

7 QUICK TIPS FOR TAKING YOUR PCB SILKSCREEN FROM ORDINARY TO EXTRAORDINARY

JOHN MCMILLAN, MENTOR GRAPHICS



P C B D E S I G N

W H I T E P A P E R

www.mentor.com

INTRODUCTION

Today, it's hard to imagine a GPS that doesn't list landmarks, street names, or even exit numbers. Every bit of visual information we can get from map symbols and text makes navigating easier and faster. The same is true for a PCB, yet we often overlook a great opportunity to provide a wealth of information about our PCB's design on the silkscreen. The information we can, and should, supply on PCB silkscreens should go beyond component outlines, reference designators, and pin 1 indicators. In this paper I'll describe and display some of the enhancements you can make to your PCB designs' silkscreens that can add value at both the bare board and assembly levels and from the test bench to the installation.

SILKSCREEN 101: THE BASICS

PCB silkscreen text follows a traditional numbering scheme from top to bottom, left to right, on the PCB. Text can be rotated (typically 90 degrees CCW) if necessary since all reference designators rarely fit in one direction. The orientation of reference designators for a PCB's physical interfaces, such as connectors, switches, and LEDs, can be at any angle necessary that benefits the end product.

Industry standards such as IPC provide designers with some guidance on spacing minimums for PCB silkscreen elements. The standard describes the basic silkscreen artwork required in order to identify the component extents and the pin 1 indicator, both of which are necessary to denote component orientation. The end goal for the PCB silkscreen is simple: to make it as easy as possible to identify the parts on the PCB during assembly, testing, and troubleshooting.

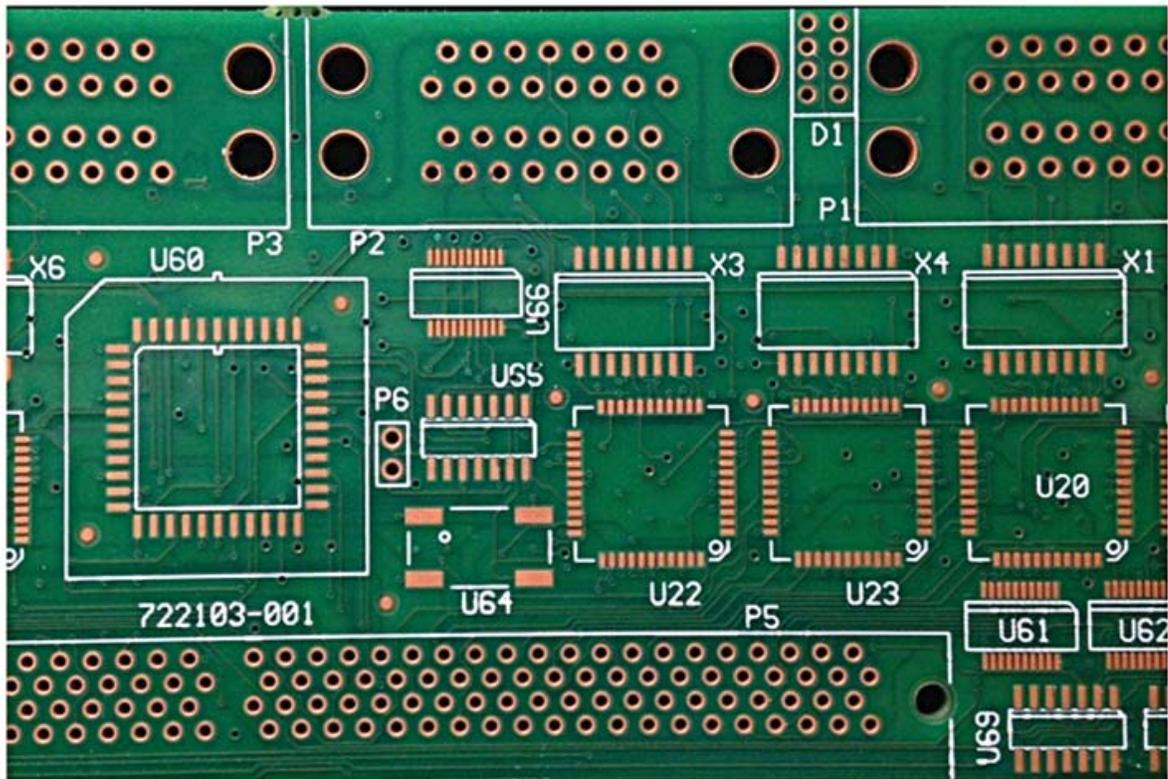


Figure 1: Basic silkscreen for component outline, reference designator, and pin 1 indicator.

Figure 1 represents a very basic silkscreen approach. The orientation and pin 1 locations of the components are captured using chamfered corners along with small circles. You'll notice that U60 has a larger secondary silkscreen outline indicating that a socket may be used in a prototype run but will likely fall into the DNL (do not load) column in the production run. The connector silkscreen for P1, P2, and P5 is very minimal and should include more informative annotations wherever possible as I'll describe in Step 1. I'm sure everyone can see many opportunities to improve this silkscreen.

TIP 1 – CONNECTOR ANNOTATIONS

My introduction makes the analogy that a PCB silkscreen should be just as informative as a GPS or map. That is, by simply glancing at the PCB the viewer should be able to figure out where and what everything is. Connectors provide designers with a great opportunity to display additional silkscreen graphics that immediately aid in the testing and troubleshooting phases without having to reference schematic diagrams or datasheets.

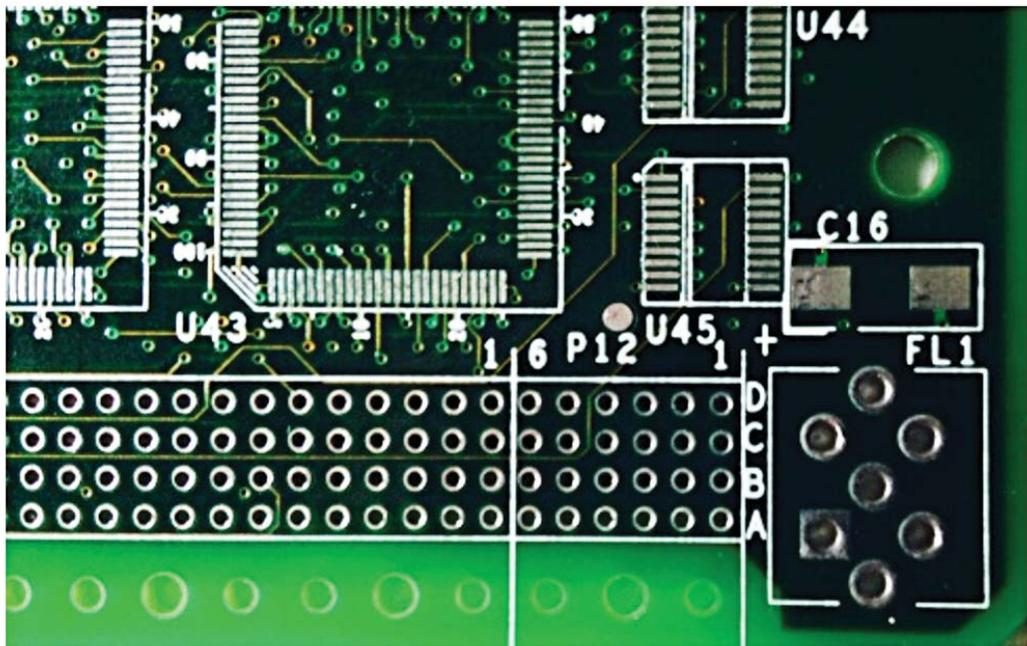


Figure 2: Additional connector annotations can aid in both troubleshooting and testing.

Looking at connector P12 in Figure 2, we can see that by including additional silkscreen for rows A, B, C, and D and column numbers 1 and 6 for the rows, we can easily identify and locate any pin on the connector. Even if annotations end up underneath components after board assembly, they're still a good practice if value is added.

Quick exercise: How long does it take you to locate P12-B3 in Figure 2 above? (I expect only a few seconds at most.)

And don't forget to take full advantage of silkscreening on both sides of the board, as applicable, for through-hole components. Annotations are extremely helpful for connectors because pin configurations may be numeric or alphanumeric, pin numbers mirrored for plugs vs. receptacles or different pin numbering schemes like Z and daisy chain pin patterns. In Figure 3 you can see clearly by the added pin number that this connector has a Z style numbering configuration.

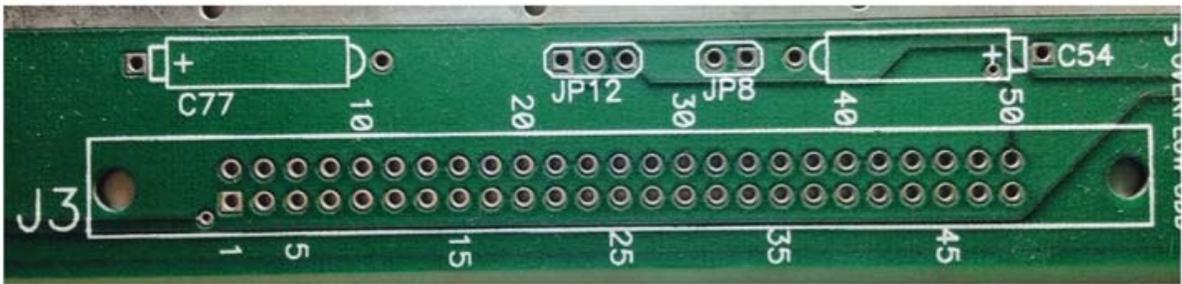


Figure 3: Connector annotations indicating Z pattern vs. linear.

TIP 2 – MULTI-LEAD COMPONENT ANNOTATIONS

There are a number of pin configurations for multi-lead components like PLCCs, SOICs, QFPs etc. For smaller pin-count devices, such as an SOIC8, it is easy enough to identify which pins are pin 1, 4, 5, 8, and everything in-between because we all know that pins are numbered CCW from pin #1. However, when we get to larger pin-count devices, like a QFP100, adding small tick marks and pin numbers can be very useful. In Figure 4 we can see that small tick marks, shorter for multiples of 5 and larger for multiples of 10, help with locating specific pins quickly without counting.

Quick exercise: How long does it take you to locate pin #45 in Figure 4 below? (I expect only a few seconds at most.)

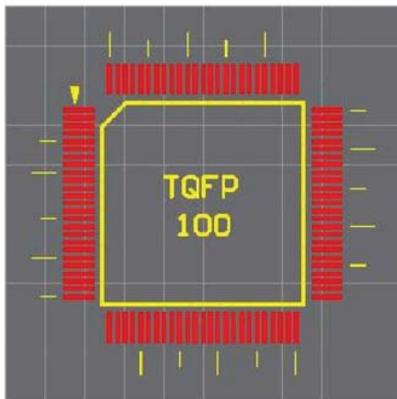


Figure 4: Tick marks at multiples of 5 and 10.

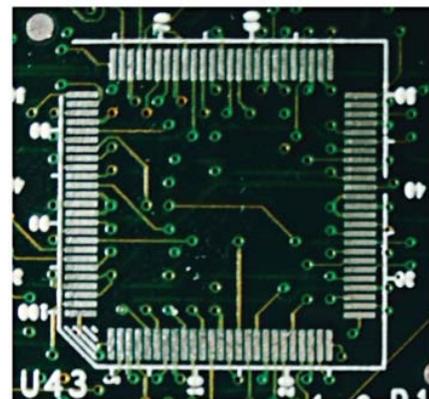


Figure 5: Tick marks at multiples of 5 and numbers on the 10s.

Figure 5 is an example of a silkscreen where actual pin numbers were added beside the tick marks for the multiples of 10. Today's designs rarely have room to provide this level of pin numbering. Something else you need to consider with very small silkscreen annotations is the resolution. Silkscreen line resolutions less than 4 mils and font heights less than 35mils high may require an additional process of exposure and developing so checking your PCB fabricator capabilities may be a good idea.

TIP 3 – POLARITY ANNOTATIONS, THOSE PESKY DIODES

Silkscreen annotations for diodes can be particularly difficult because it is not always easy to determine whether the anode goes to positive or negative. When possible, additional markings should be utilized to ensure that the component placement and rotation are correct. In Figure 6 you can see that the diodes (indicated by 'CR') have additional markings below them to aid in identifying the orientation. PCB designers should also be cautious when using a "+" symbol alone to indicate diode polarity. In Figure 6 you also see chamfered corners. These are helpful because they identify the pin 1 orientation regardless of the graphics inside the silkscreen outline (which will be covered by the part when loaded).

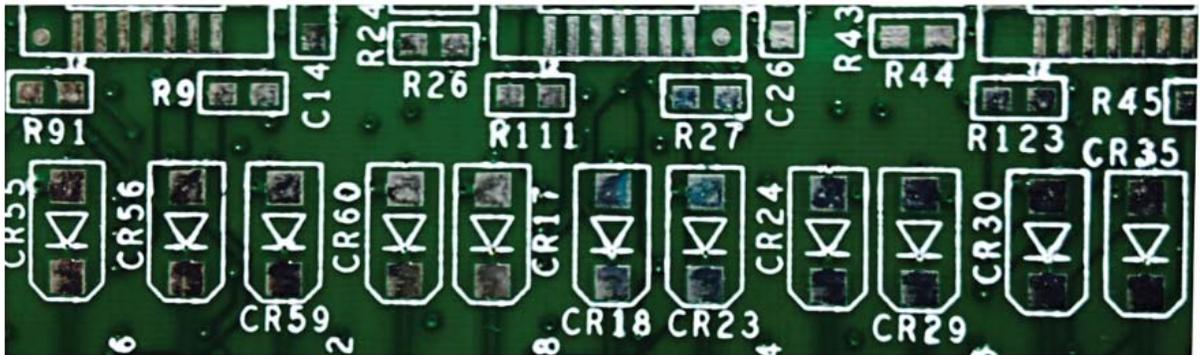


Figure 6: Silkscreen annotations for diodes (CR*) should clearly denote anode/cathode and polarity.

TIP 4 – PHYSICAL INTERFACE COMPONENT LABELS AND ANNOTATIONS

PCB designers often miss the opportunity to provide valuable information to testers, installers, and end users on the silkscreen. Whenever possible, label physical interface components like connectors, switches, LEDs, buttons, or any other device that could provide a visual indicator, physical guidance, or any other important information to the design viewer.

In Figure 7 you can see four dual LED displays, DS1 through DS4. Notice that the addition of silkscreen labels located beside each LED provides annotations for voltages and overflow odd/even, as applicable for each. Another great addition to this silkscreen that you'll notice is the white rectangular box to the left of DS3. This box enables the PCB assembler to write in, or ink stamp, a custom value, typically determined by purchase order for designs that support different configurations for that LED indicator position.

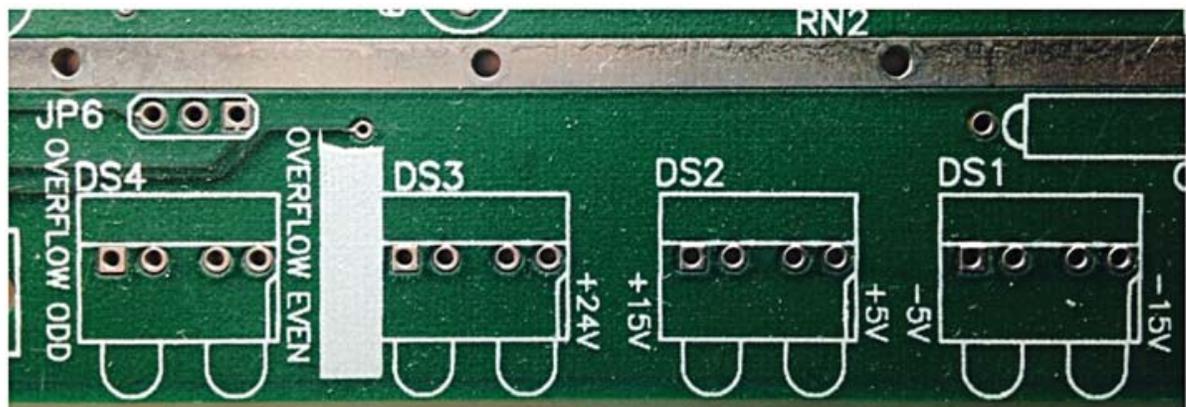


Figure 7: Adding meaningful silkscreen annotations and labels provides valuable information.

Some other great opportunities to enhance the silkscreen for physical interfaces are:

- On/Off positions for switches. Identify the switch type: power, mode, voltage, etc.
- Reset buttons, potentiometers, and fuse values.
- LED status: Fault, Error, Good, etc.

PCB designers typically consider silkscreen annotation improvements for physical interface components as a secondary step to the PCB database after the final placement is completed. PCB designs with a fabrication or environment level of Nominal and Most provide the best opportunities to enhance silkscreen annotations. For PCBs designed for a 'least' fabrication and environment level, space is at a premium and, in some cases, the silkscreen for reference designators may be omitted. In each case, physical interface annotations should be clear and meaningful but never overcrowd the board, so if you have the space, use it!

TIP 5 – REFERENCE DESIGNATOR ANNOTATIONS

In Step 4 I mentioned how the different PCB design fabrication and environment levels of Least, Nominal, and Most impact the extent of available space for silkscreen annotations. This is true for Reference designators as well. Smartphones are a great example of a "Least" environment design where reference designations may be omitted altogether. Dense networking cards are a good example of a "Nominal" environment in which PCB design space is at a premium. Depending on the design's density, different approaches can be considered for reference designator locations. See Figure 8.

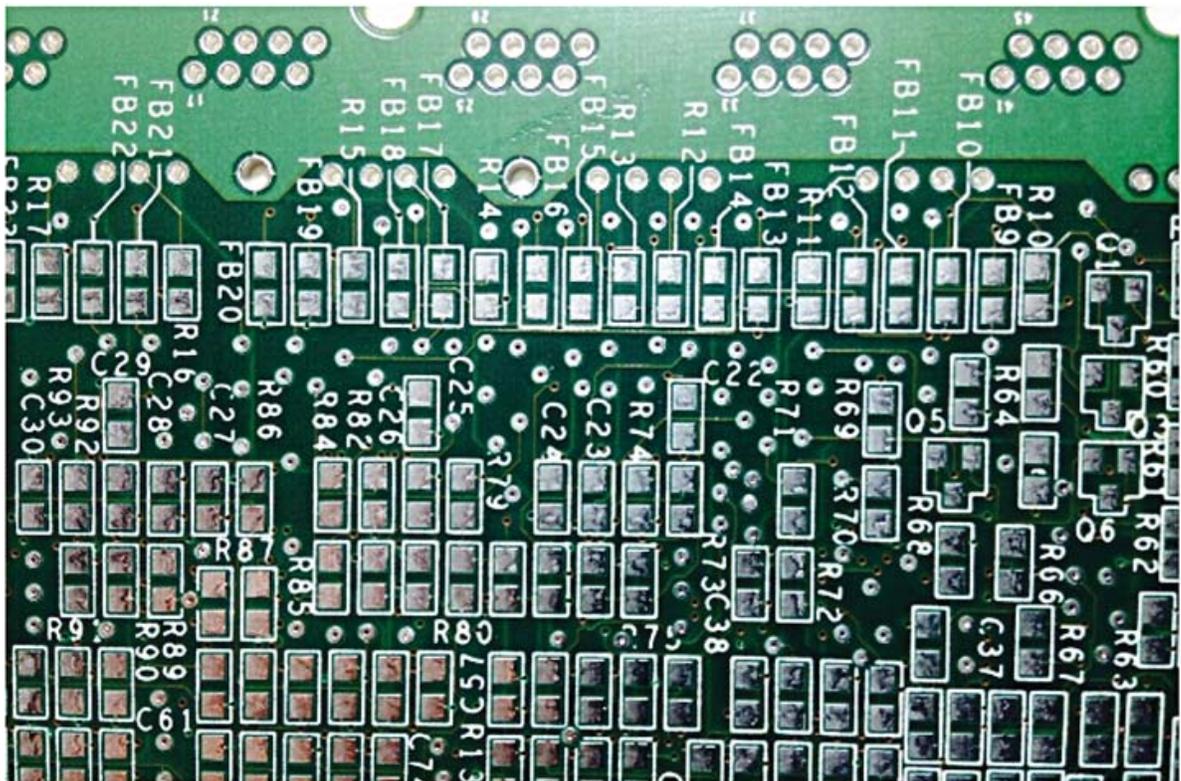


Figure 8: Silkscreened line work is utilized to lead you from the ferrite bead "FB**" reference designator to its associated land pattern.

TIP 7 – DFM ANALYSIS CHECKS FOR SILKSCREEN ANNOTATIONS

We've already established that adding useful and intuitive silkscreen annotations to PCB designs can be invaluable to the testers, installers, and technicians who handle the PCBs, but they are of no use if you can't read them. In every case, Design for Manufacturability (DFM) analysis tools like Valor® NPI should be run on PCB designs to ensure that all the silkscreen data generated pass the fabrication proximity and clearance rules.

DFM analysis tools check silkscreen graphics and flag violations of each instance where the measured distance between the silkscreen and another design object is less than the pre-defined values.

Even PCB designers who add silkscreen annotations with the best intentions may discover that a component graphic or text has been clipped or rendered unreadable. See Figure 11.

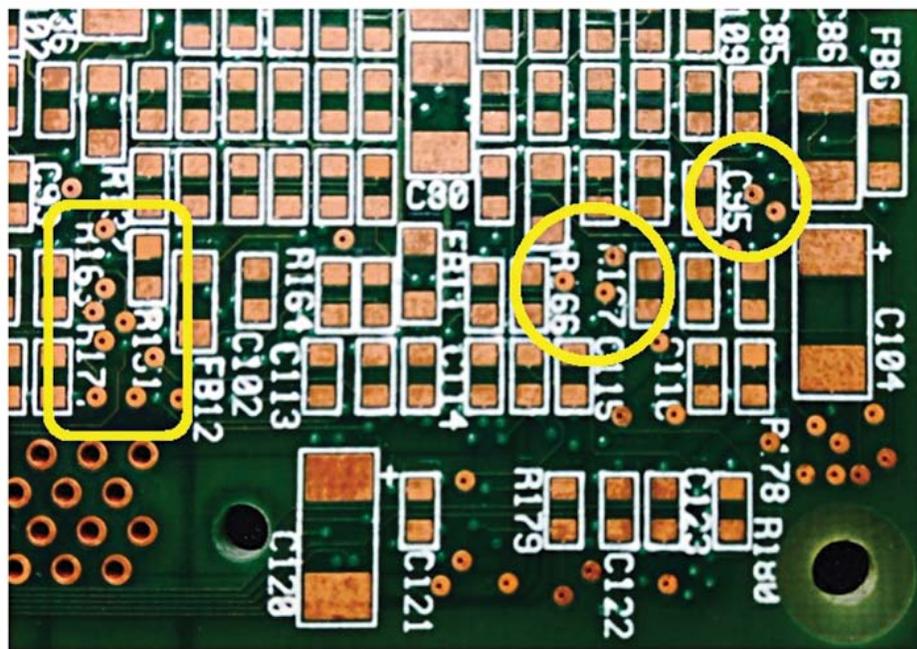


Figure 11: Circled areas show illegible testpoints that can be detected with DFM Analysis.

PCB silkscreens are generated with breaks in the text to avoid contact with pads in the design, thus preventing manufacturing errors that result from ink coming in contact with the pad area. DFM analysis provides break options that enable you to specify clearances for pads and by what distance the silkscreen must clear the pads. The purpose is to prevent the silkscreen ink, which is non-conductive, from contacting areas where soldering may be required.

Running DFM analysis on PCB designs ensures that all silkscreen proximity rules, such as silkscreen-to-pad, silkscreen-to-soldermask, and silkscreen-to-via hole/via pad rules, are adhered to. Likewise, silkscreen sizes for each instance where text line widths are greater than the user-defined maximum value or less than the user-defined minimum value are also flagged. See Figure 12.

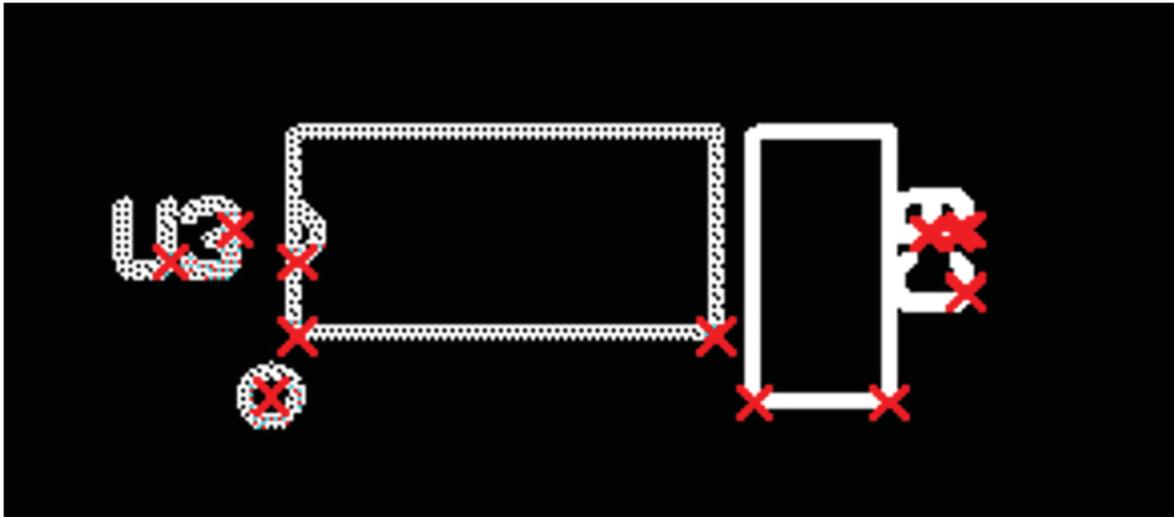


Figure 12: DFM analysis markers indicate rule violations for line and text width violations.

IS YOUR PCB SILKSCREEN ORDINARY OR EXTRAORDINARY?

Many PCB designers consider their designs a piece of art, and rightly so. It takes a lot of planning and expertise to lay out high-quality, first-pass-success designs. I hope that some of the 7 steps above provide you with new ideas that you can use on your next design, all the way from the bare board to the assembly, and from the test bench to final installation.

For the latest product information, call us or visit: www.mentor.com

©2014 Mentor Graphics Corporation, all rights reserved. This document contains information that is proprietary to Mentor Graphics Corporation and may be duplicated in whole or in part by the original recipient for internal business purposes only, provided that this entire notice appears in all copies. In accepting this document, the recipient agrees to make every reasonable effort to prevent unauthorized use of this information. All trademarks mentioned in this document are the trademarks of their respective owners.

Corporate Headquarters
Mentor Graphics Corporation
 8005 SW Boeckman Road
 Wilsonville, OR 97070-7777
 Phone: 503.685.7000
 Fax: 503.685.1204

Sales and Product Information
 Phone: 800.547.3000
sales_info@mentor.com

Silicon Valley
Mentor Graphics Corporation
 46871 Bayside Parkway
 Fremont, CA 94538 USA
 Phone: 510.354.7400
 Fax: 510.354.7467

North American Support Center
 Phone: 800.547.4303

Europe
Mentor Graphics
 Deutschland GmbH
 Arnulfstrasse 201
 80634 Munich
 Germany
 Phone: +49.89.57096.0
 Fax: +49.89.57096.400

Pacific Rim
Mentor Graphics (Taiwan)
 11F, No. 120, Section 2,
 Gongdao 5th Road
 HsinChu City 300,
 Taiwan, ROC
 Phone: 886.3.513.1000
 Fax: 886.3.573.4734

Japan
Mentor Graphics Japan Co., Ltd.
 Gotenyama Garden
 7-35, Kita-Shinagawa 4-chome
 Shinagawa-Ku, Tokyo 140-0001
 Japan
 Phone: +81.3.5488.3033
 Fax: +81.3.5488.3004

