

CALIFORNIA STATE UNIVERSITY, SACRAMENTO
COLLEGE OF ENGINEERING AND COMPUTER SCIENCE
DEPARTMENT OF MECHANICAL ENGINEERING

**DRAWING REQUIREMENTS
FOR
SENIOR PROJECTS**

RELEASE 1.1 9/27/2002

SPECIFICATIONS FOR PROJECT DRAWINGS

Engineering drawings are formal documents that convey all the information necessary to make the part shown. They must be complete enough to make the part without any other explanation or information. Drawings are made to recognized standards so that they can be easily understood by anyone who is familiar with the standards. The drawings must also indicate the allowable variation in part size and dimensions. Reference can be made to other documents, but these are always recognized standards or specification documents. Since CSUS does not have any formal specifications, only nationally recognized standards such as ANSI, ASME, or SAE standards should be used. Ideally, you should be able to mail your drawings off to a machine shop and receive back exactly the parts you wanted without ever speaking to the shop. All good engineering projects require complete and exact drawings made to accepted standards. Being able to prepare and read drawings is the mark of a skilled engineer. There are many types of engineering drawings but for senior design projects most drawings will be one of two kinds, assembly drawings or detail drawings.

We follow the ASME Y14 drafting standards and student drawings must comply with those standards.

MINIMUM DRAWING REQUIREMENTS FOR SENIOR PROJECTS

- No part may be made in Senior Project lab without an approved drawing.
- The entire set of drawings must be finished before approval. Drawings will not be approved one at a time. Approved drawings will have the instructor's signature.
- No work on the project can begin until the drawings are approved.
- A full set of detail drawings with all tolerances and dimensions is required for each part to be fabricated.
- A detail drawing is required for any part that is to be modified from a standard purchased part showing the dimensions and tolerances of the modification. Unmodified areas do not have to be fully detailed.
- Assembly drawings with parts listed are required.
- A separate bill of materials for the entire project is required, see details in this handout.
- Detail drawings are not required for purchased parts or standard hardware.
- Standard hardware, e.g. nuts and bolts, must be fully specified in the bill of materials.
- Purchased parts must be listed in the bill of materials and the vendor name and vendor part number must be provided.
- All parts including purchased parts must have a part number. The vendor part number is

not the project part number.

ASSEMBLY DRAWINGS

Assembly drawings show how a particular machine or major parts of it are assembled. These drawings indicate the relative position of the various parts and contain specific engineering information about how the parts are to be assembled. They may also provide inspection and test requirements for the assembly. Complicated machines are generally broken down into sub assemblies. The sub assemblies can be further broken down into sub assemblies as well. Complex designs may have many levels of assembly. Sub assemblies also require assembly drawings. Assembly drawings have a graphical representation of the assembly and a list of the parts by drawing number, name and quantity. Important instructions and information about the assembly are found in the notes.

Most engineering assembly drawings are two dimensional cross sections with all parts shown. Only enough views are shown to indicate the correct position of all of the parts. If a single section will show everything, that is all that is required. Additional sections can be shown if needed. Exterior view are seldom used. Exploded views were almost never used in the past but if the 2D drawings are derived from a solid model assembly exploded views may be used. They require too much work to prepare by hand in 2D and convey little information that cannot be shown in a simple section, however solid model assemblies offer the option of easily creating them. Since they are so easily prepared from a solid model they are gaining increased acceptance. Exploded views are usually used in service manuals and illustrated parts lists and have not been part of the engineering documentation until recently. Where used they must still conform to all other aspects of drawing standards.

Item numbers are used to refer to individual parts. A list of each item, its part number, and quantity required is listed on the right hand side of the drawing. Standard hardware items such as bolts, nuts, and washers are listed and may not always have drawings although they always have part numbers. If there is no drawing of these items, then enough information must be given to completely specify them. For example, a bolt should be specified as to size, thread, head type, length, and finish. A bolt would appear in the materials list as BOLT, HEX HEAD .250-20 UNC-2A X 1.00 LONG, CAD PLATED. If you can't fit a complete description of the part into the space on the drawing, you need to make a separate drawing of it. The materials list on the assembly is not the bill of materials, that is a separate and distinct document. For very complex devices, we may require specialized documents for assembly and inspection, but the assembly drawing will suffice for most senior projects. Assembly information on the drawing should be engineering requirements and not directions for how to assemble the device. If directions are needed they are usually part of separate manufacturing documents. Engineering information found on the assembly drawing includes such things as required bolt torques, lubricants, clearances that must be adjusted, and final test information.

Dimensions are seldom used on assembly drawings. The only case where they may be required is for dimensions that are adjusted or set during the assembly process. If such adjustments are required a tolerance must be specified and instructions may be included in the notes.

Assembly drawings are drawn with standard borders and title blocks, and have their own part number.

The list of materials identifies all of the parts that are used in the assembly. It is not a material specification, but rather a part list. Materials are identified in the notes on detail drawings of the parts. There is another list of parts that must be prepared called the Bill of Materials. The Bill of Materials is a separate document that lists all of the parts needed to assemble the complete machine. It will be described in more detail later.

DETAIL DRAWINGS

Most of the drawings you will need to make are detail drawings, drawings of individual parts. Only one part is shown on a drawing, and in some cases there might even be two drawings of a single part. An example of this would be a casting drawing and a machining drawing for the same part. Each of these drawings is given a different drawing number. The machining drawing would have a note that says “MAKE FROM CASTING PART NO. Xxxxx”.

Detail drawings also include inseparable assemblies. These are parts that once assembled become a single part which cannot be taken back apart. The most common examples are weldments. Weldments are welded up from individual pieces and once complete become a single part. This type of component is treated like a single part rather than an assembly, and it’s individual pieces can be shown on the same drawing.

Very large or complex parts may have several pages or sheets in a single drawing. In such cases all of the sheets have the same drawing number and are labeled “Sheet 1 of 4” and so on. In a multi-page drawing all of the notes are usually on the first page and in fact the first page may be nothing but notes. A multi-sheet drawing is considered to be a single drawing.

The number of views required for a drawing depends on the complexity of the part and the number of dimensions it takes to describe it. Generally at least one view is needed, and for a very simple part that might be enough. In other cases dozens of views, cross sections, auxiliary views and the like will be needed. Enough views must be drawn to completely describe the part and to permit all aspects of it to be dimensioned. You should draw as many views as it takes to describe the part, and no more. Drawings done on a CAD system must always be drawn full size and then scaled when plotted. Views are never labeled. Do not put “top”, “front”, etc. on your drawings. Section views are labeled with letters and the cutting plane shown unless there is only one section and the cutting plane is obvious. When labeling sections, the view itself is labeled “Section X-X” where X-X refers to the letters used to indicate the cutting plane. See any drafting text book for examples.

DIMENSIONING

Detail drawings contain **ALL** of the information that is required to make the part. Dimensions are shown for all aspects of the part, nothing is measured, calculated, or interpreted. The views of the part should be accurately drawn to scale, but it is the dimensions that govern the actual size. In general there are three types of dimensions, size, location, and form. Some general guidelines for dimensioning are:

Dimensions should be in decimal inches. Our shop is not set up to easily work with metric drawings, all of our tools are inch based.

Dimension from datum planes when possible. Select datums to reflect a functional requirement for the part.

Do not dimension to hidden lines. Hidden lines should be used very sparingly, many companies do not use them at all. If a part requires hidden lines to describe it, consider using sections instead.

All dimensions have tolerances. They must be specified or a general tolerance in the title block is provided.

Dimensions must be complete, but with no more dimensions than necessary. (No double dimensions)

Use only decimal dimensions, no fractions. Standard stock sizes (e.g. gauge size for sheet metal) can be included in parenthesis. For example, .036 +/- .007 (20 Gage) or .25 not 1/4

Dimensions should appear outside object lines.

Use ASW symbols for welded joints

Full circles are dimensioned as diameters using the diameter symbol, \varnothing Arcs are dimensioned as a radius, using a capitol R before the dimensions, e.g. R2.125

Blind threaded holes must have the tap drill depth specified and the minimum full thread depth. A good rule of thumb is to make the tap drill depth 2.5 times the thread diameter and the full thread depth 2 times the thread diameter. Tap drill diameters are not specified, they are determined from the thread size and the tolerance class called for.

Use correct thread call outs. For example .500-20UNC-2A .500 is the nominal size of the thread, the -20 refers to the threads per inch, UNC is the thread standard Unified National Coarse, and 2A is the thread tolerance class. 2 is the tolerance class used for normal fasteners and A indicates an external thread. A letter B is used for an internal thread. Metric threads are designated with an M. For example M6 X 1.0 -5g6g is a six millimeter thread with a one millimeter pitch. The 5g6g designation is the tolerance class for the pitch diameter and crest of the thread. See a recent edition of Machinery's Handbook for detailed explanations.

GEOMETRIC DIMENSIONING AND TOLERANCING

It is standard industrial practice to use GDT on detail drawings. The accepted standard is ASME Y14.5. Students are encouraged to use GDT for their projects. However, be certain that you understand it and use it correctly. If your knowledge of GDT is limited, or you are uncertain about its use then it is better to not use it. It is far worse to add geometric symbols to a drawing incorrectly than to not use them at all.

TITLE BLOCKS AND BORDERS

All drawings should have a proper title block and borders. Use the CSUS title blocks that have been created for E6 classes. They can be found on the file server in Engr. 1218. Pick the correct paper size for the part drawing, in most cases full size. Do not scale your drawing to fit a particular prototype drawing. In other words, if the drawing is to be plotted out on a C size sheet of paper do not use the border and title block for a A size sheet of paper. If the part can be shown full size on a D size sheet do not scale it down so it fits on an A size. There is a plotter in Riverside Hall 2011 that can be used to produce proper size drawings. Many blue print offices in town can plot or copy full size drawings. Make your drawings the correct size.

Standard paper sizes A size = 8 1/2 x 11

B size = 11 x 17

C size =

D size =

In the title block you must include the following information:

Drawing number

Drawing name (Same as the part name)

Name of person who made the drawing and the date drawn

General tolerances

Scale

Date

Number of sheets if more than 1, e.g. sheet 3 of 4

Signature of your instructor before the parts can be made

NOTES

Drawings should have notes which appear in the lower left hand corner of the drawing. At the minimum these should include the material specification, any heat treat, a note requiring that all sharp edges be deburred, and a note stating that all dimensions are in inches (or mm as the case may be)

Material specifications must be complete, for example Sand Cast Aluminum A356-T6. Be sure to specify the material type, the alloy designation, the basic form, and any heat treat requirements. The notes appear below a heading "NOTES" and are numbered.

TOLERANCES

All dimensions must have a tolerance. This is the allowable variation in the size from part to part. Although we seldom make more than a few identical parts for senior projects, the tolerance also indicates the size range of an acceptable part. Tolerances are part of the design. In some ways they are the most critical aspect of a design. A difference of a few thousandths or even a few ten thousandths can make the difference between a design that works well and one that fails. **Design engineers must establish the tolerances based on the functional requirements of the design.** Tolerances are not added later as an after thought. It is fair to say that drawings without tolerances are not finished designs. Failure to specify tolerances indicates that the design has not been completed. If you are not sure about reasonable tolerances ask for help.

Tolerances can be indicated in several ways:

Limit dimensions	.206 .204
Bi-lateral	.205 ± .001
Generic title block tolerances	XX= ± .030 XXX= ± .005

A note appears in the title block and all two place dimensions are toleranced ± .030 unless otherwise specified. Three place dimensions are ± .005. Generic tolerances are not to be used to avoid determining proper tolerances. Other generic tolerances can be used, these are only an example although they are typical of what is used on drawings in industry.

Tolerances are used to tell us several things. Parts which must fit together during assembly must have tolerances such that any part made to the range of sizes specified can be assembled. For example, if two parts bolt together then the hole size, bolt size, and hole location must be such that the parts can in fact be assembled. If the bolt hole is the same size as the bolt, that is a 1/4 hole for a 1/4 bolt, then the parts will only assemble if the holes are located exactly. Since it is not possible to locate the holes exactly, they must be larger than the bolt. A rule of thumb is to make the bolt hole .031" bigger than the bolts, which means that the bolt holes would have to be located within about .031" of their correct (true position) location.

Parts that interconnect require careful study. If a pin must fit in a hole, one cannot use generic tolerances. For example, if the pin is ± .010 and the hole is ± .010 with both parts the same nominal size the following range of fits is possible: .020 interference to .020 clearance. In the first case the parts could not be assembled and in the second the fit would be very loose.

Parts which have a moving connection such as a sliding fit or a bearing have even more critical dimensions. Bearing clearances are generally in the .001-.005 range, and fits for ball bearings are measured in ten thousandths of an inch.

In general you should determine which fits in your design are critical and then do a tolerance stack-up to determine the correct tolerances. If you haven't done any tolerance stack ups you are doing things wrong!!! See the attached tables that give typical tolerances for various manufacturing methods and recommended fits. A sample of a simple worst case tolerance stack up is included. Study your design to determine critical dimensions, usually interfaces between parts, and make a list. Do a tolerance stack for each dimension or fit on that list. The remaining tolerances can be selected based on the manufacturing method. Remember that purchased parts and materials always have tolerances.

Tolerances also indicate what type of machining is needed to make the part. If the diameter of a shaft is $\pm .010$ and the length is $\pm .030$, we can take a piece of stock and saw it off to length. If the diameter is $\pm .001$ and the length $\pm .005$, then we need to cut a piece of material that is over size and then turn it down in the lathe. If the diameter was $\pm .0005$, then we would cut some stock, turn it down in the lathe, and then finish grind it to size. Your tolerances answer the question "how accurate does it have to be?". Time after time the student who says "it doesn't matter" in answer to this question really means "I haven't thought about it." Tolerances always matter!!!!

LINE WEIGHTS AND LINE TYPES

Lines on drawings should be drawn or plotted in different line weights. For CAD drawings the line weight is often referred to as the pen size, even though no pen is used to plot the drawing. This is a throw back to the days when plotters always had individual pens. A larger pen makes a wider darker line. Line weights are used to make drawings easier to read and to reduce confusion. When all lines are the same weight dimension lines can be mistaken for object lines. Common line weights used are:

THICK LINES

BORDERS: Borders are the heavy dark lines around the margins of the drawing and around the title block. They are the darkest lines on the drawing and are part of the prototype drawing or title block drawing. Use a .032" line width.

OBJECT LINES are the lines that represent the outline or visible surface and edges of the actual part. They are lighter than the border but the second heaviest line. Use a .032" line width.

THIN LINES

HIDDEN LINES are dashed lines thinner in weight than the object lines. They represent

hidden edges or edges and surfaces that cannot be seen from the exterior of the object. They should be used very sparingly or not at all. Since hidden lines cannot be dimensioned and all features must be dimensioned it is usually better to avoid hidden lines and use cross sections instead. Use a .016" line width.

EXTENSION LINES/DIMENSION LINES/LEADER LINES are used to show the location of dimensions. These lines should be thin. Use a .016" line width.

CENTER LINES are used to show the center of holes and the axis of cylindrical parts. They are made up of alternating long and short lines. Use a .016" line width.

SECTION LINES are cross hatching that indicates that a view is a section or cut through the part. All cross sections should be cross hatched. Use a .016" line width.

When using Autocad, you set the line weights by color. To print a drawing you select print from the file menu which opens a print window. In this window there is a box labeled pen assignments. Picking this box will bring up a window with the color and corresponding pen number assignments. Set the line widths there. The line color in your drawing will determine the line width on the plotter.

Be sure to print out a set of drawings for the shop. No matter how careful you are, your drawings are certain to get grease and oil stains on them. Make sure you don't ruin your only set of drawings. You will also find it useful to be able to write notes, comments, and corrections on the working drawings that can be added later to the originals.

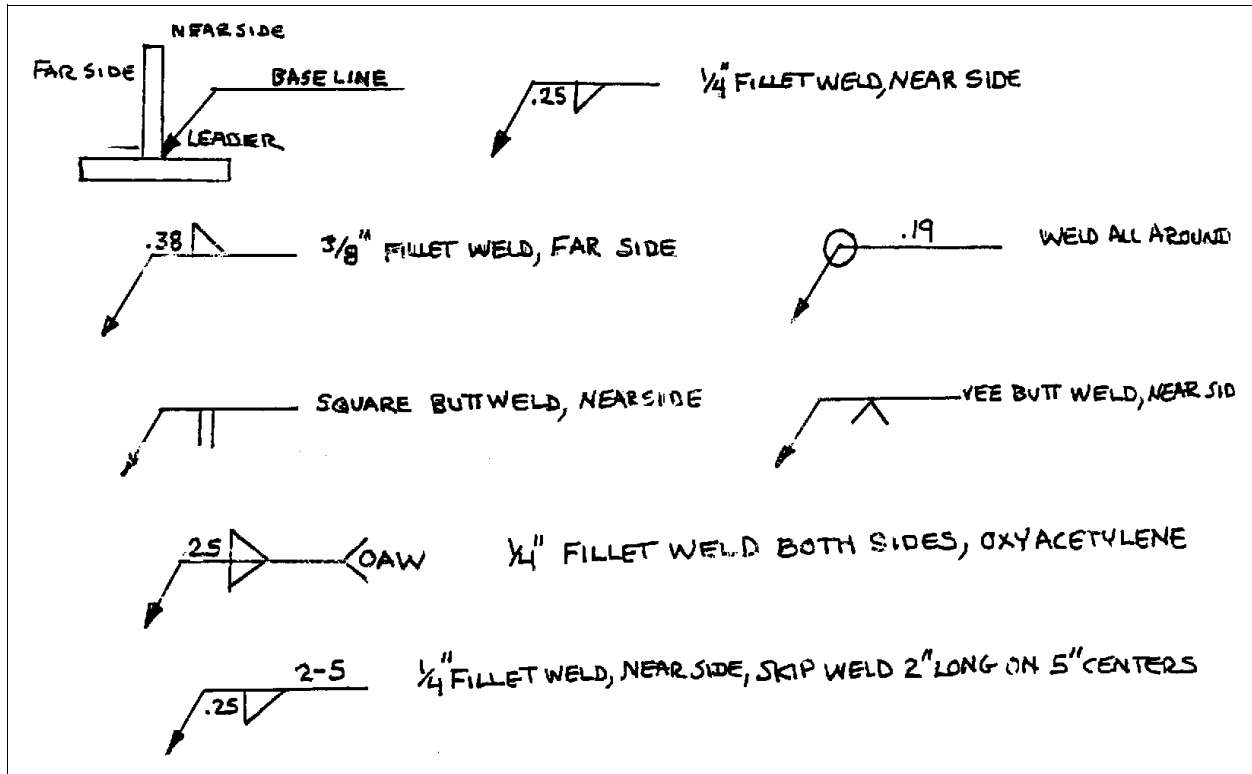
Back up all computer drawing files. Every semester someone loses a large drawing and has to start over. Don't let it happen to you.

WELDING DRAWINGS

Drawings of welded parts must use proper weld symbols and specify the weld process and the filler rod or welding rod to be used. In general we follow the American Welding Society Standards, ANSI/AWS A2.4-93. The AWS welding symbols are quite complicated, but most of the welding done at CSUS can be specified by a simplified set of symbols. The basic weld symbol is a base line with a leader pointing to the weld location. Information about the weld is shown on the base line. Information placed below the base line refers to the arrow or near side of the weld location. Information above the base line refers to the far side or opposite side. If the weld is to be on both sides then the information is repeated above and below the base line.

A great deal of weld information can be placed on the base line, but for most school projects the type of weld, and it's size are the only thing needed. Some welds are "skip" welds that have a length of weld bead followed by an unwelded portion and then another weld bead. The length and pitch of skip welds must be specified on the base line. Almost all of the welds done in the student

shop will be either fillet welds, or butt welds. For materials thicker than 1/4" it will be necessary to bevel the edge of the material prior to welding to insure good weld penetration. This type of butt weld is called a vee butt. Notice that the location of the information on the base line does not change when the leader is moved to the other end.



The type of welding process must be specified either on the symbol or as a general note. It is usually easier to use a note. If specified on the weld symbol a tail is added and the AWS designation shown. The AWS process designations for the types of welding we use at CSUS are given below.

WELDING PROCESS	AWS REFERENCE	COMMON NAME
Gas metal arc welding	GMAW	mig
Gas tungsten arc welding	GTAW	tig or heliarc
Flux cored arc welding	FCAW	flux core
Resistance spot welding	RSW	spot welding

Shielded metal arc welding	SMAW	stick or arc
Oxyacetylene welding	OAW	gas welding
Torch brazing	TB	brazing
Iron soldering	INS	soldering
Torch soldering	TS	soldering

The exact welding rod or filler material to be used is specified in the general notes.

Tolerances on welded structures will be large, usually $\pm .06$ or more. In addition, significant distortion and warping can occur.

The welded part is detailed like a single part. Enough dimensions must be shown to fully define the finished part. In a few cases it may be helpful to have drawings showing the cut dimensions of the individual pieces before they are welded together. Although it is not an “official” drawing, you may want to prepare a cut list of all the pieces giving the dimensions to which they are to be cut before welding.

Weld information can be included in the general notes if desired. For example if most of the welds are fillet welds and of the same size we could use a note instead of including the information of all the weld symbols. For example,

3. ALL FILLET WELDS .31 UNLESS OTHERWISE SPECIFIED.
2. OXYACETYLENE WELD WITH ??????? ROD

could be used instead of putting this information on the weld symbol.

Welded parts or weldments can be drawn in several ways. The best method is to show the weldment as a single part. This type of drawing is referred to as an inseparable assembly and is treated like a detail drawing rather than an assembly drawing. For this reason you can include all of the individual pieces of the weldment on one drawing (remember that a single drawing can have multiple pages). A drawing of this type would have a drawing of the completed weldment with weld symbols. The various pieces that go into the weldment can be dimension on the weldment or separately (but not both). If dimensioned on the weldment there must be enough dimensions to make the pieces. If dimensioned separately then only those dimensions that are controlled during welding would appear on the weldment drawing. If complex cuts are needed to fit the pieces together before welding then they may be dimensioned separately.

BILL OF MATERIALS

The bill of materials is one of the most important engineering documents. It provides a complete list of all parts needed to build the complete project. Parts lists are shown on assembly drawings, but the bill of material is a separate document. See the attached example. At a minimum the bill of materials must list the part number, part name, quantity, and vendor part number for purchased parts. Standard hardware such as fasteners do not have to have a drawing if they can be described completely in the bill of materials. They will still have a part number. Purchased parts do not have a drawing but will have a part number and must also have the vendor's part number listed. Do not use the vendor part number as your number.

Your bill of materials should have a column listing make or buy.

More complex projects will be made from a series of sub assemblies. In this case the bill of materials should list the assembly level as well. The complete device is called the top assembly and is level 0. All sub assemblies that go into it and all individual parts that are not in the sub assemblies will be level 1. All of the individual parts and sub assemblies that go into the level 1 sub assemblies will be level 2 and so on.

If two identical level 2 sub assemblies are used in level 1, the quantities of parts used to make one sub assembly are listed for that subassembly, and the quantity of the sub assemblies in level 1 will be 2. You multiply the number of parts in one sub assembly by the required number of that sub assembly to get the total part count. Each assembly is listed in detail only once.