How to Make Drawings

1. SVN Update (go to the topmost 2011 folder, right click, and go down to SVN update). This will download all of the drawings done by other people to your computer, as well as update any CAD files or drawings that have been changed since you last SVN Updated.
2. Check to see if the part you want to do a drawing for already has a drawing or is being worked on by going to the Google Doc listing all the parts. BEFORE YOU START, put your name into the "Drawing done by" column.

3. Determine if the part is made by Berger or Perry. All sheet metal parts are made by Berger, and everything else is made by Perry. If you are not sure, ask someone.

4. Open the part in SolidWorks, click on File, and then click on Make Drawing From Part.

5. If the part is made by Berger, select "B Landscape" for the sheet format. Otherwise select "A Landscape." (B Landscape is a 17"x11" sheet of paper. Berger can print these. A Landscape is 11"x8.5", which we can print.)

6. Drag the MOST INFORMATIVE view into the panel first. This will be the front view on the drawing, regardless of which view it is on the part. After that is done, drag to the right to put down a right view, and drag upwards to place a top view, then drag up and to the right diagonally to place an isometric view. For some parts, such as shafts, a top view may not be necessary. If this is the case, skip that step. Some parts may also need a bottom, left, or section views. A bottom of left view would be needed if the bottom or left is different from the top or right, respectively. Check the 3D model to see if a section view is needed to show any features that are not visible in any other views. Place bottom views by dragging down, and left views by dragging to the left. If you are unsure about what views you need, ask someone.

7. After you place all of the necessary views, right click the sheet, click on properties, and set it to Third Angle projection.
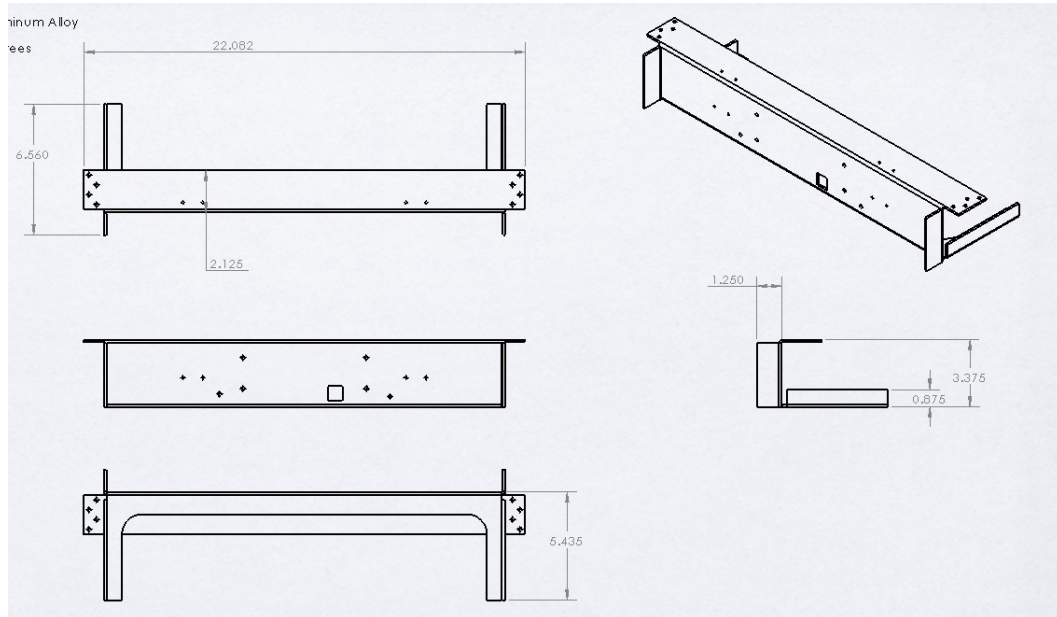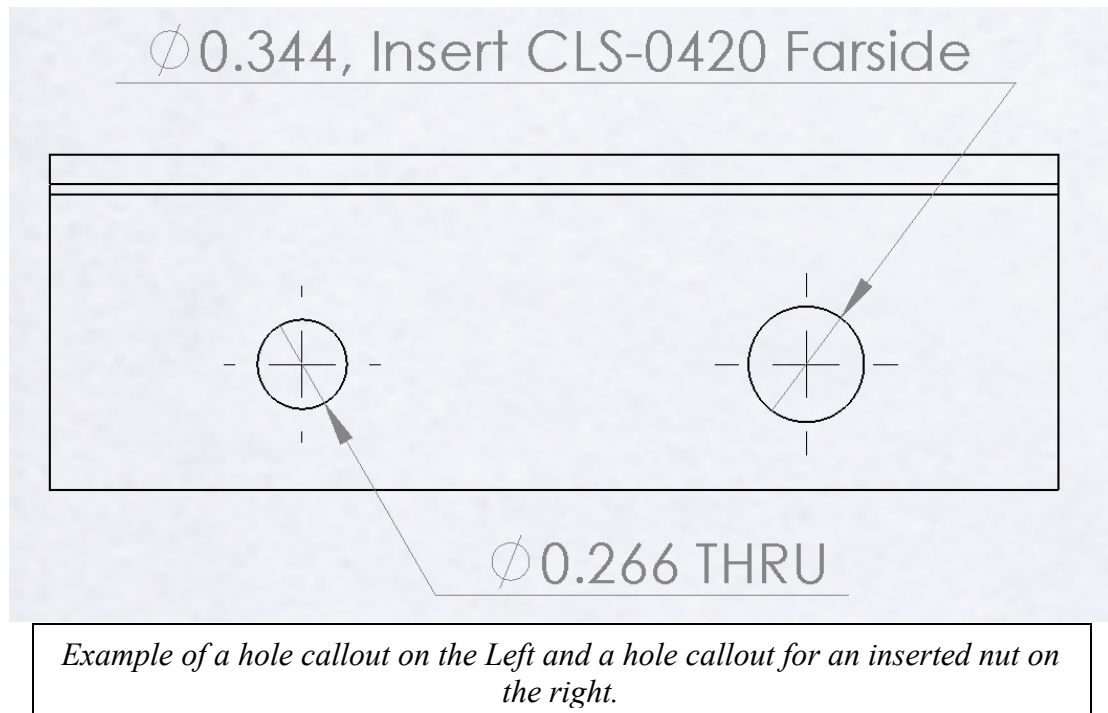
Example of a drawing with front, right, top, bottom, and isometric views.

8. Start adding dimensions using the Smart Dimension tool on the Annotations tab. Always give overall dimensions. Dimension whatever is most important to us, and give enough information for the machinist to make the part, but be careful not to double dimension. On sheet metal parts, dimension every bend that needs to be made, and give the dimension from one of the flat surfaces of the sheet metal. Make sure that you are in INCHES and have 3 DECIMAL PLACES. If you aren't, click on Tools, Options, then the Document Properties tab, Units, and change the primary units and the number of decimal places. On parts that are made by Berger, it is not necessary to dimension the locations of the holes or their sizes because the parts are laser cut from the CAD models. If there are a few very important holes in the part, those can be dimensioned. Place dimensions outside the drawing whenever possible. Give the angle of the bend if it is not 90°. Dimensioning the outside dimension is preferred.
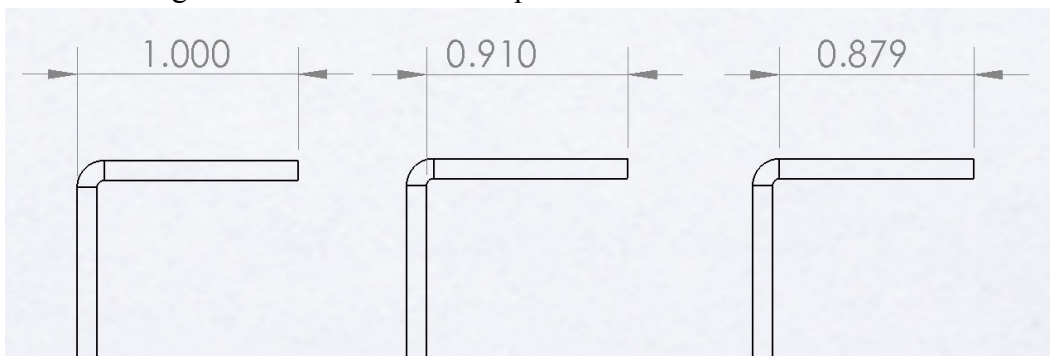
Three ways to dimension a bend. The left is preferred, the center is OK, and the right is wrong because the dimension is not to a flat surface of the sheet metal. Dimension whichever way shows what is important on the part.

9. If you need to dimension a hole, use the Hole Callout tool on the Annotations tab. You can edit the text of a Hole Callout by clicking the callout and editing the text in the Dimension Text box on the right. If there are multiple holes of the same size, instead of dimensioning each one, type "x2" as part of one of the hole callouts. If there are other holes of similar size, and it may be confusing which hole is which size, do a separate hole callout for each. For Berger parts, it does not matter too much since everything is laser cut, but for Perry parts, make sure that the size and location of each hole is clear. If a hole needs hardware inserted, such as an imbedded nut, check the robot assembly to see what the part number of the hardware is, and where it is inserted.

$\varnothing$0.344, Insert CLS-0420 Farside

$\varnothing$0.266 THRU

Example of a hole callout on the Left and a hole callout for an inserted nut on the right.

10. Add a Note to the top left of the sheet by using the Note tool on the Annotations tab. This note should list the following:
    a. Name – The name of the part from the Google Doc "_" Configuration Name. For example use names like "batteryholder_base", "line follower spacer _0.16", and "side plate_default".
    b. Number – The part number from the Google Doc.
    c. Revision level – If you check a drawing that needs changing for any reason (the first drawing was done wrong, the part changed since the drawing was done, etc.), update the revision. If you are doing a new drawing, it is revision A. Update the revision status in the Google Doc.
    d. The name of the CAD file for the part.
    e. The total quantity ordered – Can be found either by checking the Google Doc.

f. Material – For sheet metal, give the thickness of the sheet and the alloy. All of our sheet metal parts are 5052 Aluminum Alloy. (This alloy is less rigid and easier to bend than other alloys.) For other parts, say whether they are Aluminum or Steel.
g. If this part assembles to another assembly note the part number.
h. Add any additional notes which give clarification about how to make the part. For example, on a sheet metal part, you may write "All bends are 90° unless otherwise stated."

Don't write anything in the bottom right corner.

Name: Front Rail (Default)
number: 2010-068-1
Revision: C
CAD File: Front Rail 2010-068.SLDPRT
Quantity per Assembly: 1
Material: 1/8" 5052 Aluminum Alloy
Finish: None
Note: All bends 90 degrees

Example of a note in the top left corner.

11. Save the drawing in the same folder as the part with the PART NUMBER as the name of the drawing file. If SolidWorks tells you to save the part when you save the drawing, say OK. For example use names like "111-batteryholder_base.SLDDRW", "254-line follower spacer _0.16.SLDDRW", and "67-side plate_default.SLDDRW".
12. Check in your drawing by right clicking it in Windows Explorer, mousing over Tortoise SVN, and clicking Add. After this, right click in the empty space in the folder, and click on SVN Commit. Type in a log message which describes what you did, and make sure that you ONLY check the box by the files that you added (the drawing you just did). If you had to save the part, the part file will show up in the list of files to commit. Right click it and click on Revert. When the drawing is the only thing which is checked and you have entered a log message, click OK.
13. After you have checked in your drawing, enter the date that you finished in the Google Doc.
14. Do another drawing.

If you need any help on a drawing, ask someone. Scott, the mentors, and veteran team members can answer questions and are happy to do so. If I am online while you are working on a drawing, feel free to Gchat or Yahoo Messenger me. My Gchat ID is hershalspatel, and my Yahoo ID is pesky_mosquito. If you have a less pressing question, send me an e-mail.

Example of a completed drawing.