How to Install

KiCad EDA
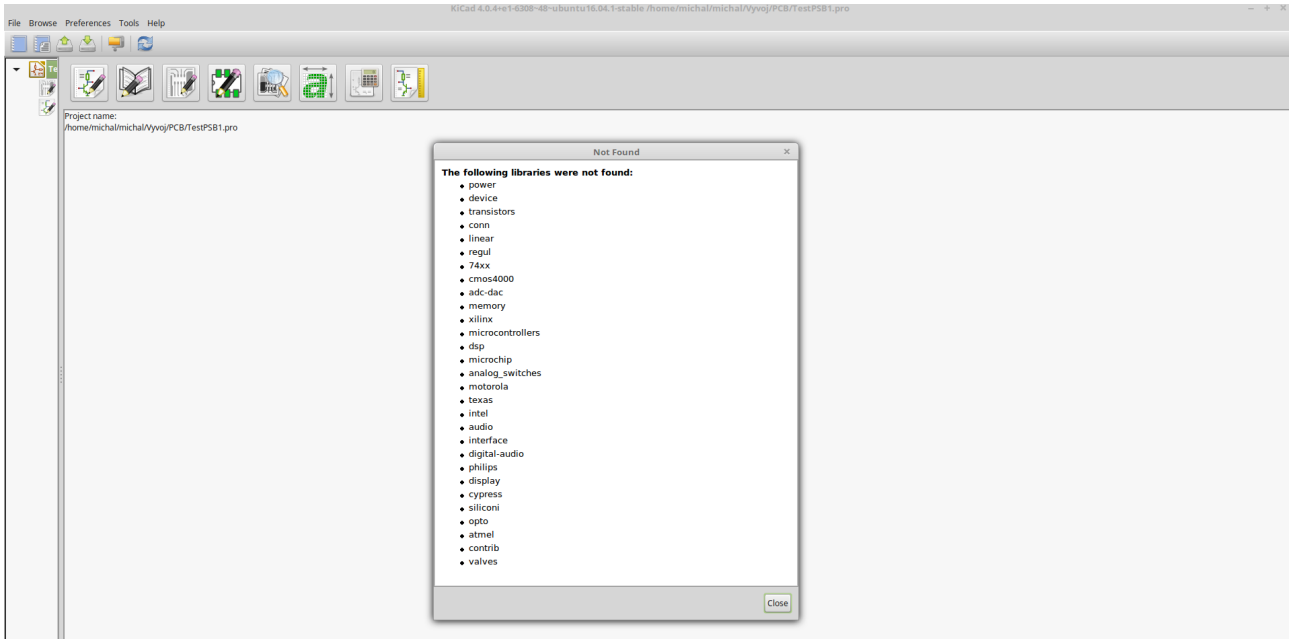**(Schematic and PCB editor)**

with

Libraries

in

Linux Mint

Have you installed KiCad this way?

`sudo add-apt-repository --yes ppa:js-reynaud/kicad-4`

`sudo apt-get update`

`sudo apt-get install kicad`

Do you have the same issue? ***The following libraries were not found…***

This should help: **sudo apt-get install kicad-library**

73! Michal OK1GHZ