

OPERATOR'S MANUAL

UltraCAD Design, Inc. PCB Trace Calculator v. 3.3

This is the latest version of UltraCAD's popular PCB Trace Calculator. Version 3.3 is updated from version 3.2 in one important way:

1. The operating temperature input in the Trace Data window now allows the entry of operating temperatures below zero. See the last page for more on this change.

This manual covers the major operating features of version 3.3. The figures and references may be to version 3.2, since the change to version 3.3, from an overall standpoint, is a relatively minor change.

Licensing:

The calculator requires a license file to operate. A license file can be obtained from this site:
<http://www.ultracad.com/ucadpcblic.htm>

Place the license file in one of two places:

1. The root directory of the system drive (probably in c:\), or
 2. In a specially prepared folder under the root folder named \ucadpcb (probably c:\ucadpcb\)
- These are the only two places the program will look for the license file.

System Requirements:

The program is written in Visual Basic version 6.0. It will run under any version of Windows starting with NT. You may or may not have the support files required for the program to run already installed in your system:

- The Visual Basic Run Time Library
- The Visual Basic controls mscomctl.ocx and msflxgrd.ocx

If you do not already have these files, an error message will display when you start the program. Included in your download package (ucadpcb3.3.zip) are the two controls and a run-time file named vbrun60sp4.exe. (These files are also available at <http://www.ultracad.com/calc.htm> if you have misplaced them.) If necessary, execute vbrun60sp4.exe and copy the control files to the \Windows\system32 folder.

* Special note for users of 64 bit systems:

Windows x64 (at least some systems) apparently protects itself from additions to the System32 folder. In such cases, do this:

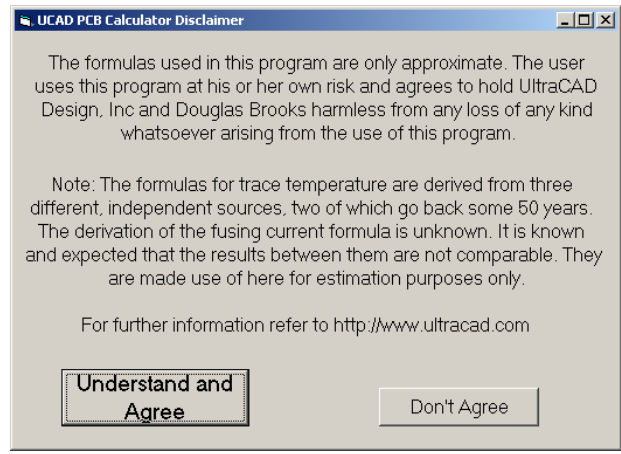
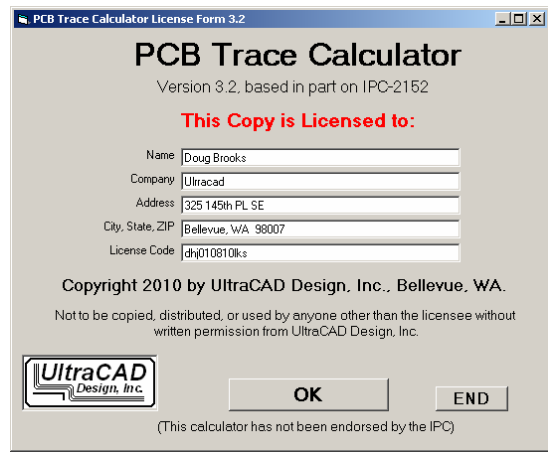
1. Place the DLL or OCX file in a convenient folder (it will probably have to stay there)
(I will use msstdfmt.dll in the root folder, c:\, as an example.)
2. Open the Command Prompt as Administrator (it is in the Accessories folder.)
(Right-click and select Run as Administrator.)
3. Navigate to C:\Windows\System32 directory (assuming Windows is installed on the C drive.)

4. Enter `regsvr32.exe c:\msstdfmt.dll` and hit return (Don't overlook the space.)
(Be sure to enter the complete path to the DLL or OCX file)
5. You should get a message that the control has been registered.

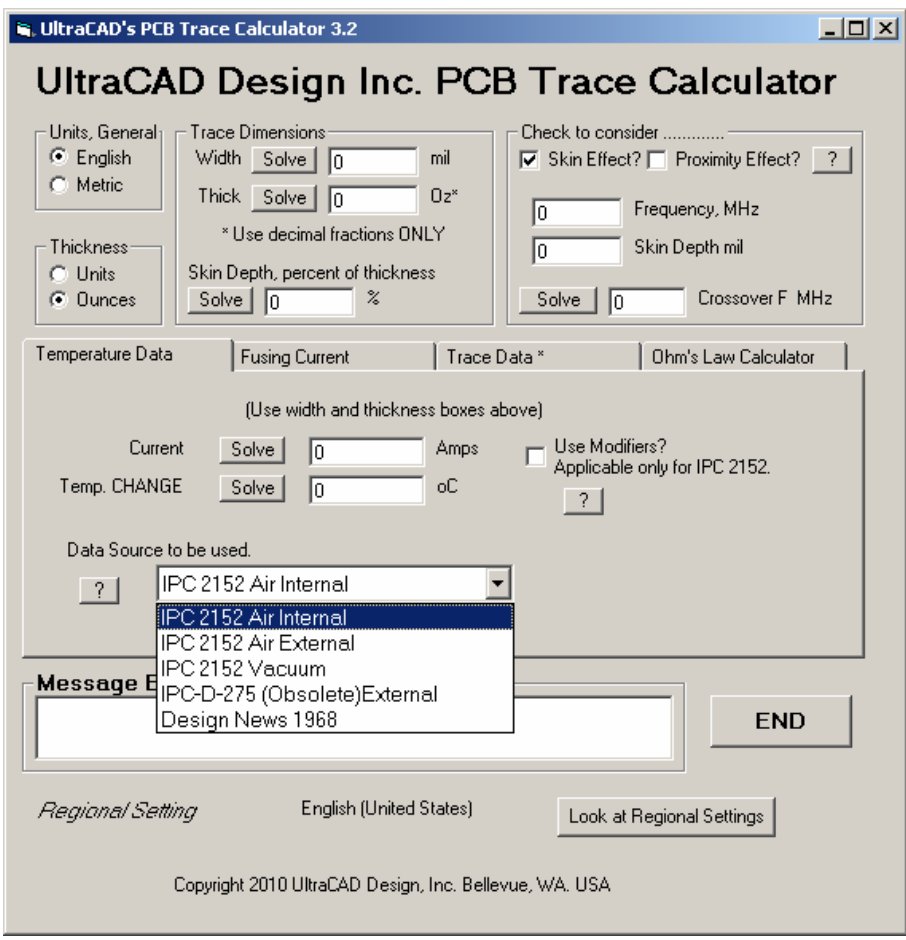
You may have to do this for two or three dlls/controls, depending on which calculators you have.

Starting the Program:

When you first start the program, the information included in the license file will be displayed. If the license is not found, an error message will say so. Following that a disclaimer screen will open. You must agree to the disclaimer to operate the program.

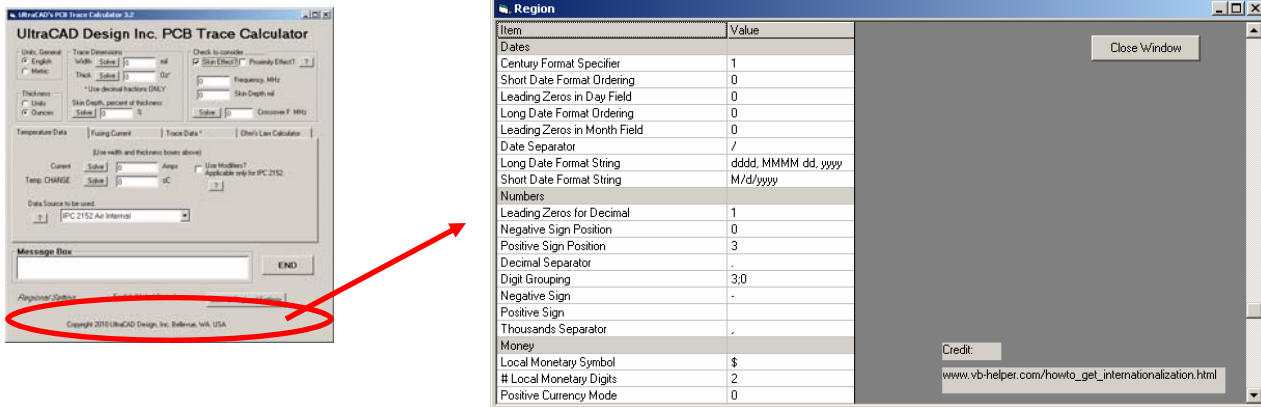


After you agree to the disclaimer, the main program window opens:



Regional Settings:

The program is “region aware.” That means it should understand, and automatically adjust for (among other things), the local conventions for the decimal and thousands separators. The local regional setting for your computer is shown near the bottom of the window, and clicking on the “Look at Regional Settings” command button will open another window showing all of the settings in effect for your region.



Skin Effect Settings:

The calculator can compensate for high-frequency skin effect. For example, suppose we enter a frequency of 15 MHz and a trace thickness of 2 Oz into the calculator. The skin depth for a 15 MHz signal is .6713 mils. As long as the skin depth is deeper than 1/2 the trace thickness, the skin effect will not limit the current. If the skin depth is shallower than 1/2 the trace thickness, it will begin to impact the current flow. For a 2 Oz. trace, the thickness is about 2.7 mils (and 1/2 the thickness is about 1.35 mils). Therefore, in this example the skin effect is limiting the current flow. The calculator shows that the skin effect is extending into each half of the trace thickness by 49.73% (.6713 mils is 49.73% of 1/2 the 2.7 mil trace thickness.) We can also see that the cross-over frequency for a 2 Oz trace (where the frequency just begins to impact the trace current) is 3.7 MHz.

The image shows a dialog box for skin effect settings. It is divided into two main sections:

- Trace Dimensions:**
 - Width: mil
 - Thick: Oz*
 - * Use decimal fractions ONLY
 - Skin Depth, percent of thickness: %
- Check to consider**
 - Skin Effect? Proximity Effect?
 - Frequency, MHz
 - Skin Depth mil
 - Crossover F MHz

For more information on skin effect, see our article at:
[http://www.ultracal.com/articles/skin effect.pdf](http://www.ultracal.com/articles/skin%20effect.pdf)

Temperature Data Calculations:

First, select the data source you would like to use. The five sources are

- IPC 2152 Air Internal trace
- IPC 2152 Air External trace
- IPC 2152 (external traces in a) Vacuum
- IPC-D-275 Older standard, now obsolete
- Design News 1968

Notes:

1. The new standard, perhaps surprisingly, states that internal traces cool *better* (more efficiently) than do external traces. This is because the thermal conductivity of the board material is better than that of air.
2. There is no difference between External and Vacuum except for very thick traces (where the form factor is more “square”).
3. For more information on how formulas are derived and what the Design News data source actually is, see <http://www.ultracac.com/articles/pcbtemp.pdf>
4. The most conservative data source is IPC 2152 Vacuum. The most aggressive data source typically is IPC-D-275.

Then enter any three of the four variables, Trace Width, Trace Thickness, Trace Current, and the Change in Trace Temperature (above 20 oC ambient) and solve for the fourth. You can use either English or metric units and enter/solve for thickness in units or ounces. In the illustration shown, for internal traces, according to the IPC 2152 standard, the required width for a 2 oz. trace carrying 10 Amps to rise no more than 25 oC above ambient is 135 mils.

If we want to consider the impact of skin effect for a 15 MHz signal, the skin effect would reduce the effective cross-sectional area of the trace. Therefore, the trace would have to be wider to carry the same amount of current. Under this condition, the calculator suggests the new, adjusted width would have to be 255 mils. A message highlights the fact that the skin effect is impacting the calculated result.

UltraCAD's PCB Trace Calculator 3.2

UltraCAD Design Inc. PCB Trace Calculator

Units, General: English Metric

Trace Dimensions: Width mil, Thick Oz*

Check to consider: Skin Effect?

Thickness: Units Ounces

Temperature Data | Fusing Current | Trace Data * | Ohm's Law Calculator

(Use width and thickness boxes above)

Current Amps, Temp. CHANGE oC

Data Source to be used: (dropdown menu)

Message Box: This calculation and skin depth... Temp CHANGE, END

Regional Setting: English (United States) | Look at Regional Settings

Copyright 2010 UltraCAD Design, Inc. Bellevue, WA. USA

Units, General: English Metric

Trace Dimensions: Width mil, Thick Oz*

Check to consider: Skin Effect? Proximity Effect? ?

Thickness: Units Ounces

Temperature Data | Fusing Current | Trace Data * | Ohm's Law Calculator

(Use width and thickness boxes above)

Current Amps, Temp. CHANGE oC

Data Source to be used: (dropdown menu)

Frequency, MHz: Skin Depth mil: Crossover F MHz:

Message Box: Skin effect is impacting results!

Calculation Modifiers:

The Appendix in IPC 2152 includes some very brief guidelines on how to adjust the results from the curves for specific special conditions. The most important of these, by far, is the proximity to an adjacent plane (Section A.4.6). Also important is board thickness (Section A.4.2). The calculator offers a method for incorporating very rough estimates of these effects by checking the “Use Modifiers?” check box.

(Note: The results obtained, while useful, are very approximate and should be used with caution.)

Using the results from the previous page, suppose we checked the “Use Modifiers?” checkbox. The window on the right would open. We calculated, on the previous page, that a trace width of 135 mils of 2 Oz copper carrying 10 Amps would result in a temperature rise of 25 oC.

Suppose now we consider that that trace is 10 mils above a 2 Oz plane, that the plane has an area larger than 25 in², and that the board is 65 mils thick. If we enter this information in the appropriate places in the window, we calculate that the *actual* (modified) temperature rise will be only 8.4 oC (Column A). This shows the impact of the plane (primarily) as well as the other factors.

But suppose we can tolerate a temperature change of 25 oC, and so we want to calculate the reduced width that the modified results will allow. Enter 25 oC in the top box of Column B (and click the “TOTAL Factor” button.) The modified temperature is 74.8 oC. This gives us a modified starting point that will result in a new trace width that, when the resulting calculated temperature change is calculated, results in a modified temperature change back at 25 oC. In this example, entering 74.8 as the Temperature change in the calculator results in a calculated width of only 50 mils. But the 74.8 temperature reduces to only 25 oC after modified to account for planes, etc.

Modifiers to IPC 2152

These modifiers are based on material from IPC 2152, 'Standard for Determining Current Carrying Capacity in Printed Board Design', Appendix A. They are very approximate. Use with caution. Thanks to Jack Olson and his 'Determining Circuit Board Current Carrying Capacity' available at <http://www.frontdoor.biz/PCBportal/HowTo2152.pdf>

Use Column A to modify a calculated value for Temperature Change.
Use Column B to enter a modified value of Temperature Change for calculation.

	Column A	Column B
Initial Temperature Change (from trace dimensions)	25	25 oC
Factor		
<input checked="" type="checkbox"/> Plane thickness > 1 Oz (1.35 mm) <input checked="" type="checkbox"/> Plane area > 25 in ² (160 mm ²)		
Proximity to Plane	Default 10 mil	0.31378 3.18694626
Board Thickness	Default 65 mil	1.065 0.93896713
TOTAL Factor	0.3341757	2.99243781
Modified Temperature Change	8.3543925	74.8109452 oC
Make Modified Width the new Trace Dimension?	Yes	Yes

Close Window

UltraCAD Design Inc. PCB Trace Calcul

Units, General
 English
 Metric

Trace Dimensions
Width 50.064 mil
Thick 2 Oz*
* Use decimal fractions ONLY

Check to consider
 Skin Effect?

Thickness
 Units
 Ounces

Temperature Data | Fusing Current | Trace Data * | Ohm's Law Calc

(Use width and thickness boxes above)

Current 10 Amps Use Modifiers?
Temp. CHANGE 74.8 oC Applicable only for IPC 2152

Modifiers to IPC 2152

These modifiers are based on material from IPC 2152, 'Standard for Determining Current Carrying Capacity in Printed Board Design', Appendix A. They are very approximate. Use with caution. Thanks to Jack Olson and his 'Determining Circuit Board Current Carrying Capacity' available at <http://www.frontdoor.biz/PCBportal/HowTo2152.pdf>

Use Column A to modify a calculated value for Temperature Change.
Use Column B to enter a modified value of Temperature Change for calculation.

	Column A	Column B
Initial Temperature Change (from trace dimensions)	24.996342	74.8 oC
Factor		
<input checked="" type="checkbox"/> Plane thickness > 1 Oz (1.35 mm) <input checked="" type="checkbox"/> Plane area > 25 in ² (160 mm ²)		
Proximity to Plane	Default 10 mil	0.31378 3.18694626
Board Thickness	Default 65 mil	1.065 0.93896713
TOTAL Factor	0.3341757	2.99243781
Modified Temperature Change	24.996342	74.8109452 oC
Make Modified Width the new Trace Dimension?	Yes	Yes

Close Window

Fusing Current Calculations:

Occasionally there are designs where a catastrophic failure can occur that can result in personal or property damage. In such cases it might be necessary for a trace to carry a very large amount of current for a very short time, perhaps until an emergency shutdown procedure can be carried out. So there may be a requirement, for example, for a trace to carry 20 Amps for 0.5 seconds, and then it doesn't matter if the trace fails (fuses) or not.

There is an important distinction between trace fusing current and a normal fuse. A fuse protects against a predictable overload and is replaceable. A fused trace results from a catastrophic event and is not replaceable. A fused trace results in a failed board which must be replaced in its entirety.

There are very few sources to work from in this area. Two individuals, W. H. Preece and I. M. Onderdonk, have published fusing current equations. They are repeated and summarized in an article:

“Fusing Current: *When Traces Melt Without a Trace*”

Available at
<http://www.ultracad.com/articles/fusing.pdf>

This section of the calculator is based on Onderdonk's equations.

Assume for example, we have a 0.5 Oz trace that must carry 20 Amps for 0.5 seconds. The question is, how wide must it be? The calculator says 56 mils. These calculations can/will be modified by frequency if the Skin Effect check box is checked.

UltraCAD's PCB Trace Calculator 3.2

UltraCAD Design Inc. PCB Trace Calculator

Units, General
 English
 Metric

Trace Dimensions
Width 55.722 mil
Thick 1 Oz*

* Use decimal fractions ONLY

Check to consider
 Skin Effect?

Thickness
 Units
 Ounces

Temperature Data | Fusing Current | Trace Data * | Ohm's Law Calculator

(Use width and thickness boxes above)

Fusing Current 20 Amps
Fusing Time 0.5 Seconds (< 10)

Message Box
Width is (0.5² * 188 / thickness) in mils

END

Regional Setting English (United States) Look at Regional Settings

Copyright 2010 UltraCAD Design, Inc. Bellevue, WA. USA

The source equations for this portion of the calculator are completely different from the source data for the temperature change portion of the calculator. Thus, it is not expected that the results will be perfectly compatible, and they are not. Nevertheless, they tend to be in the same order of magnitude, and it can be shown that, in general, they are surprisingly similar given their disparate origins.

Other Features:

The remaining features of the calculator are pretty straightforward.

The **Trace Data** window will calculate the trace resistance (Ohms/in or Ohms/cm) for a trace of the given thickness, width, and temperature (operating and/or ambient — assumed to be 20° C). It will also provide the frequency adjusted values if the Skin Effect checkbox is checked.

Effective with version 3.3 the user can now enter negative operating temperatures (in degrees C). The calculation will be approximately correct until the resistance of the conductor gets to almost zero. At this point the conductor can become superconductive (at approximately -245 degrees C, depending on what assumptions are made.) The calculator is not intended to calculate the point of superconductivity. If the calculated resistance becomes negative the calculator will continue to show zero resistance.

The **Ohm's Law Calculator** provides a basic calculational capability between various trace aspects of resistance/unit length, trace length, current and voltage drop (along the trace.) The values in this section do not carry over from prior sections. They must be entered or pasted manually.

The screenshot shows the 'UltraCAD Design Inc. PCB Trace Calculator' window. The 'Trace Data' tab is active. The 'Units, General' section has 'English' selected. 'Trace Dimensions' shows Width: 254.566 mil, Thick: 2 Oz. 'Thickness' has 'Units' selected. 'Skin Depth, percent of thickness' is 49.867%. 'Check to consider' has 'Skin Effect?' checked. Frequency is 15 MHz, Skin Depth is .6732 mil, and Crossover F is 3.7 MHz. The 'Temperature Data' section shows Ambient Temp. 20 oC and Operating Temp. 45 oC. Calculated values are: R @ Ambient: .00099 Ohm/in, R @ operating: .00108 Ohm/in, f-Adj R @ Ambient: .00199 Ohm/in, and f-Adj R @ Operating: .00218 Ohm/in. A message box contains the text 'CF is 27/thick^2 where thick is in inches.' Regional settings are English (United States).

The screenshot shows the 'UltraCAD Design Inc. PCB Trace Calculator' window with the 'Ohm's Law Calculator' tab active. It features a flow diagram for calculating resistance, current, and voltage drop. Inputs include Resistance Ohm/in (.00218), Trace Length inch (12), Total Resistance Ohms (.0262), Current A (10), and Voltage Drop V. (.262). The message box and regional settings are identical to the previous screenshot.