

Design Rules for Milling out Printed Circuit Boards in the Kista Mentorspace

If you are making a printed circuit board to mill out in the Mentorspace, here are the design rules you should use when laying out your board. Note that dimensions are in “mils” where 1 mil = 0.001 inches = 0.025 mm.

1. Except where a trace connects to a part pin such as an integrated circuit, the recommended minimum trace width is 10 mils. There is no maximum trace width limit, so you can make the trace width as big as you want within reason. A preferred signal trace size is 20 mils. If the point where a trace needs to connect to a pin is narrower than 10 mils, then you should match the width of the pin.
2. Except where forced by fine pitch part pin spacing, such as on an integrated circuit, the recommended minimum trace spacing is 10 mils. There is no maximum spacing limit, so you can space them apart as much as you want. If the pins on a part you are connecting to have a spacing less than 10 mils, then you should match the spacing of the pins, however you should increase the spacing of the traces as soon as practical away from the part pins. In general, it is best to space out your traces as evenly as possible.
3. Keep parts and traces at least 20 mils away from the edge of the board.
4. Holes less than 125 mils in diameter are drilled. Holes larger than 125 mils are contour routed. We usually have the following drill sizes in the Mentorspace:

22 mils
31 mils
35 mils
37 mils
40 mils
125 mils

If you use a hole size less than 125 mils that is not on this list, then use the closest size that we have instead.

5. The milling machine does NOT create plated through holes. Keep that in mind when creating vias because you will have to use bits of wire to connect the two ends of the via and solder them all by hand. Dealing with large numbers of vias is a pain, so you may want to hand route your board in order to minimize the number of vias created. Autorouters often are somewhat reckless when it comes to putting down lots of vias.
6. The recommended size of vias is 70 mils in diameter with a 22 mil hole. This is to make them easy to solder. To avoid the chance of drill breakout, vias should not be less than 55 mils in diameter. Although vias can be any shape, the best choice is round.
7. Vias under components should be avoided, especially for surface mount parts. If you put a via under a surface mount part, you may have a hard time soldering the part onto the board because the solder blob on any vias under the part will interfere.

8. Your design must include a board outline of non-zero width. Without it the CAM tools used to prepare your board for milling won't be able to generate a tool path to cut out your board.

9. Be careful with through hole connectors. If you route traces to the connector pins on the same side of the board where the body of the connector is placed, then you may find that the body blocks access to the pins after it is placed on the board. If that happens you will have trouble soldering the pins. Remember that holes drilled by the milling machine are not plated though, so if the connector body blocks access to the pins then you should route traces to the connector on the side of the board opposite to the one where the connector is placed. If the connector is angled or completely open frame (no body), or if the connector is surface mount then none of this applies and you can route the traces to it on any side of the board you want.

After you have finished your board layout, you need to generate the following files in the indicated format. Most good circuit board CAD tools can generate these formats.

a) One file that defines the bottom (solder side) of your PC board. It needs to be in Gerber 274X format.

b) If you have a two sided board, then you also need to provide one file that defines the top (component side) of your PC board. It needs to be in Gerber 274X format.

c) One file that defines your board outline. It needs to be in Gerber 274X format.

d) One file that defines the holes that need to be drilled. This file needs to be in Excellon format. This is a standard format for drill files. If your board is very simple and has no holes or vias, then you don't need to generate this file.

When you have these files ready, then you are good to go. In the spirit of the Mentorspace, if you don't know how to use the milling machine or the CAM tools to generate the tool paths then ask! For sure you will find someone in the Mentorspace who will help you. Or, send me email. From time to time we have seminars on how to make printed circuit boards. Email me if you would be interested in attending one.

One last thing is that the milling machine is capable of really good work. The rules above will help you make a usable board, but there are all sorts of advanced techniques that will let you make boards with very thin traces and spacing and controlled impedances. We also have methods to include writing on the board showing part outlines, logos, and other useful information. If you want to get good on any of the advanced techniques, just send me email!

Mark