WHAT EVERY MECHANICAL DESIGNER NEEDS TO KNOW ABOUT PRINTED CIRCUIT DESIGN

OVERVIEW

One of the more challenging parts of managing today's electronic product development process is integrating the various players in a project. PCB designers often find themselves sandwiched in the middle between Industrial or Mechanical designers and Electrical designers. Each side presents different and conflicting requirements, and it often falls to the PCB designer to resolve these differences.

The purpose of this article is to detail how to describe board mechanical information in a way that is easy and straightforward for a PCB designer to use, eliminates common sources of error, and facilitates checking.

PCB DESIGN SOFTWARE IS DIFFERENT FROM MECHANICAL CAD SOFTWARE

Compared to Mechanical CAD programs, PCB Design software has very limited mechanical design capabilities:

- Most PCB design software can only view a board in a single orientation and from a single direction. Boards are viewed from a single side and opposite side features and components are viewed as an "X-Ray" through the board.
- PCB design software typically has much less flexible dimensioning capabilities. In particular, most PCB software lacks the automatic linked-in dimensioning that mechanical CAD users take for granted.
- Most PCB design software has less flexibility in measuring and documenting distances between features. Even the lowest-cost Mechanical CAD software implements extensive relative measurement and feature "snap to" capabilities.

AGREE ON THE GROUND RULES

The most important step for a mechanical designer and a PCB designer is to agree on the ground rules for working together. Establishing a common coordinate system, orientation, dimensioning, and units is key to getting off to a good start.

COORDINATE SYSTEM

In the circuit board industry, it is standard to place the origin at the lower left hand corner of the circuit board, with the horizontal axis as the X axis and the vertical axis as the Y axis. Some designers prefer to use the lower left mounting or tooling hole as the origin. For boards with cutouts or chamfers in that corner, place the origin where the corner would have been without the cutout or at either remaining corner. For circular or radially-symmetrical boards, place the origin at the center of the board. Establishing a useful origin for unusual shaped boards may take some creativity.

ORIENTATION

If practical, orient the longest axis horizontally to allow the largest view area on a standard landscape computer monitor. Create all board drawings from the same viewpoint, unless there are compelling reasons to show them in another orientation. If drawings need to be viewed from the reverse "mirrored" side, be sure to keep the same origin and to provide notes to help the designer definitively establish which edge is which.

DIMENSIONING

PCB designers are used to working with Ordinate Dimensions. Writing all dimensions in ordinate style removes a common source of error by eliminating the need to do math to figure out feature and component locations.

UNITS

Agreeing to use the same units may sound pretty basic, but writing the units on the drawing makes sure that no one ends up making embarrassing assumptions.

DESCRIBE THE BOARD SHAPE, DESIGN RESTRICTIONS AND COMPONENT PLACEMENT

The Mechanical Designer needs to communicate the following to the PCB Designer:

- Board Physical Description: shape of board, including slots, cutouts and thickness, location and diameter of mounting and tooling holes.
- Design Constraints: maximum component height for both sides and keep-out areas.
- Location and part number of critical components, especially connectors.

Except for the simplest of boards, this is a lot of information to place in a single drawing. Cluttered drawings can be difficult to make and error-prone to read. When in doubt, create separate drawings for different information. Be sure to keep the orientation consistent and unambiguous!

PHYSICAL REQUIREMENTS

Draw the shape of the board to scale and detail slots, cut-outs, chamfers, mounting/tooling holes and hole clearances. For curved sections, show the centerpoint and radius, plus the location of the curve end-points. PCBs are milled to shape so concave corners need to be rounded. If there are board dimensions that can be expanded or shrunk, indicate them on the drawing. Be sure to note the required board thickness (0.062”, 0.093” and 0.031” thicknesses are most common).

Creating a system-level drawing can be a great way to communicate physical constraints. Show the board superimposed on an enclosure or in relation to other boards in a system.

DESIGN CONSTRAINTS

Indicate design constraints such as whether parts may be placed on both sides, and the maximum component heights for each side. Show areas that need to be free of components or free of traces. Place an arrow and a note on the drawing to indicate the direction of system airflow.

COMPONENT LOCATION

Specifying component locations can be one of the more challenging parts of a PCB design project. The geometric center is the best reference point for many parts such as chip resistors, capacitors, diodes, transistors and IC’s. However, other components such as connectors don’t always have obvious points of reference. Here are some guidelines:

- Use the geometric center of the mating surface as the reference point for vertical mount connectors.
- For right-angle connectors, dimension the center line and front edge.
- For through-hole connectors, include a set of reference dimensions for at least one pin (pin 1). Note that using the pad as a reference point does not always work for surface mount parts as the pad center may not correspond to either the pin or lead center.
- For all connectors, show the relative position of both pin 1 and for pin 2.
- For FFC/FPC (Flat Flex Cable/Flat Printed Circuit) connectors, indicate the cable conductive side and cable plug-in direction.

In general, don’t be anxious about including additional “reference” dimensions. When in doubt, add a note explaining the chosen reference points.

Finally, when positioning parts – especially connectors – remember that the PCB pads may extend considerably beyond the part body.

DON'T OVER-SPECIFY!

Start out with a quick discussion drawing before moving to a fully detailed drawing. Discussing the project with the PCB Designer before creating a completely specified drawing can save everyone a lot of work and frustration. There's nothing like spending a day drawing up every last detail of a board design only to find out that there is no way that a huge part can fit or that the cutout in the middle of the board leaves no room to place parts that need to be kept together.

As the design moves forward, be sure to try to give the PCB designer as much flexibility as possible. If practical, specify how the board size can be changed, show ranges for critical component locations and indicate both "nice to have" and "must have" requirements.

DESIGN CHECKLIST

_____ Discuss the project with the PCB designer before creating detailed drawings.

_____ Clearly indicate the date and/or revision level on all drawings.

_____ Maintain the same origin and orientation for all drawings.

_____ Specify the drawing units: English (inches or mils), Metric (meters or mm).

_____ Clearly indicate viewing direction and reference board edges.

_____ Use ordinate dimensions.

_____ For curved board sections, show the curve centerpoint and radius, plus the dimensions of the curve endpoints.

_____ Specify mounting and tooling hole locations, diameters and clearances.

_____ Indicate connector orientation by specifying the location of pins 1 and 2.

_____ For FFC/FPC connectors, indicate the cable conductive side and cable plug-in direction.

_____ Specify component and trace keep-out areas for both sides.

_____ Specify maximum component heights for both sides.

_____ Indicate how close components may be placed to the board edge.

_____ Don't put too much on any single drawing. Use multiple drawings to minimize the clutter.

_____ Specify the location of critical components. When unambiguous, use the component center as the reference point.

_____ When specifying the location of connectors, also include reference notes showing pins 1 and 2, and reference dimensions showing center lines, pad locations and/or front edges.

_____ Don't over-specify! Give the PCB Designer as much sizing and positioning flexibility as possible.