

PCB Current Carrying Capacity

BusBoard Application Note AN0004

The maximum current rating for prototyping PCBs depends on many factors, such as:

- Track width and shape
- Hole sizes and unplated/plated
- PCB thickness and number of layers
- Construction methods
- How close heat-producing parts are
- Number of parallel (or closely spaced) track carrying high current
- Airflow and cooling, ambient temperature

Here are some guidelines that will help you plan you project.

1. The current carrying capability of a copper track depends on the width and thickness of the track, how heat is conducted away (via air and/or the PCB laminate), and what temperature rise can be tolerated.

A conservative number is to allow a 10 degree rise in track temperature due to the electrical current. Some designers allow for a 20 or 30 degree rise.

The ambient temperature that the board is operating in is also a factor.

2. The IPC-2152 standard provides a way to estimate the current carrying capacity of PCB tracks. However, the standard is somewhat complicated to apply. We recommend using software to apply the IPC-2152 track width calculations.

The Saturn PCB calculator is a great tool which will do the IPC-2152 calculations and allow you to plug in different numbers and see the results. (It also has many other useful calculators.)

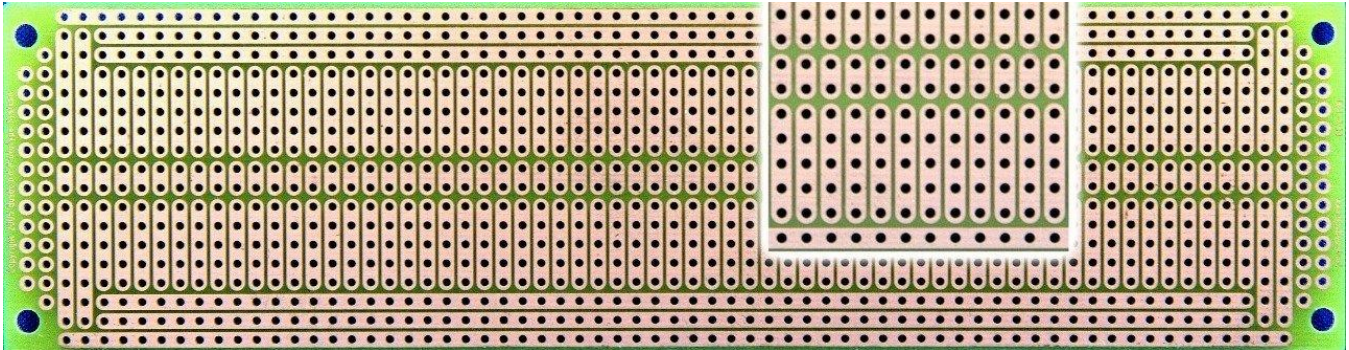
See http://www.saturnpcb.com/pcb_toolkit.htm

Install it and click on the **Conductor Properties** tab for Conductor Current calculations.

Note that the older IPC-2221A standard is now considered obsolete and it may not have been conservative enough for external tracks.

3. Current Example #1 – BR1, ST3U and POW3U Prototyping PCBs

The track width is 85 mils for BusBoard [StripBoard \(ST3U\)](#) and [PowerBoard \(POW3U\)](#) and [BR1 Solderable PCB BreadBoard](#). The width is actually slightly less because holes are drilled in it, so the temperature rise will be a little higher unless you cover the holes with solder.



BR1 Solderable PCB BreadBoard

BPS prototyping boards use 1 oz. per square foot of copper which is 1.37 mils thick. (1 mil = 1/1000th of an inch, so it is 0.00137" thick). Enter it into the calculator as 0.5oz base copper weight plus 0.5 oz plating.

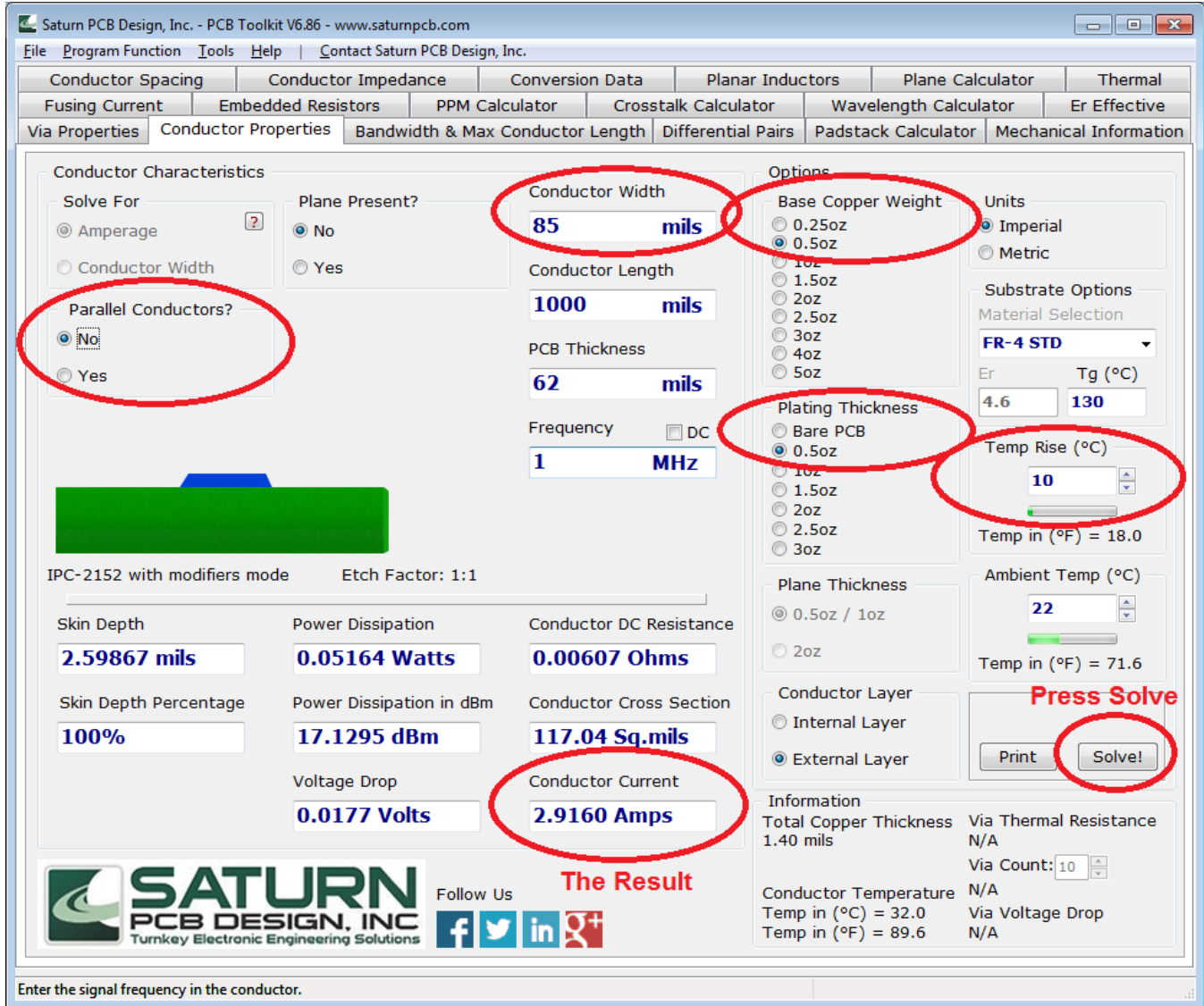
Plugging those numbers into the Saturn Calculator gives the following currents:

- 2.92 Amps for a 10° C. rise, 1 conductor
- 0.98 Amps each for a 10° C. rise, multiple parallel conductors
- 3.96 Amps for a 20° C. rise, 1 conductor
- 1.33 Amps each for a 20° C. rise, multiple parallel conductors

The 1 conductor number is the expected temperature rise of a single track by itself with the heat allowed to dissipate into free air. The multiple parallel conductors number is lower because there are multiple tracks generating heat, so the 10° or 20° C rise occurs with less current (per track).

These numbers do not include derating to account for real-world situations where tracks and components are located more closely together and air flow may be restricted.

The screen capture on the next page shows the software and the setting used.



Saturn PCB Design, Inc. - PCB Toolkit V6.86 - www.saturnpcb.com

File Program Function Tools Help | Contact Saturn PCB Design, Inc.

Conductor Spacing | Conductor Impedance | Conversion Data | Planar Inductors | Plane Calculator | Thermal
 Fusing Current | Embedded Resistors | PPM Calculator | Crosstalk Calculator | Wavelength Calculator | Er Effective
 Via Properties | Conductor Properties | Bandwidth & Max Conductor Length | Differential Pairs | Padstack Calculator | Mechanical Information

Conductor Characteristics

Solve For: Amperage Conductor Width

Plane Present?: No Yes

Parallel Conductors?: No Yes

Conductor Width: **85 mils**

Conductor Length: **1000 mils**

PCB Thickness: **62 mils**

Frequency: **1 MHz**

Options:

- Base Copper Weight: 0.5oz
- Plating Thickness: Bare PCB
- Plane Thickness: 0.5oz / 1oz
- Conductor Layer: External Layer

Units: Imperial Metric

Substrate Options: Material Selection: **FR-4 STD**

Er: **4.6** Tg (°C): **130**

Temp Rise (°C): **10**

Temp in (°F) = 18.0

Ambient Temp (°C): **22**

Temp in (°F) = 71.6

IPC-2152 with modifiers mode Etch Factor: 1:1

Skin Depth: **2.59867 mils**

Power Dissipation: **0.05164 Watts**

Conductor DC Resistance: **0.00607 Ohms**

Skin Depth Percentage: **100%**

Power Dissipation in dBm: **17.1295 dBm**

Conductor Cross Section: **117.04 Sq.mils**

Voltage Drop: **0.0177 Volts**

Conductor Current: **2.9160 Amps**

Information:

- Total Copper Thickness: 1.40 mils
- Via Thermal Resistance: N/A
- Via Count: 10
- Conductor Temperature: N/A
- Temp in (°C) = 32.0
- Via Voltage Drop: N/A
- Temp in (°F) = 89.6

The Result

Press Solve

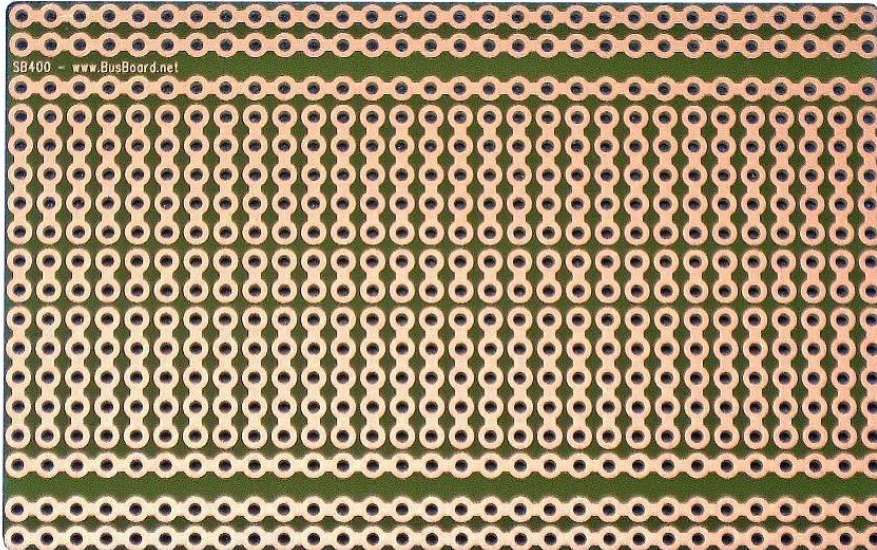
Print Solve!

Enter the signal frequency in the conductor.

Your construction methods can greatly affect the thermal characteristics of the tracks. Derate the numbers to have a safety margin for your system. A test with an IR camera to see the actual temperatures is recommended if you are pushing the limits.

4. Current Example #2 – SB404 Solderable PCB BreadBoard

The SB4 and SB404 use tracks that are narrower between the holes to make it easier to cut tracks.



The track width between holes is 40 mils (0.040"). The pads are 85 mil circuit with 42 mil holes, which leaves 43 mils of copper next to the holes. Therefore, we use 40mil track width for the calculations.

Plugging those numbers into the Saturn Calculator gives the following currents:

- 1.87 Amps for a 10 C. rise, 1 conductor
- 0.56 Amps each for a 10 C. rise, multiple parallel conductors
- 2.53 Amps for a 20 C. rise, 1 conductor
- 0.76 Amps each for a 20 C. rise, multiple parallel conductors

5. A way to be conservative is to thicken the tracks to handle higher currents. Run a bead of solder along the length of track to increase the thickness and the current carrying capability.

Running the current through 2 tracks in parallel is another way of increasing the current carrying capability and decreasing the temperature rise..

Other Resources

Here are some other resources that are helpful for understanding trace width vs. current carrying ability. They discuss the older IPC-2221A standard, but they still provide useful insight.

"Determining Circuit Board Current Carrying Capability" by Jack Olson

<http://frontdoor.biz/PCBportal/HowTo2152.pdf>

<http://frontdoor.biz/PCBportal/HowTo2152.xls>

"Calculation Of PCB Trace Width Based On IPC-2152" by Lazar Rozenblat

<http://www.smps.us/pcb-calculator.html>

Revision History

Rev.	Date	Description	Author
1	16 Nov 2015	Created.	Scot Kornak

Print Date: 16 November 2015

© 2015 BusBoard Prototype Systems Ltd.

This document may be freely copied and distributed in unmodified form.