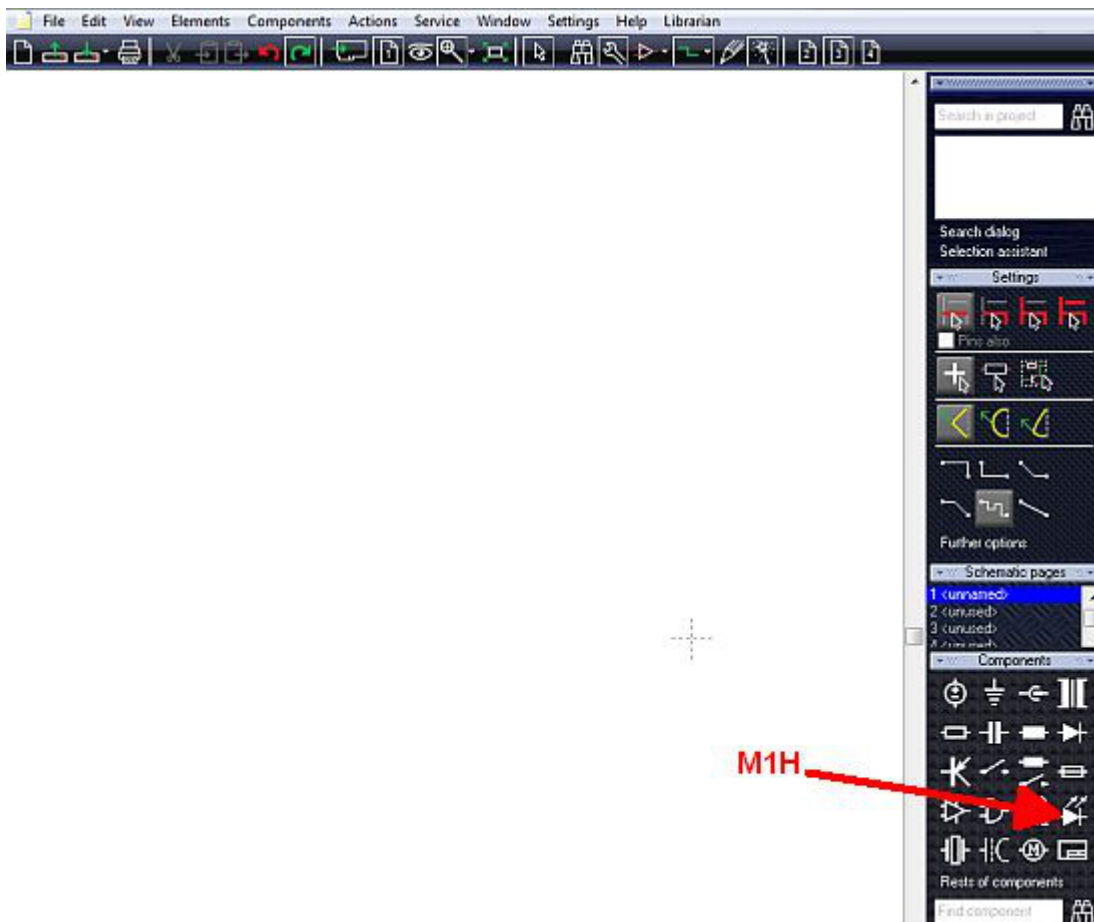


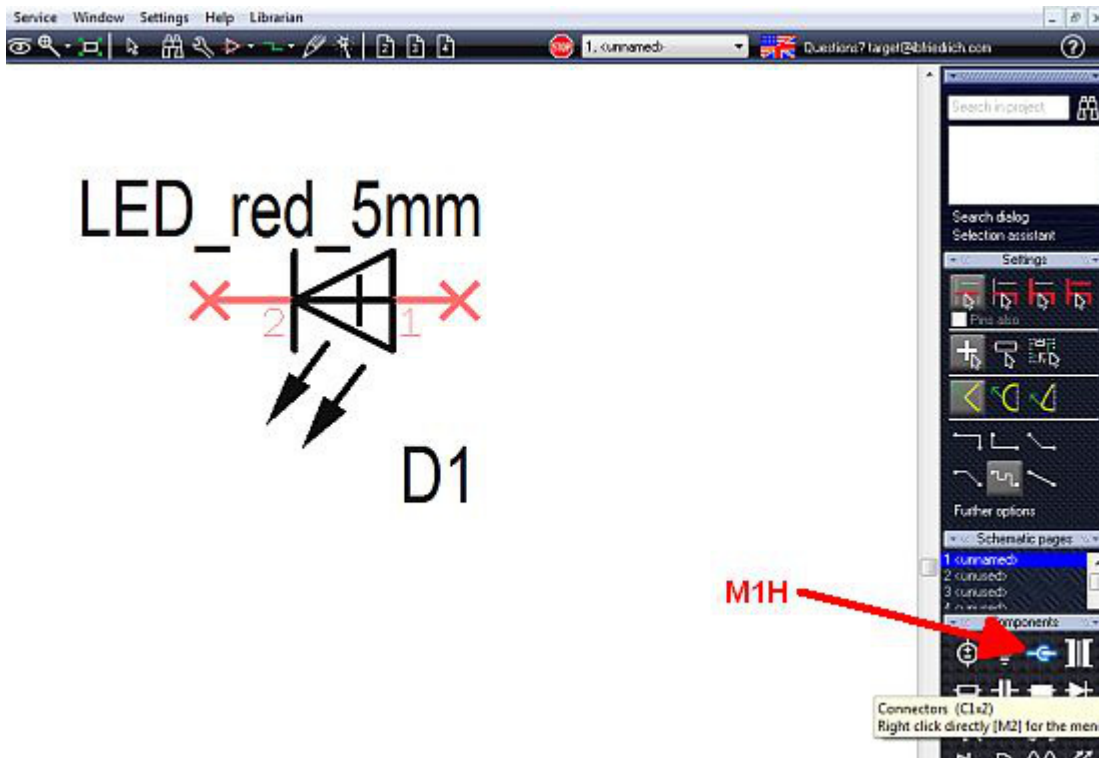
Import a component symbol to the schematic

The following image shows an empty schematic page. Now we import a symbol from the sidebar. Click **M1H** upon the symbol pictogram of a LED and bring it into the schematic by drag and drop. M1H means: "Hold left mousekey"

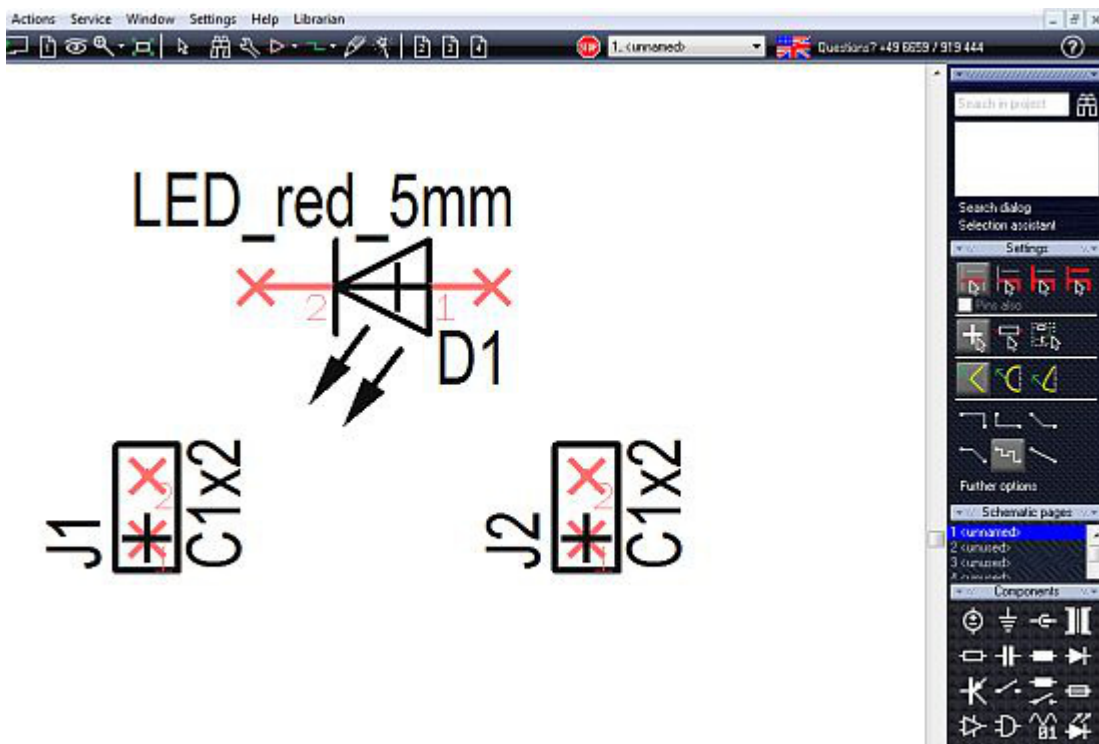


A click **M1** on the pictogram will open the component browser to import further parts of this component family. In case the component value appears too big, you might change it by **M11** upon it's handle cross. The appearing dialog allows to modify the entries. "LED_RED_5mm" stands for Component Value. "D1" stands for "Component Name".

M1 means: "Click left mousekey". M11 means: "Click left mousekey double".



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



Place them at a convenient location. Rotate any symbol prior to dropping by M2.

M2 means: "Click right mousekey,"