on the Drawing toolbar.

4. In the PropertyManager, select **Tutor.sldasm** 🟩 then click **OK** ✔.

5. Reposition the views on the sheet as shown below.
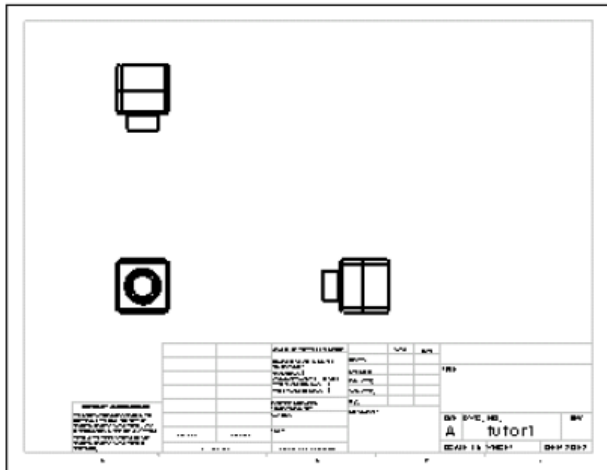
💡 You can use **Standard 3 View** 🖼 to add all three standard views to a drawing at once, or you can use **Model View** 🗔 to add one view at a time. The resulting views are the same.

Inserting Another View

You can add more views to drawings to show the model in different orientations.

In this topic you add a standard isometric view of the assembly.

1. Click **Model View** on the Drawing toolbar.

2. In the PropertyManager, do the following:

   a. Under **Part/Assembly to Insert**, select **Tutor** .

   b. Click **Next** .

   c. Under **Orientation**, click **Isometric** under **Standard views**.

   d. Under **Display style**, click **Shaded With Edges** .

   e. Under **Scale**, select **Use sheet scale**.

   The pointer changes to

3. Click in the sheet to place the view.

4. Click .
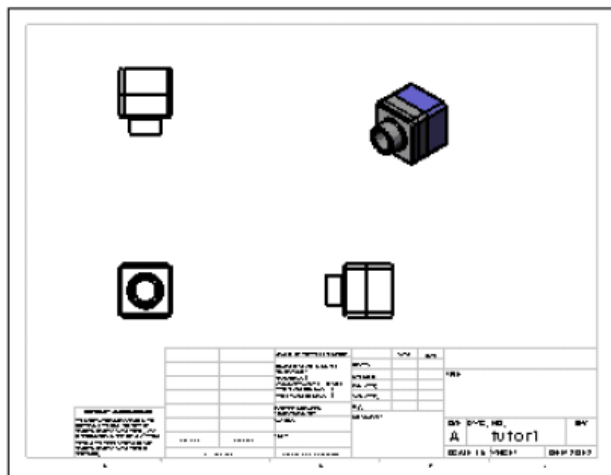
Printing the Drawing

1. Click **File**, **Print**.

    The **Print** dialog box appears.

2. Under **Print range**, select **All sheets** to print both sheets.

3. Click **Page Setup**.

    The **Page Setup** dialog box appears, where you can change printer settings such as resolution, scale, paper size, and so on.

4. Under **Resolution and Scale**, select **Scale to fit**.

5. Click **OK** to close the **Page Setup** dialog box.

6. Click **OK** again to close the **Print** dialog box and to print the drawing.

7. Click **Save** on the Standard toolbar.

8. If the system notifies you that the model referenced in the drawing has been modified, and asks if you want to save it, click **Save All**.

9. Close the drawing.

Congratulations! You have completed this lesson.