

Design & Manufacturing Tips

Following is a compilation of design & manufacturing knowledge you should accrue in this class. You should follow these rules for all parts designed and drawings submitted in this course and industry. Ask for clarification on any of these points as you work on your projects for this course. [Here's a link to a more comprehensive version of the document to help you beyond this course.](#)

Detail Drawing Essentials

1. **Finish designing the part before detailing it.** This simple rule is often overlooked. Is the part ready to be detailed (dimensioned)? How does the part mount to the overall assembly? How do other parts attach to the part? Is there adequate room for tools necessary to assemble the part (i.e. screwdrivers, wrenches, sockets, etc.)? Is the size of the part justified (i.e. can it be designed smaller to use less space and reduce raw material cost, if threaded, are the fastener sizes reasonable, etc.)?
2. **Include necessary dimensions for every part feature.** The purpose of a detailed part drawing is for someone to be able to manufacture the part without further communication with the designer. Check that **every** feature is dimensioned **completely**. Every feature requires at least two dimensions for location.
3. **Include proper tolerances for each dimension.** A dimension is useless without a tolerance, because the tolerance communicates the range within which the feature geometry is acceptable. Most companies maintain a list of the typical tolerances achievable by their manufacturing facility (whether in-house or outsourced). Typical tolerance ranges for part features designed for the project in EML2322L are:
 - $\pm 0.0005''$ to $\pm 0.001''$ for **critical** hole locations or sizes (i.e. dowel pin locations or holes)
 - $\pm 0.002''$ to $\pm 0.005''$ for **important** hole locations (i.e. wheel hub & motor attachment bolt patterns)
 - $\pm 0.005''$ to $\pm 0.020''$ for **normal** hole locations (i.e. holes to attach motor mounts to your robot)
 - $\pm 0.020''$ to $\pm 0.050''$ for **non-important** hole locations (i.e. bracket holes to arbitrarily mount a release chute motor to your robot or to specify holes through the 80/20 angle brackets)
 - $\pm 0.001''$ to $\pm 0.005''$ for flatness specification on critical surfaces (i.e. the hub face the wheel attaches against)
 - $\pm 0.020''$ to $\pm 0.050''$ for overall size of machined parts (this isn't usually important to part function)
 - $\pm 0.060''$ for (manual) sheetmetal parts produced in the lab (usually marked by eye with a tape measure and sheared)
 - $\pm 0.060''$ for typical pieces of 80-20 cut to length (this tolerance can be achieved by cutting with the bandsaw)
 - ± 3 to ± 5 degrees for angular bends or cuts (usually marked by eye with a protractor)Smaller tolerances require more time to produce the feature; this adds expense to a part, so always use as large of a tolerance as possible that still allows the part to perform its required function.
4. **Proper hole and thread notes.** This is a common weakness for new designers, so use what you learn in the laboratory. You've never seen a drill bit denoted by its radius, so never dimension holes using their radii, but rather their diameter (only arcs are dimensioned by their radii). Specify properly sized clearance holes using close or free fit standards off the tap chart. Select the correct type of thread (i.e. coarse threads in weaker materials). Include all required information for threaded holes: (1) the tap drill size and depth, (2) the full thread callout (thread specification and depth), and (3) the number of identical features on the part (if more than one). Always dimension hole depths in the same leader as the rest of the hole note. Since drills create cone-bottomed holes, denote on each drawing whether hole depths are dimensioned with respect to the point or the larger diameter of the cone. For blind threads, always specify at least one extra fastener diameter on the drill depth so the tap has room to cut the allowable thread depth (i.e. 35mm drill depth for 25mm of usable M10 threads). Always design threaded holes with five or more threads of engagement to prevent thread shear failure.

5. **Part designer's and drawer's names.** The designer's and drawer's names must be on every drawing (detail & assembly) so the manufacturer knows who to contact with questions. Both should serve as competent and trustworthy checkers to catch mistakes that can be costly in wasted time or money.
6. **Drawing units.** You will only think this is funny until you receive a part that is 25.4 times too small or too large that you have to pay for. Denote units on every drawing you ever make.
7. **Part quantity.** Parts will be made differently depending on the quantity desired (i.e. 1, 10, 100 or 1000). Knowing how many of each part to manufacture is critical, as producing more parts will generally result in lower cost-per-part. Denote part quantity in the general notes on the drawing or in the RFQ.
8. **Material specification.** Whoever is manufacturing your parts cannot read your mind. Denote the type of material and the raw stock size in the drawing's general notes. For example, 3/16" 6061-T6 aluminum flat bar, 2" Acetal (Delrin) round bar or 0.065" 1018 steel sheetmetal.
9. **Part name.** This might seem trivial, but the part name is important because it's referenced in the bill of materials and will be discussed with the manufacturer. Referring to a part as "*the small one with 4 holes in one end and two slots in the other...*" is unprofessional and will lead to mistakes.
10. **Debur notes.** Debur notes are important to prevent injury when assembling the manufactured parts or using the product. The drawing note "debur all edges" is adequate to instruct the manufacturer to remove sharp edges. Anytime you use a cutting tool (i.e. on a milling machine, lathe, bandsaw or drill press) a sharp edge will result that requires debur instructions.
11. **Surface finish specifications.** This is a difficult point for many to grasp and can result in parts which are unnecessarily or even outrageously expensive. When designing and manufacturing parts, there are usually two points of concern: (1) the features are within the allowable tolerances, and (2) the surface finish is acceptable. Surface finish specs are analogous to tolerances: without them, we don't know how good is good enough. You don't want to finish every surface on a part without reason, as doing so (or specifying it) will only add excessive expense to your part. (We did this on the first parts made in lab simply to demonstrate what is achievable on the mill and lathe; this is rarely done in industry.)

As with tolerances, we must always explicitly specify surface finishes on every surface to denote how nice of a finish is required. We can specify a general surface finish spec in the drawing notes section and then explicitly denote any surfaces which require a different finish. Qualitative finish specifications range from bandsawed edge to milled, turned, ground or polished surface. Other common finish notes include *only finish noted surfaces*; *raw material finish okay*; *finish surfaces with abrasive cloth* (i.e. by hand with sandpaper); *satin finish okay* (i.e. sandblast or vibratory finishing); etcetera.
12. **Title block and drawing notes.** Each drawing (detail, assembly or BOM) should have a consistent title block that includes the part name, designer's name, drawer's name, drawing scale and revision number. Each detail drawing should also contain a tolerance table in the title block and a "NOTES:" section that includes units, material and quantity specifications, debur notes and other notes specific to the drawing.
13. **Miscellaneous dimensioning tips.** Never dimension to hidden lines; if unavoidable, use a section view instead. Never show hidden lines or dimensions in isometric views, but always show tangent lines. Conversely, always show hidden lines in orthographic views. Never shade isometric or orthographic engineering drawings. Ensure each part drawing is easy to read and of good print quality. Scale dimensions proportionally to the rest of the drawing and ensure all text, leaders and dimensions have consistent line weights and font sizes. Never place text on top of part views. Spread multiple views over separate pages for large or complex parts and denote as "1 of 3", "2 of 3", etc. (as with writing, clarity is as important as content). [HERE IS AN EXCELLENT LIST OF DIMENSIONING RULES.](#)

Design Tips

1. **Fasteners.** Use the correct type of threads in the proper type of material (coarse in weak; fine in strong). Specify correct threaded hole notes and tap drill sizes based on the tap chart standards. Never use smaller threads than necessary (avoid threads smaller than #6 or M4 whenever possible, as the tools to produce such threads are weak and add significant manufacturing time). Specify threads to the proper depth (typically 2.0X the nominal fastener diameter, but NEVER less than one or more than three fastener diameters). Specify proper sized clearance holes using close or free fit standards off the tap chart.
2. **Clearance holes versus line fits.** If you want to put a hole through a part for a 1/4" fastener, never use a line fit (i.e. 0.250"), but rather, a clearance fit. A fastener requires clearance to pass through the part without threads. In this case, you should specify a 0.257" or 0.266" diameter hole, because these are the industry standard sizes recommended in the tap table for (close or free fit) clearance holes.
3. **Sheetmetal design notes.** The lab shear can cut up to 0.065" thick steel sheet. Sheetmetal cannot be threaded for fasteners, because you always need at least 5 threads in the workpiece. As an example, a 10-32 threaded hole would require a workpiece thickness of ~0.15" (1/32"x5). To attach thin sheets of metal together, design parts with additional flanges on the ends which can be welded or drilled and permanently riveted together using 1/8" rivets; substitute standard fasteners in lieu of rivets if the parts require disassembly between lab periods. If the parts require welding, make them out of steel because thin aluminum sheet is very difficult to weld. If a part has a shape that resembles a deep channel like this: |_|, or if a part has several bends, make it out of multiple pieces that are subsequently welded or fastened together. Obtain assistance during the design phase so someone with more experience can explain how to design parts to be made using equipment in the laboratory (or at a company).
4. **Material selection.** In general, machining time is proportional to material strength, and cost is proportional to machining time. Knowing this, good engineers should always select the weakest material that is strong enough for the design; this ensures the cheapest material and lowest manufacturing costs. If you ever select a stronger material because you didn't know how to (or care to take the time to) calculate the required strength, you will quickly gain a reputation for being ignorant or apathetic. Prudent material selection is pivotal to good part design. In addition to material strength, manufacturability and cost, did you consider material stiffness, ductility, toughness, corrosion resistance or thermal properties?
5. **Final design details.** Have you thought about other details that make the difference between a good design and a poor one? Are you sure other parts will not interfere with the one(s) you have designed? Did you leave room for tools to access fasteners which attach the part to the rest of the assembly? (An accurate assembly drawing is not just an academic exercise!) Can you simplify manufacturing by combining multiple parts into one or splitting one part into two or more? Can you simplify the part design to reduce the number of manufacturing steps? **If you aren't sure about all the steps necessary to make the part, you are not ready to finalize the design.** Take time to answer these questions now, before they are more costly to fix once manufacturing commences. You want your reputation associated with good designs, not poor ones.
6. **Leverage DFM knowledge.** The difference between a CAD specialist and design engineer is the later applies manufacturing knowledge to the design. Think about what you're drawing (or checking). In general, square holes cannot be made using traditional processes. Up to this point every hole you made was created using rotating tools that produce round features. Fillets and radii look nice in CAD but think about how you will manufacture them without the use of more expensive CNC equipment. **Good designers place function and part cost before appearance.** Refer to the following section for valuable tips on reducing part manufacturing time and cost via proven DFM tips.

Designing for Manufacturability (DFM) of Machined Parts

1. **Anderson's Law.**¹ Never design a part you can buy out of a catalog unless you can clearly justify the choice (i.e. to save weight (if that's an important design goal), to reduce size for improved packaging, to use an alternate material, etc.). Off-the-shelf (OTS) parts are significantly less expensive considering the cost of design, documentation, prototyping, testing, improving and the overhead cost of purchasing all the constituent parts. Suppliers of off-the-shelf parts are more efficient at their specialty, because they are more experienced on their products, continuously improve quality, have proven reliability records, design parts better for DFM and have dedicated production facilities that can produce parts at lower cost (it's difficult to compete on the price of 20 parts with a company that manufactures the same part by the thousands). Using OTS parts helps us focus on our real mission: designing and building products.
2. **Design parts to take advantage of nominal raw material sizes.** Students entering this course cannot understand this important point, but after making parts on mills and lathes, this should make sense. **Always design parts around nominal raw material stock sizes.** A piece of 2" round bar might measure between 2.005" and 1.995" in raw stock size. During lab it was necessary to machine 0.010 - 0.015" off the hub's outside diameter (OD) to make it truly cylindrical, leaving it between 1.995" and 1.980". If the OD is specified as $2.000 \pm 0.005"$, this target will not be achievable; but if the OD is specified as $1.980 \pm 0.005"$, it easily can be achieved. For the hub manufactured in lab, the exact OD has absolutely no effect on the function of the part, so if the designer is smart, (s)he would specify it as: $2.000 \pm 0.020"$ and place a note on the drawing that it does NOT need to be a finished surface. That way any 2" round raw stock would work and the person making the part would not need to waste time finishing surfaces that don't improve the part's function. On the other hand, if the OD of the part was important, the designer could specify the OD as $1.980 \pm 0.001"$, which would allow the part to be cut to final size in one pass, whereas specifying the OD as $2.000 \pm 0.001"$ would require the purchase and use of the next larger raw material stock size (2.5"), as well as many passes to cut it down to the final $2.000 \pm 0.001"$ size.
3. **Avoid designing mirror image (right or left hand) parts.**¹ When designing paired parts like motor mounts, design with symmetry to save manufacturing time (since parts can be stacked and machined in unison) and assembly time (because there's no right or left to track). If identical parts cannot perform both functions, add features to both right and left hand parts to make them the same. This tip also reduces design time (half as many part models and drawings to create) and manufacturing cost (making twice as many of the same part is always cheaper than making two half-size batches of different parts).
4. **Use larger feature tolerances.** $\pm 0.010"$ is a lot easier to achieve than $\pm 0.001"$, so use looser tolerances whenever possible and always ask why the design tolerances cannot be made larger.
5. **Use fewer and/or coarser surface finish specifications.** Never request "*finish all surfaces*" or "*finish all over*" because it's very rarely needed. Make sure you can justify EVERY finished surface on a part. Like finer tolerances, better surface finishes add more time and expense to each part.
6. **Use fewer dimension datums.** Each reference datum requires edge finding to locate a zero. Using fewer datums decreases setup time, reduces error (tolerance) stack up and lowers the chances for mistakes.
7. **Use nominal part dimensions.** If making the part manually, it's much easier to read nominal dimensions off a part drawing (i.e. 2.000" or 1.125") than arbitrary dimension (i.e. 2.019" or 1.131").
8. **Use weaker materials.** Weaker materials generally have higher machinability, so use them whenever possible. In addition, weaker materials typically have a lower cost, which can be substantial.
9. **Use thru-bolted holes.** Drilled clearance holes require less manufacturing time than threaded holes, so use thru-bolted holes whenever possible to reduce part cost. On the flip side, when using thru-bolted holes, you must be able to access the back of the part for assembly.

10. **Specify cone-bottomed holes.** Cone-bottomed holes are produced by drills; flat bottom holes are produced with end-mills. Drills are much faster for producing holes and should be used exclusively unless you have a very good reason to do otherwise.
11. **Make the part smaller.** If there's no good justification otherwise, make the part smaller; this reduces material cost, manufacturing cost and leaves more space for other components in the assembly.
12. **Design for minimum raw-stock removal.** It takes less time to remove less material. Better designs start with material that is near net shape and minimize the amount of machining operations.
13. **Avoid small cutting tools.** Larger tools are stronger and remove material faster without vibrating or breaking. Time is money when it comes to manufacturing, so try to avoid designs requiring small tools.
14. **Design for favorable tool stiffness.** Since the strength and stiffness of cutting tools limit productivity, maximize stiffness by minimizing each tool's required length (L) relative to its diameter (D). L:D ratios should be under 3:1 for milling and 8:1 for drilling whenever possible; smaller is always better.
15. **Design around standard cutter sizes.** If you can design features to use standard cutter sizes, you can often make parts on manual machines that otherwise would require CNCs. CNCs cost more per hour to operate, so for prototyping, parts that can be produced on manual machines are typically cheaper.
16. **Avoid unnecessary fillets.** Fillets look nice in a solid model but can add a LOT of expense in secondary operations. Make sure fillets are justified (i.e. in areas of high stress) because they can significantly increase part cost and demonstrate ignorance or apathy if specified without cause.
17. **Show Cartesian coordinates on detail drawings.** When dimensioning bolt circles (polar arrays), include Cartesian coordinates so hole centers can be easily located when machining or programming a CNC machine. If the manufacturer takes time to calculate the coordinates, you pay for that time; so reduce part cost by including coordinate dimensions on drawings as well as bolt circle diameters.
18. **Reduce the total number of parts.**³ The reduction of the number of parts in a product is probably the best opportunity for reducing manufacturing costs. Less parts implies less purchases, inventory and handling. A part that does not need to have relative motion with respect to other parts, does not have to be made of a different material, or that would make the assembly or service of other parts extremely difficult or impossible, is an excellent target for elimination. Some approaches to part-count reduction are based on the use of one-piece structures and selection of manufacturing processes such as injection molding, extrusion, casting, and powder metallurgy (which are beyond the scope of this course).
19. **Good designs are elegant in their simplicity.** As stated eloquently by [Dr. Kevin Craig](#), create designs that are explicitly simple. Keep complexity intrinsic, buried, and invisible. The less thought and less knowledge a device requires for production, testing and use, the simpler it is.
20. **Treat each drawing you create as a resume.** Good shops that manufacture parts for customers will always have enough work to stay busy; in other words: they don't need your business. Your drawings always compete against others as job shops decide which to take on. Many shops will refuse to quote parts that appear to be drawn by someone who is inexperienced, ignorant or apathetic; OR they will add a nuisance cost multiplier of 150% - 300% realizing you don't know what you're doing and you're going to require hand-holding to get the parts your project needs. So realize the impression drawings make on others and invest time to present yourself as intelligent, competent and organized.

Assembly Drawings & Bill of Materials (BOM)

1. **General assembly drawing notes.** Assembly drawings use a combination of orthographic and isometric views to clearly communicate how individual parts combine to create assemblies. Good assembly drawings are like Lego® manuals: a 10 year old should be able to follow them. Assembly drawings must include dimensions showing the location of different parts with respect to each other; however, they should not contain individual part dimensions, which belong in the detail drawings instead. Never show hidden lines in assembly views, but show all tangent edges. All drawings should be easy to read and uncluttered, so print on multiple sheets when necessary.
2. **Exploded views in assembly drawings.** Exploded views are required when parts of an assembly are not clearly distinguishable by the BOM balloon leaders.
3. **General BOM notes.** Bill of materials come in several varieties depending on the use for which they are intended. A BOM can define products as they are designed (engineering bill of materials), as they are built (manufacturing bill of materials) or as they are maintained (service bill of materials). EML2322L makes use of the manufacturing BOM, which provides a list of all components used on the final design, including fasteners. A BOM should contain a clearly formatted table with at least four columns: (1) unique and sequential item number, (2) part name, (3) part description, (4) quantity used in the assembly. For each item in the BOM, there should be a balloon and leader clearly pointing to the corresponding part in the assembly drawing(s); this will require multiple views. Label all item numbers sequentially so there are no duplicates across all BOM tables.
4. **BOM fastener notes.** BOM fastener descriptions require three pieces of information: the thread specification, fastener length and head type (i.e. *1/4-20 x 0.5" button head cap screw* or *M6x1.0 x 25 hex head bolt*). When the information listed is provided in the BOM, detail drawings are not necessary for common OTS fasteners, so do not include them for the course design project unless the fastener(s) require modification(s).

Closing Notes

A world-renown international intercollegiate design judge once said, “*a good engineer can do for a dollar what anyone else can do for ten*”. As engineers, most of us will be tasked with designing components to fulfill particular functions. Only by designing parts with the above tips in mind will you be able to produce cost-effective solutions that will keep the company you work for in business after their competitors have shut down because their engineers didn’t understand how important this is.

Works Cited

1. Anderson, David M. *Design for Manufacturability*. n.d. webpage. <http://www.design4manufacturability.com/DFM_article.htm>.
2. Bralla, James G. *Design for Manufacturability Handbook*. McGraw-Hill Companies, Inc., 1999.
3. Greenlee, Bob. *Design for Manufacturing - Guidelines*. n.d. <<http://www.unm.edu/~bgreen/ME101/dfm.pdf>>.