EML2322L – Design & Manufacturing Laboratory

SolidWorks Tips

The following tips offer suggestions for working more efficiently with SolidWorks. The goal is to help you communicate your ideas more effectively.

1. **Know your tool.** Your effectiveness in SolidWorks, like any other tool, improves when you understand its capabilities. Review the SolidWorks Resources available under the Project Resources submenu on the main course webpage.

2. **The internet is your friend.** If you think that SolidWorks *should* do something or the way you are doing something seems overly difficult, use a search engine to check if there is a better way. ***Quite often 15 minutes of research can save hours of frustration.***

3. **Use SolidWorks' sheetmetal tools for sheetmetal parts.** SolidWorks has a built in sheetmetal tool that allows you to create bent sheetmetal parts and flatten them (or vice versa) as required for the detail drawings of sheetmetal parts. Some video tutorials are available under the SolidWorks Resources page on the course website.

4. **Build (sub)assemblies from the bottom up.** Create simple subassemblies that are mated into more complex subassemblies and eventually into the full robot assembly. Doing so allows individual team members to work simultaneously on the assemblies (each on a different, unrelated sub-assembly) and simplifies creation of (sub)assembly drawings later. Many groups consciously avoid using proper subassemblies in CAD because they (incorrectly) feel it adds time to their work, when that couldn't be further from the truth! ***Use (sub)assemblies liberally, as doing so makes your lives so much easier when working with multiple teammates on fairly large assemblies.***

5. **Spend adequate time organizing.** I have never heard a group comment, "We just spent too much time organizing," but have frequently heard (and experienced) the opposite. Create a clear file structure for part and assembly storage, employ logical naming conventions, and use organized part numbering conventions. Always avoid duplicating files.

6. **Use Dropbox carefully.** There is no requirement to use Dropbox for this course, but many groups intelligently choose to do so. Dropbox stores files in different locations on different computers (your various Dropbox folders) while SolidWorks records the file locations *of the files as of the last save*. If the file path changes (which it often will using Dropbox on different computers), SW will not be able to locate the files and will lose data like exploded views. One solution to this problem is to designate a consistent file path on each team member's computer for SW files such as C:\EML2322L\... . Duplicate the Dropbox files into this location prior to editing them and copy updated files back into the Dropbox folder.

7. **Use reference geometry such as planes, lines, and points to facilitate proper mates.** This topic should have been covered in the CAD course, but it's a common area of weakness with new designers. When drawing a part you should always think about how it attaches to other parts and include necessary reference geometry to facilitate mating. Failure to do so is probably the leading cause for incomplete, unrestrained mates. Once all mating is performed, one simply turns off the associated reference geometry under the View menu to unclutter the final assembly.

8. **Mating to a sketch.** Elements in part model sketches (lines, points, etc.) can be mated within an assembly model. To make part model sketches visible, right click them in the feature tree (on the left of the screen by default) and click the glasses. This will hide / show the sketch. Sketches can be created in a part model solely for mating in an assembly model. This is often used for aligning the caster wheel along the simplified 80/20 extrusion.

9. **Implement configurations for assemblies with dynamic parts.** Multiple configurations for a part or an assembly can be created to reduce the total number of files created for the same part. For example, various lengths of 80/20 can be generated by extruding the same cross-sectional sketch to different lengths, or a lifting arm can be mated in different orientations to show the full range of motion. Each configuration will suppress or unsuppress specified associated features or mates.

10. **Use mechanical mates to maintain dynamic assemblies.** Mechanical mates are another powerful tool which can allow you to visualize the full range of motion of the dynamic portions of a design. By implementing mechanical mates such as gears, slots, or hinges, an assembly can be fully defined (as is good practice) while still allowing the relative motion of certain parts which will better allow your TA to provide adequate feedback during design review. A good example of when to use mechanical mates is a moving arm, for which you would need to check clearances throughout its designed range of motion.

11. **Use the simplified 80/20 part model or print your assembly drawings on B-size sheets.** A simplified part model of 80/20 is provided for use on the course website. The simplified model does not contain the four grooves and associated tangent edges, so it prints a lot cleaner on A-size (8.5x11") sheets and it also speeds up rendering of the complete assembly model. So we recommend using the simplified version for your project, or planning to print your assembly drawings on B-size (11x17") sheets for clarity.

12. **Edit the tolerance table.** The standard tolerance table provided on the drawing templates may not suit all of your parts. This table can be edited by right clicking the sheet drawing area, selecting "Edit sheet format," and then editing the text in the tolerance table. To resume editing the drawing, right click the sheet drawing area and select "Edit sheet."

13. **Aligning dimensions in drawings.** Aligning dimensions in drawings improves the readability of your drawing. Select multiple dimensions in a drawing. Right click any of the dimensions and hover over "align." The dimension alignment choices are listed. Choose the appropriate alignment.

14. **Using layers to hide unwanted sketch elements.** SolidWorks supports drawing layers. Drawing layers act like transparencies on which details of a drawing are kept. Each layer can be formatted (color, line style, thickness) and hidden. If you require sketch elements (points, lines, etc.) to be added to your drawing to properly dimension it, these elements will, by default, appear when you print the drawing. Placing the unwanted elements in a layer and hiding that layer will prevent the elements from appearing when printed. To use layers, enable the layer toolbar (right click the toolbar area and select "layer"). Click on the  symbol on the layer toolbar to open the layer properties. Create a new layer and call it "hidden" so you remember to hide it later. Color the hidden layer red so it is obvious when something is on that layer. Select a dimension, point, or line and change the layer toolbar dropdown menu to "hidden." The dimension, point, or line is now on the "hidden" layer. Again, open the layer properties. Click the lightbulb next to your hidden layer. Click OK. Everything on that layer is now invisible (but still present). CAUTION: pay attention to the layer dropdown while creating new drawing elements. If your hidden layer is shown, all new elements are automatically placed in the hidden layer.

15. **Custom properties.** Custom properties are user defined properties that belong to a part model or assembly model and can be accessed by other files that reference that part / assembly model. To access, create, and edit custom properties, click file, select properties and navigate to the "Custom" tab. A common use of custom properties is to simplify creation of proper bill of materials (BOMs). For example, the "Item No.", "Part/Assy No.", and "Part/Assy Description" columns of the BOM can reference the custom properties of the parts and subassemblies present in the drawing.

16. **EML2023 computer lab.** If you have problems running SolidWorks on your personal computer you can use the workstations in the 3D printing lab on the 3rd floor of MAE-B.

17. **Effectively communicate with your TA.** The TAs have seen a lot of ideas succeed and fail. Their job is to stimulate discussions so you can generate lists of pros and cons for the decisions you make. Therefore, it is essential you communicate well with your TA. 3D PDFs can facilitate communication by allowing others to quickly view and rotate part or assembly models using Adobe Acrobat. To generate 3D PDF files, click Save As… >> Select PDF as File Type >> Check 3D PDF checkbox.