

# Making a Drawing

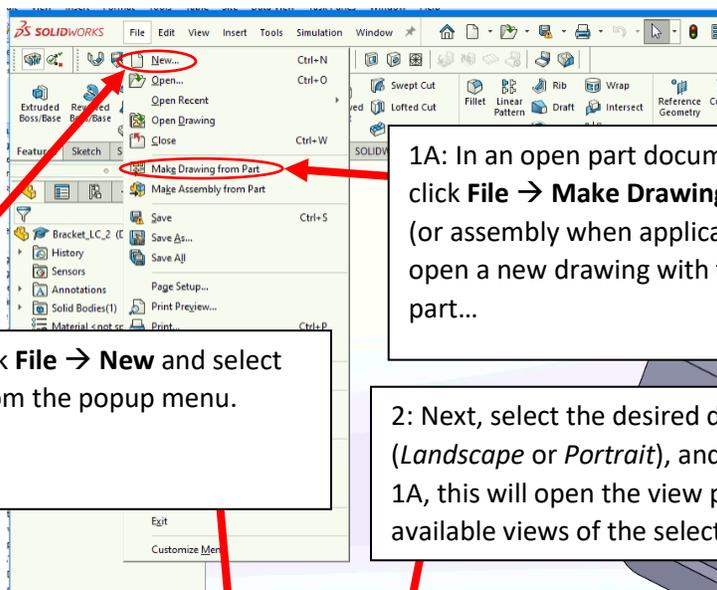
Understand how to:

1. Create a new drawing.
2. Select a sheet template.
3. Insert and edit model views.

**Drawing Purpose:** Allows a user to make a two-dimensional representation of parts and assemblies. Drawings are created to convey to the designer, client, and or manufacturer the requirements of a part or assembly such as size, complexity, and cost. The ultimate goal of a drawing is to convey enough information to manufacture the part or parts with ease.

*Note: The United States employs third angle projection, as will this course, while Europe uses first angle projection.*

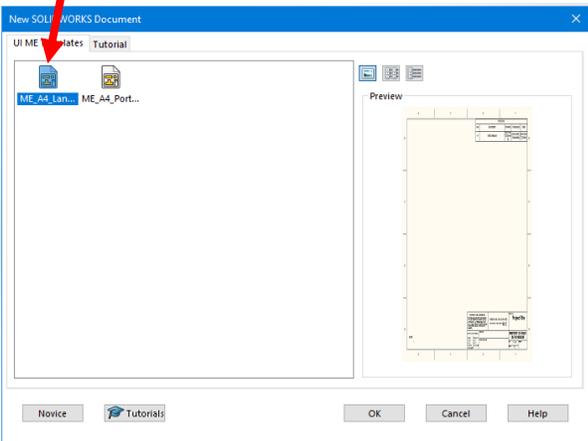
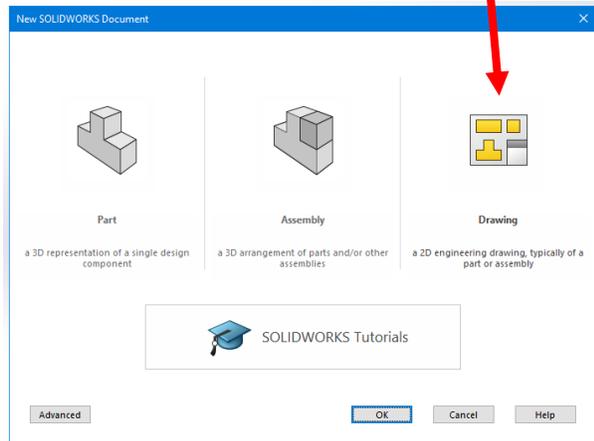
## Creating a Drawing for a Single Part



1A: In an open part document, left-click **File** → **Make Drawing from Part** (or assembly when applicable) to open a new drawing with the current part...

1B: ...or click **File** → **New** and select **Drawing** from the popup menu.

2: Next, select the desired drawing template (*Landscape* or *Portrait*), and click OK. From 1A, this will open the view palette with the available views of the selected part.



## Choosing a Sheet Template

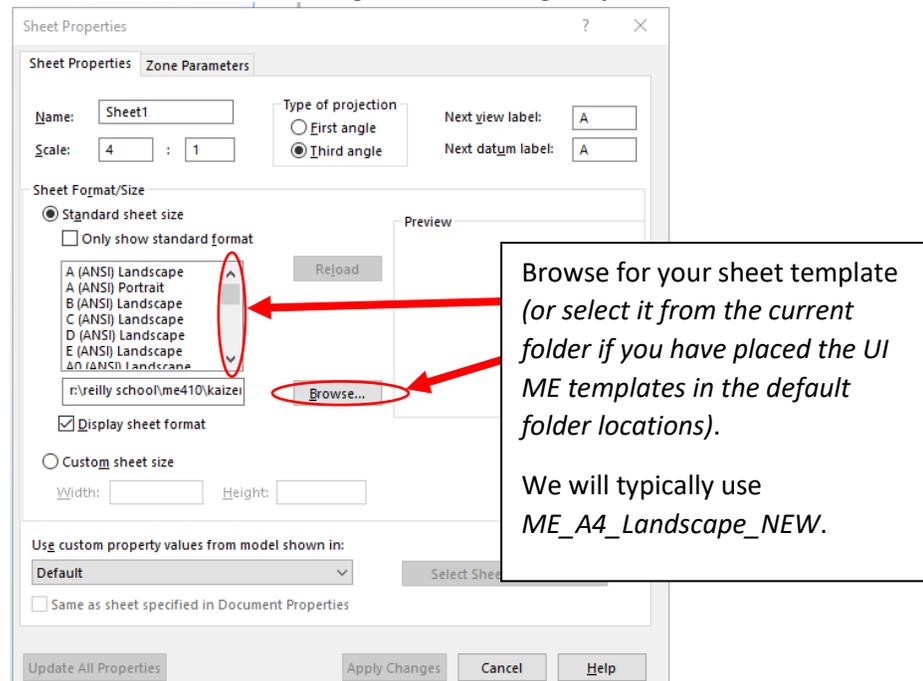
*Note: The Mechanical Engineering department at the University of Idaho uses a standard sheet template that can be downloaded in a zip file from the Mindworks course page (on Day 6), or from the Mindworks Course Resources page.*

Included in the zip file are drawing templates in Landscape and Portrait. We will typically use ME\_A4\_Landscape\_NEW.drwdot which loads the sheet format from ME\_A4\_Landscape\_NEW.slddrt. Locate the files and save a copy to a personal folder for you to use in upcoming assignments.

*Instructions are also available to place the files into the default folders where SolidWorks will automatically look for them (if you have Admin rights on your computer).*

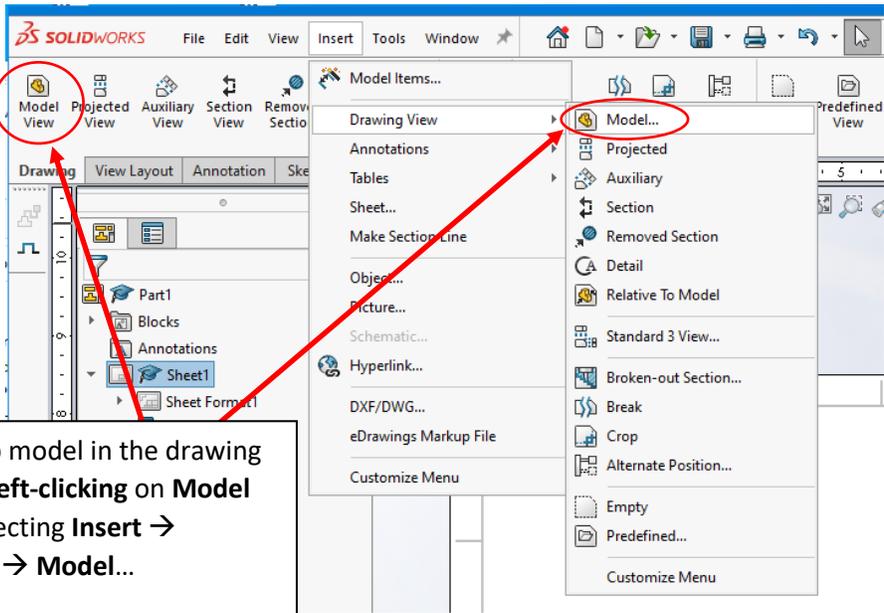
## Sheet Format/Size

If your drawing doesn't have a sheet format or is using the wrong one, you can edit it by right-clicking on the sheet tab or the sheet name in the Feature Manager and selecting **Properties**.



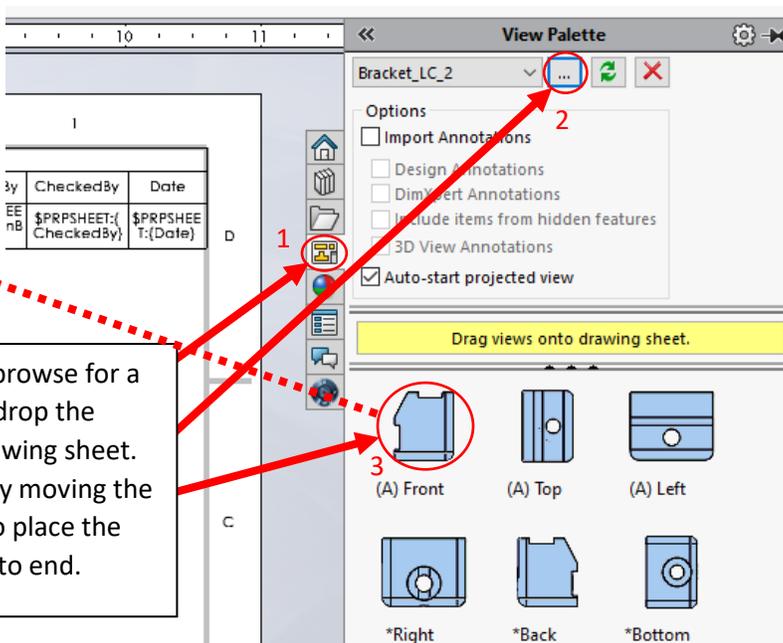
# Inserting a Model

## Methods 1 and 2:



Insert a part to model in the drawing document by **left-clicking on Model View** or by selecting **Insert → Drawing View → Model...**

## Method 3:



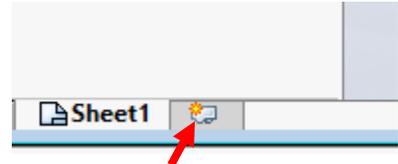
Use the View Palette to browse for a part, and then drag and drop the desired view into the drawing sheet. Continue placing views by moving the cursor and left-clicking to place the desired view. Right-click to end.

# Adding Additional Sheets for Additional Views and Parts

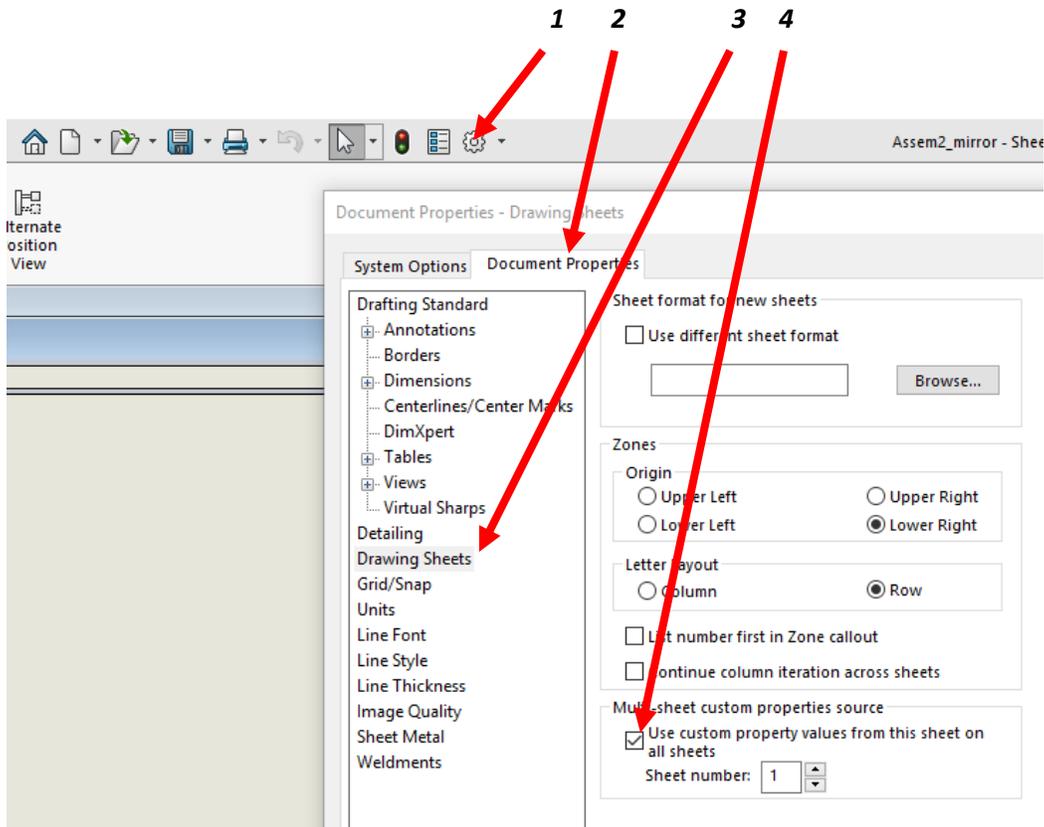
Add additional sheets to the drawing by clicking the *New Sheet* icon (  ) at the bottom of the drawing window to the right of “Sheet1”.

This can be used for:

- a) complex parts that require multiple views, or
- b) adding additional parts to the drawing package.

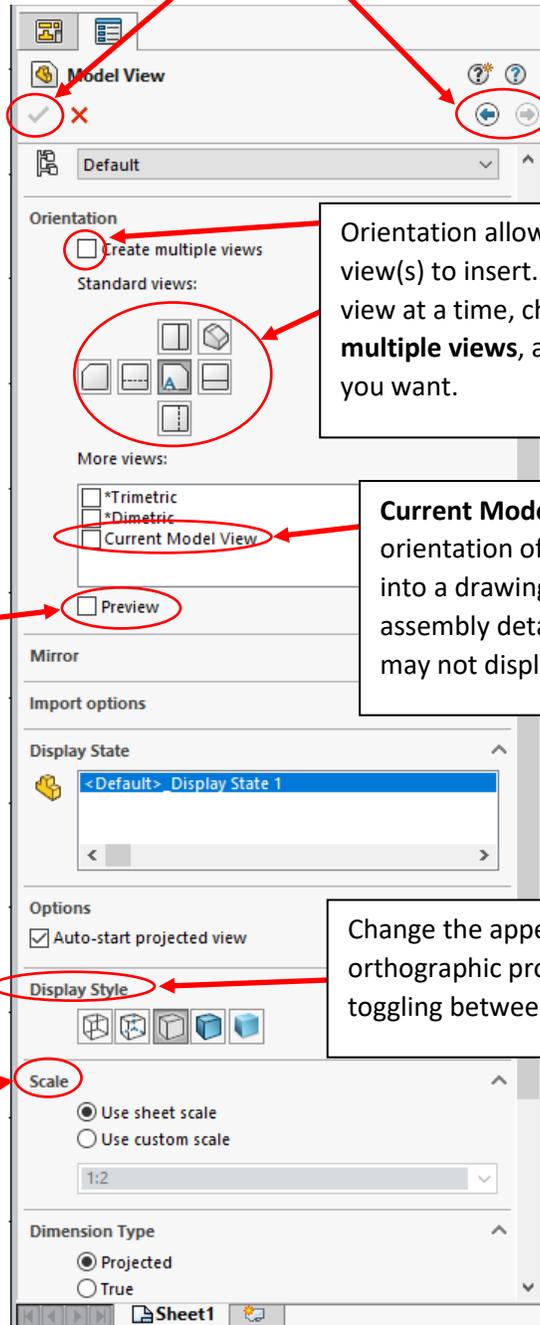


*Note that when you include multiple parts in the same drawing file, you might first need to uncheck a default setting that makes all title blocks dynamically linked to the part on Sheet 1 regardless of the part shown on a particular sheet. This isn't what you want for a drawing package with multiple parts (i.e., you want different information to propagate into each sheet's title block based on the part you bring in to that sheet). To make sure this setting is unchecked, go to Options → Document Properties → Drawing Sheets. Then make sure in the “Multi-sheet custom properties source” section, the box is unchecked for “Use custom property values from this sheet on all sheets”.*



# Model View Property Manager

Press the back arrow to select a different model and the green check to accept the view changes.



To see a preview of the part orientation on the sheet, check the box next to **Preview**.

Orientation allows user to specify which view(s) to insert. To insert more than one view at a time, check the box by **Create multiple views**, and then select the views you want.

**Current Model View** inserts the current orientation of the open part of assembly into a drawing. Useful for showing part or assembly details that an isometric view may not display as well.

Change the appearance of the 2D orthographic projections on the model by toggling between the choices.

Change the scale of the model – use either the user-specified sheet scale, or create a new scale.

*Note: if custom scale is selected ensure that the title block displays the true overall scale. For singular drawing views that differ from the main scale of the drawing, a note stating the scale of the view is sufficient.*