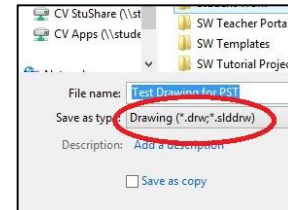


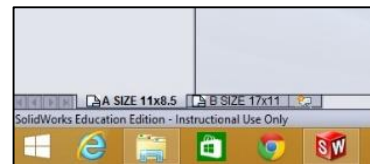
How to Make and Print Drawings Using SolidWorks 2013 and a Pre-made Template

1. Open Solidworks.
2. Use the OPEN command from the FILE menu to open either the *CVHS INCH TEMPLATE* or *CVHS MM TEMPLATE*. Which one you open depends upon the units you intend to use. Your instructor has either provided these files to you or has told you where they can be found.

3. Immediately go to the FILE menu and save your drawing as a drawing file with a unique name. Make sure the file type is a "Drawing" which should give your file a .slddrw extension. **IF YOU DON'T DO THIS STEP, YOU WILL OVER-WRITE YOUR TEMPLATE FILE (which has a .drwdot file extension).**

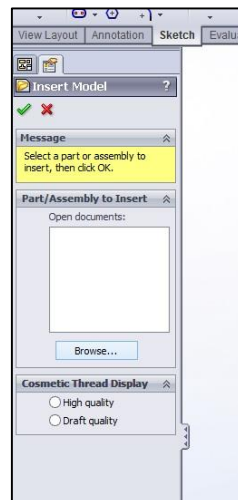


4. On the bottom of the screen there are two tabs: "A SIZE 11x8.5", and "B SIZE 17x11". Make sure the tab for the paper size you wish to use is selected. This determines the paper size you will be printing your drawing on.

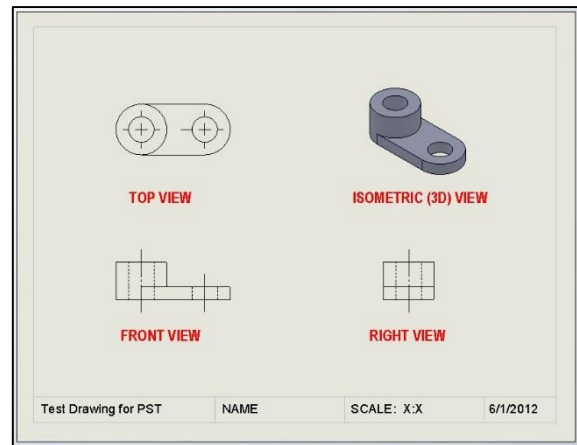


5. A blank page with four dashed-outlined boxes should be in the graphics area. Right click on the bottom left box and select the "Insert Model" option from the pop-up menu.

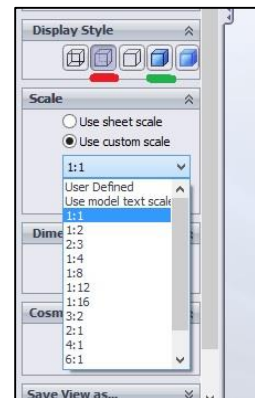
6. In the window on the left side of the screen (known as the Feature Manager area), you should see the options shown at left. Click on the BROWSE button and navigate to either the part file or assembly file that you wish to insert into your drawing. When you either double click the part file or select the INSERT button, the four blank boxes on the drawing sheet should be populated with various views of your part (or assembly).



7. Likely you will need to relocate the views. However, the various views are linked to one another – you can't just move them anywhere you want on the sheet. Also, each of the four views depict a specific view of your part as shown here. These limitations can be changed, however the settings built into the template are designed to satisfy most of your needs for this class. Please make sure each of the views shown to the right are positioned as shown on the drawing to the right. The red lettering is for illustration only and does not need to be on your drawing.



8. You may need to adjust the scale of your part if your part is too small to see well, or if the part is too big to fit on the drawing. To do this, click on the front view of your part. Options will appear on the left hand side of the screen in the Feature Manager. Scroll down to the SCALE option. When you select the “use custom scale” option, it allows you to pick a different scale to display your part. If you want to pick your own scale, you can select the “User Defined” option. You must scroll up in the pull-down menu to see this. To accept your change, click on the green check mark at the top of the Feature Manager window.



9. A part can displayed in various fashions. When you click on a view, you can change how it is displayed by selecting a different option under the “Display Style” portion of the feature manager. The “Hidden Lines Visible” (underlined in red in the picture) and “Shaded With Edges” (underlined in green in the picture) are the two options most often used. Your front, top and right views should have hidden lines visible. Your isometric, 3D view should be shown as a solid using shaded with edges. Your template will set your display states correctly, but now you know how to change them if necessary.

10. If your part comes into your drawing with the wrong orientation (i.e. the front view isn't what you want the front view to be), there is a way to change this. It is wrong because of the way you created it. To change, click on the view that should be the front view. The Feature Manager will again have options appear on the left side of the screen. Select a different view from the ones circled in red shown in the picture to the right here. It may ask you if you want to make the change because this view is linked to other views. Go ahead and say yes. You may need to try multiple views to get the one you want. Also, if your part is at an angle, it is likely because you didn't draw things horizontal, vertical, or on the Front, Top or Right planes. This can be changed as well, but it requires changing the part file.



11. To modify the information in the title block, you need to edit the sheet format itself. To do this:
- Right click somewhere on the drawing.
 - Select “Edit Sheet Format”. Your drawing views will disappear. Don't panic. Go ahead and modify the text in the title block as necessary by double clicking on the text and making the needed changes. Change the Title, Name, Scale, and Date text using ALL CAPS.
12. When finished, you can revert back to your drawing by right clicking somewhere on the drawing and selecting “Edit Sheet” (notice the Edit Sheet Format has changed to this option). You can toggle back

and forth between your border and your drawing in this manner anytime you wish. Sometimes you will want to include dimensions in your drawing.

13. **(OPTIONAL)** While you can insert your own dimensions manually, it is easier to start by importing them into your drawing first. Under the ANNOTATIONS tab, select MODEL ITEMS. In the Property Manager box:
 - a. Under SOURCE/DESTINATION select “Entire Model”.
 - b. Check the box that says “Import Model Into All Views”.
 - c. Under dimensions, usually you want every option selected, and DO NOT check the “Eliminate Duplicates” box. It is better to import too many dimensions at first and then delete redundant ones. SolidWorks, when it eliminates duplicates, does not always choose the proper dimension to import while it eliminates duplicates.
 - d. Under ANNOTATIONS, check the “Select All” box.
 - e. Under REFERENCE GEOMETRY, check the “Select All” box.
 - f. Under OPTIONS, check the “Include Items From Hidden Features”. You don’t generally want to dimension to a hidden line, however sometimes it is helpful to have the dimension first show up before changing it to comply with our class drawing standards.
 - g. Click on the green check mark at the top of the Property Manager. Dimensions will be imported into the three projected views.
14. **(OPTIONAL)** You almost always will need to reposition and modify the dimensions that SolidWorks provides. Use the SKETCH tab and SMART DIMENSION to add dimensions. Use the NOTE command under the ANNOTATIONS tab to add general notes on your drawing.
15. **(OPTIONAL)** You almost always will need to add center lines on some features. Insert them by using the “Centerlines” command under the ANNOTATIONS tab. Center marks can be added in the same fashion.
16. **To Print your Drawing:**
 - a. Always do a print preview first. If it doesn’t look correct here, it won’t look correct when it comes off the printer.
 - b. For A-Size drawings, be sure to print on the F11-LJ-4050 printer (located in the NW corner of F-11).
 - c. For B-Size drawings, be sure to print on the F11-LJ-5200 printer (located in the SE corner of F-11).
 - d. Be sure to enable the **print all text in black** option under the Advanced Tab in the printer properties. Also, under Page Set-Up, be sure to select **print in black and white**. If you don’t do these things, some of your lines will appear faint on your print.