

KiCAD

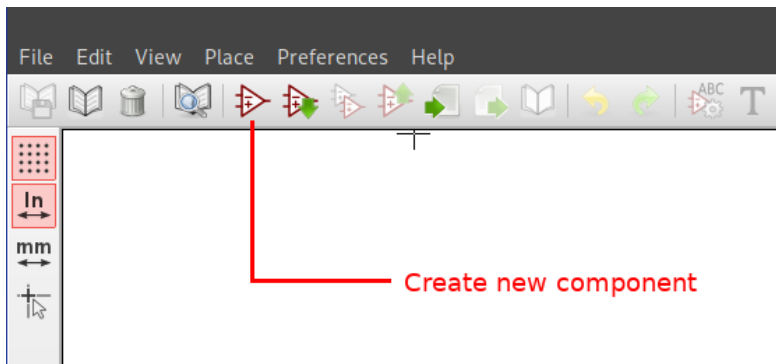
Creating a schematic symbol

Katharina Fey

17. March 2018

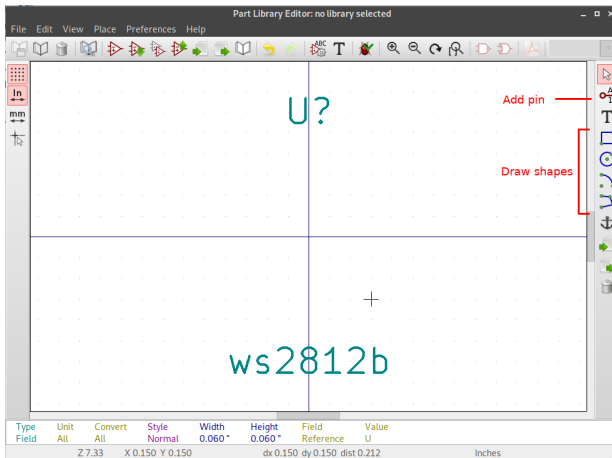
Create new symbol

- ▶ Open the schematic symbol editor
- ▶ Create a new symbol
- ▶ Alternatively, you can of course load an existing library and add a symbol there



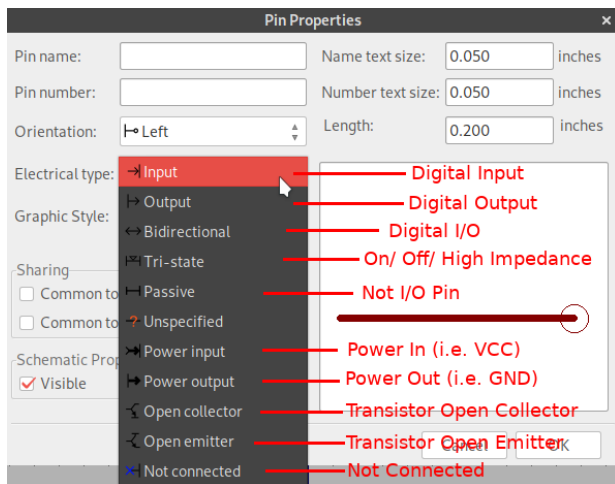
Create new symbol

- ▶ Create pins according to the datasheet
- ▶ Use shapes to make it look nice



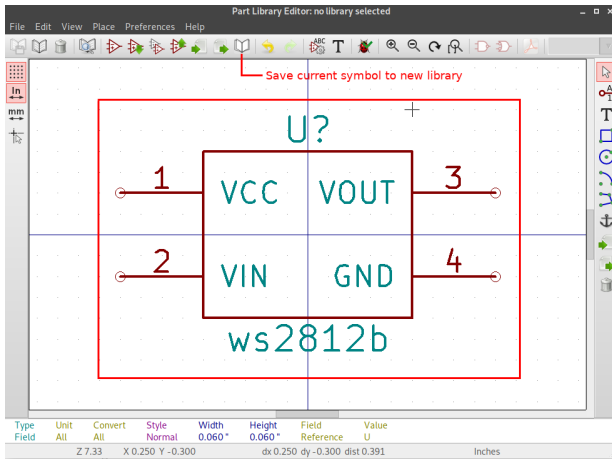
Create new symbol

- Make sure the new pin is set as the correct type



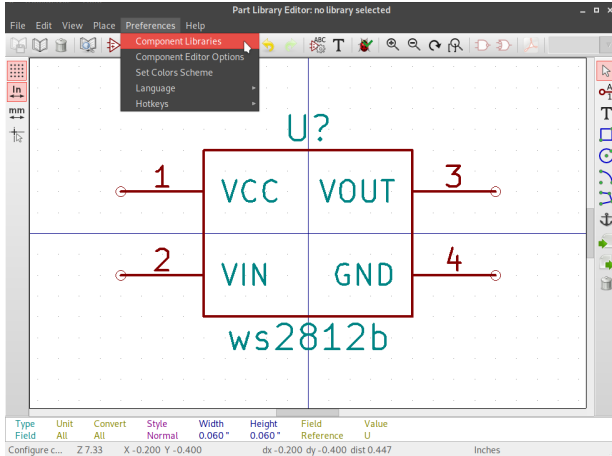
Save as new library

- Because this is the first symbol, save it as a new library



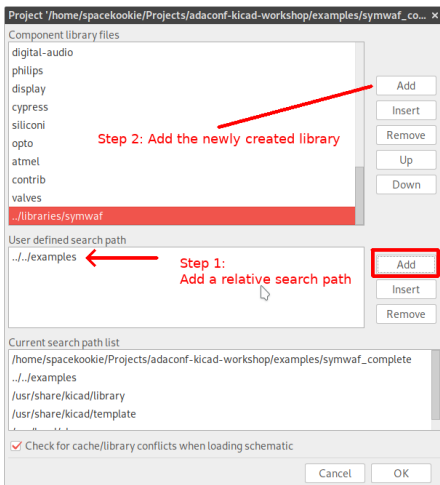
Include new library

- Open library include settings



Include new library

- Add the new library to the include path



Set library as active

- Load the newly created library to work on it further

