Hierarchic Desing with gEDA

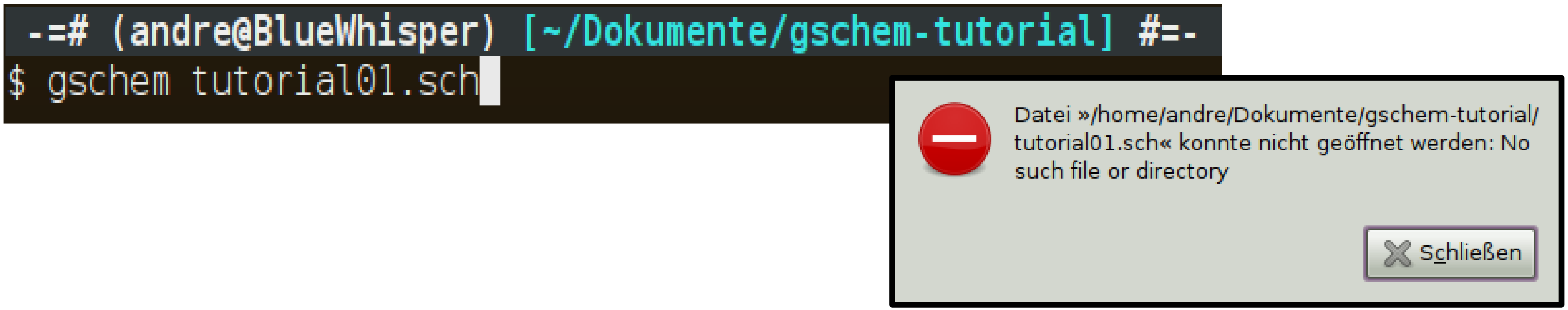
How to make a design with hierarchic levels

Requirements

- functional gEDA installation
  - i.e. for Ubuntu best practise: install all gEDA packages via „Synaptic“

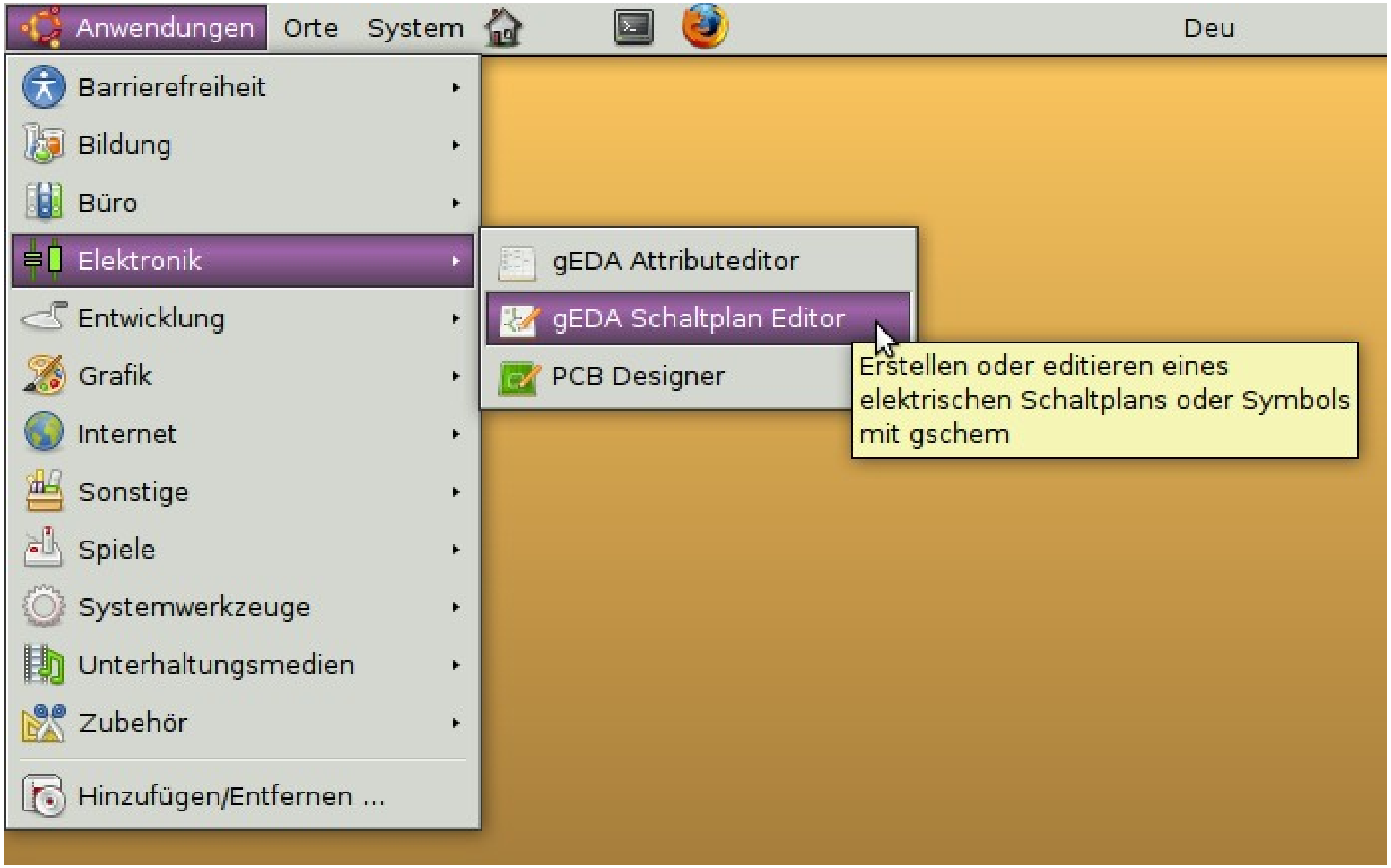
Starting

- Start gschem via console
  - Change to the directory of your project
  - Enter „gschem schematic-name.sch“
  - gschem will open and will present a message, that the file does not exist
    - for a new file ignore that message
    - the file will be saved with that name when saving



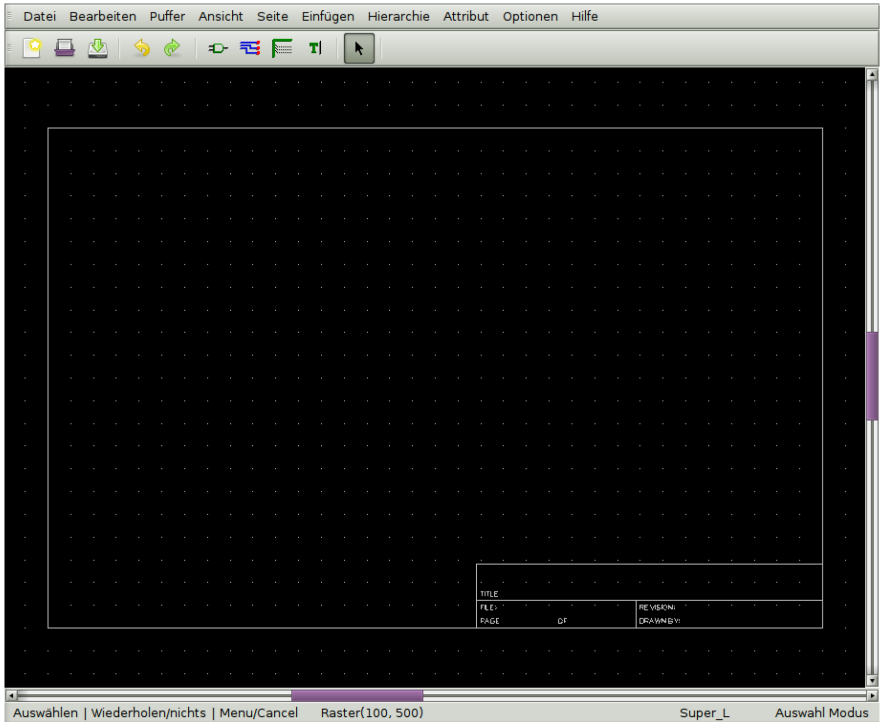
Starting

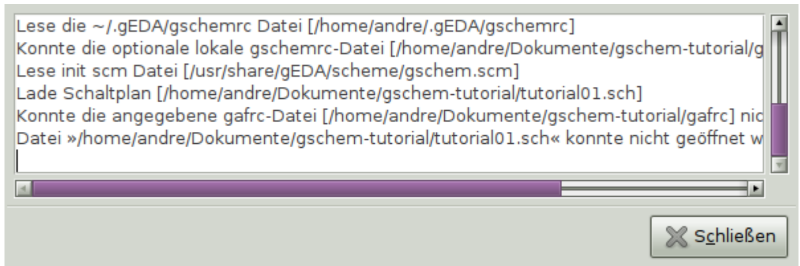
- via the Ubuntu gnome desktop menu



Starting

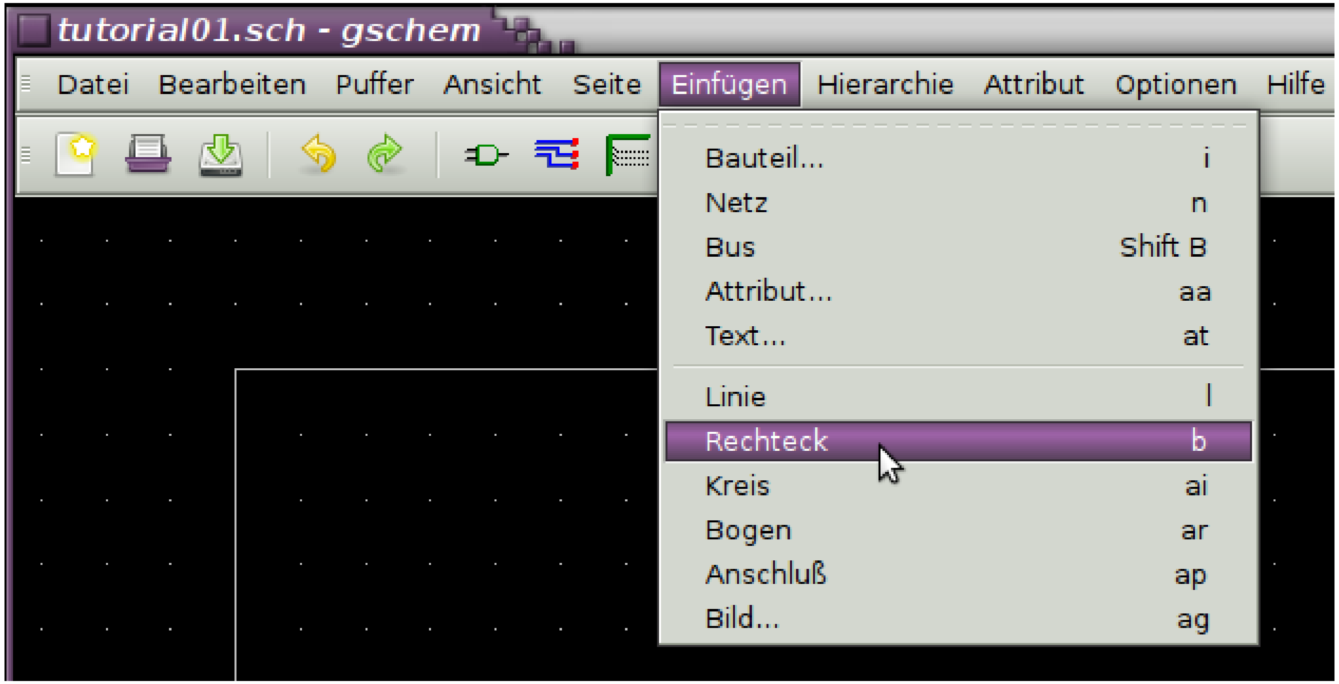
- a schematic window will pop up
- an additional window will pop up, that informs you that some files could not be found
  - ignore that message

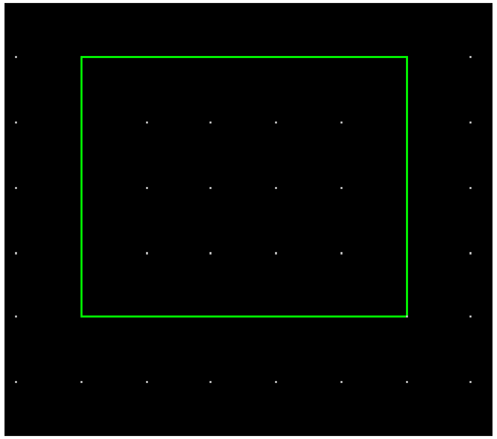




create symbol for hierarchic modul

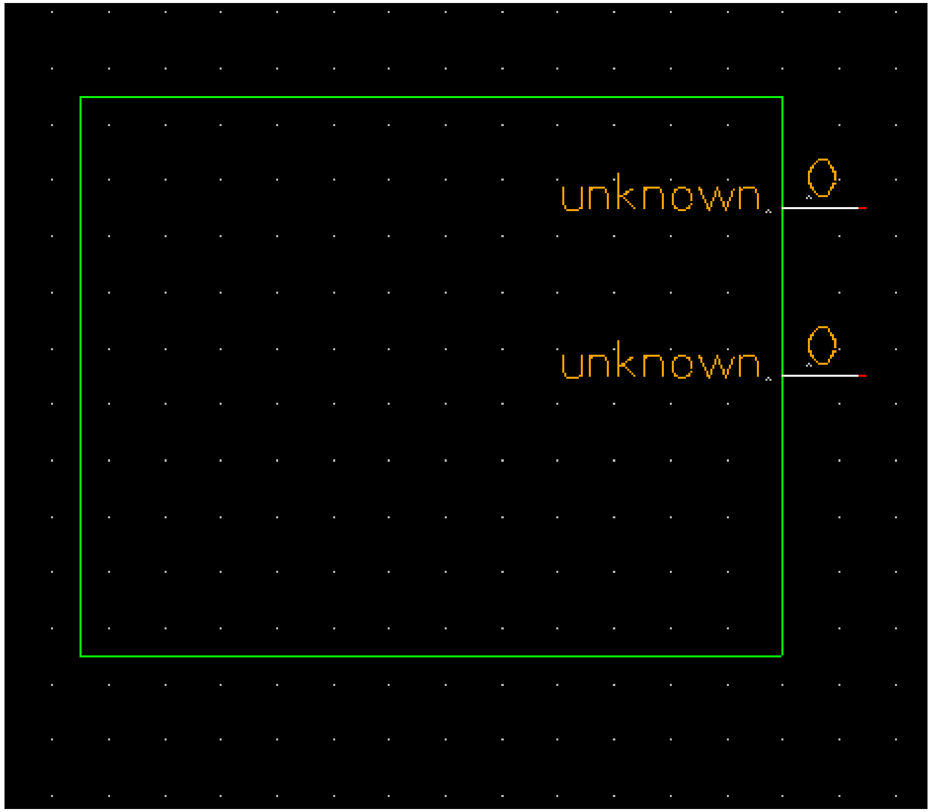
- lets start with a power supply module
- draw a rectancle





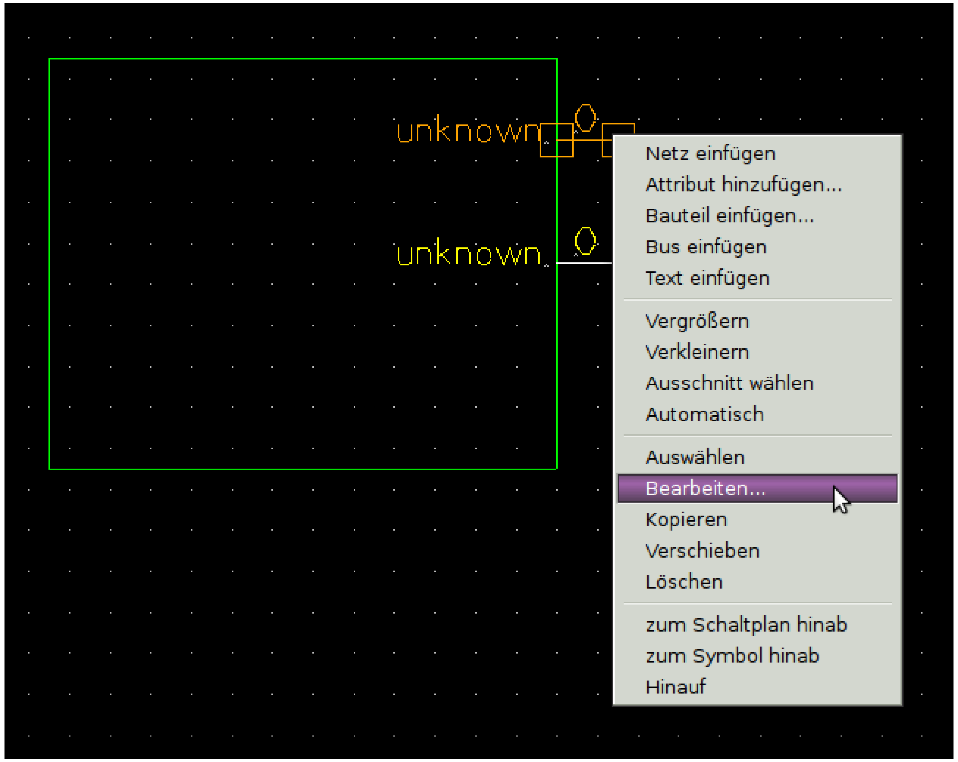
create symbol for hierarchic modul

- add some pins according to the style guide (300 mil long)
  - for this example we will have 3.3 V and 5 V that come from this module



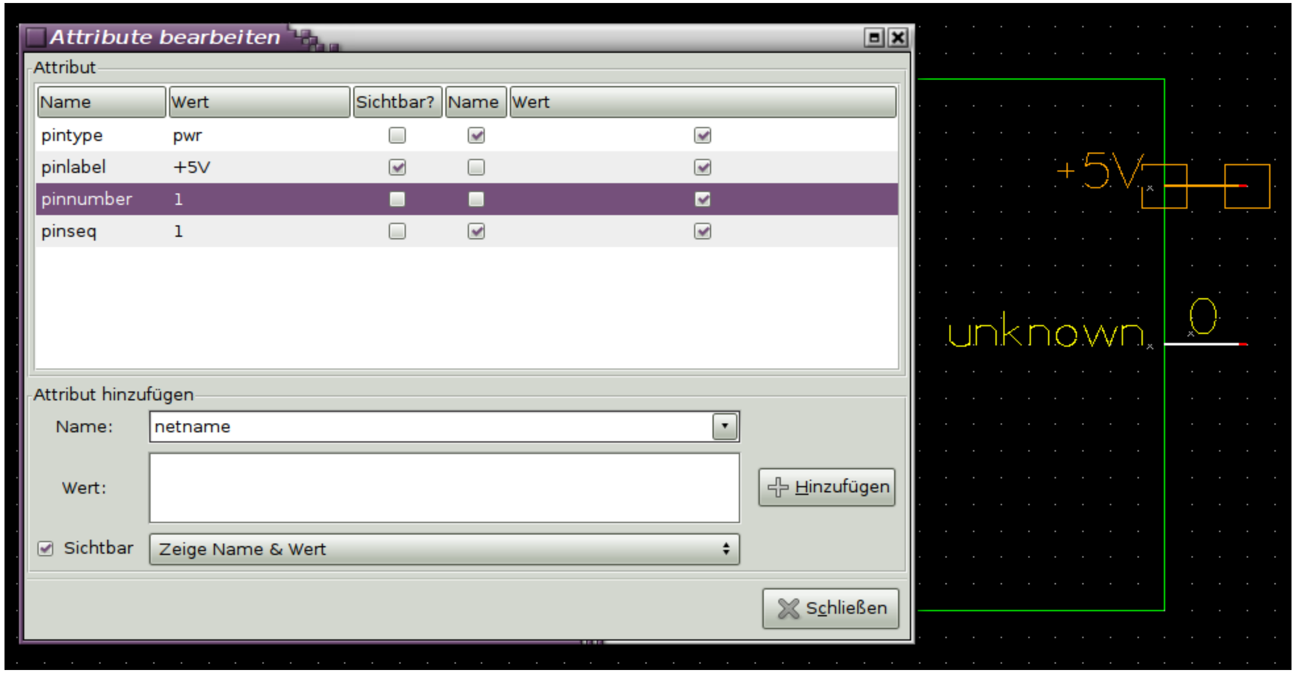
create symbol for hierarchic modul

- change attributes of the pins
  - mark the pin
  - click right mouse button
  - choose „edit...“



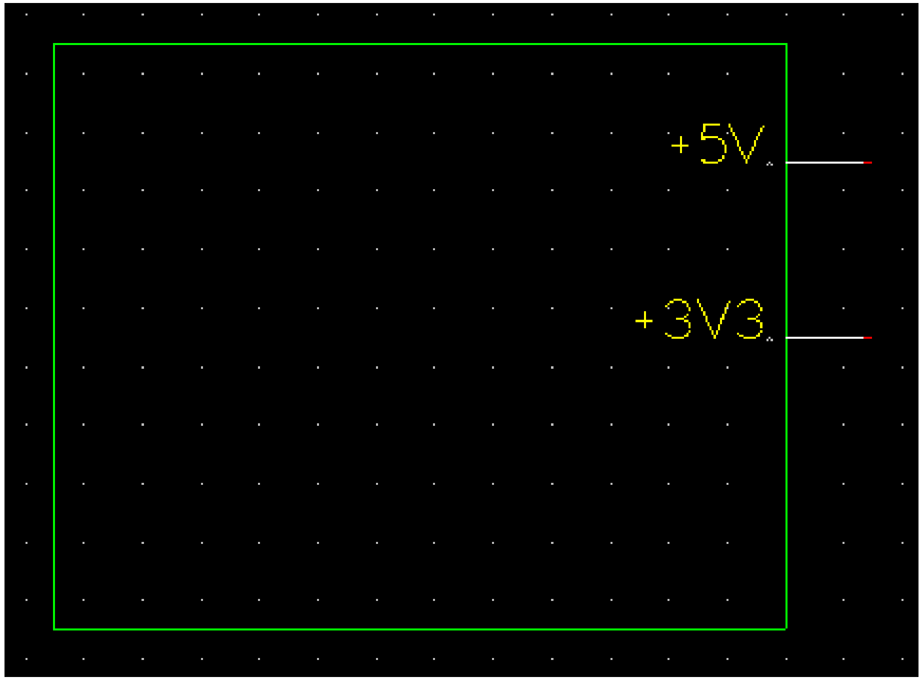
create symbol for hierarchic modul

- change the values for
  - pin number (e.g. „1“; invisible)
  - pin seq (should be the same as pin number)
  - pin label (e.g. „+5V“)
  - pin type (e.g. „pwr“)



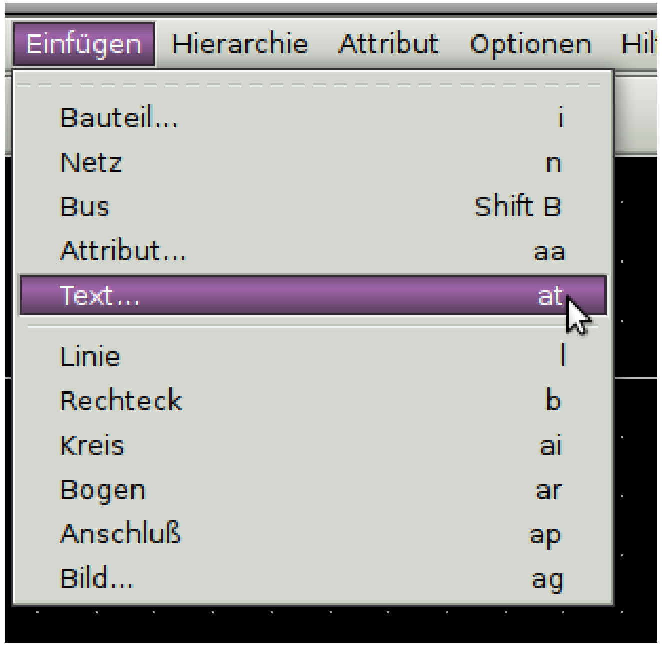
create symbol for hierarchic modul

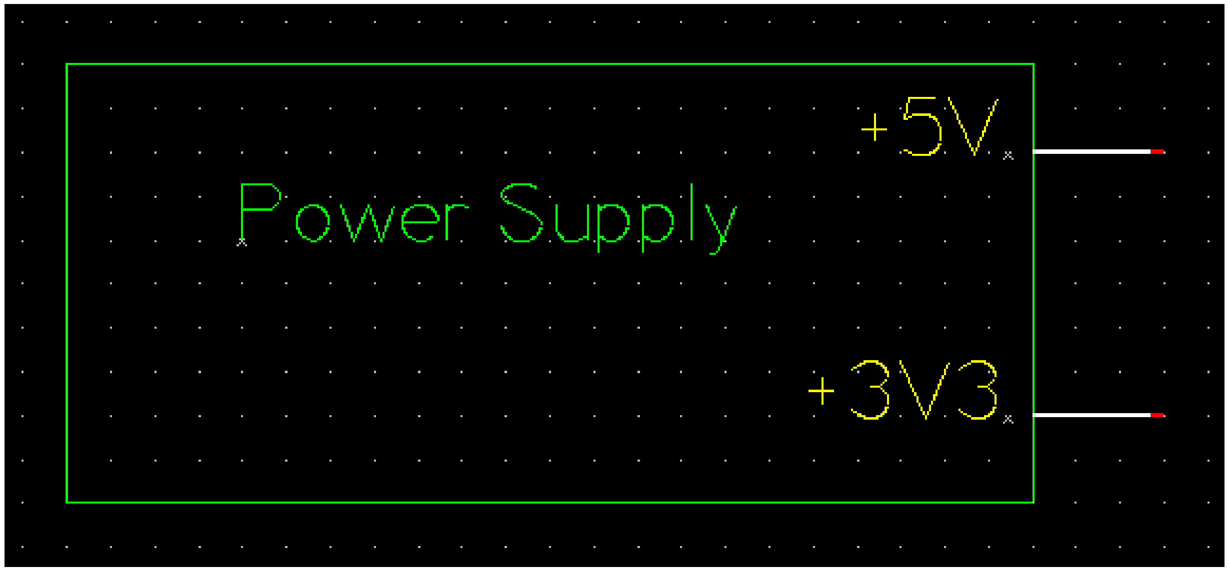
- apply the same steps to the second pin



create symbol for hierarchic modul

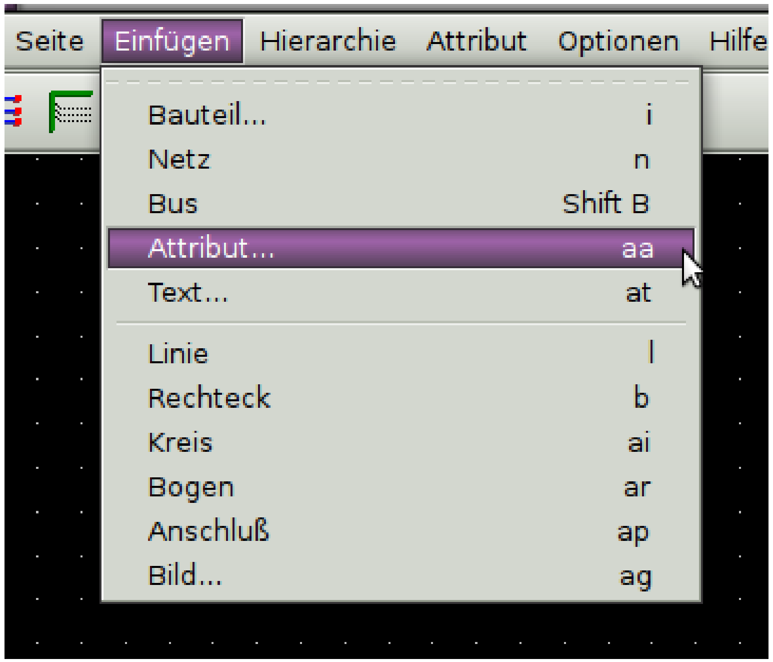
- now we add a text for the symbol
  - enter the text and place it in the schematic
  - modify the rectangle so that the shape is not too big

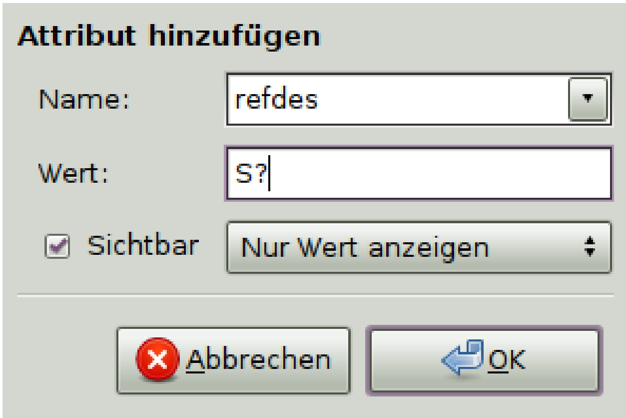


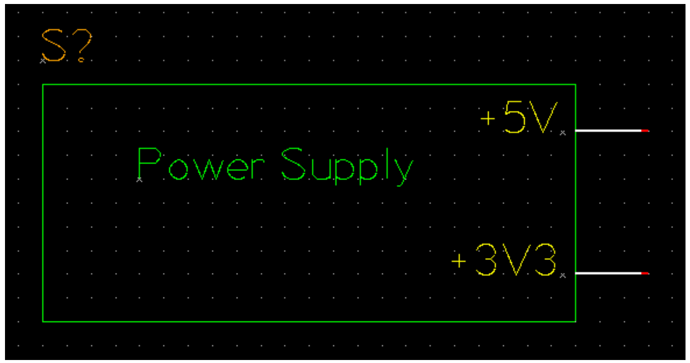


create symbol for hierarchic modul

- now we need to add a reference designator to the schematic
- check that nothing is marked
- go to menu „insert“ → „attribute“
- in the window enter „refdes“ and „S?“
- the attribute will be placed in the leftmost lower corner of the schematic → place it on the top left of the symbol

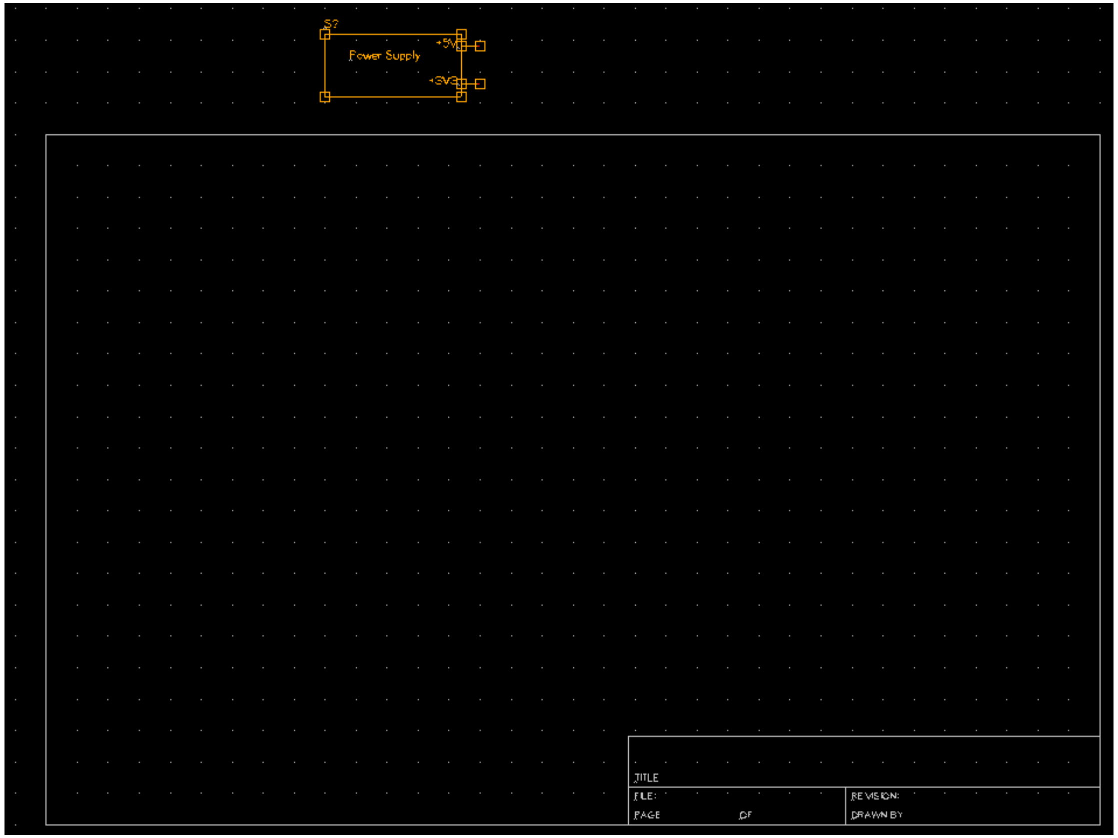






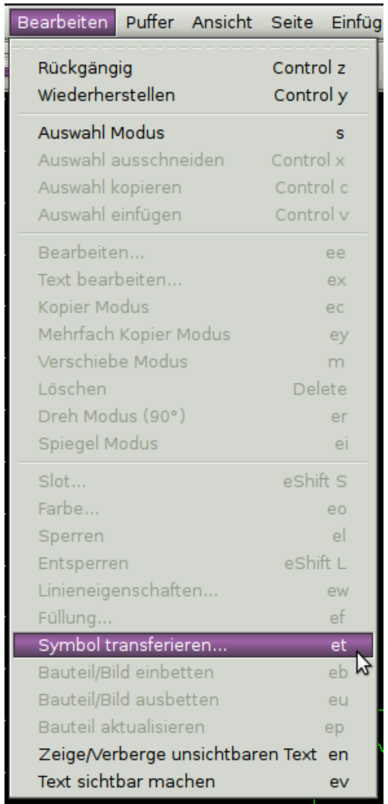
create symbol for hierarchic modul

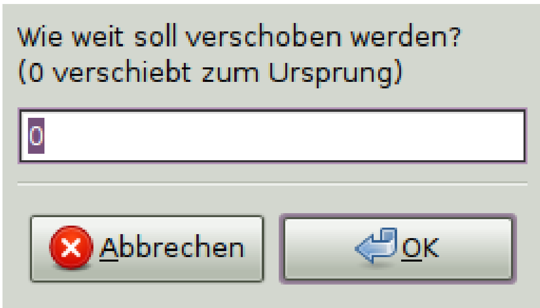
- let us finish the symbol
  - mark all elements in the drawing and put them outside the frame that gschem puts automatically in every new drawing
  - mark the frame and remove it

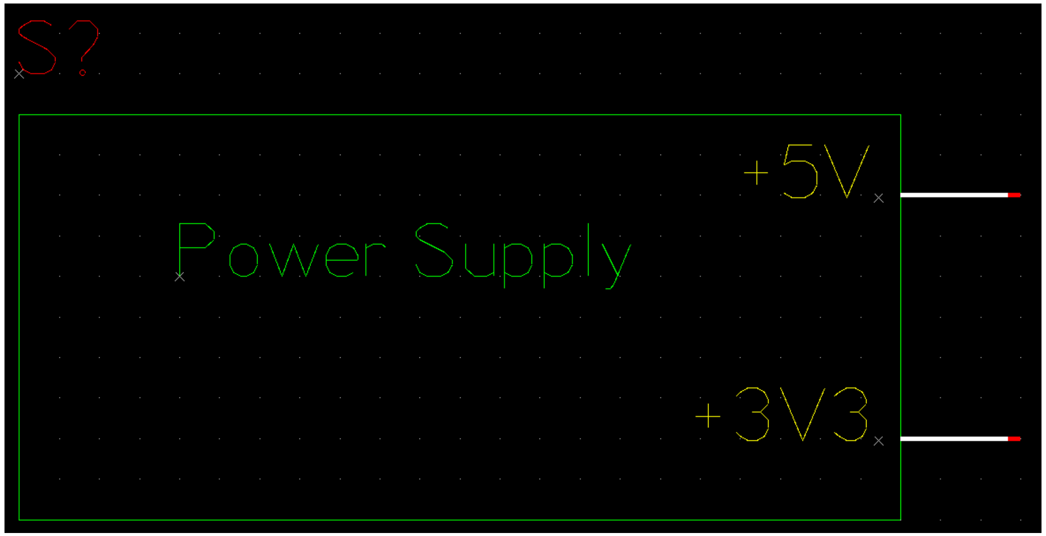


create symbol for hierarchic modul

- let us finish the symbol
  - we need to put the symbol to the origin of the drawing area
  - this is done by selecting „transfer symbol“ from the „modify“ menu
  - save the symbol in your project directory as „powersupply.sym“







create gafrc file

- this file contains at least some paths for the design
- this step is necessary so that the created symbol can be used in the design
- on the console go to the directory of the project and enter the following code
-
-

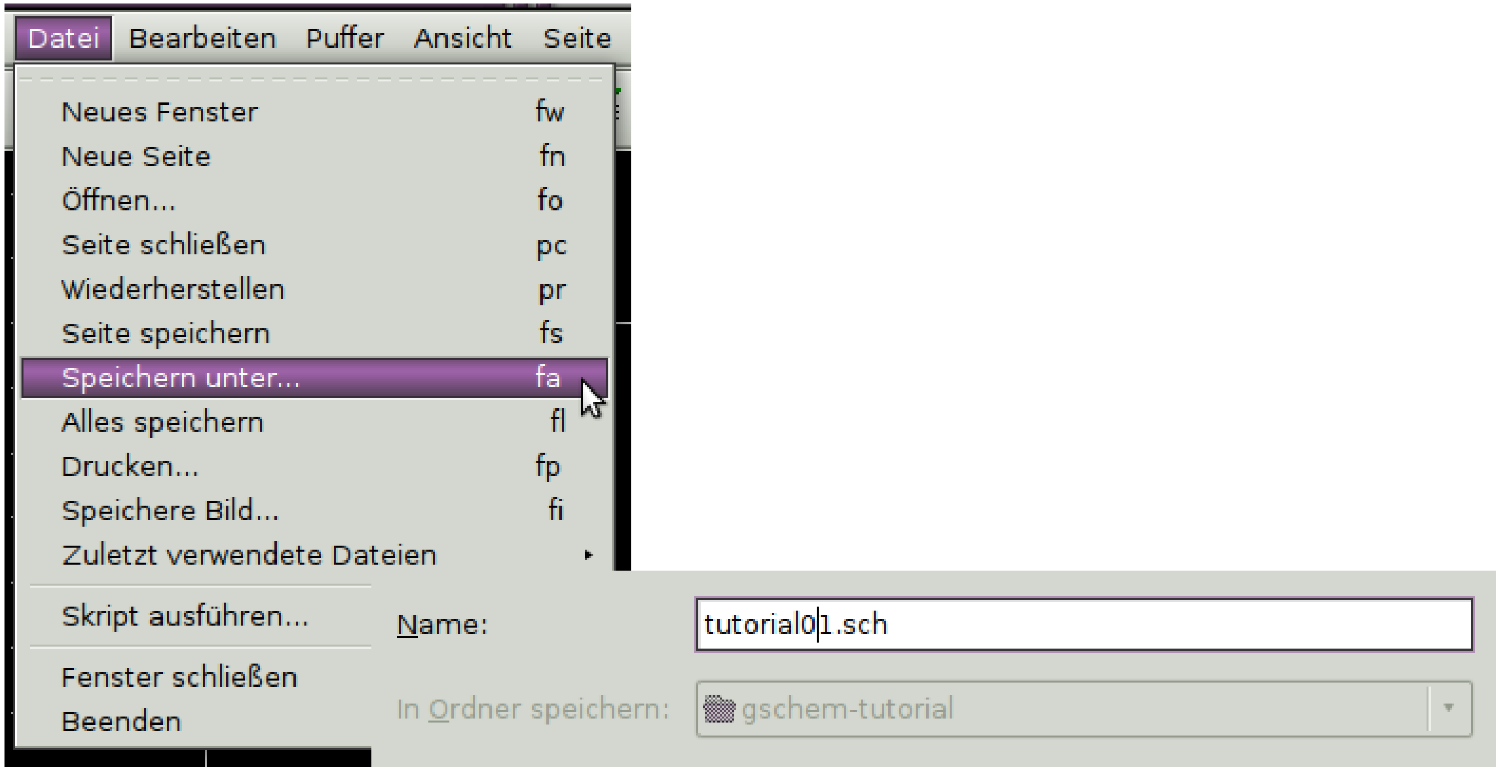
```
echo "(component-library \".\")" > gafrc
echo "(source-library \".\")" >> gafrc
```

- this will create the file „gafrc“ that contains the necessary entries

```
(component-library ".")
(source-library ".")
```

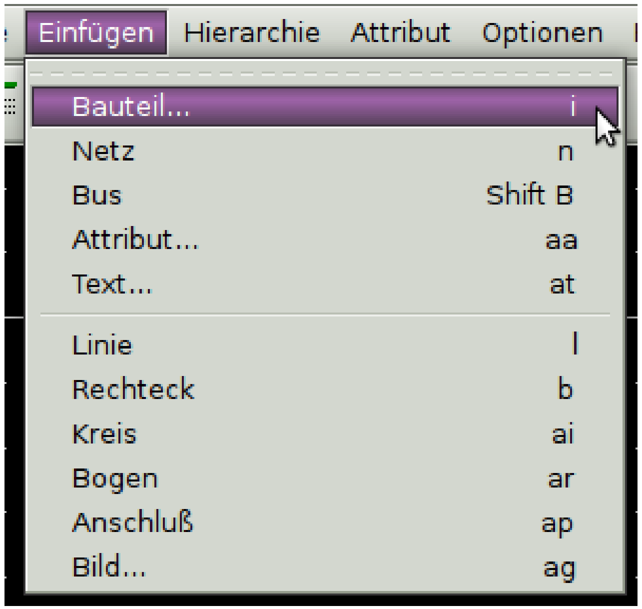
create sheet for hierarchic design

- create a new drawing in gschem
  - select „file“ → „new sheet“ or „new window“
- save it in the project directory as „tutorial01.sch“
- after saving the file, open it again via „file“ → „open“
  - this will cause, that the gafrc file in this directory is read too

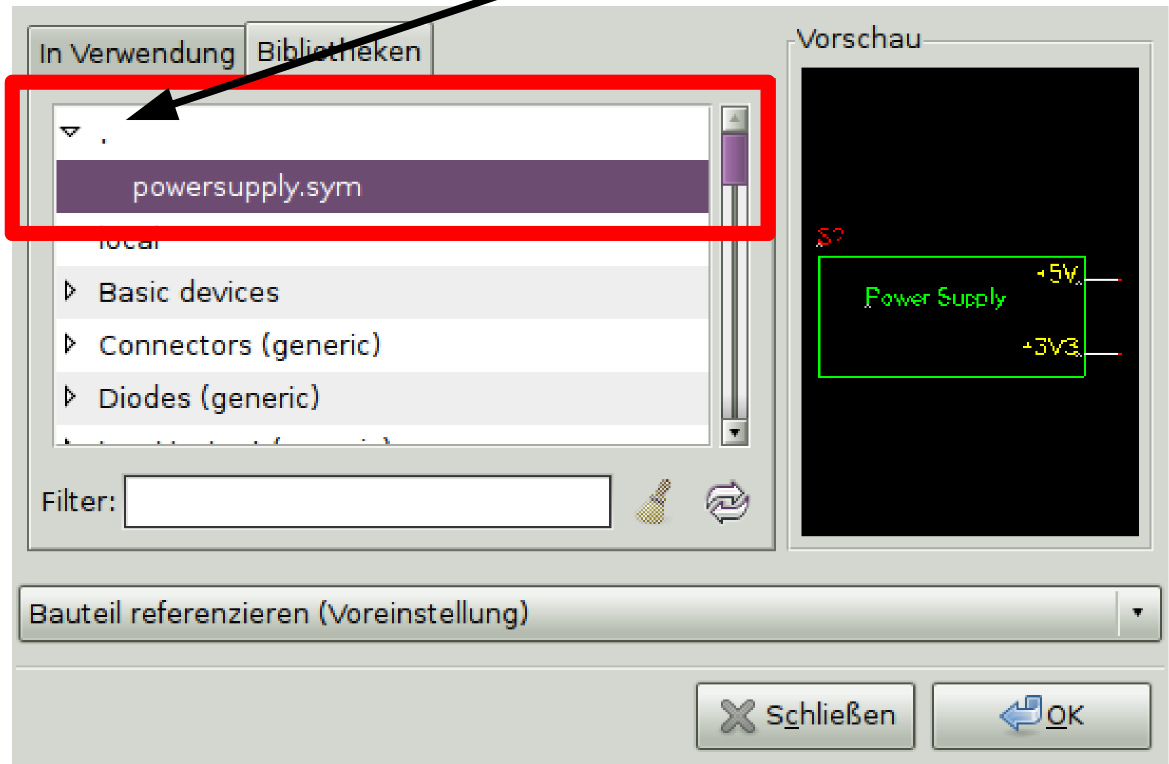


create sheet for hierarchic design

- add symbol for hierarchic block (i.e. „powersupply.sym“) from „insert“ → „component“ in the schematic

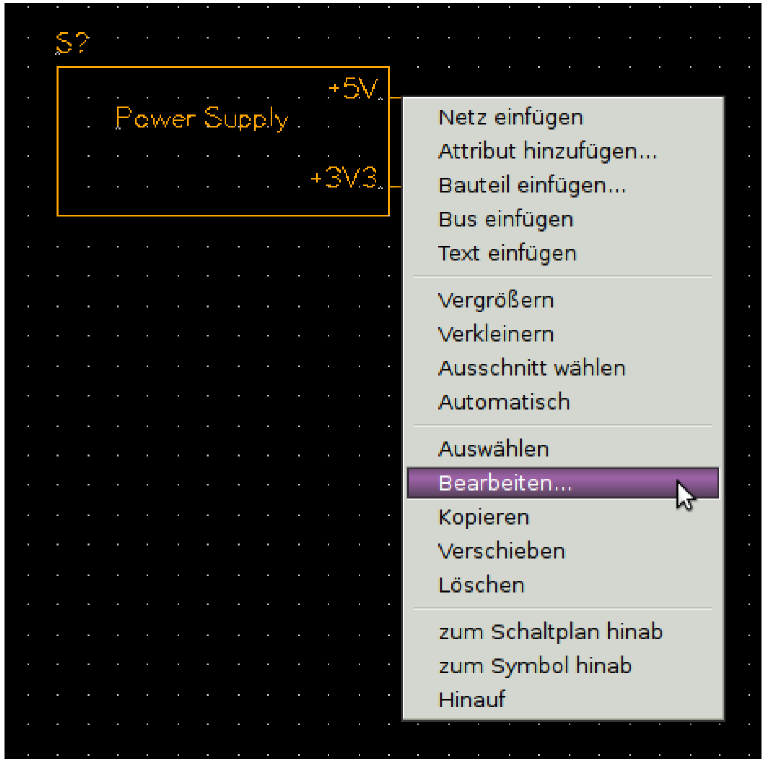


current directory



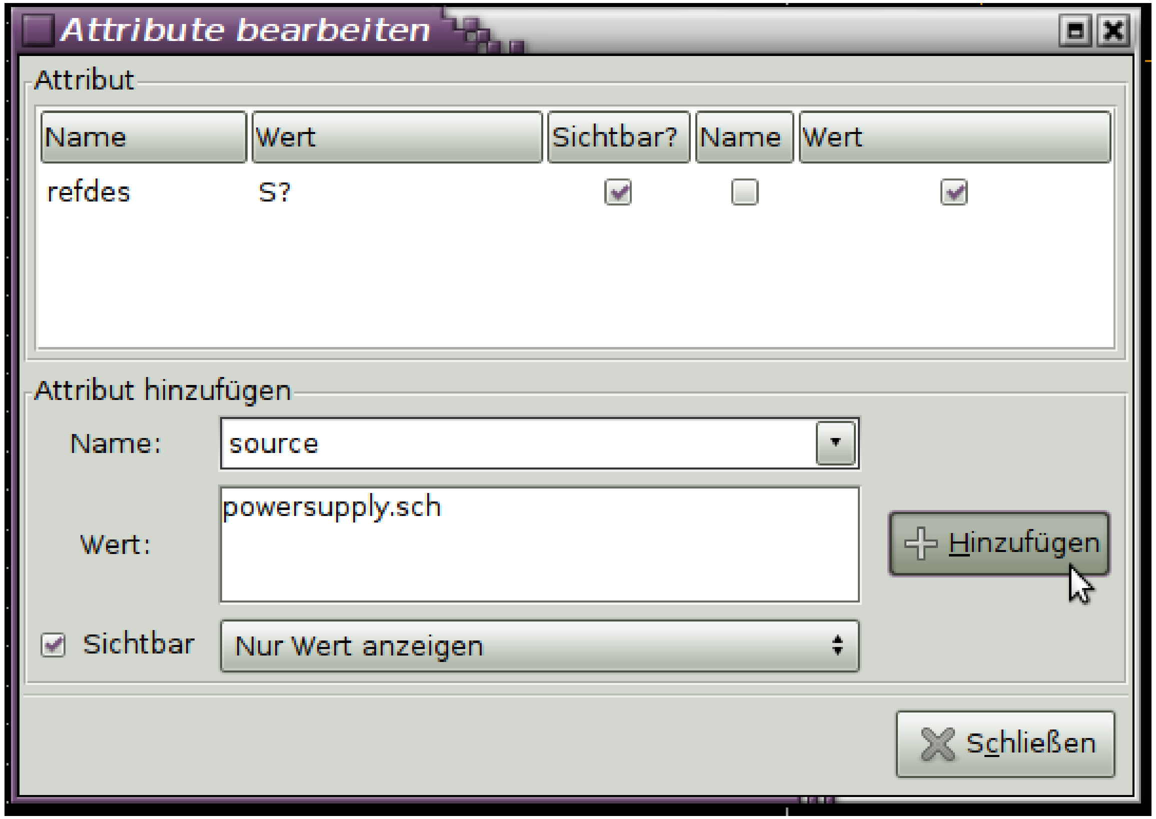
create sheet for hierarchic design

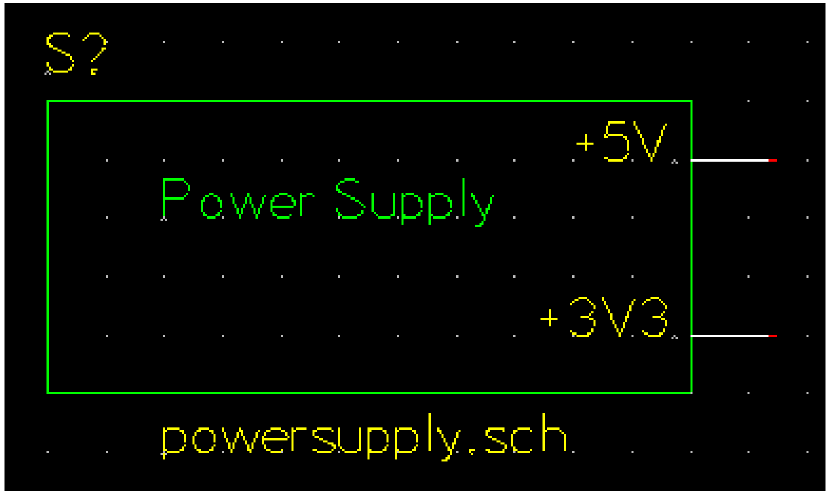
- now the symbol must be connected to a schematic that contains the actual circuit of the power supply
- mark the symbol and choose from the „right click menu“ the entry „edit...“



create sheet for hierarchic design

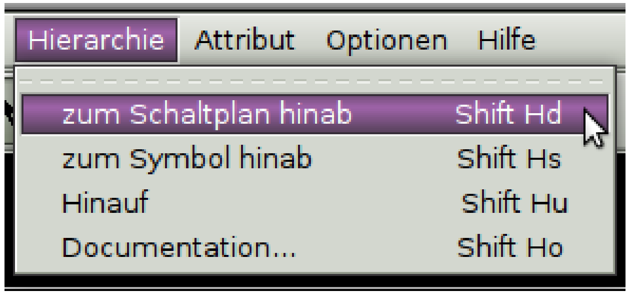
- add attribute „source“ with the value „powersupply.sch“ and place the attribute below the symbol in the schematic





create sheet for hierarchic circuit

- create a new scheet and save it as „powersupply.sch“
- now change to the „tutorial01.sch“
- mark the symbol for the power supply and choose from the menu „hierarchy“ → „down to schematic“
- „powersupply.sch“ should now open



Congratulations!

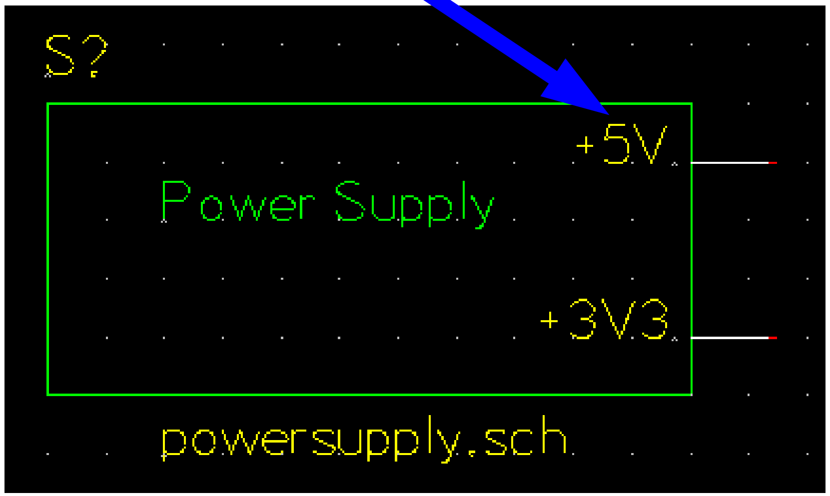
- you now know
  - how to create a symbol for hierachic designs
  - how to place that symbol in the design
  - how to connect that symbol to a schematic

- to connect the actual circuit to the signals on the symbol you need to use the same signal names from the symbol in the schematic
  - insert a „output“ from the library
  - change the refdes of the output to the name of the signal

Finishing

- to connect the actual circuit to the signals on the symbol you need to use the same signal names from the symbol in the schematic
  - insert a „output“ from the library
  - change the refdes of the output to the name of the signal

Name on symbol pin



Name on schematic output pin