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) tracks carrying high current
- Airflow and cooling, ambient temperature

To provide general guidance on PCB track current capacity, we provide the following suggested maximum rated currents for prototyping PCBs. These values are the conservative IPC-2152 standard values.

BPS Part# (Track Width)	Recommended Max Current, Single Conductor (10º C rise)	Recommended Max Current, Multiple Conductors (10ºC rise)
BR1 (0.085”)	2.9 Amps	1.0 Amps
ST1, ST2, ST3U (0.085”)	2.9 Amps	1.0 Amps
PR1, PR2, PR3U (0.085”)	2.9 Amps	1.0 Amps
PR2H1, PR2H2, 3U (0.085”)	2.9 Amps	1.0 Amps
SB300, SB400, SB830	1.9 Amps	0.6 Amps

1 PCB Current Rating Guidelines

Please consider the following PCB current rating guidelines and practices if you would like to carry high currents through the PCB tracks for your 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 at 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.

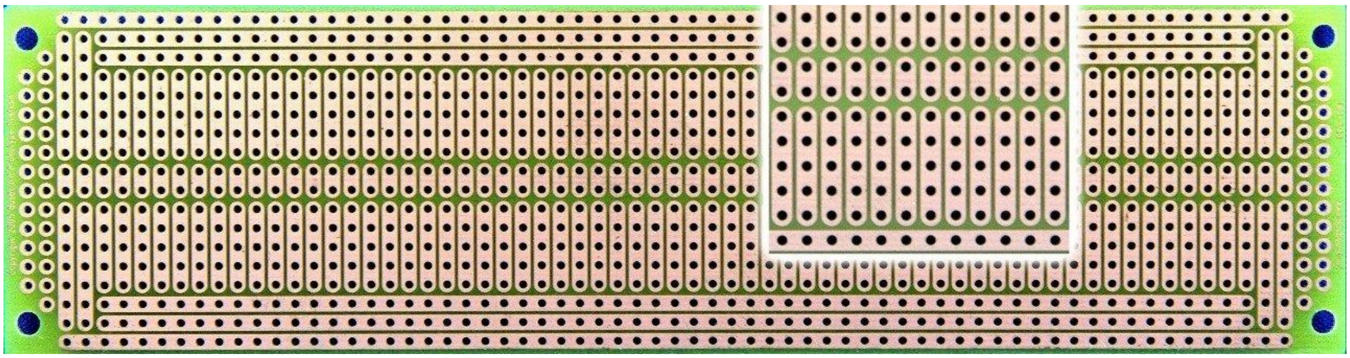
   The Saturn PCB calculator is a great tool which will do the IPC-2152 calculations and allow you to plug in different numbers and see the results. (It also has many other useful calculators.)
   See http://www.saturnpcb.com/pcb_toolkit.htm

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 0.5 oz base copper weight plus 0.5 oz plating.

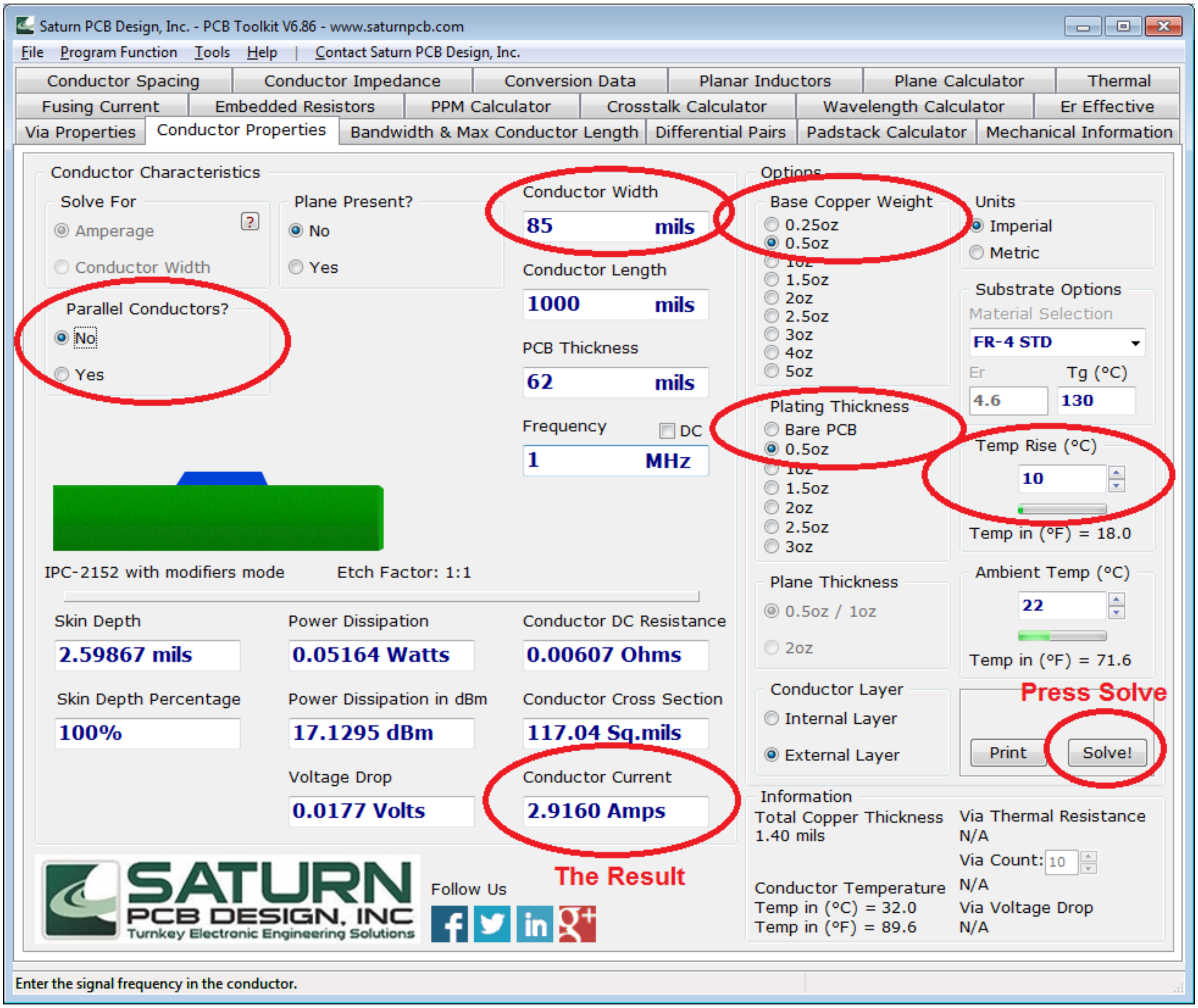
Plugging those numbers into the Saturn Calculator gives the following currents:

- 2.92 Amps for a 10º C. rise, 1 conductor
- 0.98 Amps each for a 10º C. rise, multiple parallel conductors
- 3.96 Amps for a 20º C. rise, 1 conductor
- 1.33 Amps each for a 20º 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 10º or 20º 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 closely together or an enclosure may restrict cooling air flow.

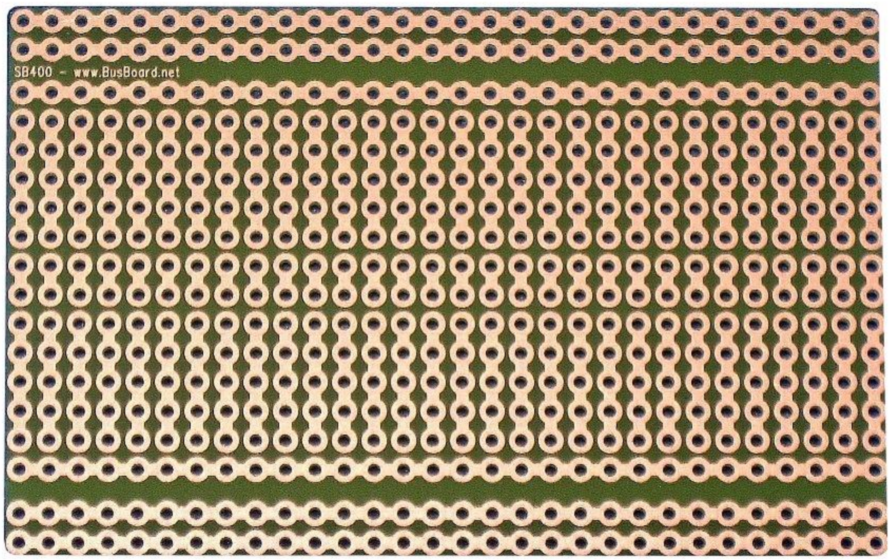
The screen capture on the next page shows the software and the setting used.



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

Increasing Current Capacity

To carry higher currents, you can make the tracks thicker with solder. Run a bead of solder along the length of track to increase the thickness and fill the holes. This increases the current carrying capability and decreases the temperature rise.

You can also connect 2 or more tracks in parallel to carry the current through multiple tracks to increase the current carrying capability and decrease 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
2	2-Nov 2016	Part# table with suggested current rating added to page 1.	Scot Kornak

Print Date: 2 November 2016