

Activity 5.1.C Pegboard Toy Working Drawings


Introduction

In the last activity, you made a virtual 3D model of a pegboard toy. In this activity, you will make for each part a working drawing, which is a multi-view drawing with specified dimensions. Such drawings are called “working drawings” because they are typically used as instructions to make the prototype. You will also make an assembly view and an exploded view, which show how the toy is to be assembled.

Procedure

1. Open Autodesk Inventor Professional.
2. New > Standard (in).idw > Create or New > Drawing

In Inventor, the default sheet size is C (17” x 22”). Here we want to change the sheet size to A (8.5” x 11”), to print more easily. When we change the sheet size, we also need to change the title box. Follow the instructions in the presentation *5.1.C Creating Drawings CAD* to change the sheet size, change the title box and create the drawing.

- a. In the browser, right click on *Sheet:1* and select “Edit Sheet...” Set *Size* to A, and click OK.
 - b. In the browser, below *Sheet:1*, right click on *ANSI – Large* and select *Delete*. This deletes of the C-sized title block.
 - c. In the browser, expand the folder *Drawing Resources* (by left clicking the “+”), and expand the subfolder *Title Blocks*. Double click on *ANSI A*, which places the A-sized title block on Sheet 1
 - d. Click on the Application Button (“I PRO”) in the upper left corner, and select *iProperties*. On the *Summary* tab, enter the Author with your first initial and last name, the Company as “SHS” and the Title as “Pegboard Toy”. Click OK and this information will be saved in the title block.
3. From the Place Views panel, select Base.
 4. Drawing View pop-up menu will appear. Select Open an Existing File.  Find and select your Pegboard.ipt.
 5. In the Drawing View window, set Scale to 1/2 and make sure Front View is highlighted. Lastly, select OK.
 6. Move the Pegboard to the lower, left hand corner and left mouse click to place the view.
 7. From the Place Views panel, select Projected.
 8. Click on the Front view of Pegboard and move your mouse straight up from the Front view then left click to place the Top view.
 9. Move your mouse down and the to the right of the Front View then left click to place the Right Side view.

10. Finally, move your mouse straight up from the Right Side view and left click to place the Isometric view.
11. After all views are placed then right click and select Create.
12. Double click on the Isometric view and set the Style to Shaded. Double click the Front view in the lower left, set the Style to Hidden Line and turn Shading off.
13. Your drawing should look similar to Figure 1. Note that dashed lines are used to show the location of internal structure, and are called “hidden lines”.

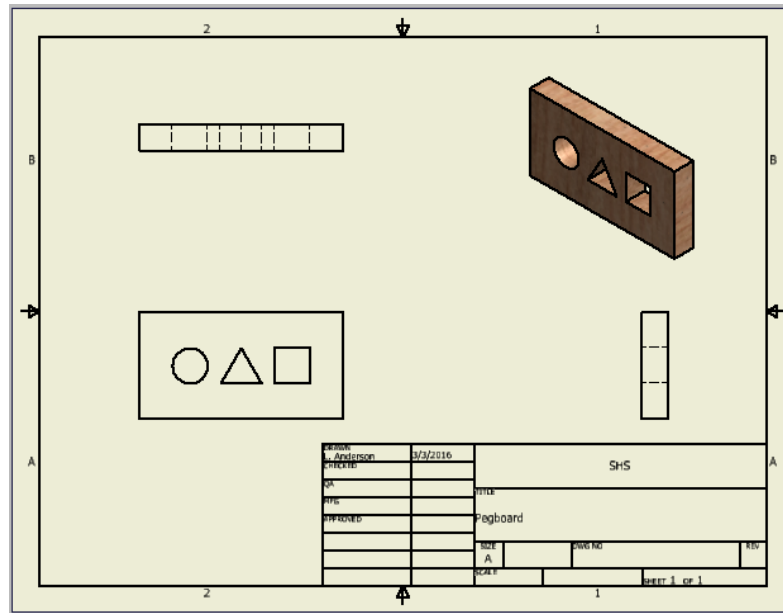


Figure 1.

14. Before continuing, save your drawing as Pegboard.idw.
15. Select the Annotate tab. Using the tools Dimension, Leader Text, Text and the symbol Center Mark, do your best to make your drawing match what is shown in Figure 2. Remember, the working drawing is used by someone to manufacture the part, so complete and accurate information is extremely important. Note that the symbol \varnothing is used to indicate a diameter, and leader lines are at an angle.

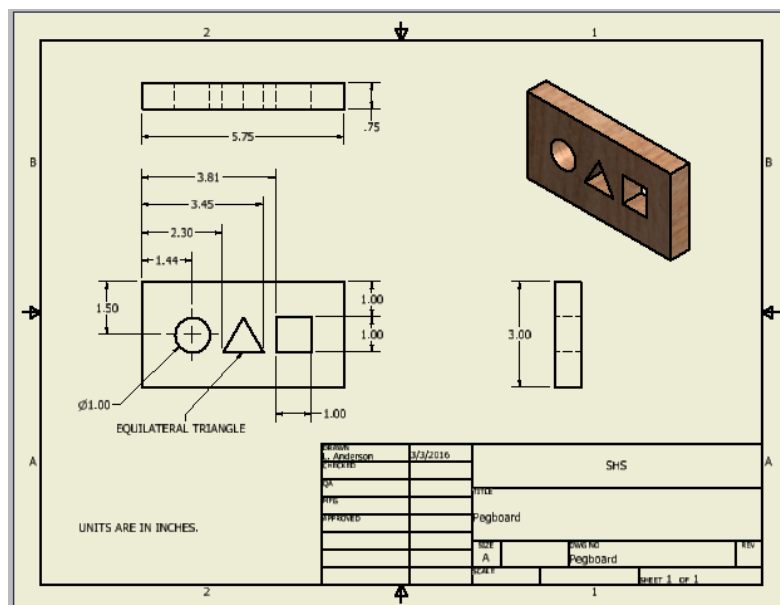
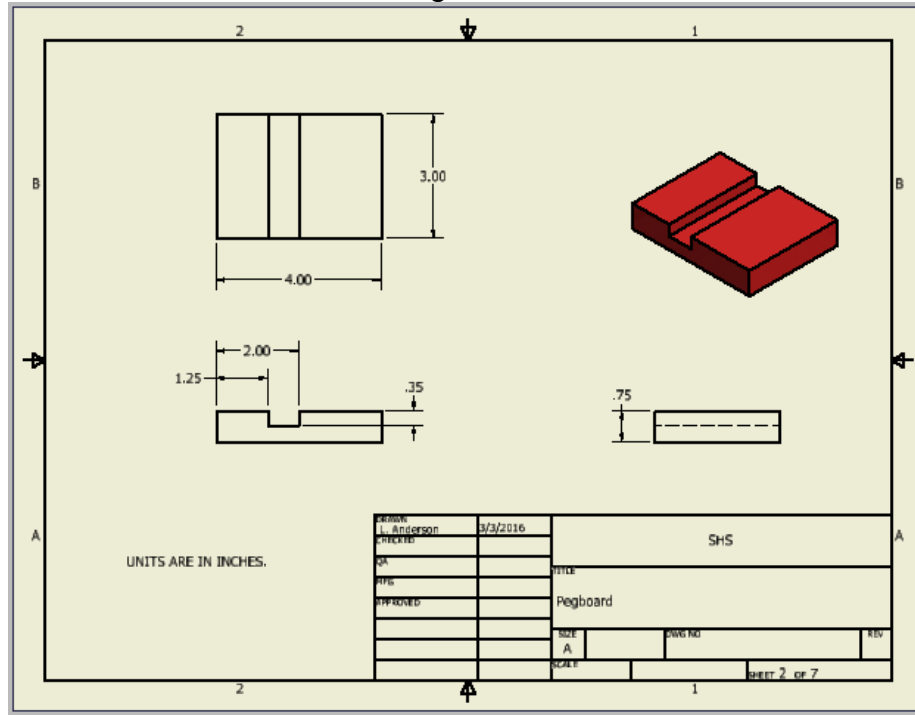


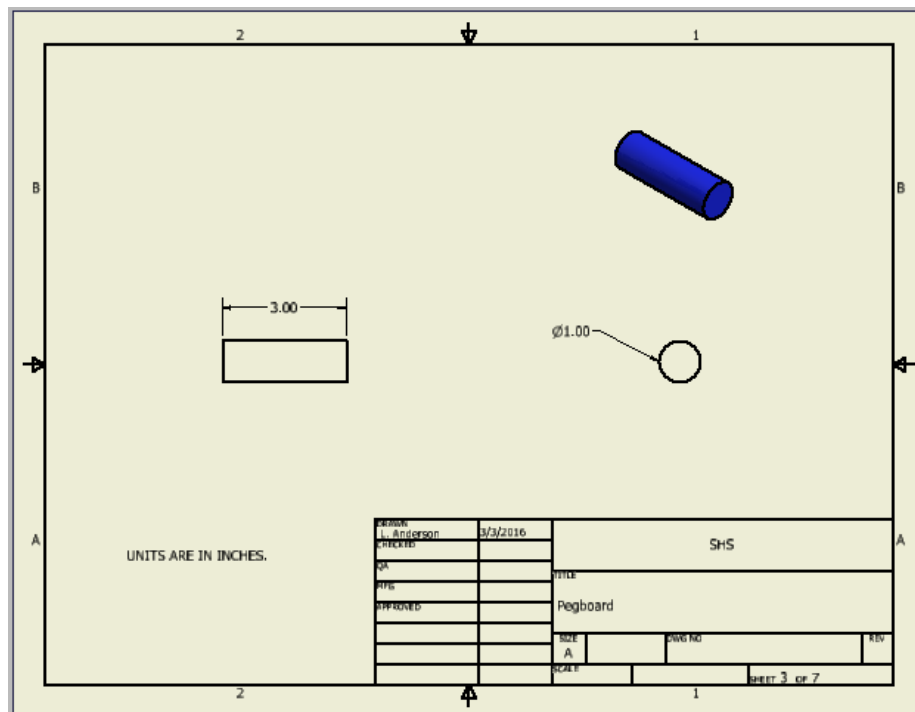
Figure 2.

16. Similar to a Microsoft Excel or Google Sheets document, an Autodesk Inventor drawing document can have multiple sheets. The convention is to have all the drawings necessary for an assembly within the same document, with each part on a separate sheet. Make a new sheet by selecting the *Place Views* tab and clicking on *New Sheet*. Note the page size and title block are kept the same.

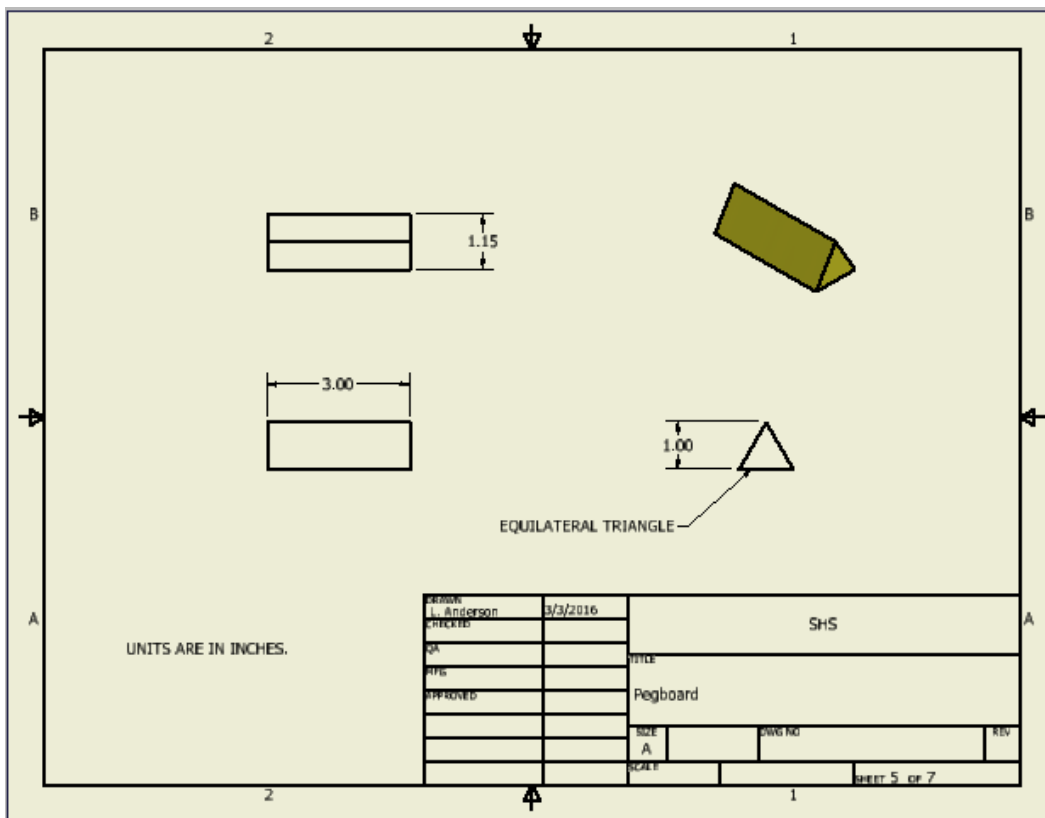
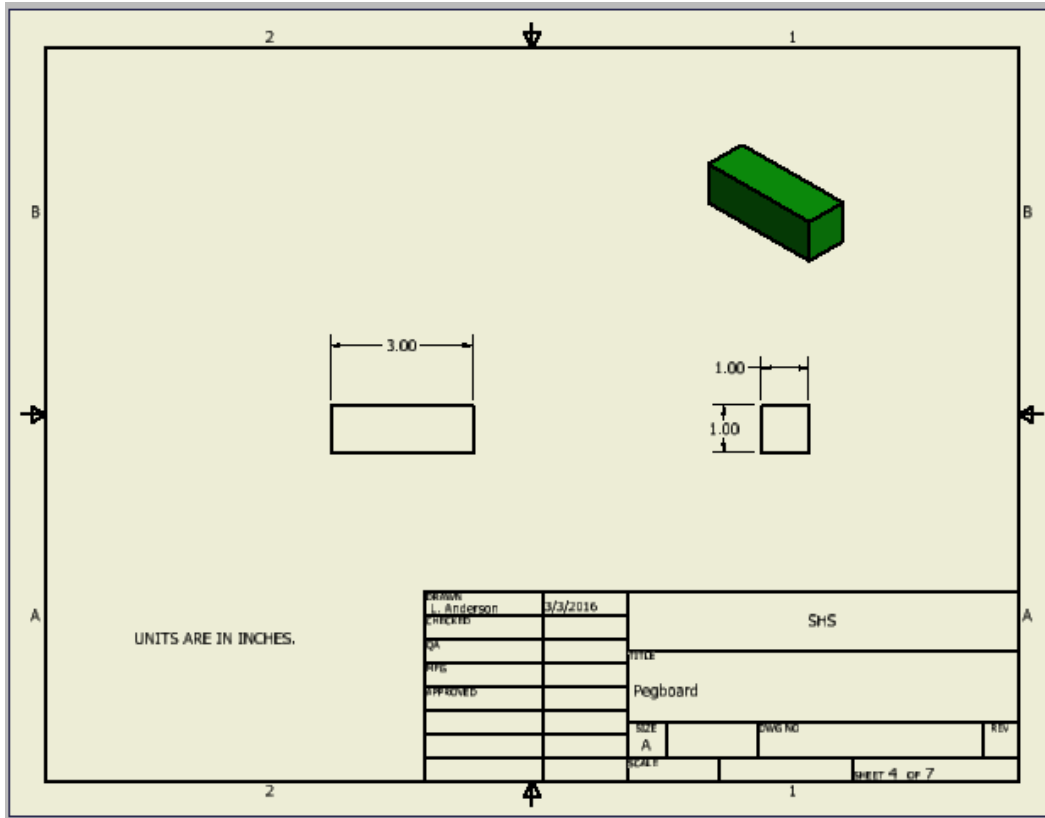
17. Using the same method described above, Place your End.ipt, setting the Scale to 1/2. Create a drawing similar to the one below.



18. Select the *Place Views* tab and click on *New Sheet*. Create drawings of the Round Peg, the Square Peg and the Triangle Peg, as shown below.

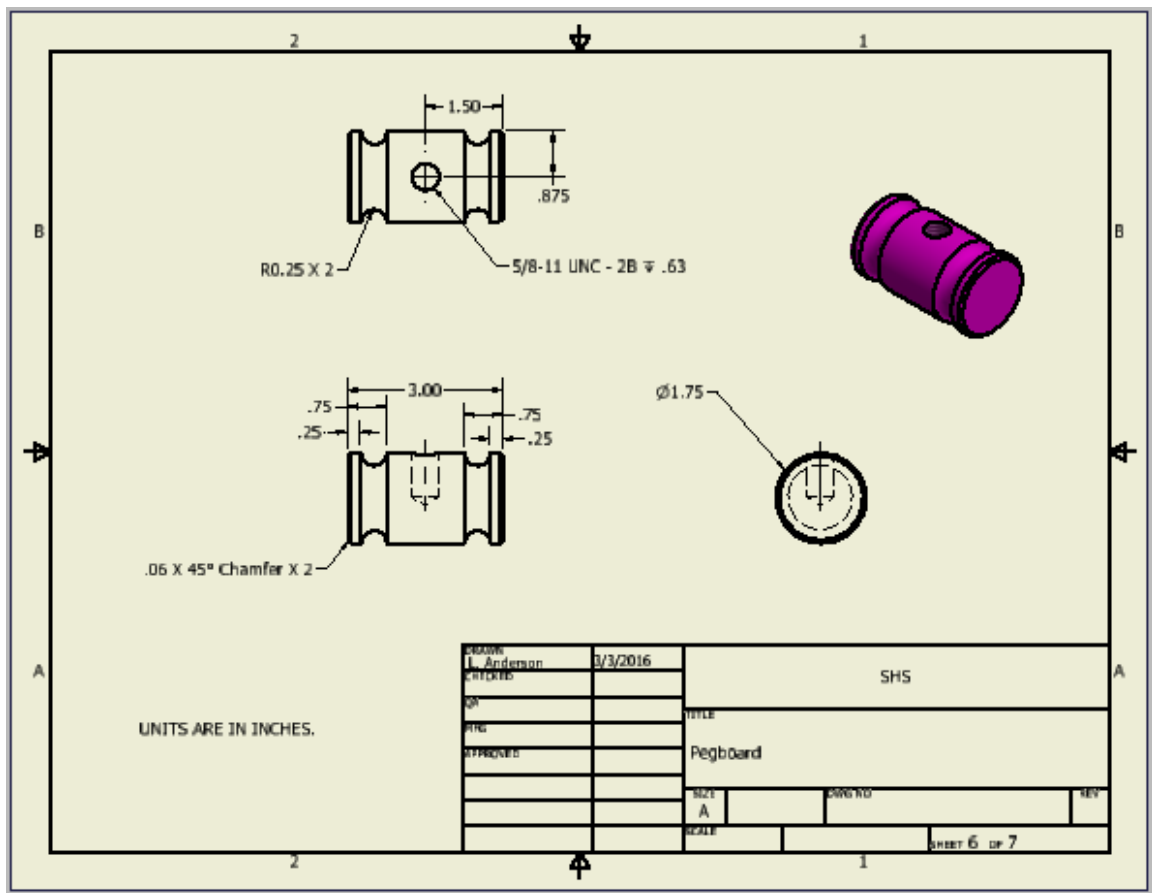


Note that for the round and square pegs, only two views are needed, because the Top view is identical to the Front view. Tip: Save your file after every new drawing!

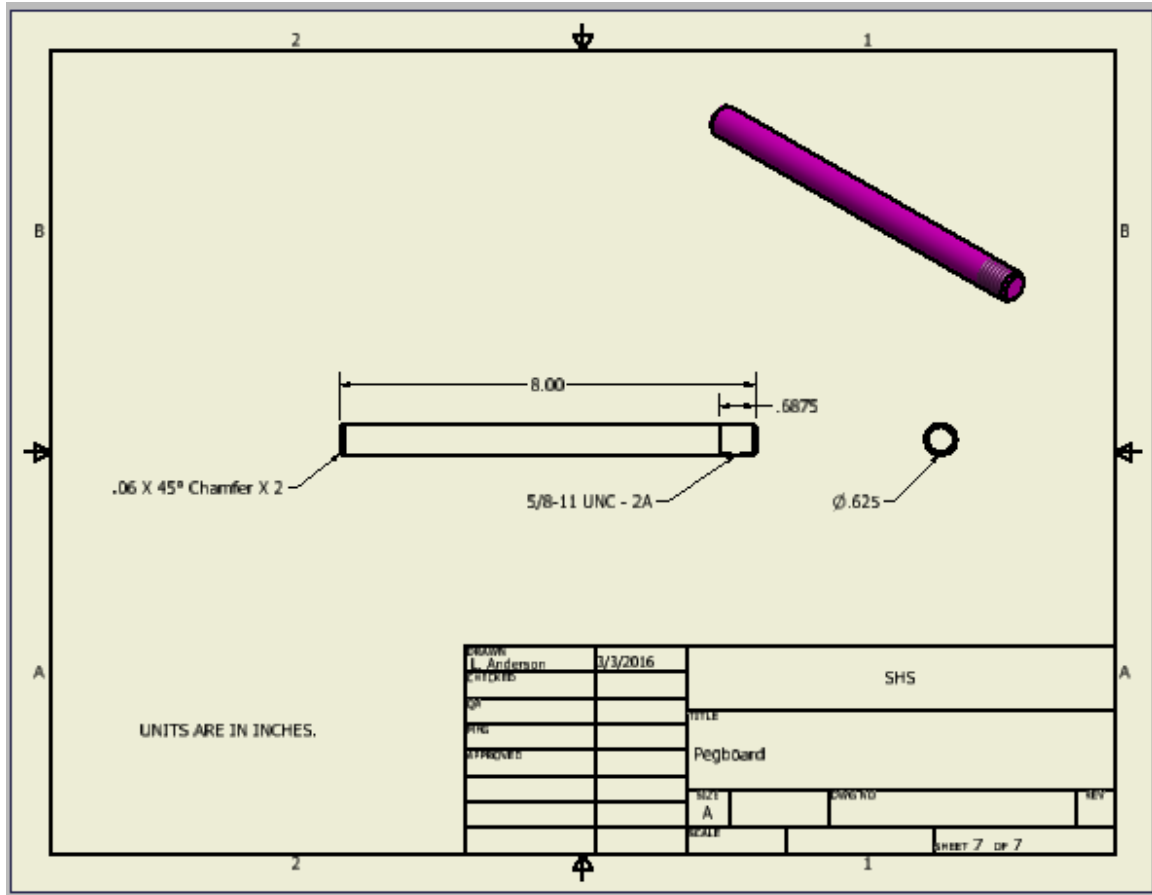


19. Mallet Head:

- Top View: Use the symbol *Center Mark* to specify the center of a circle as seen from above, which is used to locate the position of the hole. (Think of a person with a drill positioning the drill bit.) Use the tool *Hole and Thread* to indicate the hole characteristics. A circular arc (i.e. a partial circle) is specified by its radius of curvature, which is signified by the letter “R”. The “X 2” indicates that the other curve is identical (to avoid the clutter of duplicate text). Double click on a number to change the number of digits after the decimal.
- Front and Side View: Use the symbol *Centerline Bisector* to specify the axis of a round hole as seen from the side. To indicate the chamfer characteristics, select the *Chamfer* command, and double click on the chamfer (you may need to zoom in). Double click on the text to add “X 2”.
- Keep in mind that you need to use the Escape Key to exit a command, which you need to do before you can move or edit text or change sheets.



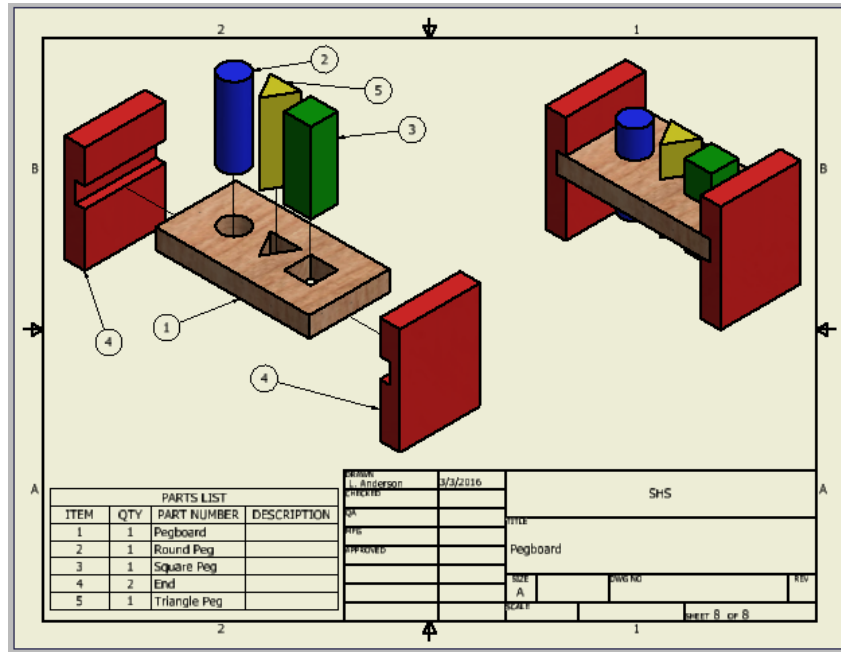
20. Mallet Handle: Because the Top view is identical to the Front view, only two views are needed. To show the threads, double click the Front view, select the Display Options tab and check Thread Feature. To indicate the tread characteristics, use the *Hole and Thread* command. To indicate the chamfer characteristics, use the *Chamfer* command, and double click on the chamfer (you may need to zoom in).



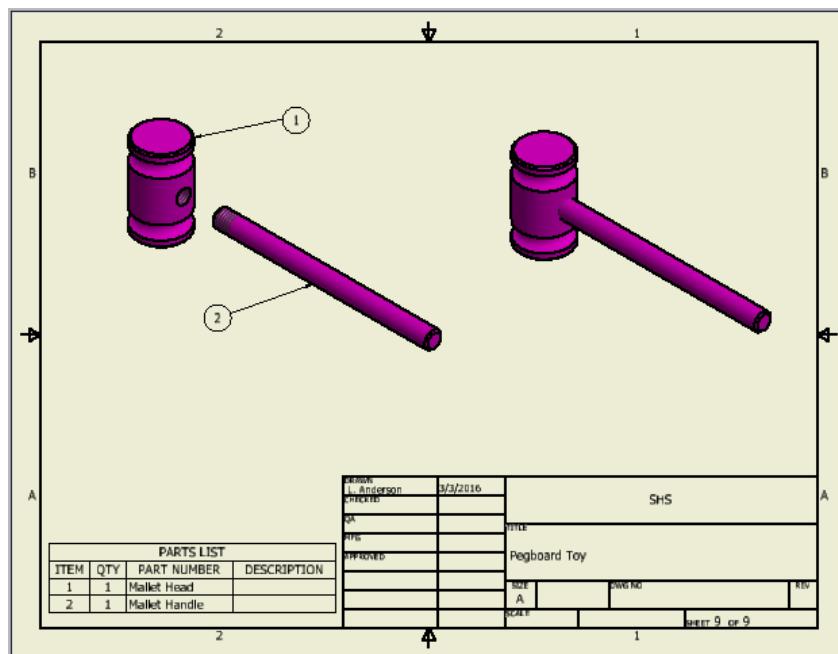
21. Finally, we will make drawings of the assembly and of the exploded view presentations, which often can serve as assembly instructions. From the *Place Views* tab, select *New Sheet*.
22. Select *Base* and import the pegboard assembly (Pegboard Toy.iam). Set Scale to 1/2, and rotate to an isometric view. Note the *Change view orientation* button is useful to do this.
23. Place the assembly view on the right side of the drawing. Shade it.
24. Select *Base* again, and import the presentation file (Pegboard Toy.ipn). Rotate the view to the same as the assembly view (or vice versa). Place the exploded view on the left side of the drawing. Shade it.
25. From the *Annotate* tab, select *Parts List*. Under Source, browse for and select the presentation file (Pegboard Toy.ipn). Click OK.

26. Place the parts list in the bottom left corner of the drawing. The parts list can be resized by clicking and dragging a corner. Resize it so it does not overlap the title block. Note the parts list can be edited by double clicking on it.
27. On the Annotate tab, select the Balloon command. Click on the edge of a part, move the cursor away from the part, left-click to place, and then right-click and click *Continue*. When finished, hit the *Escape* key to exit the *Balloon* command.

The numbers in the balloons correspond with the “Item” numbers in the parts list. Note the leader lines for the balloons should be placed at angles (i.e. not horizontal or vertical, to help avoid confusion with dimension lines).



28. In a similar manner, create an assembly view for the Mallet with parts list and balloons.



29. Save your drawing!
30. Move these two Assembly Drawing Sheets to be the first two sheets in the document. Do this by clicking on a Sheet icon in the Browser and dragging it.
31. Save the document as a PDF, by clicking on the Application Button (“I PRO”) > Export > PDF. Click on the “Options...” button at the bottom, and set the *Print range* to *All sheets*---otherwise only the first sheet will be saved to the PDF. Then choose a file name (e.g. Pegboard Toy) and click Save.
32. Show your drawing file (all sheets) to your teacher for credit.

Teacher's initials_____

33. Your PDF file could then be printed, or emailed to a manufacturer. Additionally, your presentation file animation could be “recorded” and saved as a .wmv or .avi video file, and posted to the internet to help illustrate the assembly process.

In this class we will be using the 3D printer, which requires parts in STL format. For printing purposes, it is best to print one part at a time.

In Inventor, open each part individually, and save in STL format by

Export > CAD Format > Save as type: STL Files (*.stl)

34. Open your STL files in MakerBot Desktop, to check on their size and orientation for printing on the Replicator 2X. If needed, change the orientation for best printing, and save.
35. In this manner, anything you create in Inventor can be saved as an STL, uploaded to your shared Google Drive folder, and sent to the 3D printer!