

# Define a PCB outline

## From IBF-Wiki

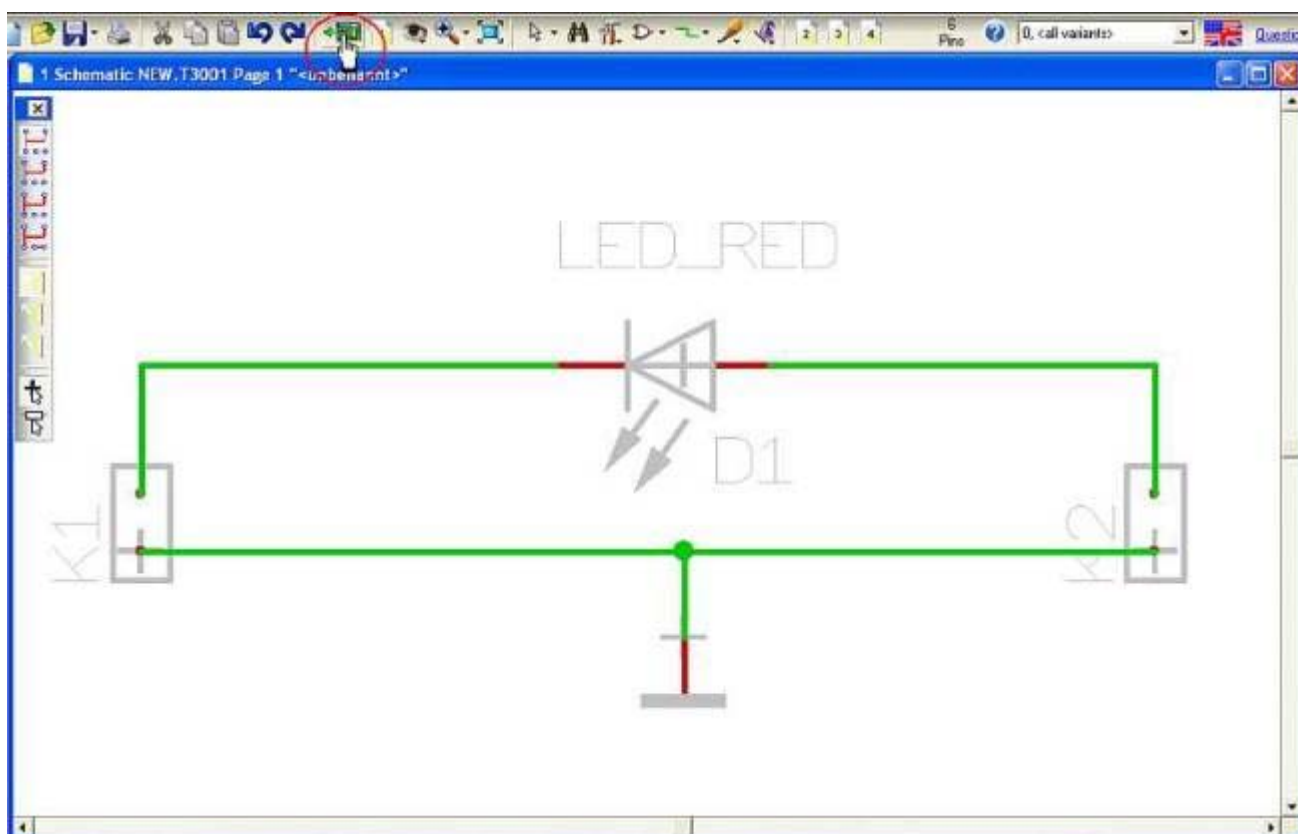
This article is part of an introduction tutorial called CrashCourse2

Download this article as PDF-File

Please switch over to the PCB layout view by the icon "Go to PCB view" (see cursor) or by the key F3.

More  
details?

Outline  
Layer

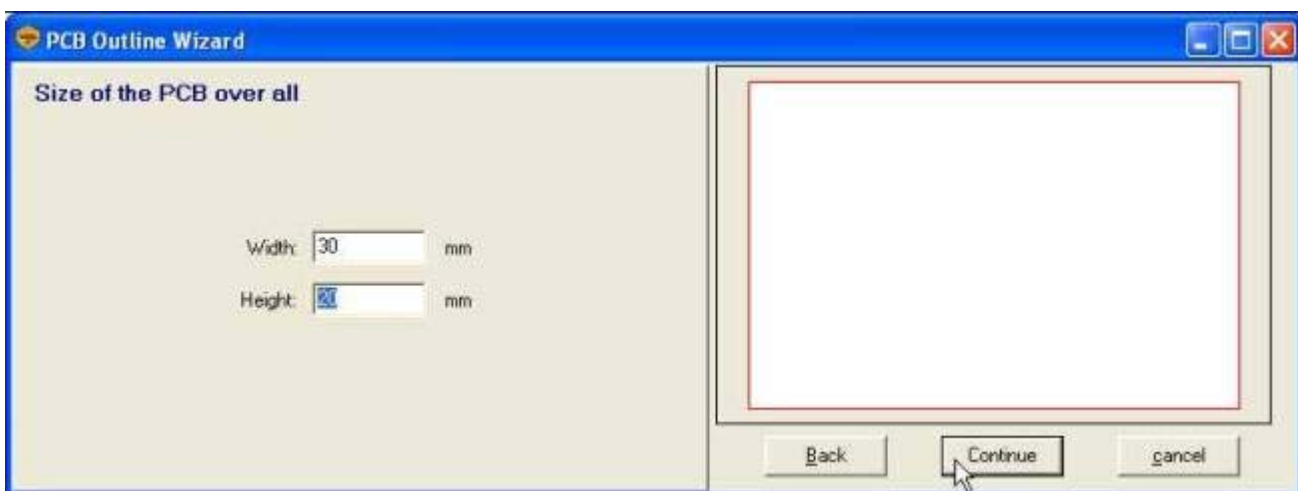


What you get is the Layout view showing a standard PCB outline in light red. It has the measurements of a "eurocard" (W=160mm, H=100mm). Those measurements are much too big for our little project. Any modification of the outline can automatically be done by the PCB outline wizard in menu "Actions/PCB Outline Wizard":

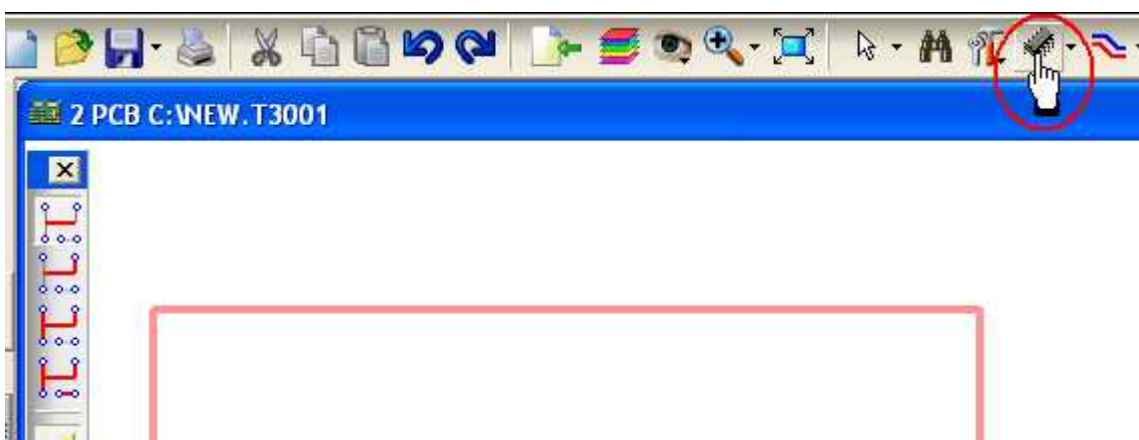




Immediately the assistant opens and we enter Width=30 and Height=20. The fact that we work in metric measurement is predefined by standard. It can be adjusted to imperial measurement by the icon "Adjust View..." (=the eye). The grid is set at the same location.



Because we don't have further breakouts at the edges or within the square, we click on button "Continue" as long as the desired shape appears on the layout screen. Now we can import the component packages (the footprint pattern, or landmarks), see cursor...



More details?

Drawing  
Drag  
Line  
Text



[Next step](#)

[One step back](#)

[Back to overview](#)

Retrieved from "[http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Define\\_a\\_PCB\\_outline](http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Define_a_PCB_outline)"

Categories: [Actions](#)

---

- This page was last modified 09:52, 30 August 2007.