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MEC222 Frequently Asked Questions

This page answers some common questions about drafting and CAD, as well as some administrative matters.

For more general questions, see the general FAQ, Salustri's grading policy, and the University policies listed in the course outline.

Which parts of assignments can I do in pencil?

Some parts of an assignment must be done in pencil; other parts must be done in pen.

The rule is simple: all engineering drawings/sketches must be entirely in pencil; everything else must be in indelible ink.

Read the rules for cover pages.

How do I choose a good front view?

It can be difficult to decide which view is the best *front view* for a given drawing. There are some guidelines, but sometimes - depending on the complexity of the part - they can conflict. Sometimes, there is no one “best” front view; in these cases, graders will take the ambiguity into account. The guidelines are:

Natural bottom. Many parts have a “natural bottom” face. Imagine holding the part, and placing it on a tabletop. You will likely choose the flattest and largest face as the bottom, to ensure the piece won't tip over. That's the “natural bottom” of the part. This also identifies which view will be the top view.

Most interesting geometry. The front view is the view that should have as much “interesting” geometry as possible facing forward. The more holes, edges, faces, chamfers, slots, lugs, flanges, etc are visible, the better.

Fewest hidden lines. While this may seem to be the same as *most interesting geometry*, it often isn't. For instance, because an automobile body is generally symmetrical left-to-right, but asymmetrical front-to-back, one may argue that the best front view is that of the *side* of the automobile (because it will have the fewest hidden lines.

Fewest hidden lines in *other* views. If the first three guidelines aren't enough, one can choose a view as the front view such that it will lead to the fewest hidden lines in the *other* (side & top) views.

What is a technical sketch?

Besides CAD, MEC222 also focuses on *technical sketching*.

A technical sketch is a carefully drawn *freehand* sketch. Sketching is covered in Chapter 2 of the textbook. It is expected that by the end of the semester, students will have learned to make freehand sketches of parts and assemblies.

You can use your drafting instruments to help you execute a sketch, but in only certain limited ways. For instance, if you need to draw a line between two points that are 5cm apart, you can use a ruler, straight-edge, and/or compasses to determine where those two points should be. However, you must learn to draw the actual line freehand. Likewise for circles and other curves: by the end of the semester, you should be drawing all these shapes freehand.

To encourage students to learn to draw freehand, marks will be *deducted* for lines and curves *not* drawn freehand. Deductions will follow a sliding scale. At the beginning of the semester, no marks will be deducted. However, as the semester progresses, grading will become more and more stringent.

It is easy to use quad-ruled paper to help you sketch. Use two paperclips to fix a sheet of blank paper onto your pad of quad paper. You will be able to see the quad paper's grid through the blank paper. You can use the grid to help you draw freehand. The grid size of typical quad paper is 1/4in or 1/2cm, so it's quite useful for estimating dimensions too.

Drawings on the final exam will be expected to be done freehand; grid paper will be provided.

There is a whole chapter (ch 2) on sketching in the textbook. Read it.

Also, Table 5.1 (page 232), the *Alphabet of Lines*, enumerates all the different kinds of lines we use in sketching, with examples. You should put a post-it on this page till you know it by heart.

What is required of a drawing?

For technical sketches

A *technical sketch* or just *sketch* is done free-hand, but follows all the rules of engineering drawings as described in lecture and in the textbook. A drawing that violates the rules below is *always unacceptable* and will get a failing grade.

- Sketches appear on blank (both sides), white, unlined paper only.
- There must be a border, approximately 1cm wide, all around every page.
- Your drawing must be entirely within the page border.
- All text in hand drawings must be in engineering block. (Refer to the textbook for further details.)
- There must be a title block of some type on every page.
    - The title block must include your full name, your student ID, your section number, the assignment ID (including the page and question number from the textbook), the units of

measurement, and the date you submitted the work.
- Always specify a scale, even if it's 1:1.
- All sketches must be done in pencil, including the content of the title block. Indelible ink must *never* be used in technical sketches and drawings.

You *may* leave construction lines on your drawings so long as they are faint enough that they cannot be mistaken for anything else. This is rather like “showing your work” and can help the TA understand what you were trying to do.

- You can also use a light blue pencil (like this one) to draw construction lines. The light blue is often used in industry because it does *not* show up on photocopies and computer scans - so it saves you the trouble of having to erase them.

See also other drafting notes for further information.

For CAD drawings

A *CAD drawing* is created via CAD software, and is a proper engineering drawing; that is, it is composed of views of object(s), has a proper border and title block, etc. as described in lecture and in the textbook.

There are some rules that **must** be followed for both technical sketches and CAD drawings. A drawing that violates these rules is *always unacceptable* and will get a failing grade.

- CAD drawings appear on blank (both sides), white, unlined paper only.
- There must be a border and title block on every page.
  - The format of the border and title block is whatever the CAD package provides.
  - The border and title block must be drawn by the CAD package.
- Your drawing must be entirely within the page border.
  - The title block must include your full name, your student ID, your section number, the assignment ID (including the page and question number from the textbook), the units of measurement, and the date you submitted the work.
  - The title block must be filled in using the CAD package. You **cannot write** manually (pen, pencil, etc.) on a drawing.
- If the scale of your drawing is not 1:1, you must include the scale in your title block.
- Students may assume that whatever font is used by the CAD package is acceptable.
  - However, all CAD text must be in ALL CAPS.

For everything else

A *solid model* or *rendering* is **not** a drawing; there will also be conventional homework questions. These can be treated as conventional homework questions in any other course.

Use *indelible ink* for all non-drawing homework.

What units of measurement do I use?

In mechanical/industrial drafting there are really only two units used: *millimetres* if the drawing is in SI and *decimal inches* (e.g. 1.375) if the drawing is in American units. **Never mix the two.**

Here are three rules to help you determine what units to use. Check each one in the order given here.

1. The assignment may specify which units to use explicitly.
2. If units are not given explicitly, and if there is at least one dimension given as a fraction or a decimal number, then the units are inches.
3. If you still cannot tell what units are being used, consider the general shape and size of the part. If it were in millimetres, would it be unreasonably small? If yes, then it must be inches.

How do I scale my drawings?

A *scale* is a ratio that is used to shrink or expand the size of the image you draw so that it fits on the page.

Scales are always written as ratios of integers, where the left number represents a unit on the drawing and the right number represents a unit in real life. So a scale of `1:2` means one unit on the drawing is two units in real life - so-called “half-scale.”

Scaling a drawing by 1:2 does **not** mean you also divide dimensions by 1/2. Dimensions are always indicated at 1:1 scale. For instance, if you have to draw a 10cm line at a scale of 1:2, you draw a line 5cm long, but you dimension it as 10.

There are no units of measurement in scales. One must *never* write something like `1 in : 1 cm`. Units of measurement are *completely different* from scales.

If the scale is 1:1, it can be omitted. Conversely, if no scale is given on one of your drawings, we will assume you intended 1:1.

Historically, only a few scales have been used: 1:1, 1:2, 1:5, 1:10, and their inverses. However, because we typically draw on letter-sized paper in this course, scales of 1:3 and 1:4 are also acceptable.

To select a good scale for your drawing:

1. Block out the views you expect on scrap paper; estimate how much room on the page each view will need.
2. Based on the alloted space on the page and the size of the actual part you are drawing, select the scale that will keep the part as large as possible and yet small enough to fit into the allocated space.

All parts on one sheet must be to same scale.

Different scales can be used on different sheets.

What is a "working set of drawings?"

Refer to Chapter 10 of the textbook for detailed information.

A *working set* of drawings applies to assemblies of parts. They consist of:

1. An assembly drawing.
2. A Bill of Materials (BOM).
3. Part drawings for all parts to be manufactured.

Here are some further guidelines about each of these elements.

Assembly Drawing. An assembly drawing is used by someone who has to assemble the product.

- It must be sufficiently detailed so that person can see where every part goes.
- There must be at last one, but usually two, principal views, section views to show what the product looks like internally, and possibly other views that show the necessary detail.
- Every part (or subassembly) must be labelled with a numbered balloon. The number in the balloon must correspond to an item in the Bill of Materials that describes the part (or subassembly), and also corresponds to the part drawing number.
- Fasteners and “off-the-shelf” parts may be shown in highly simplified or stylized form.
    - For instance, an electric motor that you decide will be purchased “off-the-shelf” need not be shown in great detail. This is because you're buying the motor, not designing it; so you're not responsible for its details, but you *are* responsible for indicating how it is assembled into your product. You should render just enough detail that the person who will assemble the product containing the motor knows where the motor goes and how to orient and attach it.
- Dimensions shown on the assembly drawing are those that are not available from any part drawing. For instance, the overall length of an automobile cannot be read from any one part drawing, so one dimensions the automobile's overall length on the assembly drawing.
- Certain other key dimensions may also appear on the assembly drawing. For instance, the location of mounting holes for an electric motor are usually shown on the motor's assembly drawing, because those dimensions will be needed by someone mating the motor to some larger assembly.

Bill of Materials. The Bill of Materials (BOM) lists in order of part number (see Assembly Drawing, above) every part needed to make the product.

- Each entry includes the part number, the part name, the quantity of those type of part, and (optionally) a description of the part such as an engineering specification (for bolts, screws, welds, etc.) or a make and model for *off the shelf* parts that can be purchased rather than manufactured.
- In “real life,” BOMs are placed on the Assembly Drawing. (Refer to Chapter 11 of the textbook for examples.)
- Because in this course we draw on letter sized paper, students are allowed to put the BOM on a separate page, right after the Assembly Drawing. In this case, the BOM is not a drawing per se and so needs no border or title block. However, it should appear as a table of information.
- There's nothing wrong with having the BOM on the assembly drawing, *if* you can make it all fit legibly; but for the reasons noted above, it's not required to be on the assembly drawing.

Part Drawings. These are drawings of the individual parts needed to make the product described in the

Assembly Drawing.

- One can place more than one part on a page, so long as each part is:
  - clearly separated on the page from other parts; and
  - is properly named, including the *part number* from the Assembly Drawing and BOM.
- Part drawings are needed only for parts that must be manufactured. This typically *excludes* fasteners like bolts and screws, washers, gears, etc. *Don't make part drawings for bolts, screws, washers, gears, etc.*
- A *minimal* sufficient number of views are necessary. This includes at least one principal view, plus section and auxiliary views as required. It is up to the student to decide which views are necessary.

How many views of a part are needed?

In some cases, the assignment will clearly state how many - and what kind of - views you must draw.

If this is not given, then draw the fewest principal views needed to show the entire geometry. If you are drawing a sphere, you only need one view. If you are drawing a flat plate, you only need one view, and specify “constant thickness” with a dimension in a note on the drawing. If two views are enough to show all sides of the geometry, then only draw two views.

If the “interesting” geometry of a part lies on an oblique plane in all principal views, then you need to provide an auxiliary view of the oblique features.

If the part has “interesting” geometry that is hidden or “inside” the part, then you need to add a section view.

Remember, a drawing may consist of multiple *sheets*. If you need five views in a drawing, you can use more than one sheet to show them.

It is up to each student to learn and decide how many, and what type of, views to draw.

How do I dimension my drawing?

Refer to this Google Doc.

What does U.O.S. mean?

U.O.S. = *Unless Otherwise Stated*.
This abbreviation is often seen on drawings after default values for common dimensions.
Example: `All fillets 0.25 R U.O.S.`

Do I need to sketch/draw threads?

Threads (on bolts, nuts, screws, etc.) are almost always *omitted* from sketches in practice. You do not need to draw threads in your sketches.

The same is true of CAD drawings. Unless you are actually drawing a bolt or other threaded item for the sake of its manufacture, you need not draw threads on bolts in CAD drawings.

How do I treat section views?

- Hatch marks must be evenly spaced.
- Hatch marks must be at an angle NOT found elsewhere on the drawing.
- Section lines do not *have* to be identified (A-A) unless there's more than one section in the drawing.
- Section views may have dimensions, but only such dimensions that are **not** evident on principal or auxiliary views.

Where do section/auxiliary views go?

If you are sketching, it is useful to place section views and auxiliary views in locations that facilitate drawing them. A section view usually goes near the view upon which it is based, and an auxiliary view is usually aligned with the projection lines from which the auxiliary view is based. Refer to the textbook for examples.

If you are using CAD, you can place the views in any reasonable location.

What's the difference between first angle and third angle projections?

See section 5.4.2, page 266 of the sixth edition of the textbook. We use third angle projections in Canada.