

How to Make and Print Drawings Using SolidWorks 2015

1. Open the template file you have been given. (If you need to download this file, be sure to extract it to your H-drive first, and then open it from within Solidworks).
2. Immediately do a "SAVE AS", located under the FILE menu. Save it to your H-drive with a new name, and change the file type to a drawing (slddwg extension).
3. Right click on the bottom left empty view. Select "INSERT MODEL" and open up the part or assembly that you want to make a drawing of. Views of your part should appear in the empty view boxes.
4. Adjust the scale of your views as required so that most of the sheet is used.
5. If the isometric view does not appear, delete the empty view (the dashed square).
6. Under the "VIEW TAB", use the "PROJECTED VIEW" option to insert a new isometric view, using the bottom left view (what should be the front view) as the source view.
7. Reposition the isometric view by moving the cursor over the view until a move symbol appears. Click and reposition as necessary.
8. Hover your cursor over the isometric view and right click. In the Property Manager, change the view style to SHADED WITH EDGES.
9. Confirm that the front, top, and right side views have a view style of HIDDEN LINES VISIBLE.
10. Verify that the front, top, and right side views are aligned to one another.
11. Insure your front view is actually your front view. If completing drawings of objects provided by your instructor, your front view and the instructors should match. If they do not, click on the lower left view and change the source view until it looks correct. This all will depend upon how you created your part in Solidworks. If you find you need to rotate the view as well, you can do this by selecting the view, right clicking, selecting ZOOM/PAN/ROTATE, and then selecting the ROTATE option.
12. Insert dimensions from the ANNOTATION tab using SMART DIMENSION. Alternatively, sometimes 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.
13. You almost always will need to reposition and modify the dimensions that SolidWorks provides when you import dimensions from the model. Use the SKETCH tab and SMART DIMENSION to add dimensions. Use the NOTE command under the ANNNOTATIONS tab to add general notes on your drawing.
14. Delete all redundant dimensions – they just clutter your drawing.

15. Insure features requiring centerlines and center marks have them. Both can be added manually from the ANNOTATIONS tab as well as the menus.
16. To modify the information in the title block, you need to edit the sheet format itself. To do this:
 - a. Right click somewhere on the drawing.
 - b. 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.
 - c. 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.
17. 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. To get it to look correct, insure you have:
 - b. The F11 printer selected (located in the NW corner of F11) – this may not be the default printer.
 - c. Drawing color is marked as BLACK AND WHITE in the PAGE SETUP Window.
 - d. After selecting PRINT, in the print window make sure that CURRENT SHEET is selected for the Print Range. If you don't do this, sometimes it will print an extra blank page which just slows printing down (and wastes paper).
 - e. If you make any changes to your printer's settings, look at the print preview again and insure it is what you want – you only get one chance to print and then it will be graded – correct or incorrect.
 - f. For B-Size drawings (17x11), be sure to print on the HP-5200 printer (located in F12). To do this, you will also need to use the 17x11 tab of your drawing template (located at the bottom).