Import a component symbol to the schematic
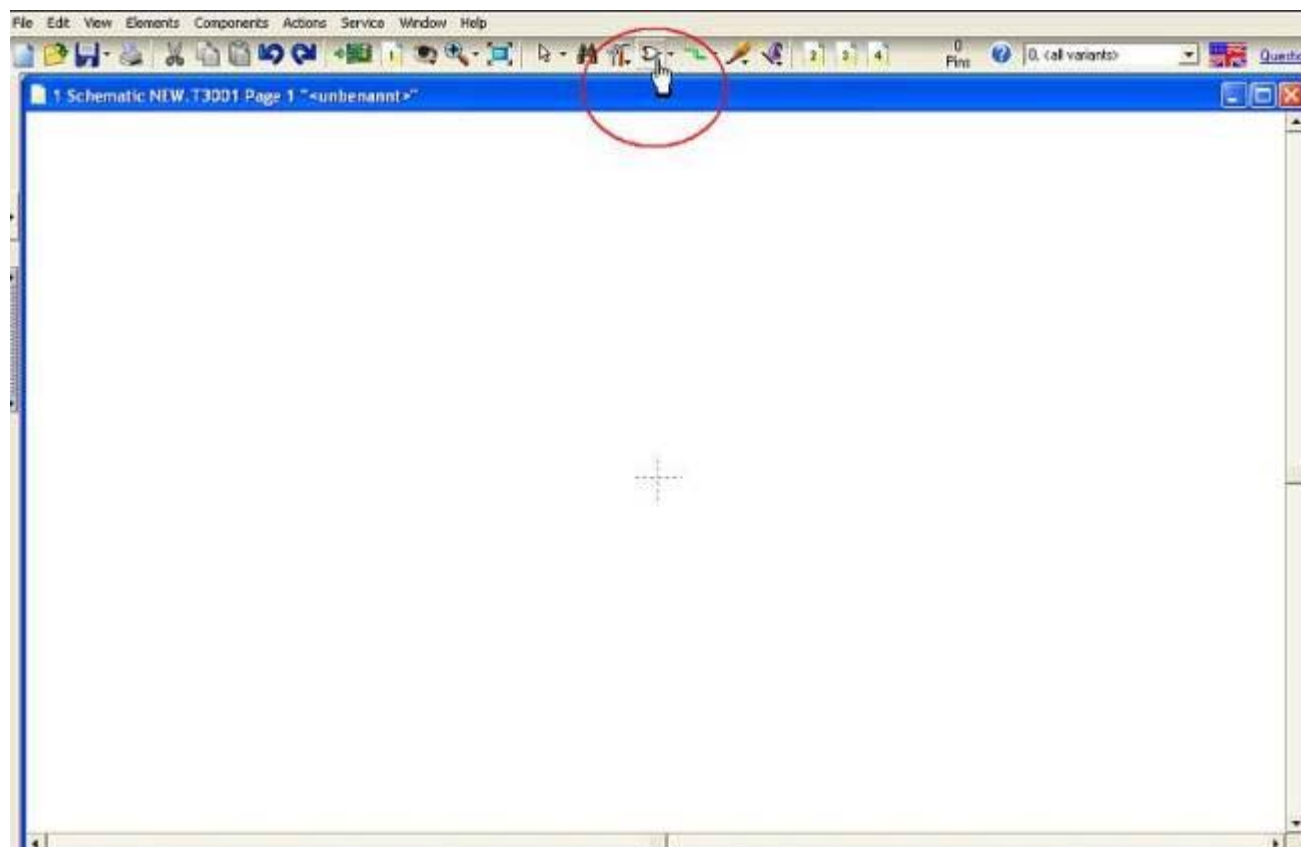
From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

The following image shows an empty schematic page. Now we import a symbol from the component library. Click **M1** upon the symbol button on the icon bar. You might also press button **[Ins]** from the keyboard.
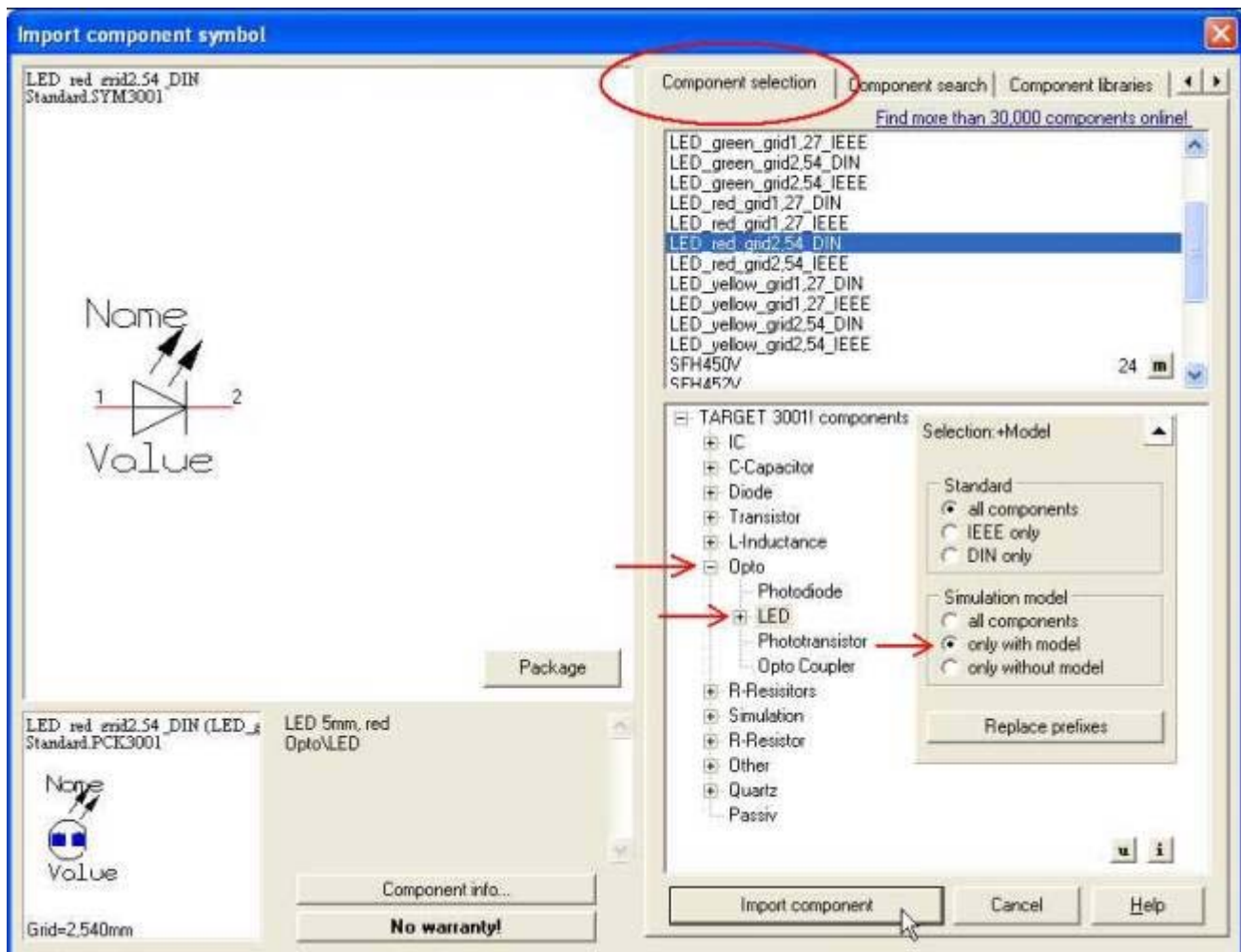
More details?

Origin
Grid
Mousekeys
Key combinations
Zoom

The library browser opens and allows component search and import. In our case we pick a **LED red, 5mm**.
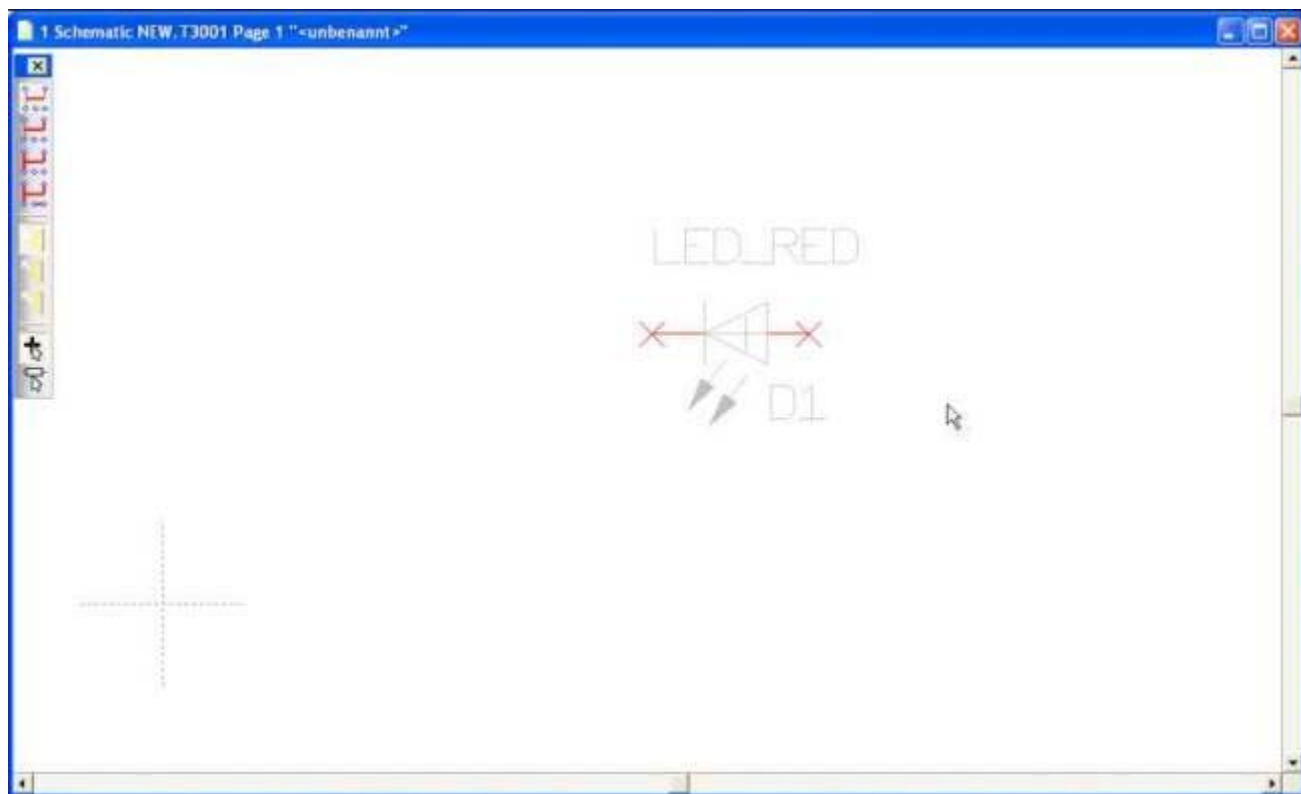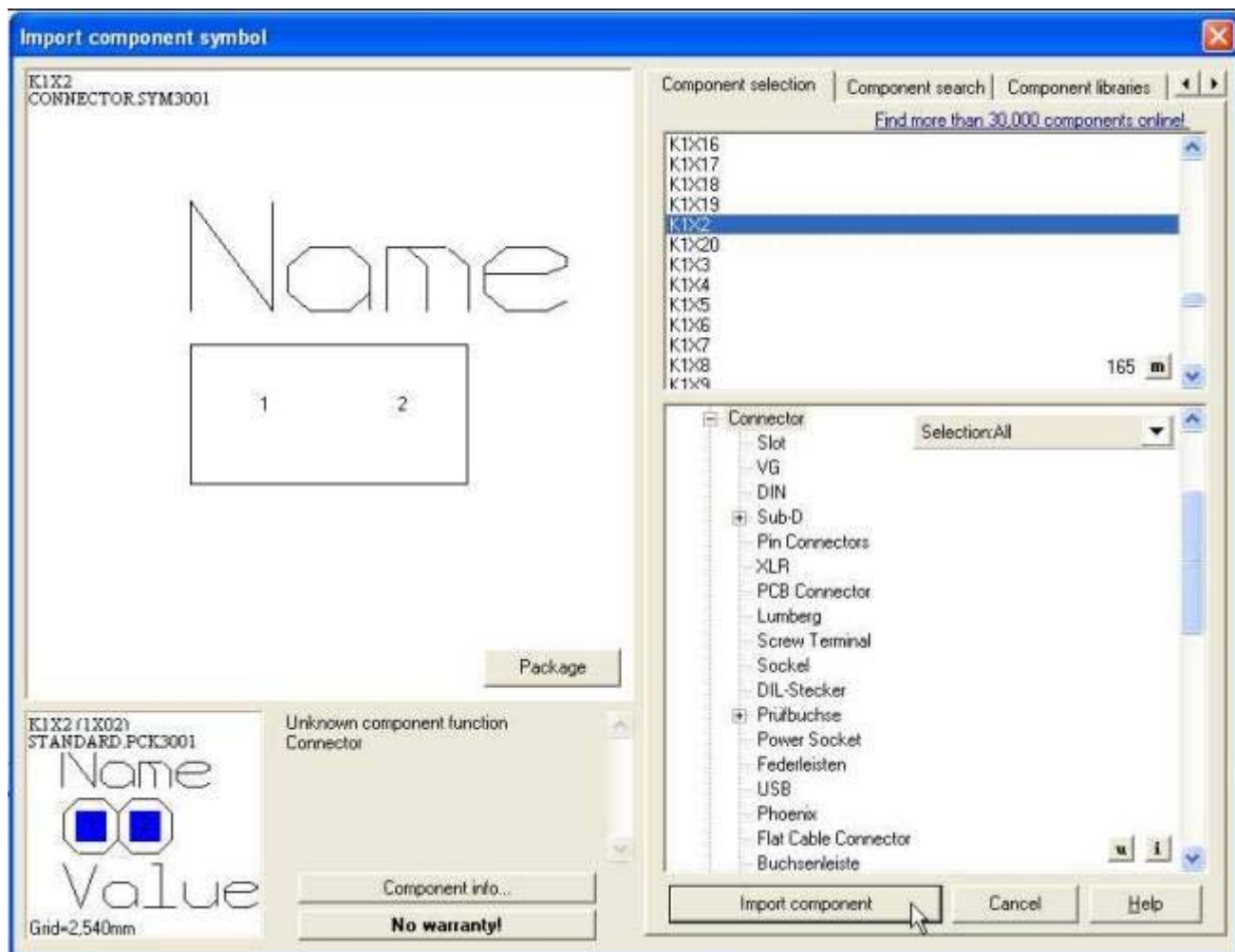
More details?

Libraries

After clicking **"Import component"**, place the symbol on the schematic by M1 preferably within the first quadrant of the underlying coordinate system. In case the component value appears too big, please change it by M11 upon the handle cross. In the appearing dialog modify the entry at "Component Value" to zu LED_RED. "D1" is the "Component Name".

More details?

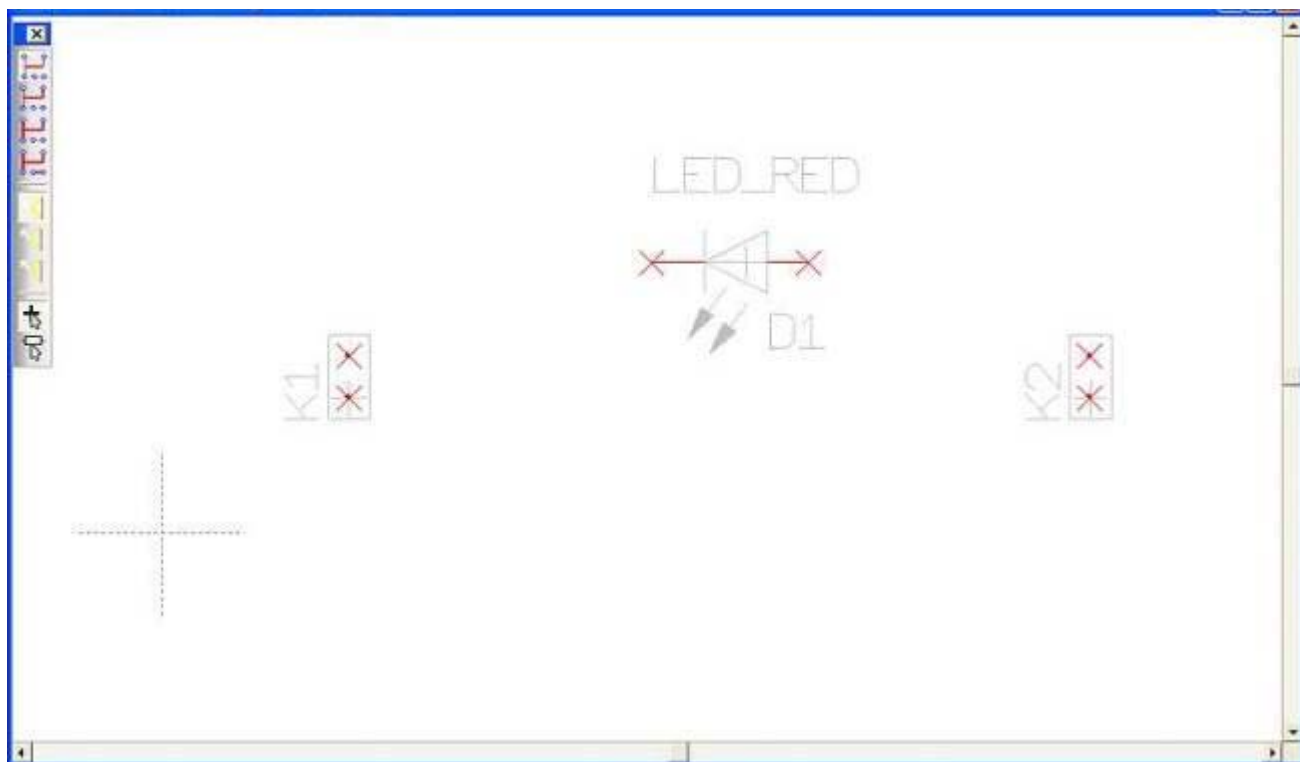Select Displace / Move / Relocate from

Now get in two dual pole connectors to the schematic by the same procedure...

...and place them at a convenient location. Rotate any symbol prior to placement by **M2**. That's the way our schematic might look like by now:

Next step
One step back

Back to overview

Retrieved from
"http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Import_a_component_symbol_to_the_schematic"

Categories: Components/Libraries | Schematic

- This page was last modified 09:40, 30 August 2007.