

Tips on designing PCBs for the Seneca PCB lab

INTRODUCTION

Differences in manufacturing processes (milling, industrial, etching):

PCB milling process (**Figure 1**) is a subtractive mechanical method used to prototype printed circuit boards without chemicals or masks. A CNC Machine uses high-speed carbide cutting tools to remove copper from selected areas of a copper-clad laminate, leaving behind the desired traces, pads, and circuit geometry.

In contrast, industrial high-volume PCB manufacturing optimizes for throughput, consistency, and multilayer complexity. Industrial processes typically use photolithography, automated drilling, copper electroplating, solder mask application, and silkscreen printing. They can produce very fine trace widths, tightly controlled impedances, and complex multilayer boards at a much lower per-unit cost when manufacturing thousands of boards. However, setting up masks, tooling, and production lines adds lead time, making industrial fabrication less suitable for rapid design changes during prototyping.

Compared with chemical etching, milling eliminates the need for photoresist materials, UV exposure, etchants, and chemical waste disposal. Chemical etching removes unwanted copper by dissolving it chemically after a masking process, which can achieve finer features and smoother trace edges than milling. However, it requires handling hazardous chemicals and raises environmental concerns. Milling is generally cleaner, safer, and more convenient for laboratory use.

A main limitation of PCB milling is its lower resolution compared to chemical etching. Mechanical tool diameters limit the minimum trace width and spacing that can be reliably produced, making it more challenging to fabricate high-density circuits. Milling can also result in rougher trace edges.

This document provides tips on how to make your boards better suited for the milling manufacturing process used at the Seneca PCB Lab. Milling machine-ready boards have many advantages, including:

- Higher tool and raw material efficiency.
- More reliable and easier-to-solder boards.
- Faster delivery and reduced reprints due to soldering-related issues.

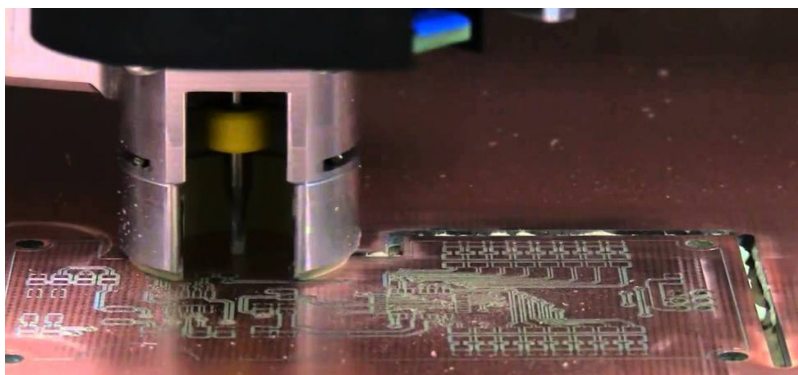


Figure 1: PCB Milling Process description.
Source (LPKF)

TIP #1 – BOARD SIZE

Students must use the PCB footprint efficiently. The board outline must be no more than 100 mils from any outer component. Projects not meeting this requirement will be returned to the student. **Figure 2** shows a non-optimal design, and **Figure 3** shows the expected board outline for this design.

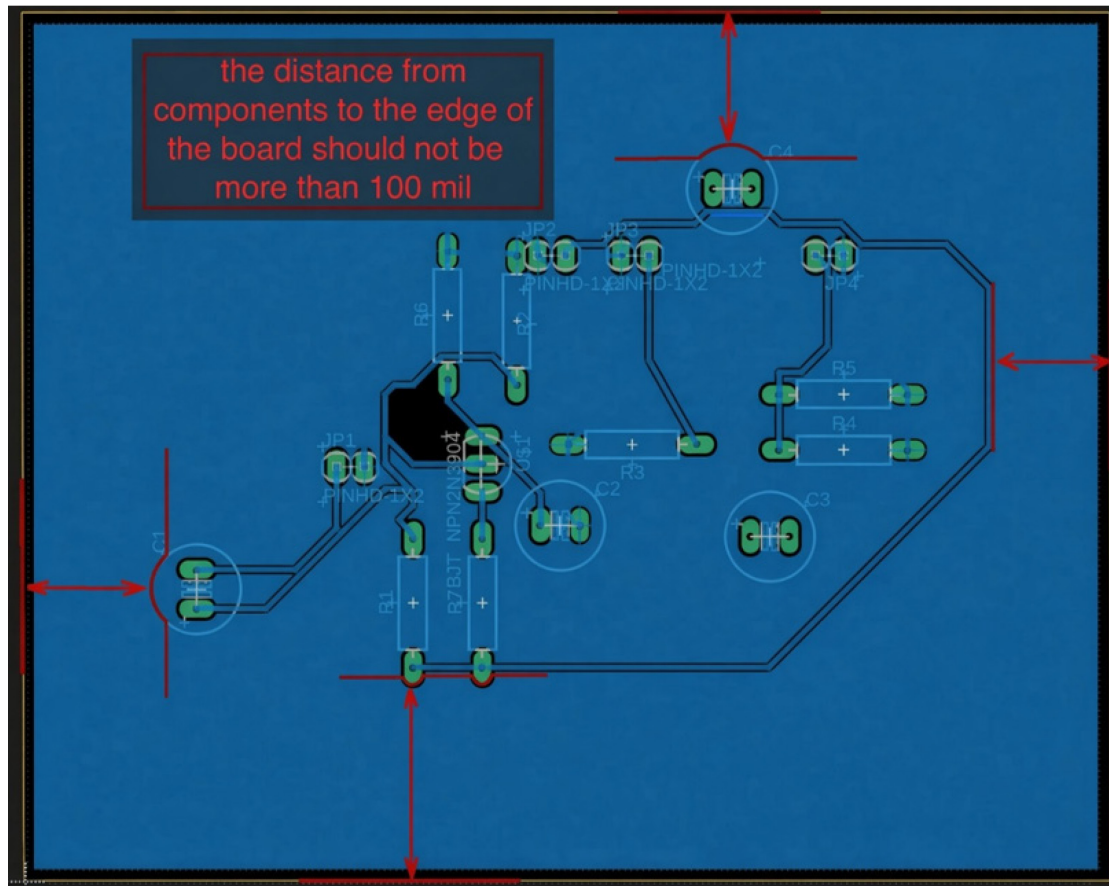


Figure 2: PCB Design with non-optimal board area utilization.

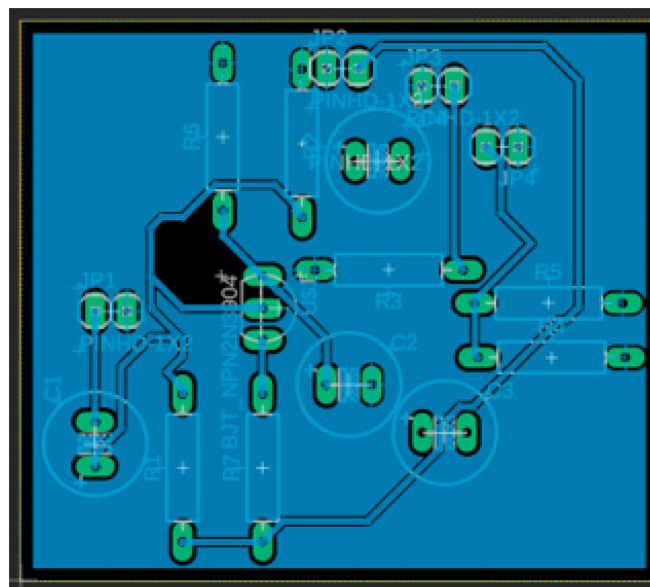


Figure 3: Proper utilization of the board's area.

TIP #2 – TRACE ANGLES

During manufacturing, milling tools can remove excessive material based on their diameter. This problem is more pronounced on thin traces or trace intersections where the connection angle is less than 90 degrees. **Figure 4** shows two problematic intersections and how to fix them.

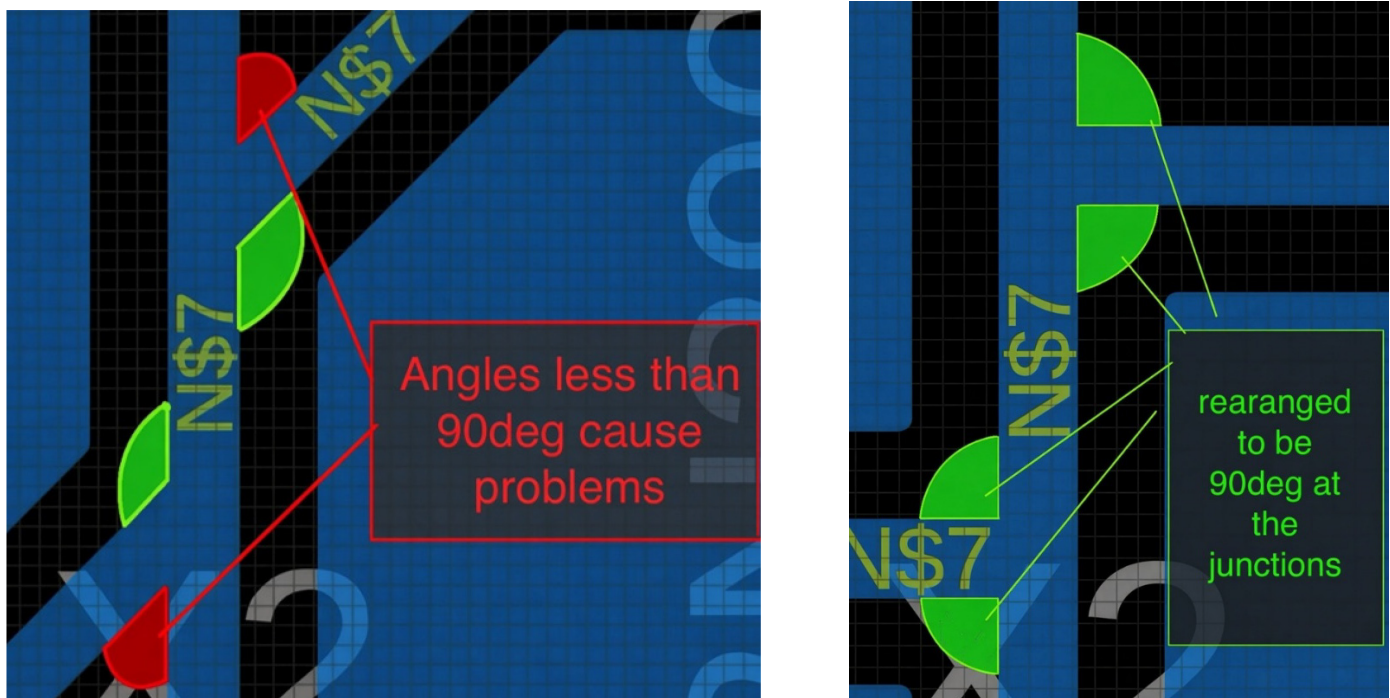


Figure 4: Issues with intersection angles and the recommended solution.

However, 90-degree angles should be avoided in single-trace routing (**Figure 5**).

single-trace

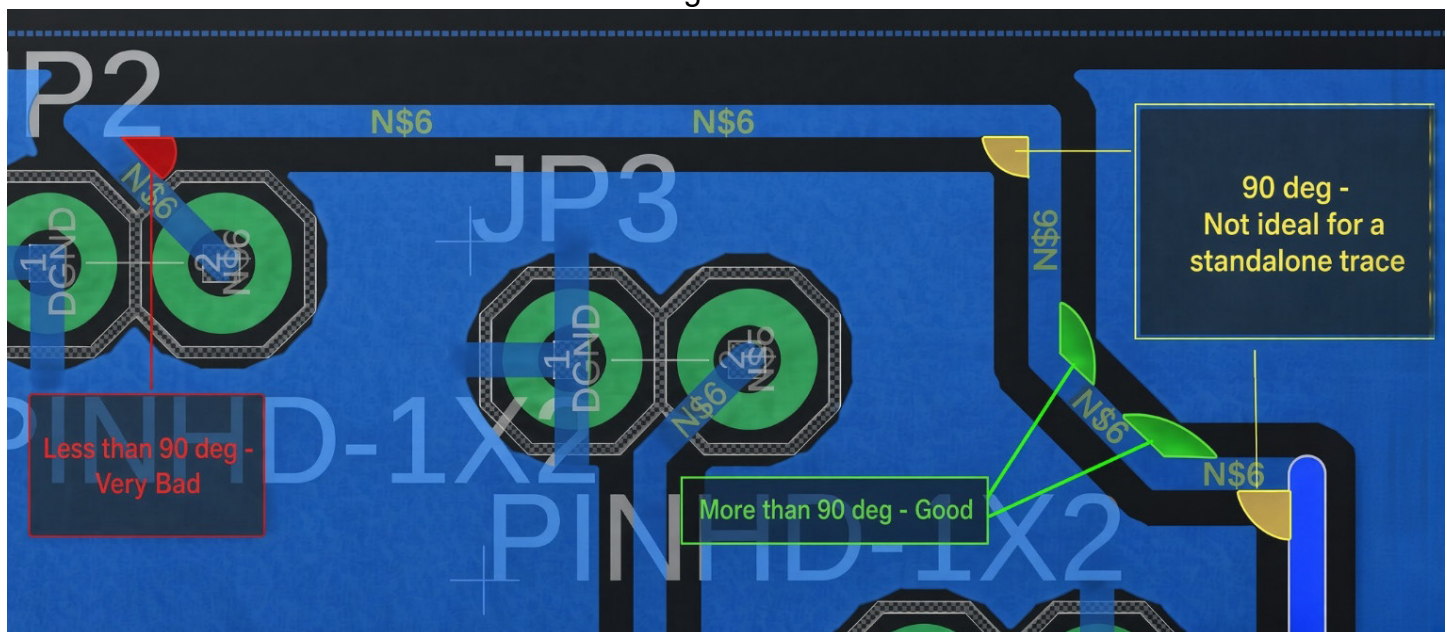


Figure 5: Angle recommendations on single-trace routing.

TIP #3 – PAD TO TRACE CLEARANCE (SAME SIGNAL)

Traces should originate at the centre of the pad and come out perpendicular to the surface. For an elongated slot-style pad like the ones used in the SEMET library, it should be 90 degrees out of the long side **OR** 90 degrees out of the short side (**Figure 6**). Any angle is acceptable for circular pads.

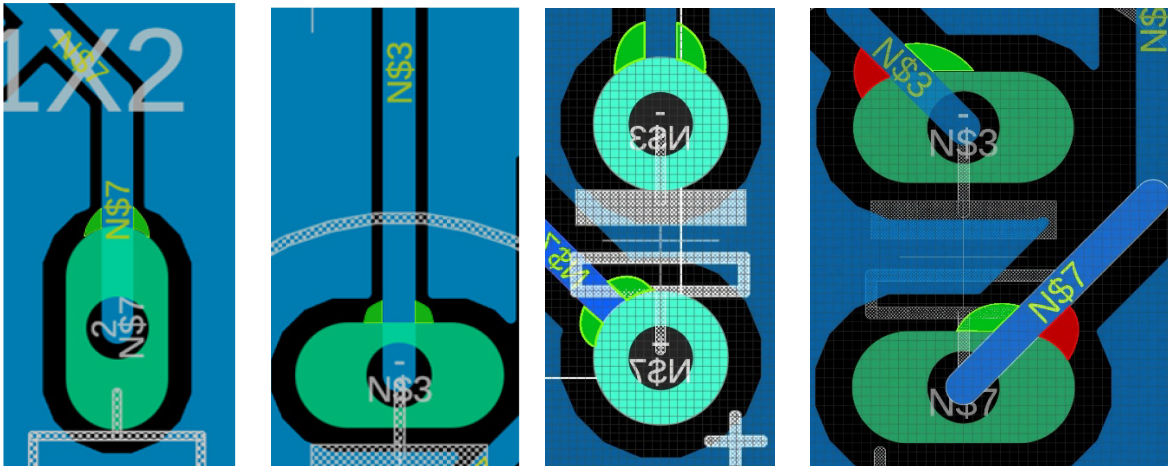


Figure 6: Correct trace placement on pads compared to incorrect placement (rightmost).

Like the example in Tip #2, milling tools can remove excessive material from pads and traces if the rule above is not followed (**Figure 7**).

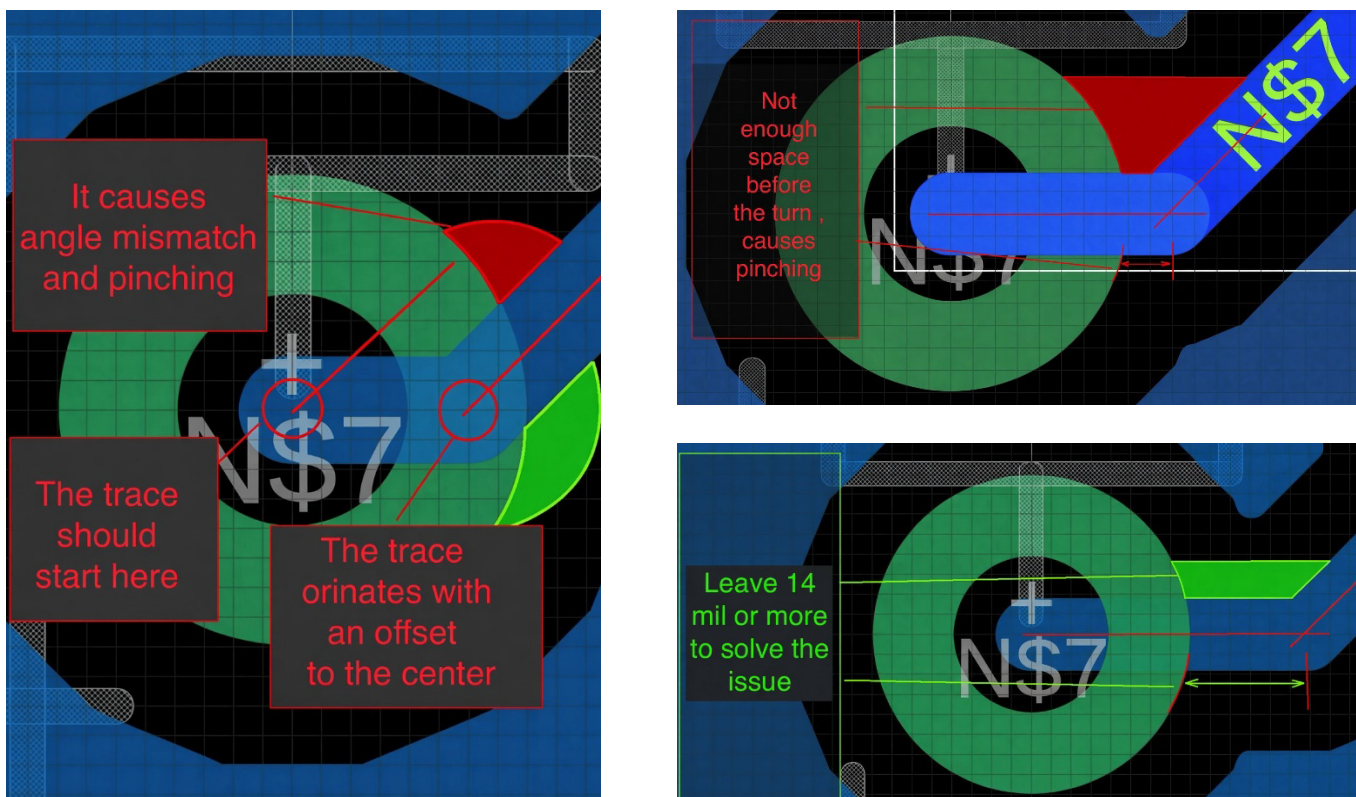


Figure 7: Another example of incorrect trace placement and its fix.

TIP #4 – PAD SIZE (PAD STACK)

Pad sizing depends on the shape of the pads. SEMET library elongated slot pads have two dimensions: the long side and the short side (**Figure 8**). The minimum Pad Size, 20 mil, applies to the **Long Side** only. SEMET library, by default, comes with corrected pads. Circular pads only have one dimension.

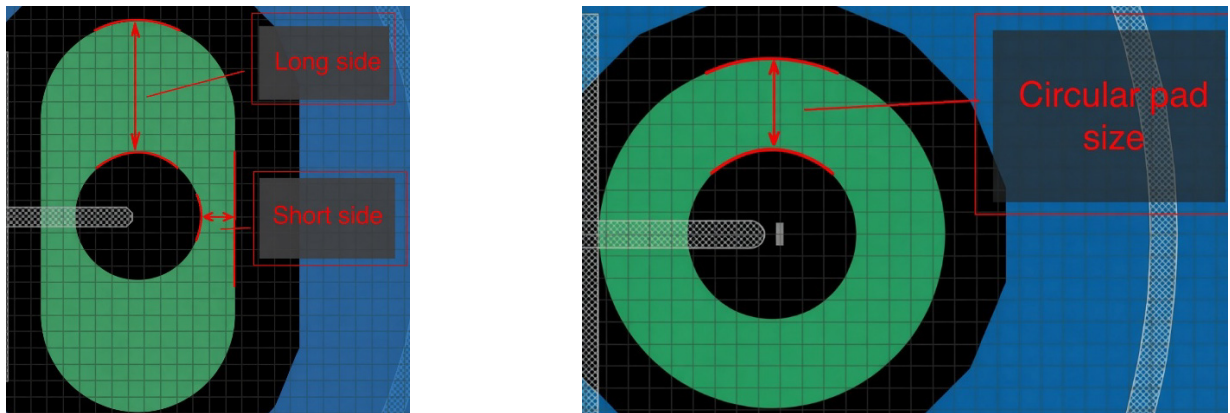


Figure 8: Slot and circular pads compared.

TIP #5 – TRACE CLEARANCE

Trace clearance refers to the distance between two traces (**Figure 9**) on the board. This distance is part of the DRC Check performed by Fusion, and it is controlled by the “**Wire – Wire Clearance**” rule. The one exception is for same-signal traces. Fusion DRC Check does not flag it as an issue, so the student must flag it manually (**Figure 9**).

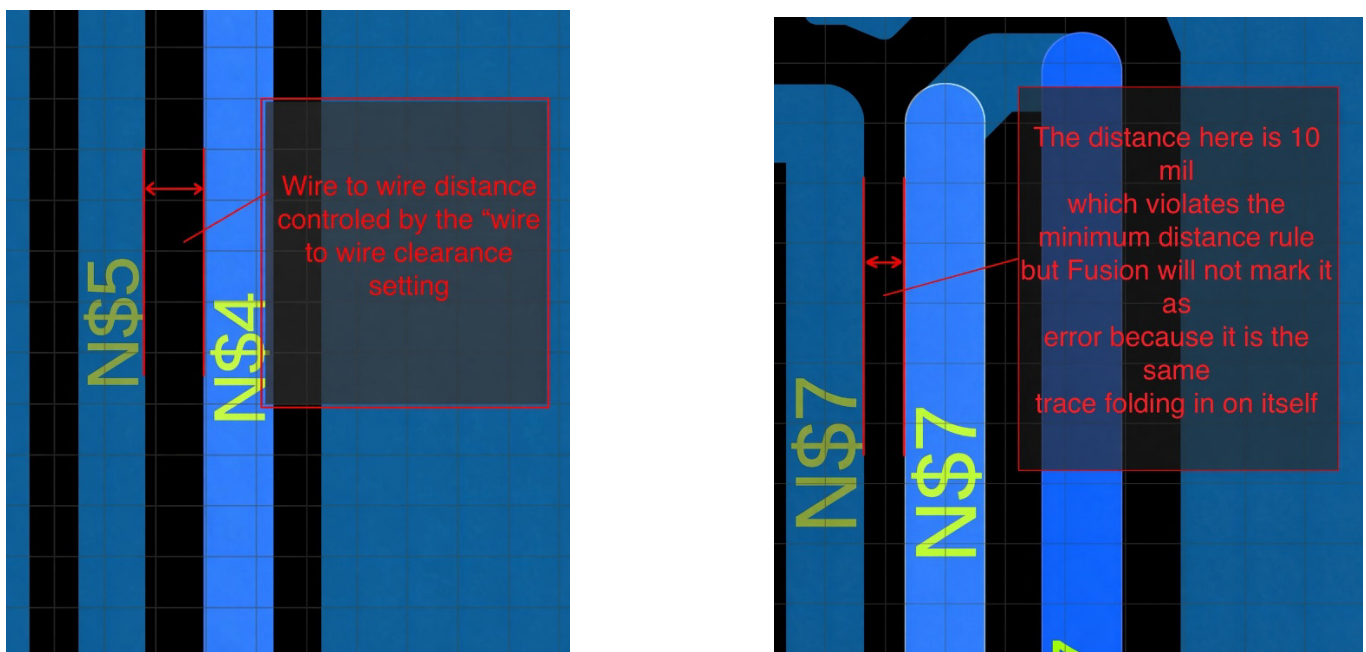


Figure 9: Wire Clearance Rule example.

TIP #6 – COPPER POUR (POLYGON)

For Copper Pour, use these recommended settings in **Figure 10**. The minimum values can be found in the [PCB Page](#).

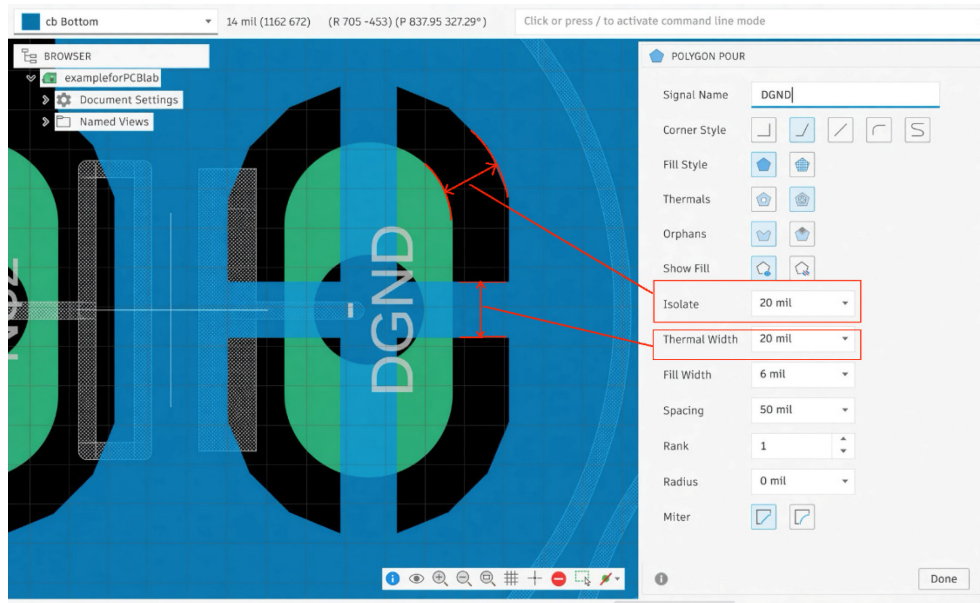


Figure 10: Copper Pour Settings.

TIP #7 – MOUNTING HOLE

The minimum hole diameter our machine can produce is 1.0 mm (39.37 mil). For mounting applications, the SEMET library includes a convenient parameterized component called MOUNT-HOLE (**Figure 11**). This component is designed with DRC-safe settings and can be easily configured to match the required hole size for the mounting hardware being used. A mounting hole diameter of 3.2 mm is typically used for M3 hardware, while 4.3 mm is recommended for M4 hardware. By adjusting the component parameters, you can match the hole diameter and clearance to the selected screw size while maintaining compliance with PCB design rules.

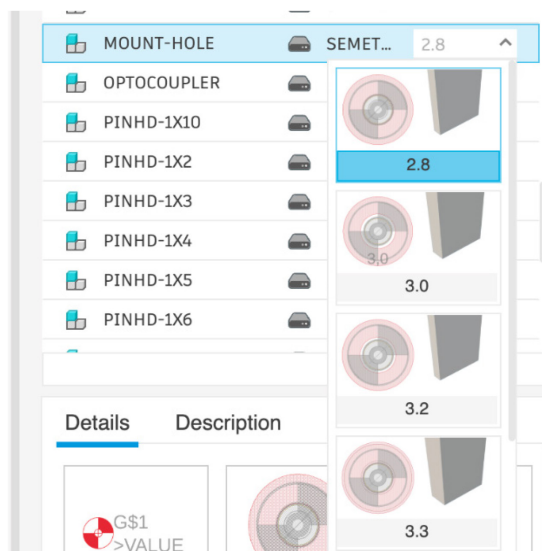


Figure 11: MOUNT-HOLE Library component.