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.





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

More details?





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...

Import component symbol	Sec. 1997
K1X2 CONNECTOR SYM3001	Component selection Component search Component libraries
Nome 1 2 Peckage	K1X16 K1X17 K1X18 K1X19 K1X2 K1X20
	K1X3 K1X4 K1X5 K1X6 K1X7 K1X8 K1X9 165 M
	Connector Selection:All
KIX2/IX02) STANDARD.PCK3001 Connector	DIL-Stecker Prufbuchse Power Socket Federleisten USB Phoenix Flat Cable Connector Buchsenleiste
Grid-2,540mm Component info No warranty!	Import component Cancel Help

...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.