Uedem, the 31th. of August 2009

How to create and modify symbols for EESchema / KiCad. Rev.C

by Bernd Wiebus.

Preliminary and uncomplete!
No responsibility ist taken for the correctnes of this informations!
Use careful!
No warranty!

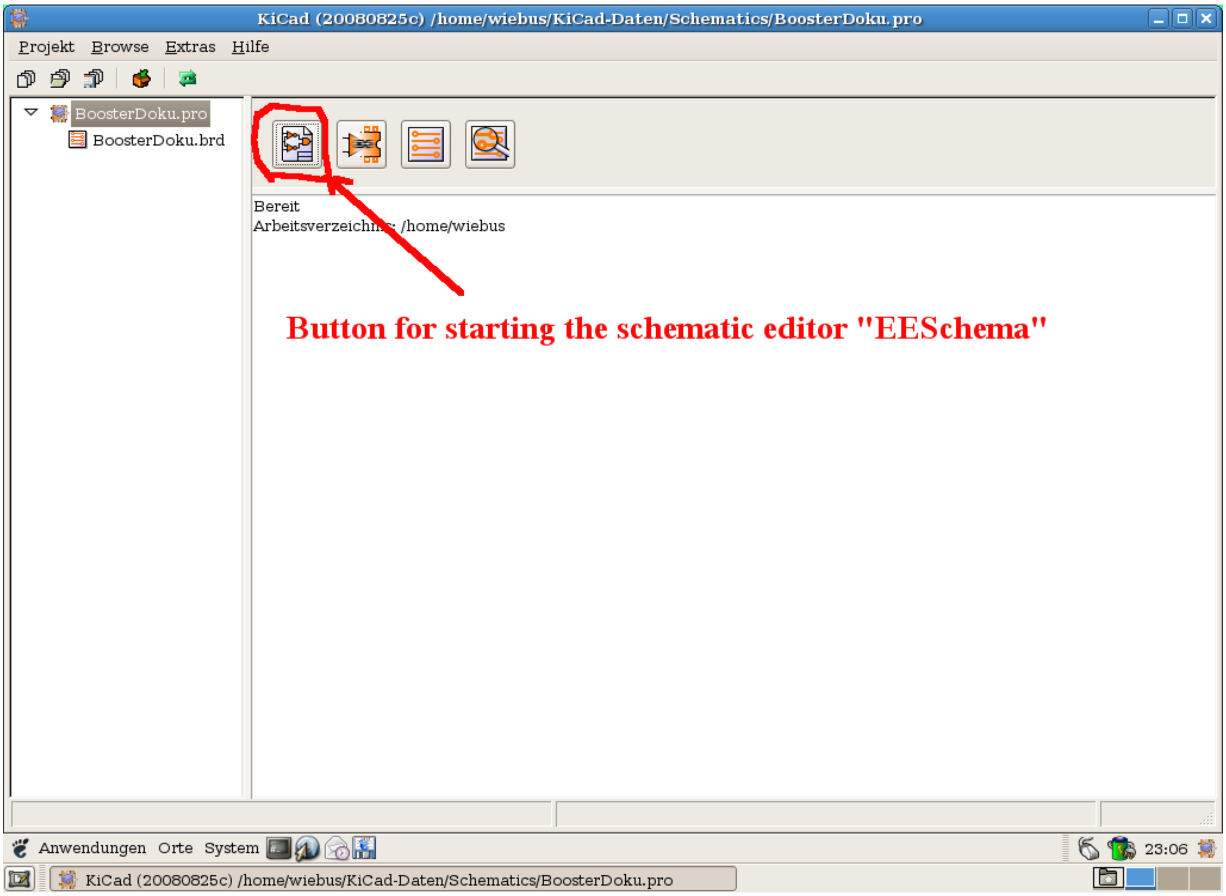
I had trouble to get the right settings of KiCad to get an English version , because i first installed it as German version in a German enviroment. So some of my screenshots are a mix of Englisch and German.
I hope to fix this in future versions of this document. Sorry.

In the following text i want to show, how to create and modify schematic symbols for EESchema/ KiCad. EESchema is part of KiCad, a GNU programm for creating schematics and layouting PCBs.

The following statements bearing on: EESchema: Build Version: EESchema (2006-08-28) - Unicode version, and EESchema: build Version (20080825c-final). Operating system Linux „Etch“ bzw. „Etchnhalf“.
But the behavior with windows XP is just the same. Due to the fast progress in the work of Jean-Pierre Charras, this manual could be obsolete in some aspects.

1th. Start KiCAD and EESchema.

After starting up KiCAD you get the KiCAD-Screen, where you find also the Button to start the schematic editor EESchema. Look at picture 1.

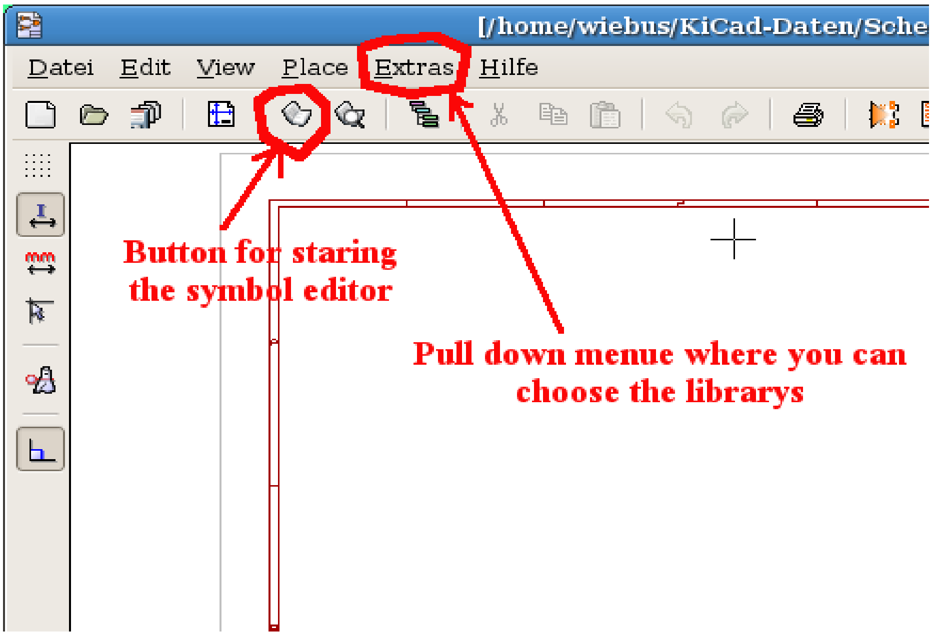


Picture 1: Starting EESchema.

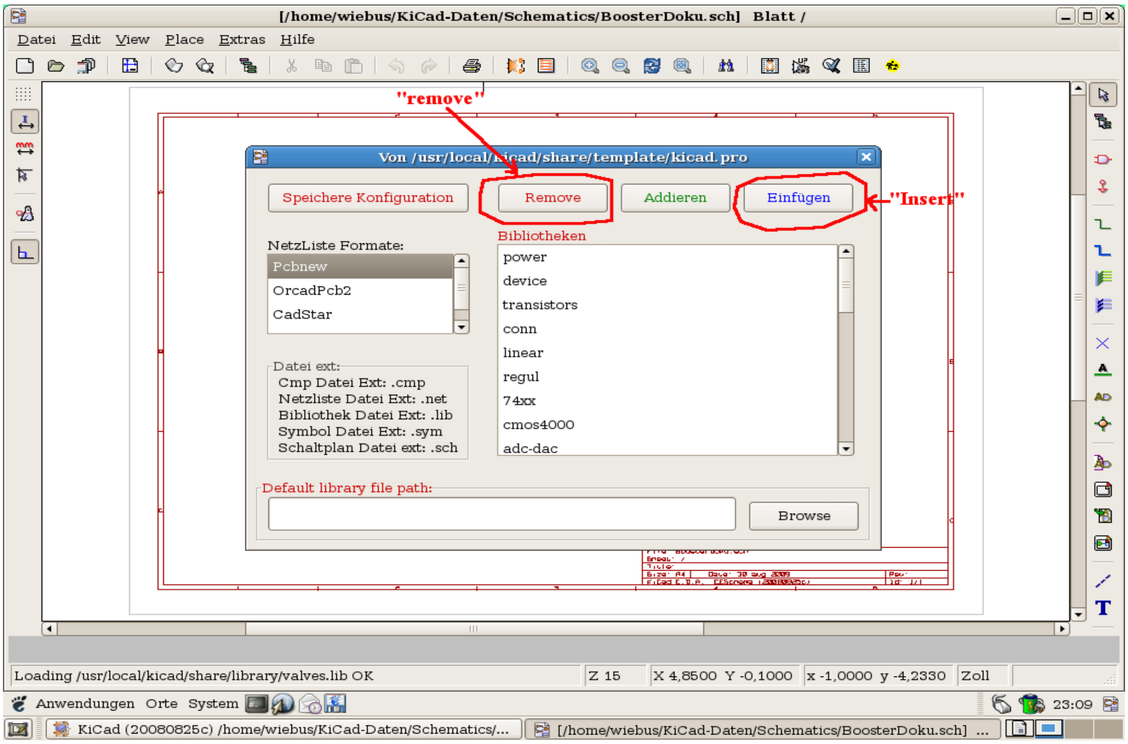
2th. Embedding of librarys.

Now you get a new screen, the one of EESchema. Look at picture 2. For editing librarys, they have to declared at EESchema. For this, you have to choose the menu item „Extras“, and then the subitem „Library“.
You get a new window, look at picture 3, where you see a list of already embeddet librarys. Normally you see here at the first opening after new start of the Program a list of librarys embeddet by default. New librarys get embeddet by using the Button „Insert“ at the upper riht side. this will lead to an opening window, where you can edit the folder and its path to your desired new library (with the suffix „.lib“). After this, you should see the new library (with the whole path) at the list.
„**New library**“ means here, a homemade or somewhere gotten library, which is not at the default, but NOT a library, you want to create. Only existing librarys. With the Button „Remove“ you can remove a library from the list. Also you can remove librarys, which are in the list at

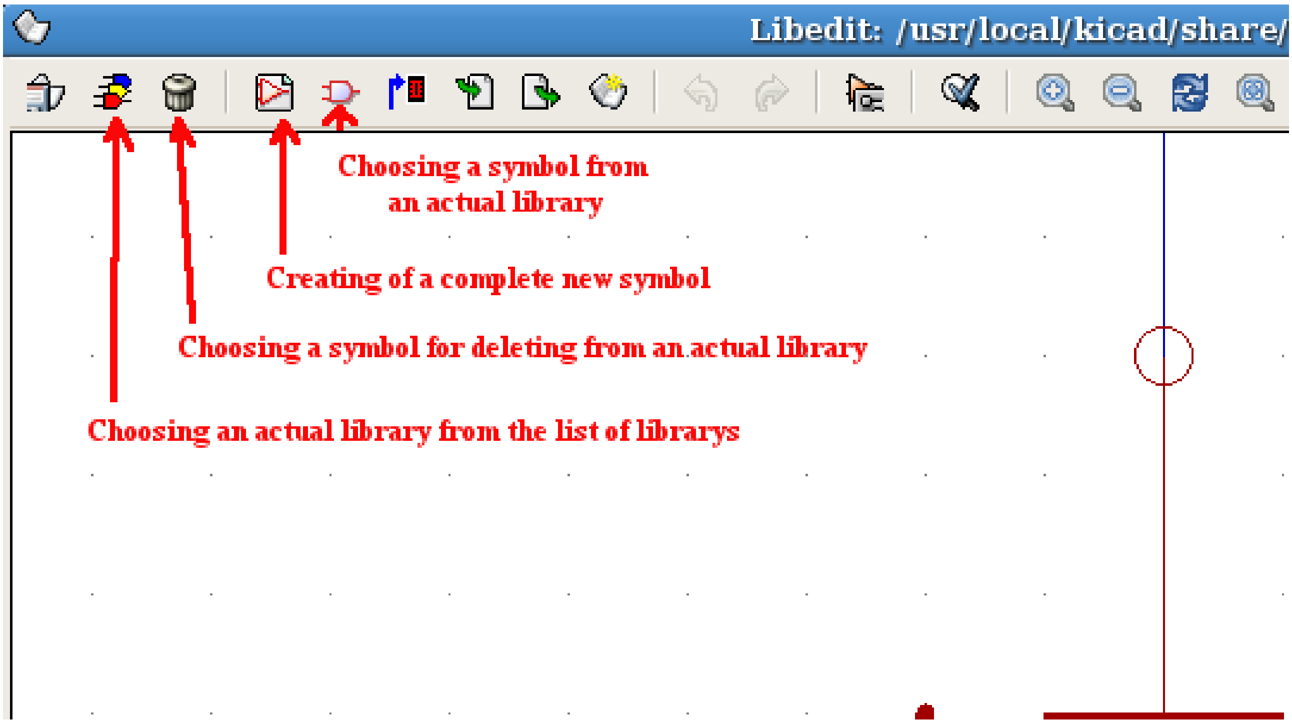
default. But with the next startup of EESchema, you get the original default librarys. For changing de original default values, you have to edit a file by hand somewhere. Make shure you have all rights to get access to the library file. If you have embedded all desired librarys, close the window with the button at the upper right, who shows a black St. Andrews cross.



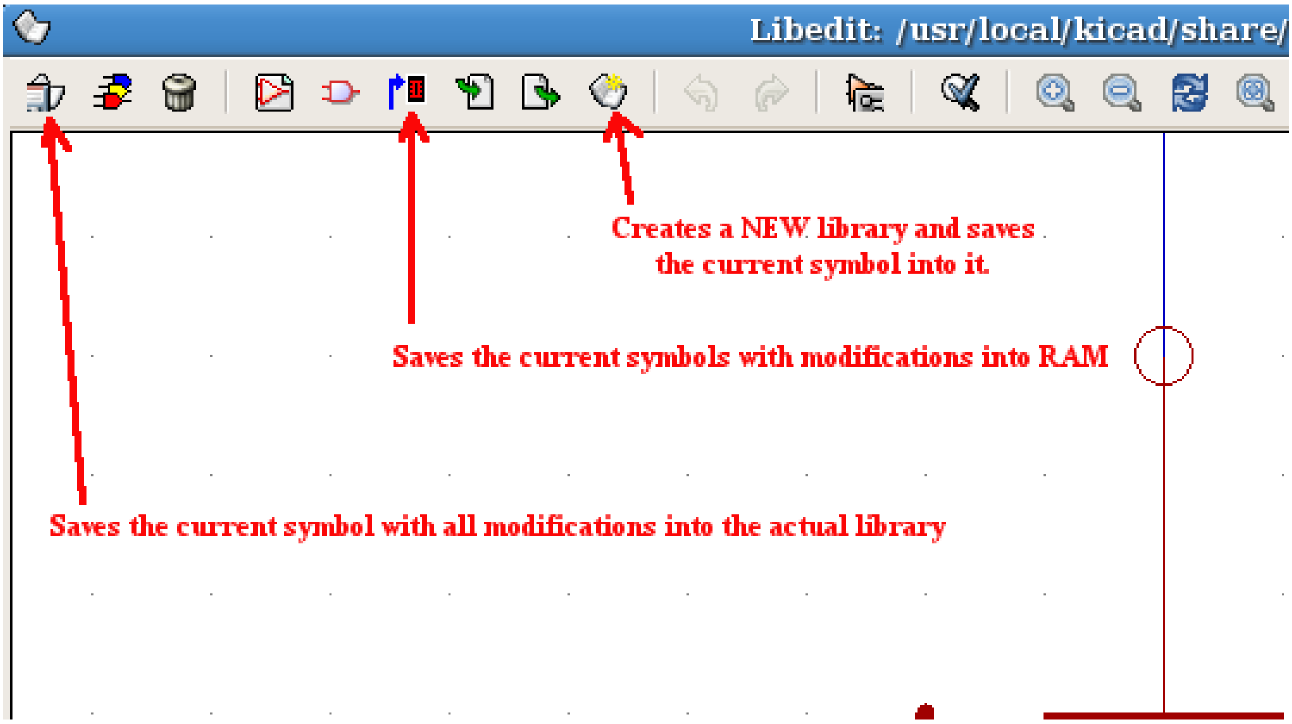
Picture 2: Choosing the librarys and start button for symbol editor.



Picture 3: Inserting and Removing librarys from the list.



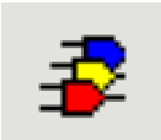
Picture 4: Buttons at the symbol editor.



Picture 5: Buttons at the symbol editor.

3th. Library Editor

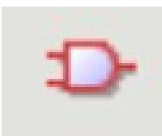
At picture 2, you see the button, who starts the library editor, which is used to manipulate the symbol library. A new window starts. At the upper side, there is a row of buttons. You see it at the pictures 4 and 5.



This Button allows you to choose an actual library to edit from the list of embeddet librarys. From this library, you choose the symbols, who you want to edit, and new or manipulated symbols will saved into this library. You can only choose librarys, which are already embeddet in EESchema.

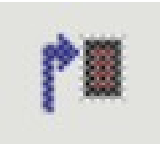
This Button allows you to choose a symbol from an actual libray for deleting.

This button allows you, to create a complete new symbol. There is an another way to create a symbol. Open a symbol for editing, and chance its name. Then save it back to the library or into another library.Dieser Button bietet die Möglichkeit, ein **komplett neues Symbol** zu erschaffen.



This button allows you to choose a symbol for editing from the actual library.

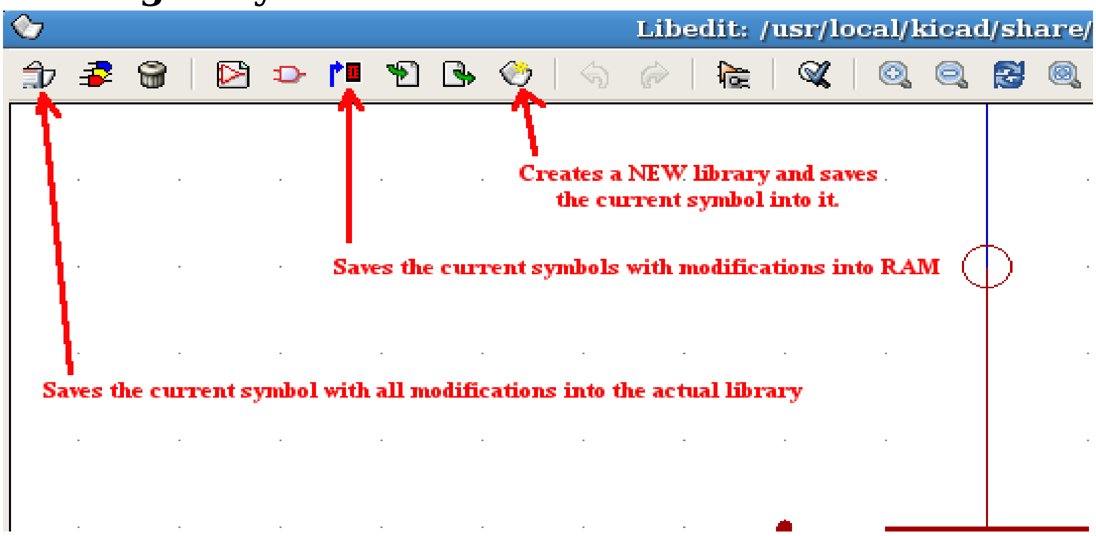
„Save all“. The actual edited symbol is saved to the aktual library at the hard disc.



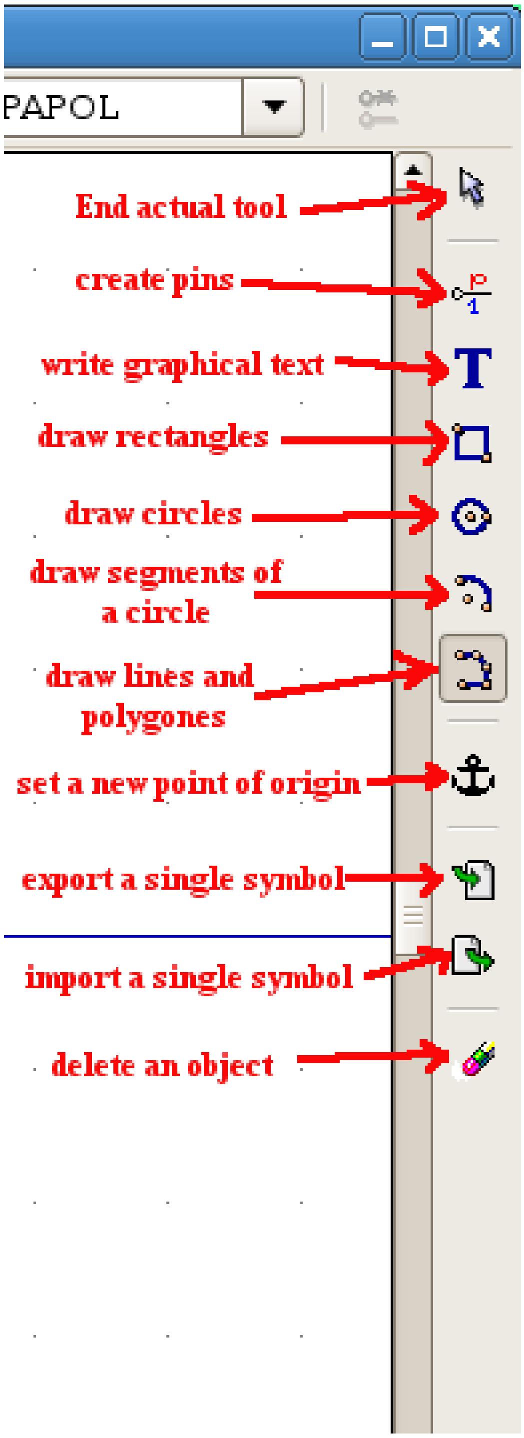
The actual edited symbol is saved with all changes into a non-permanent buffer storage (RAM). This gives you a way, to modify a symbol and use it modifyed at the schematic witout changing the symbol permanently at the harddisc.
Only for small, special changes. Be careful! The button „Save all“ will now change the symbol permanently at the harddisc.

This Button creates a new library and saves the actual edited symbol into it. This is the way to create a new library. This new created library is not acessibly for later savings, unless you will embedding it to the list of librarys of EESchema.

5th. Editing of symbols.



Picture 6: Buttons for editing symbols (upper bar)



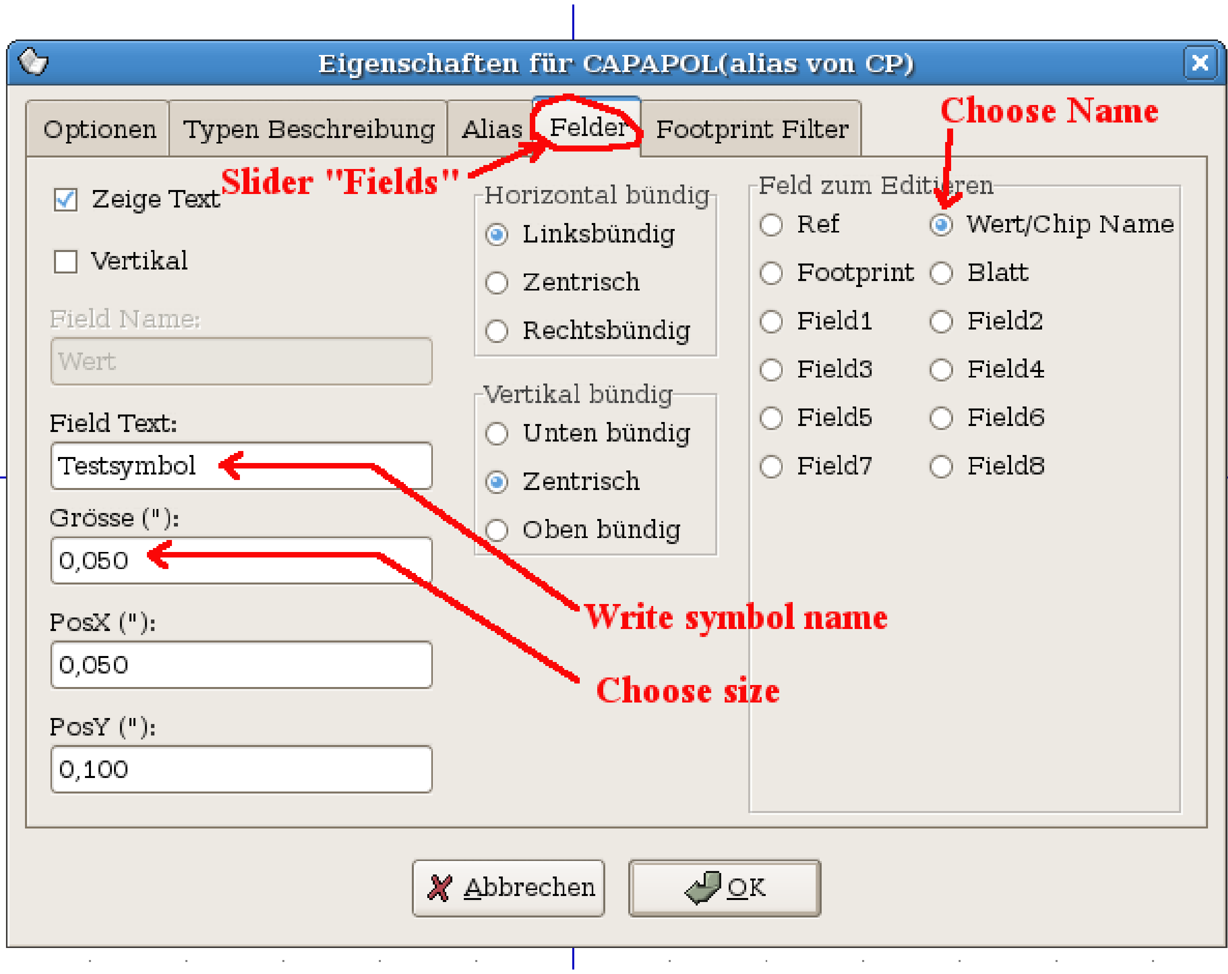
Picture 7: Buttons for editing symbols (right bar)

You see an overview of buttons for editing symbols at the pictures 6 and 7.

Symbols can only edited by Libedit, either by choosing an already existing symbol from an already existing library (Button), or by creating them entirely new (Button).

If you open already existing symbols for editing, you should give them a new name for saving them, for not destroying the old version accidentaly. Allocating new names to a symbol is done by using the button „device properties“:

Using this button, you will open a window like at picture 8.



Picture 8: „device properties“

You have to choose the flag/slider „Fields“ , then to activate the button „Wert/Chip Name“, and now you can write the desired new name into „Field Text“ .

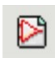
Very very important: There must be NO SPACE CHARACTER in the name!

Reason: At the definition of the symbol you will save, a space character is used at some points to separate the name from following definitions and properties. A space character at the name will make the program to recognitze the end of the name at the space character, and read the following characters as definitions or properties. But at most cases, this will be bogus and you will get an error message Wrong DEF format in Line xyz". You can save the symbol with the bad name witout any problems, but you cannot read it back into EESchema (but of course all other correct symbols of that library). Then you will get this error message, but will never be able to fix the error by Libedit. Look at the attachmend: Error: blank at the symbolname.

The „Name" is the name, which is used to save the symbol to the actual library, und by whom it can be found later at EESchema, if you want it to place it at your schematic. As an example „Resistor" or „PNP-Transistor". It is choosen normally very abstract. A special value like „470 Ohm" or „BC327" will be given later at Eeschema for an special device.

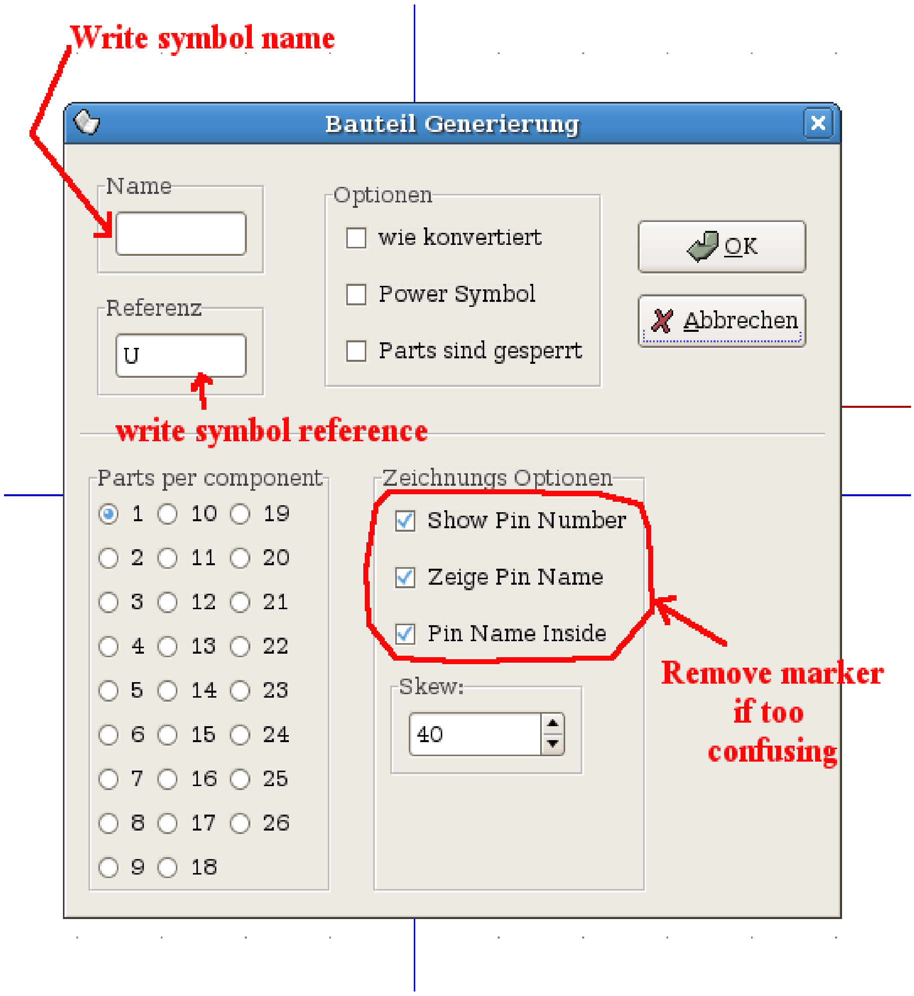
If you choose an symbol from an existing library, you should alter this name first, therefor you cannot overwrite and destroy the original symbol by saving.

The „numbering Name", wich shows the special device with this symbol at the schematic, like „R15" for a resistor, should allocated with the Button „Ref" (Reverence?) . This is the button just left of the button „Wert/Chip Name". Here you put normally only one or two letters in, which are used for numbering of the devices. Like „R" for a resistor or „C" for a capacitor. In EESchem a you will see there an „R?" or „C?" with a questionmark. You have to put in the number by hand.

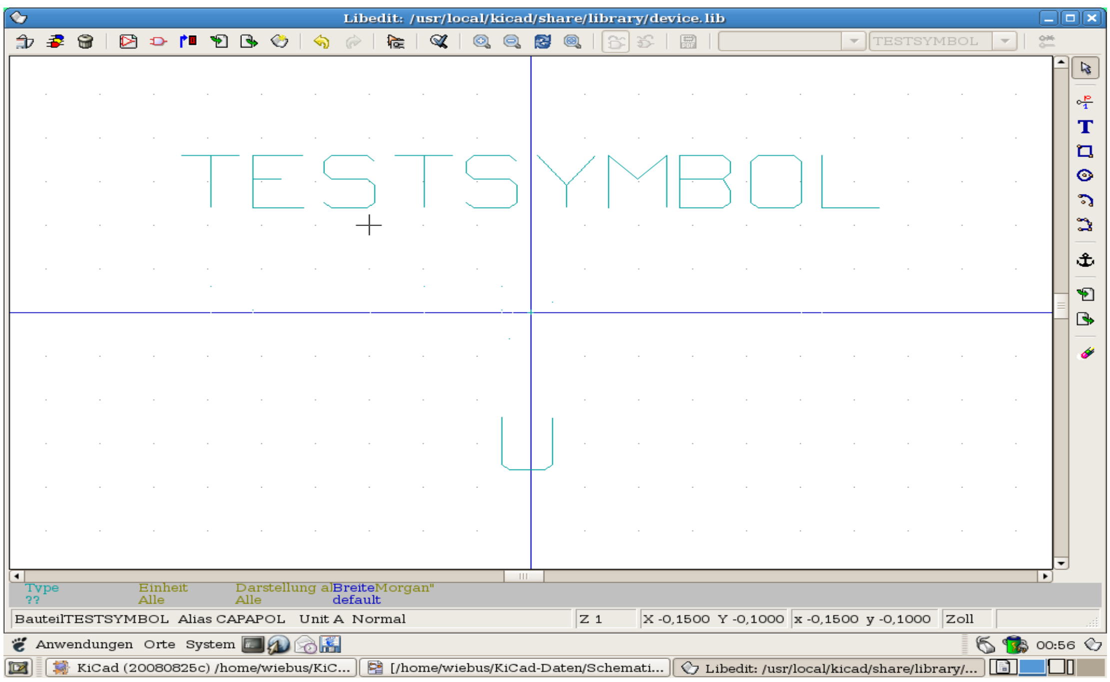
You should give a name or a reference also to entirely new symbols. You can use the button 

for making new symbols, it will open a window like you see at picture 9.

At the upper side left, you can write „Name" and „Reference". The markers at the field „Zeichnungsoptionen" can be removed, if the symbol or the schematics get too confusing. Personally i remove them all with the exeption of „zeige Pin Name" , but the markings you see at picture 9 are the default markings. Finally, you have to quit with the „OK" Button at the upper right side.
The window closes now, and you get an symbol, which is empty with the exeption of the text for „Name" and the „Reference" (At the example, i used the symbolname „Testbauteil" and the reference „U".) See this at picture 10.



Picture 9: Creating a very new symbol

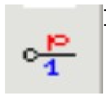


Picture 10: Testsymbol with reference „U“.

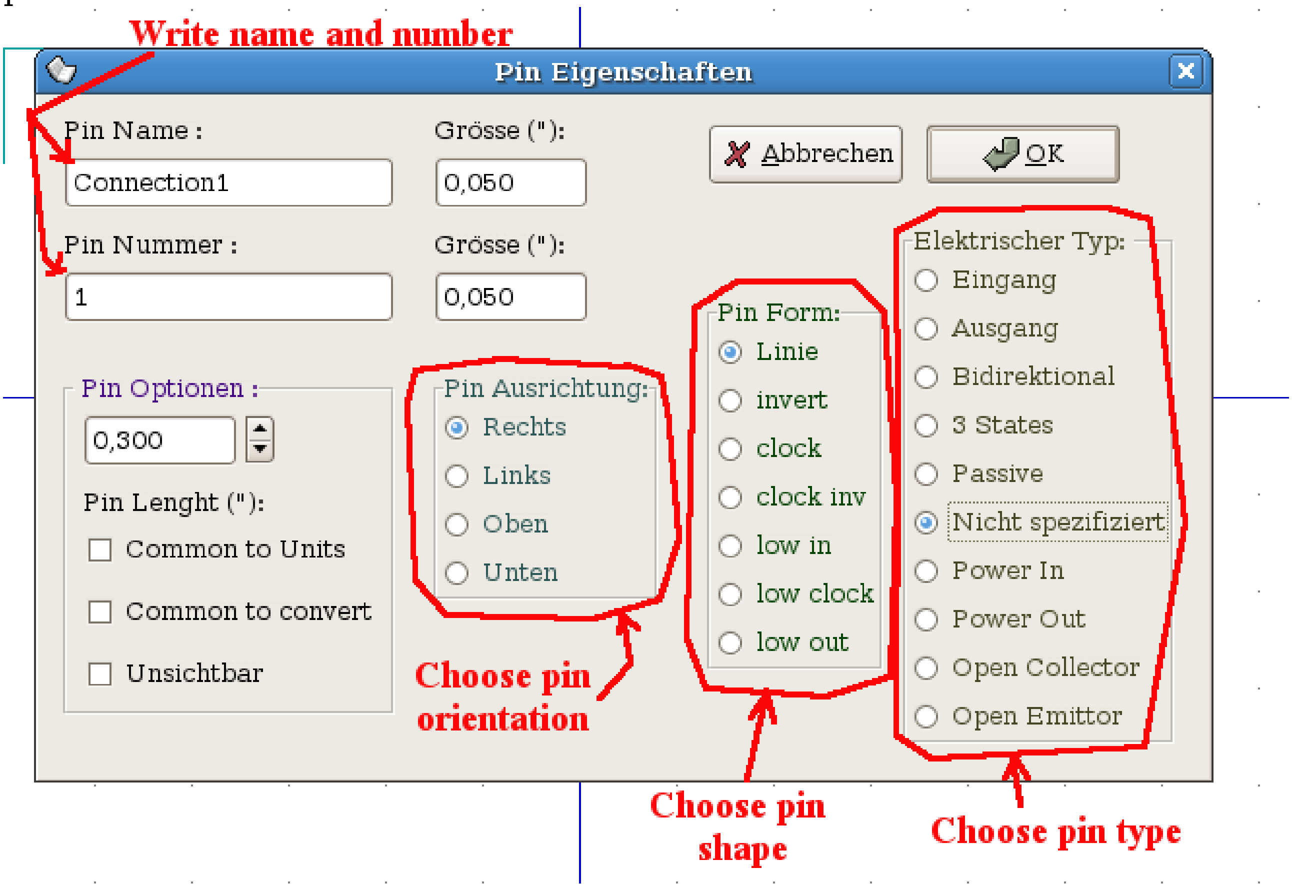
If you click at this textfields with your right mouse button, you will open a window with more properties and editings of the textfields. As an example „Move Field“. Now we choose „Move Field“, to move this cumbersome but needful text out of the middle of the screen.

There are more commands like „rotate field“ und „edit field“, which are doing mostly the same as the name says. Also there are additional commands vor zooming, refreshing ant something like this. Especial carefull you have to use the command „raster“.

Connection Pins:

It is opportune, to go on with the placement of the connection Pins of the sympol. Therefor you choose the icon „insert pin“  from the buttonbar at the right side.

Roughly at the desired position of the pin we schould click with the left mouse button into our schematic. A window „pin properties“ will now occur like at picture 11.



Picture 11: Pin properties as like name, number, orientation, shape and type.

At the field „Pin Name“ , you can write the name of the pin. I have choosen the name „Connection1“ for the testsymbol. But i could have choosen „base“ or „Drain“ for a transistor as an example, or perhaps „Anode“ for a diode or a tube, or „Vcc“ at an IC. It is beneficial to choose a name, who describes the properties of the pin very well and is short. Blank characters at the name will be replaced by underline characters.

At the field „Pin Number“ you have to choose a pin number. But every pin number can be used only once at a symbol. The ideal case is, if the pin number of the symbol corepspondences with the pin number from the data sheet. This will be easy at Standard ICs, but difficult for some Transistors, because there are transistors of the same type but from different vendors with the same name, same housing and pinning, but other pin numbering.

At the field „Pin orientation“ you can choose, which orientation of the pin is desired.
As a rule of thumb, you should remember:

„right“ means, the symbol is right of the pin.
„left“ means, the symbol is left of the pin.
„top“ means, the symbol is in top of the pin.
„bottom“ means, the symbol is at the bottom of the pin.

At the field „Pin shape“ you can choose different shapes of connection pins.
I do not know, wether this different shapes of connection pins have any meaning at further properties.

At the field „pin type (electrical type)“ you can make decisions about the meaning and electrical properties of the pin. This effects to the connectability, the den ERC (Electrical Rule Check) and the netlis. But „not specified“ should allow connectability at every time, despite perhaps some problems with the ERC.
The types „Power in“ and „Power out“ are used for power connections and ground connections (Power ports) verwendet. But for this, this symbols have to be inserted in the library „Power“.
Choosing „not specified“ and leaving the symbol in another library than „Power“ allowes it to use ground and power connection symbols without errors from the ERC. But errors, like the fatal connections between different power sources or shortcuts between power sources and ground cannot be detected by the ERC.

Clicking to „OK“ closes the window and shows a pin attached to the mouse pointer, which can be moved. Clicking the left mouse button will place the pin. Clicking the rigth mouse button will open a window „Pin-Tool“, where you can choose different ways to edit the pin.

„end tool“ closes the Pin-Tool.

„move pin“ allowes you to move the pin to a new position.

„Edit Pin“ allowes the editing and changing of pin-name and pin-number, pin orirntation and further properties.

„Delete Pin“ deletes the pin.

I do not know the meaning of „Global“. Perhaps you can import or export some pin properties.

„Zentrisch“ moves the visible window for getting the pin into the middle. This command moves only the window, not the pin or the point of origin.

With „Zoom in“ and „Zoom out“ you can switch the scale of the visualisation a step up or a step down.

With „Choose Zoom“ you can choose a zoomfactor from a list. Default value is 4.

„Zoom ganze Seite“ chooses automaticaly the scaling factor to a value, that you can see all objects in the hole screen.s

„Redraw“ redraws the visualisation, if after some actions some things are remaining at the screen, or are have vanished.

„Choose raster“ chooses the raster, in which you place the pin. THIS COMMAND HAS TO BE USED VERY CAREFULLY! Default is a 50mil raster.

„close“ closes the window.

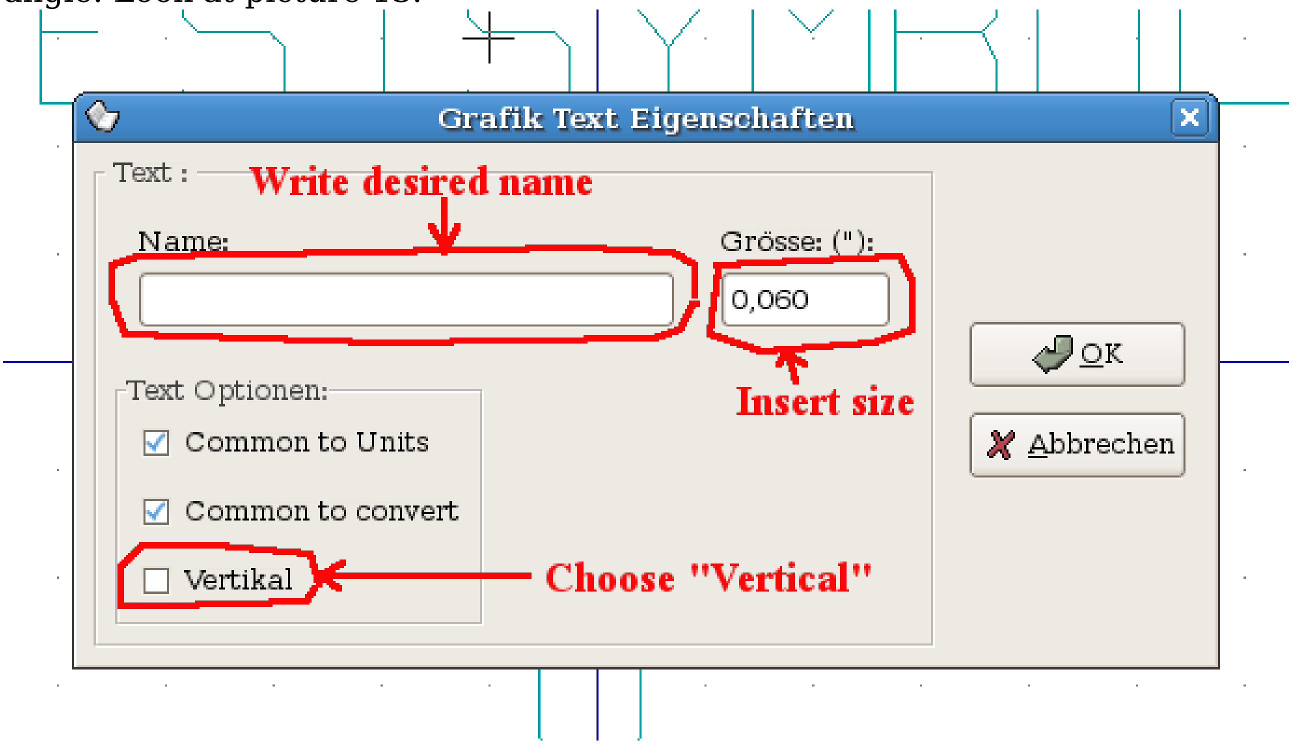
There are some remarks about „Choose raster“. Special for pins the choosen raster is very important for pins, because the raster is very important for connecting the pins during drawing the schematic. You can only connect to a pin, if the raster is right. As an example: if you have placed the pin at the symbol at the 50 mil raster, it can be placed and connected at the schematic also with the 50mil raster and all finer raster types. But if you have placed the pin at the symbol as an example at the 25mil raster, it can be placed and connected at the schematic also with the 25mil raster and all finer AND Fitting raster types. So you can choose also the 5mil and the 1mil raster. The 50mil raster will not work, because it is too rough, and 10mil and 2mil will not work because you cannot divide 25 by 10 or 2 without remainder.
A akwardly choosen raster will force you to change the raster more often during connecting in the schematic. This means a lot of unnecessary additional work.
This is the reason, **WHY IT IS EXTREME USEFUL** (but not forceful necessary), **TO PLACE PINS AT THE 50 MIL RASTER ONLY!**

The additional design of the symbol can be done with any liked raster.
But it happens very often, that you have to move a pin again, during drawing the symbol in a fine raster, and you forget to switch the raster back to 50mils, when you move the pin.

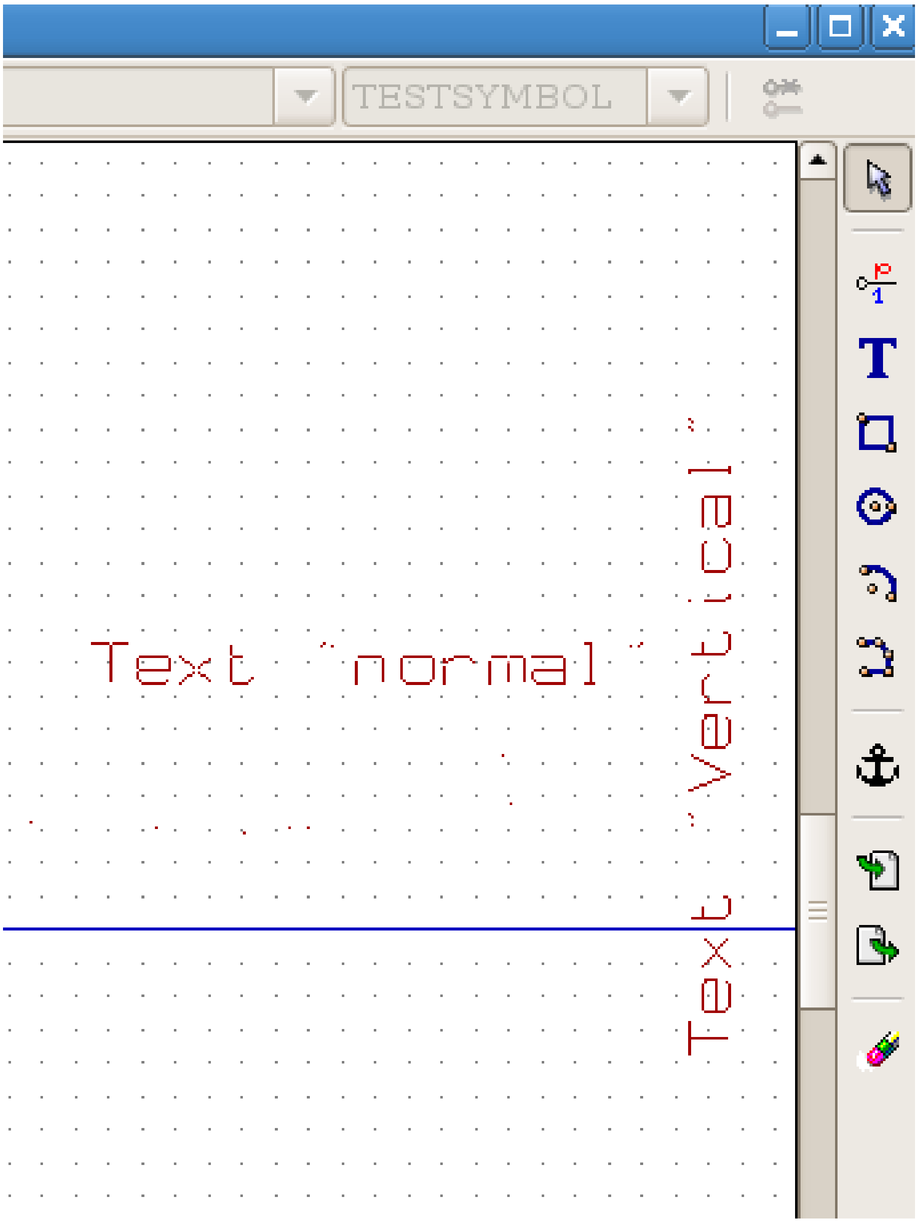
The further design of the symbol with drawing and text can be done with the buttons at the right side. Look at picture 7.

Text:

The button Text  allowes you the design of „graphical text“. This means text, who appears only as a remark in the symbol witout any function (like it will be with the name or the number, which are used for identification by the program).
If you have choosen this button, a click with the left mouse button into the drawing will show you a window, like you see at picture 12. At the field „name“ you can insert the desired text, and at the field „Größe“ you can insert the size of the letters (default 0,060).
If you want to show the text at 90 degrees turned counterclockwise (so that it can be read easyly from the right side), you have to mark the field „vertikal“.
„vertikal“ means not the letters on the top of each other, but turned by a right angle. Look at picture 13.



Picture 12: Writing text.

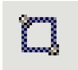


Picture 13: Text „normal“ and text „vertical“.

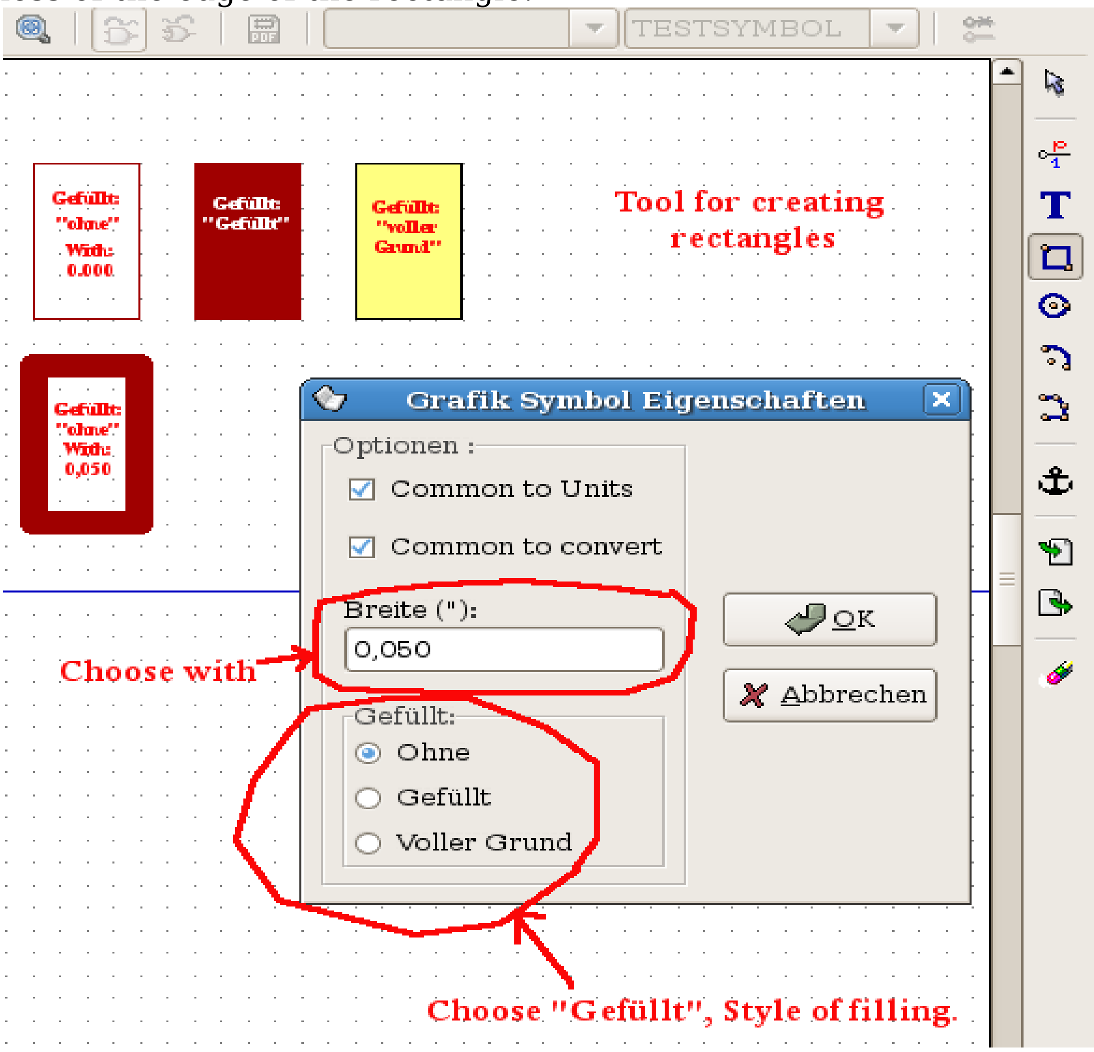
If you click by your right mouse button to a textobjekt, then a window will open, where you can choose different options. Some of the options are „Move Text“, „Text Editor“, „rotate Text“ and „Delete Text“ .
„Move Text“ allowes you to shift the text, „Text Editor“ opens the same window like at picture 12 for a new text for later changings. „rotate Text“ has the same meaning like the setting or unsetting of the marker at the field „vertical“. „Delete Text“ will delete the text objekt.

All further options like „Zentrisch“, „Zoom in“, „Zoom out“, „Choose zoom“, „Zoom ganze Seite“, „redraw“, „choose raster“ und „close“ are the same like you have learned at creating the pins.

Rectangles:

The button „Rectangles“  allows you the drawing of rectangles. After you have activated the button by clicking it with your mouse, a **DOUBBLECLICK** with your left mouse button allows you to make a rectancle by moving the mouse. The ending and the fixing of the rectangle is done by a single click with your left mouse button. A single click with your right mouse button to the edge of the rectangle (pay attention to the choosen raster!) opens a window. At this window, there are several options:

„Tool End“ ends the tool.
„Move Rect“ allowes the shifting ans moving of the rectangle.
„rectangle options“ allowes you the changing of some properties of the rectangle. Look at picture 14. At „with (Breite)“ you can choose the line thickness of the edge of the rectangle.

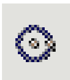


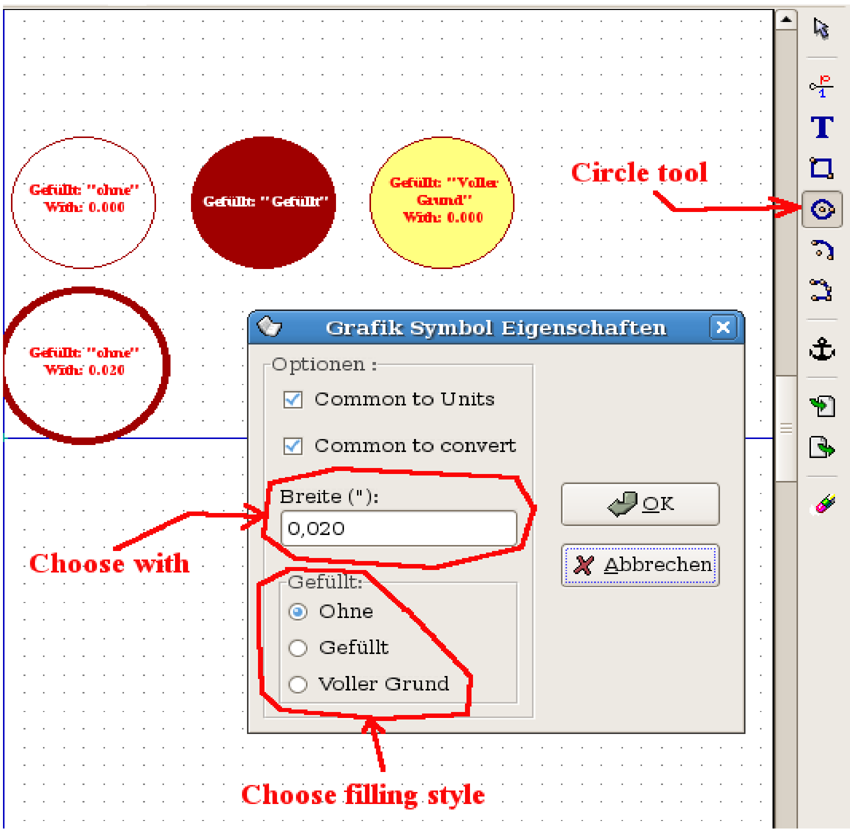
Picture 14: Choosing rectangle properties from the window „graphic symbol properties“.

Default value is 0.000. An example for 0.050 is also shown at picture 14.
At „Gefüllt“ you can choose by marking, wether the rectangle will be shown only by the edge („ohne“) , or wether it will be shown complete filled („gefüllt“). „voller Grund“ means, the rectangle is shown by the edge and by a tinted filling. You can see examples as picture 14. The window „graphic symbol properties (Grafik Symbole Eigenschaften)“ is shown in similar form also for rectangles, circles, segments of a circle and polygons. REMARK: If the window not contains the „options“ property, you did not catch the opject correct with the cursor. The most easyly way to catch the objekt with the cursor at cornerpoints, start- and endpoints and centerpoints.

All further options like „Zentrisch“, „Zoom in“, „Zoom out“, „Choose zoom“, „Zoom ganze Seite“, „redraw“, „choose raster“ und „close“ are the same like you have learned at creating the pins.

Circles:

The button „Circles“  allowes you the drawing of circles. After you have activated the button by clicking it with your mouse, a single click with your left mouse button will fix the centerpoint at the drawing. Moving with the mouse increases the radius of the circle, until you end the tool with a second click with your left mouse button. A single click with your right mouse button to the edge of the circle (pay attention to the choosen raster!) opens a window. At this window, there are several options:



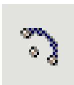
Picture 15: Choosing circle properties from the window „graphic symbol properties“.

„Tool End“ ends the tool.
„Move Circle“ allowes the shifting and moving of the circle.
„circle options“ allowes you the changing of some properties of the circle. Look at picture 15. At „with (Breite)“ you can choose the line thickness of the edge of the rectangle. Default value is 0.000. An example for 0.020 is also shown at picture 15.

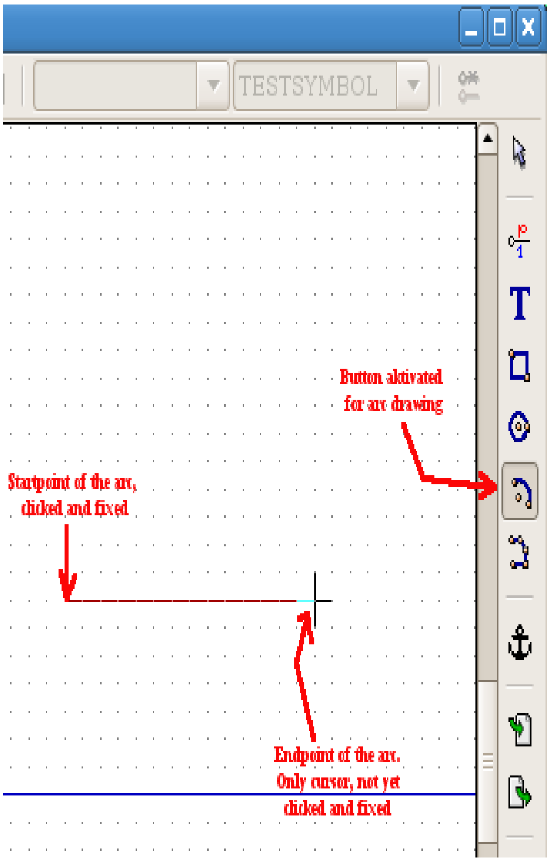
At „Gefüllt“ you can choose by marking, wether the circle will be shown only by the edge („ohne“) , or wether it will be shown complete filled („gefüllt“). „voller Grund“ means, the circle is shown by the edge and by a tinted filling. You can see examples as picture 15. The window „Grafik Symbole Eigenschaften“ is shown in similar form also for rectangles, circles, segments of a circle and polygons. REMARK: If the window not contains the „options“ property, you did not catch the opject correct with the cursor. The most easyly way to catch the objekt with the cursor at cornerpoints, start- and endpoints and centerpoints.

All further options like „Zentrisch“, „Zoom in“, „Zoom out“, „Choose zoom“, „Zoom ganze Seite“, „redraw“, „choose raster“ und „close“ are the same like you have learned at creating the pins.

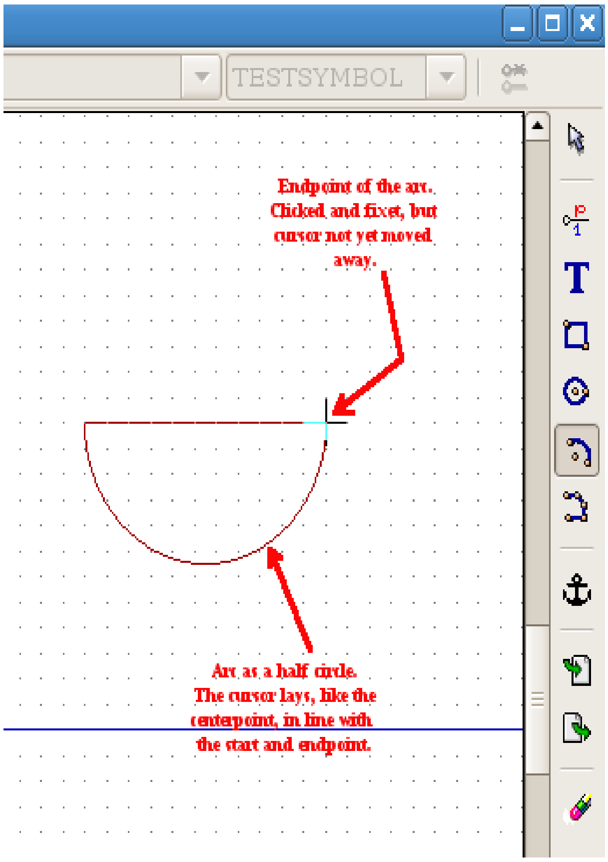
Segments of circles:

The button „Segments of circles“  allowes you the drawing of circle segments. After you have activated the button by clicking it with your mouse, you will choose by a first click with your left mouse button the start point, and with a second click with your left mouse button you will choose the endpoint of the circle segment. First, it looks, like you would draw a line between the start and endpoint of the circle segment. Look at the pictures 16 and 17.

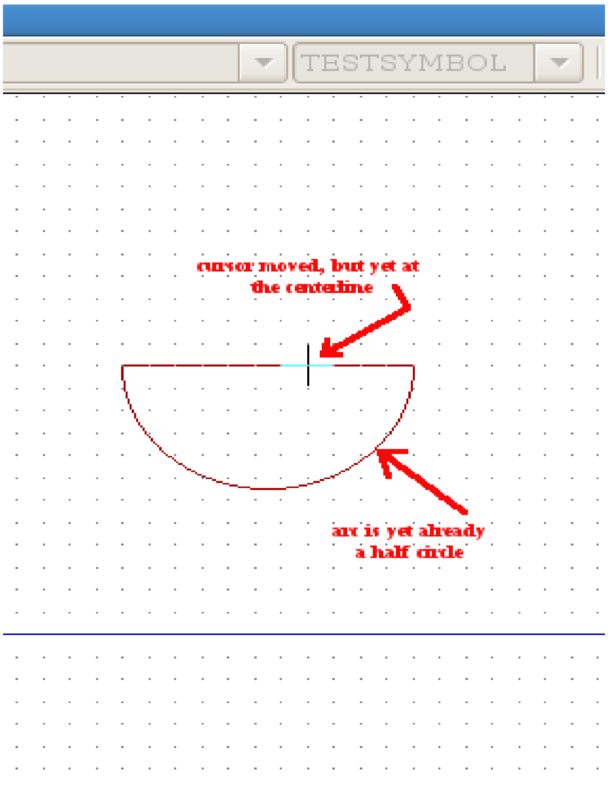
Straight after you have fixed the and point with a second left mouse click, you **move the centerpoint** of the circle by moving the mouse cursor, which will **change the orientation and with of the bowing**. The centerpoint can only be moved in a right angle to the line between start and endpoint. The **orientation is changed by moving the cursor across the the line** between start and endpoint. Look at the pictures 18, 19 and 20.



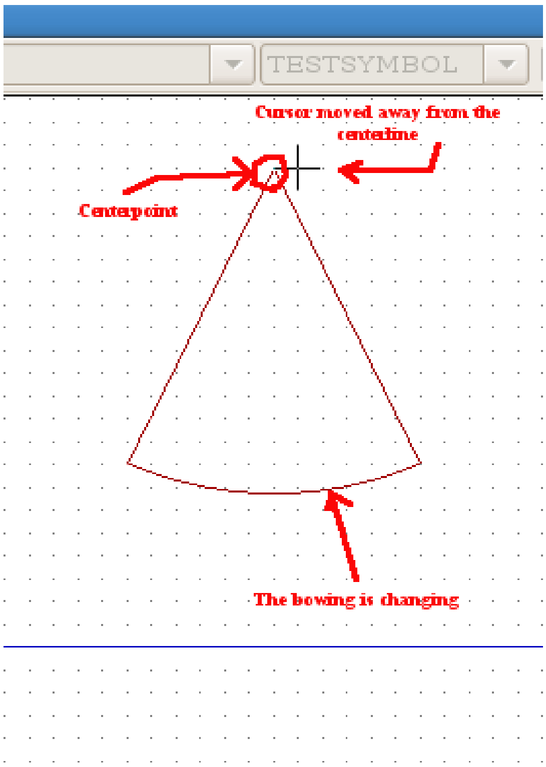
Picture 16: Startpoint and endpoint of the arc.



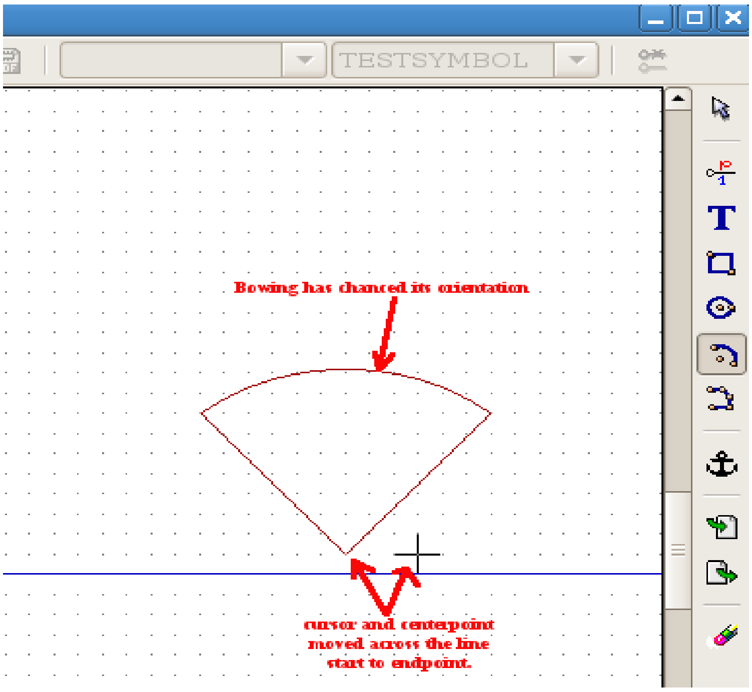
Picture 17: Creating the arc by fixing the endpoint.



Picture 18: Changing the bowing.

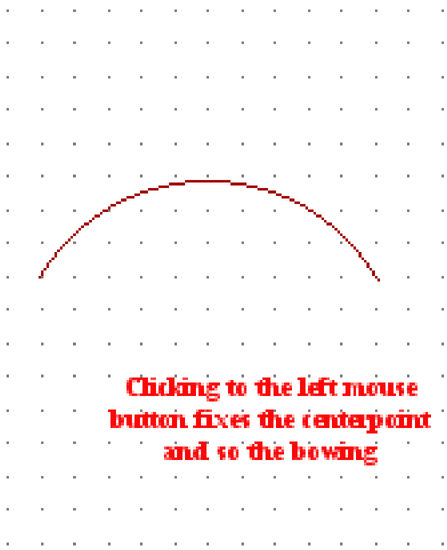


Picture 19: Changing the bowing.



Picture 20: Changing the orientation of the bowing.

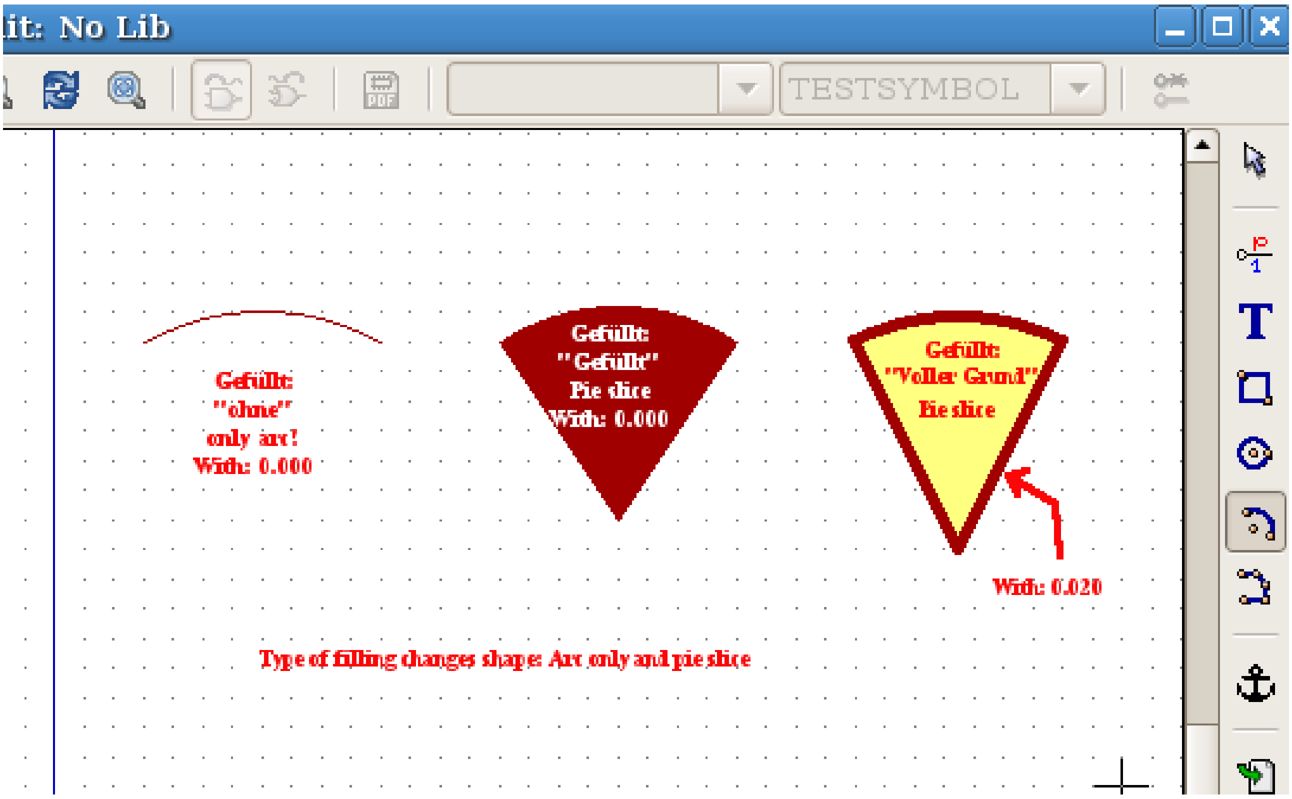
An ending left mouse click fixes the centerpoint and therefor the hole segment of a circle. Look at picture 21. But if you clicked **during the active working** at the circle segment to the **right mouse button**, a tool would open with the Standard options and the adittional option „circle options“. At this option, you can edit some properties of the segment of the circle.



Picture 21: Finishing the arc.

The segment of the circle is therefore seen as a „pie slice“. At „With (Breite)“ you can choose the line thickness of the edge of circle segments. Default value is 0.000. An example for 0.010 is also shown at picture 22.

At „Gefüllt“ you can choose by marking, wether the segment of the circle will be shown only by the edge („ohne“) , or wether it will be shown complete filled („gefüllt“). „voller Grund“ means, the segment of the circle is shown by the edge and by a tinted filling. The window „ graphic symbols properties (Grafik Symbole Eigenschaften)“ is shown in similar form also for rectangles, circles, segments of a circle and polygons. REMARK: If the window not contains the „options“ property, you did not catch the opject correct with the cursor. The most easyly way to catch the objekt with the cursor at cornerpoints, start- and endpoints and centerpoints.

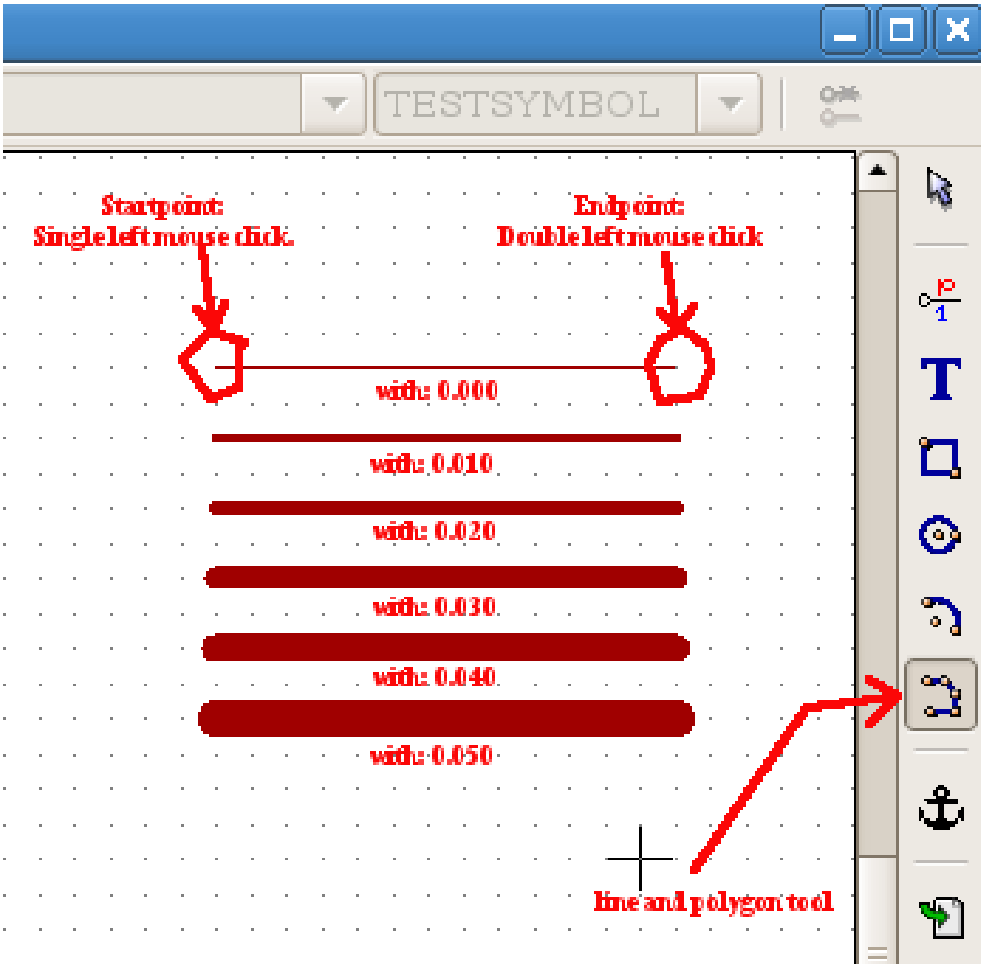


Picture 22: Filling of the segment of a circle leads to an arc only or to a pie slice shape.

But here at the segments of a circle, the funktion of the filling property „ohne“is a little bit different. At this case, ONLY THE ARC OF THE CIRCLE is displayed, but NOT THE RADII to the centerpoint. Look at picture 22.

Polygon:

The button „Polygon“  allowes the drawing of lines, which are chained together for the shape of a polygon or traverse. It is very similar to the rectangle and circle tool, but much more complicated.
For the drawing of lines:Zum Zeichnen von Linien:
After you have activated the button by clicking it with your mouse, you will choose the startpoint by a click with your left mouse button. Then you move the mouse cursor to the end of the line, and fix it with a doubble left mouse click. The next line is startet at the end of the previous line by a single left mouse click, and so on. So you can draw a sequence of lines. They have not necessary meet another. You can chance the with of the lines by clicking the right mouse button. Now a window will open (like at the rectangle or circletool), where you can use at the option „linien Optionen“ the well known Window „Grafik Symbol Eigenschaften“. There you edit at the field „breite“ a desired with. Look at picture 23.

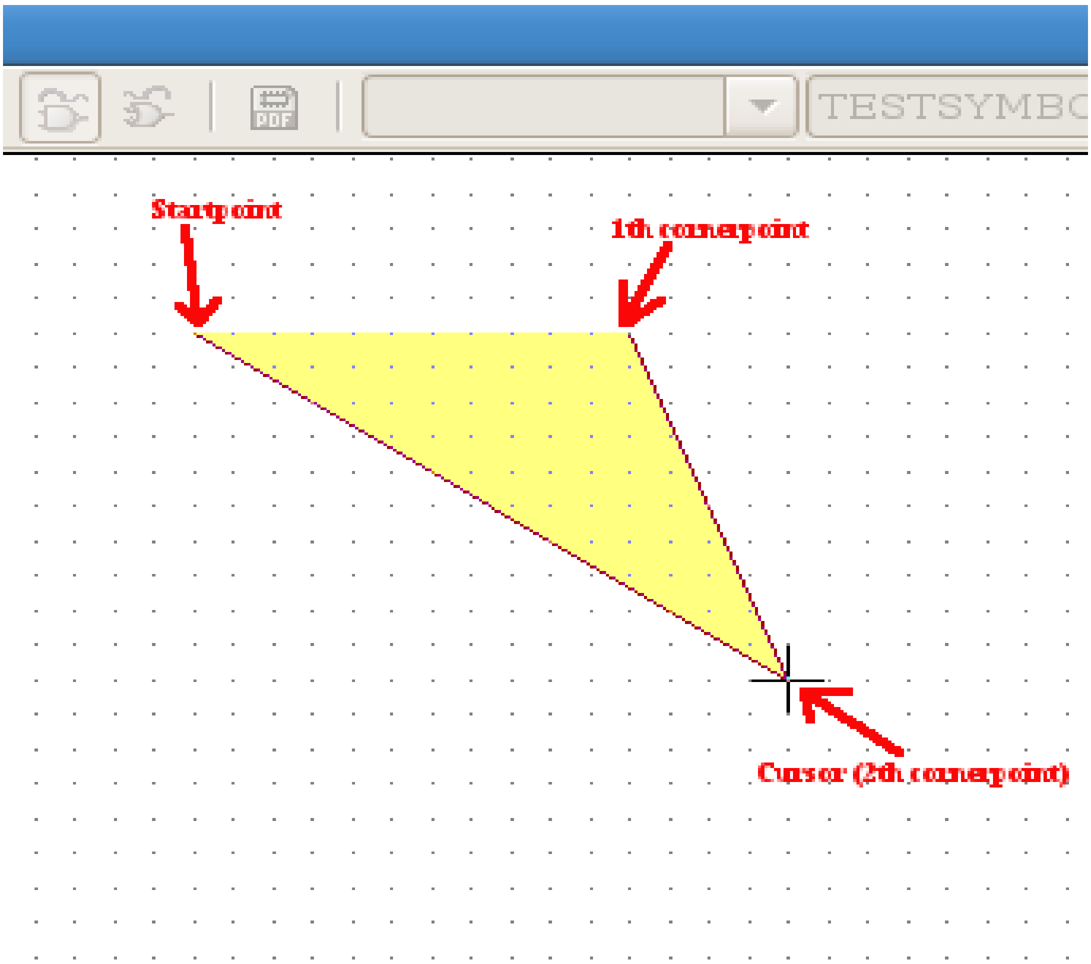


Picture 23: Drawing of lines at different with.

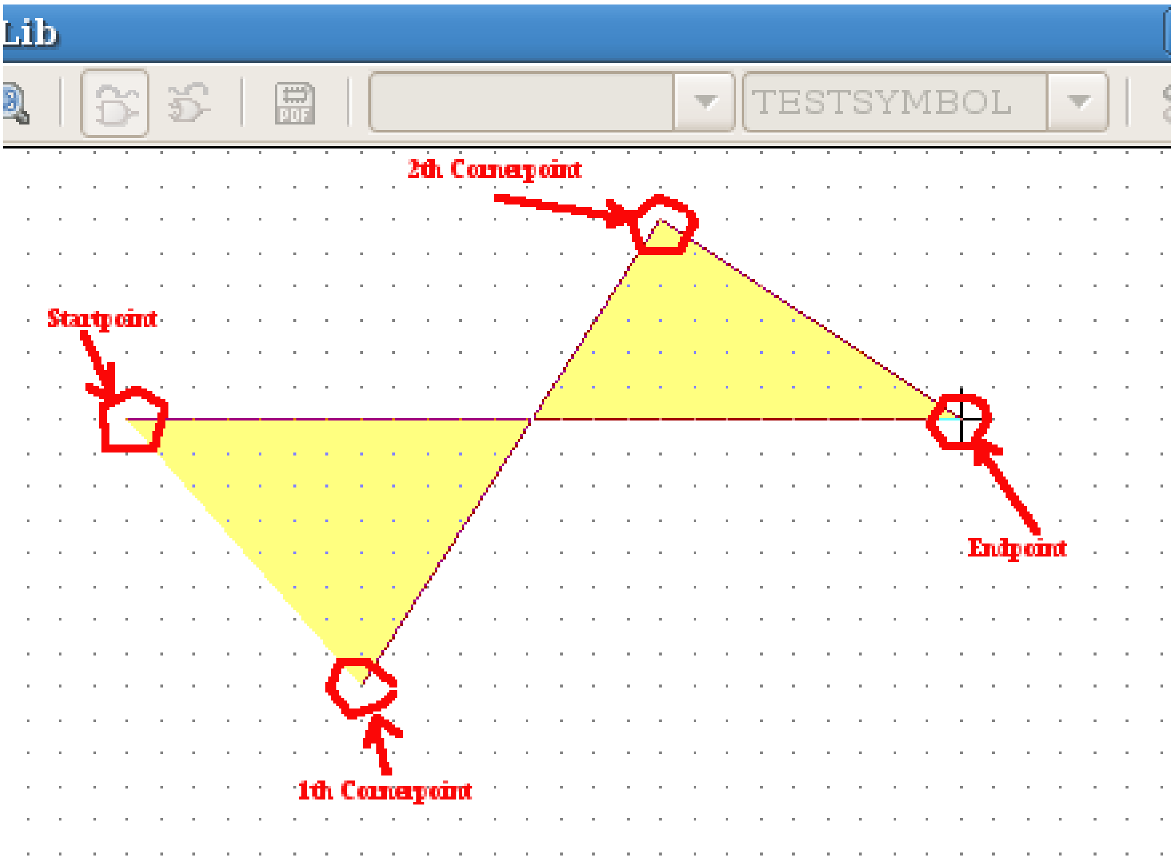
But if you not end a line with a left doubleclick, but a left single click, and start the line again with a single left click, you got a sequence of chained lines, like a traverse or a polygon, if the end of the last line meets ste start of the first **(DANGEROUS!?)**. The last point of the polygon has to be endet again with a left doubleclick. Polygons are mainly a way to create planes, and not lines.

This is the reason, why at example picture 24 the startpoint and endpoint are identical. You can see that the object is a plane, if you choose the window „Grafik Symbol Eigenschaften“ by clicking the right mouse button. you get an option „Linien Optionen“. With „Gefüllt“ you can coose „Ohne“, „Voll“ and „Gefüllter Grund“ like at rectangles or circles. You can see it at picture 24.

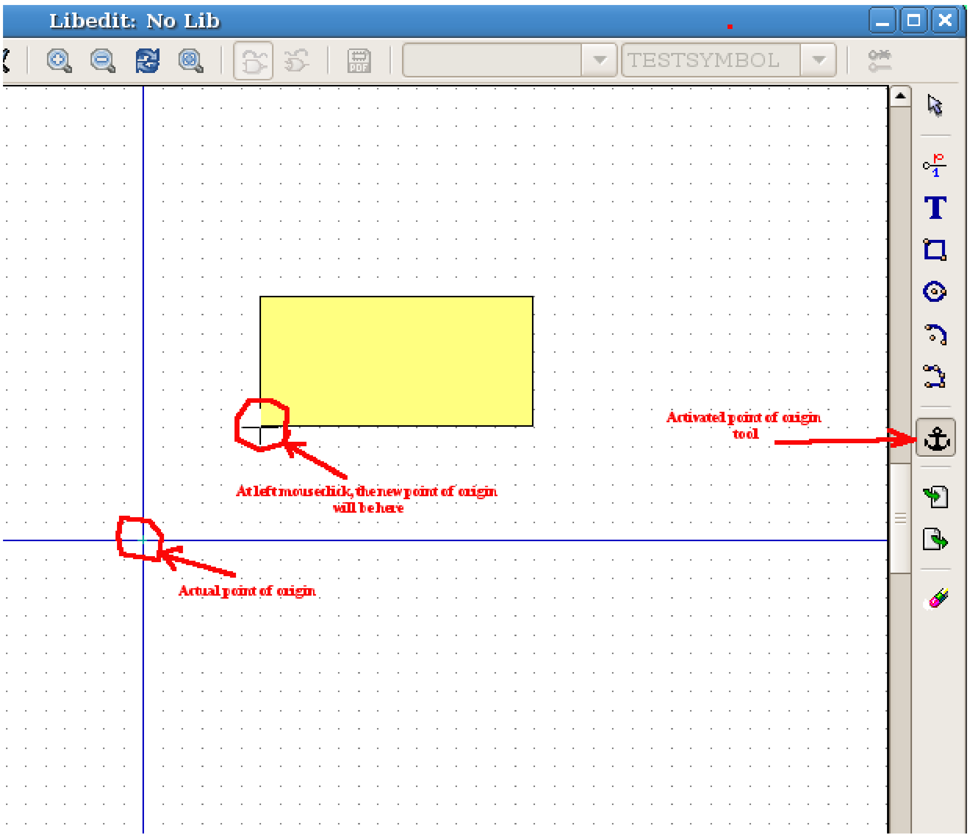
Startpoint and endpoint of a choosen polygon have not necessary to be identical. This tool creates a plane, if there are more than two points of the polygon exists, which are not in one straight line. Look as an example to the pictures 25 and 26. This is also the reason, because i think, it could be possibly dangerous to draw chained lines, which should not make a plane, as a polygon line instead of different independent lines.



Picture 25: Three points making a plane.

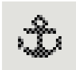


Picture 26: Four points are making a plane.



Picture 27: Point of origin tool.

Point of Origin:

The button „Point of origin“  allowes to shift the point of origin of the symbol. Its use is very simple: After activating the tool with a mouse click to the button, a click at the left mouse button fixes the point of origin to the actual cursor position. Look at picture 27.

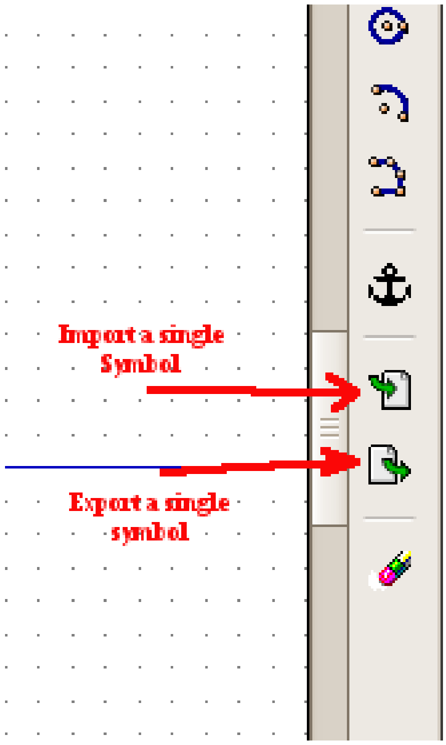
Delete/Erase:

The button „Delete“  allowes the deleting of objects out of the symbol. After activating the tool by clicking the button with the left mouse click, you can choose the object you want to remove with the cursor and click to the left mouse button. The easyest way to catch the objects is to click them at edges, cornerpoints, start or endpoints and centerpoints (circles and segments of circles).

Importing and exporting Symbols:

The button „export Synbol“  will export the hole current symbol to a file with the ending „.sym“. A window will open, where you can choose path and file name.

The button „import Synbol“  will import a symbol from a file with the ending „.sym“. Normaly a file created by the above export tool. The symbol will be added to the current symbol as a group, and can be placed as a group. After placing, all elements of the group can choosen and edited normaly. Look at picture 28. The buttons are not only at the right side, but also at the upper tool bar. Exporting and importing are commonly used for creating new symbols by modifying old symbols.



Picture 28: Importing and exporting single symbols as *.sym file.

Group:

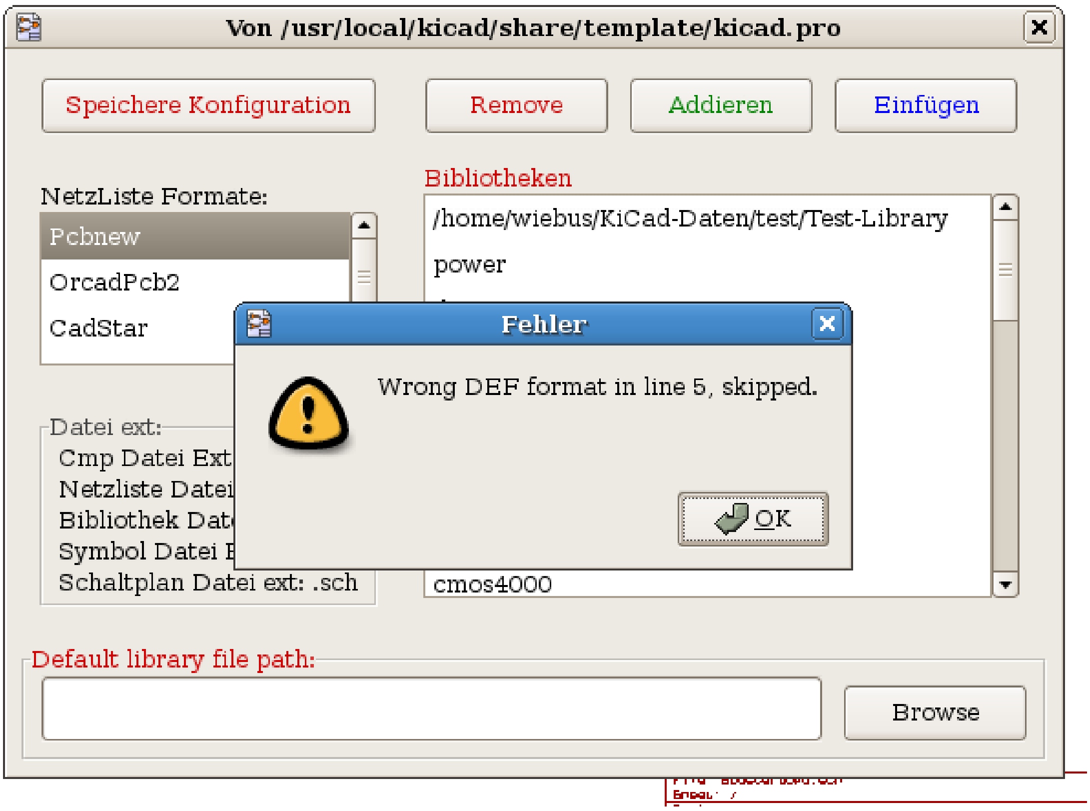
No button is associated to this tool. You can, without any tool/button activated, click with your left mouse button into your window, keep the left mouse button pressed and draw a rectangle open by moving the mouse.
Alle objects in this rectangle a grouped together and can be moved together.
Clicking at the right mouse button will open an attached menue.

Attachment

Error: Blank at symbol name (Wrong DEV format in Line xyz, skipped).

To show you what happens, if you create a symbol with a blank in the name, i will create this error as an example and fix it again.

Therefor we choose any symbol at the symboleditor, here as an example „TRANSISTOR-P-MOSFET-ENHANCEMENT“ and save it into a new library as an examle „Test-Library.lib“. Than we save it again with a new name, as an example „TRANSISTOR-P-MOSFET ENHANCEMENT2“ into the same library. Look especial to the error of an blanc between MOSFET and ENHANCEMENT2. Saving this symbol will not create any error. Attention: Do not forget to ad the library „Test-Library.lib“ to the list of used librarys at Eeschema! Now close the symboleditor, Eeschema and the whole KiCad. Then start KiCad again.



Picture 29: Error Message „Wrong DEF format.......“

If we now add the library „Test-Library.lib“ to the library list at Eeschema, we will get an error message „Wrong DEV format in Line xyz, skipped.“ (xyz is at the example 5). We can now quit this error message by clicking „OK“. Look at picture 29. If we look now into the library, there is ONLY one symbol „TRANSISTOR-P-MOSFET-ENHANCEMENT“. The other symbol, with the blank at the name „TRANSISTOR-P-MOSFET ENHANCEMENT2“ will not be displayed and is not accessible by the symboleditor. So you cannot fix the error, but will get the error message every time you add the library to the library list.

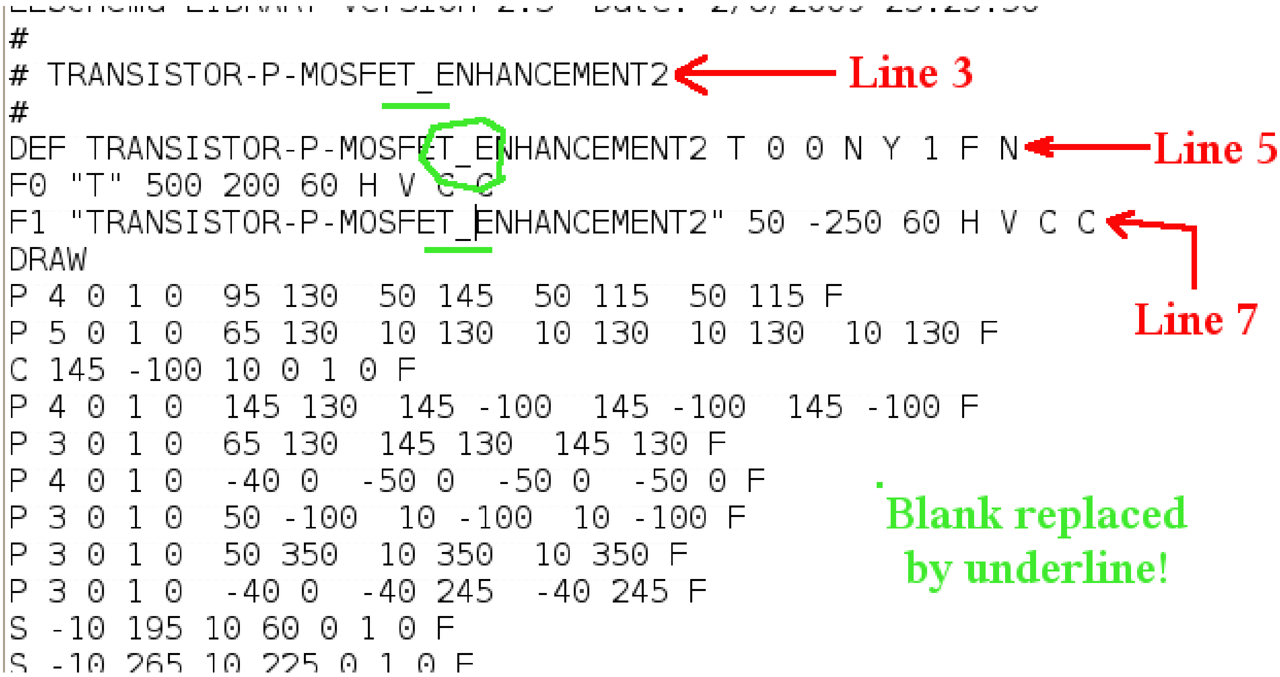
A way, to get rid of this error, is to save all symbols one by one into a new library and delete the old library. By this way, you will lost the complete symbol with the mishap at the name, and it is a lot of work.
But there is a simpler way, who also allowes the rescue of the symbol:

Take a simple text editor and look into the file „Test-Library.lib“. See this at picture 30.

```
EESchema-LIBRARY Version 2.3  Date: 2/8/2009-23:25:50
#
# TRANSISTOR-P-MOSFET ENHANCEMENT2          Line 3
#
DEF TRANSISTOR-P-MOSFET ENHANCEMENT2 T 0 0 N Y 1 F N   Line 5
F0 "T" 500 200 60 H V C C
F1 "TRANSISTOR-P-MOSFET ENHANCEMENT2" 50 -250 60 H V C C
DRAW                                                   Line 7
P 4 0 1 0  95 130  50 145  50 115  50 115 F
P 5 0 1 0  65 130  10 130  10 130  10 130  10 130 F
C 145 -100 10 0 1 0 F
P 4 0 1 0  145 130  145 -100  145 -100  145 -100 F
P 3 0 1 0  65 130  145 130  145 130 F
P 4 0 1 0  -40 0  -50 0  -50 0  -50 0 F
P 3 0 1 0  50 -100  10 -100  10 -100 F
P 3 0 1 0  50 350  10 350  10 350 F
P 3 0 1 0  -40 0  -40 245  -40 245 F
S -10 195 10 60 0 1 0 F
S -10 265 10 225 0 1 0 F
S 10 -115 -10 30 0 1 0 F
S -10 365 10 265 0 1 0 F
X S 3 350 -100 300 L 50 50 1 1 U
X D 1 350 350 300 L 50 50 1 1 U
X G 2 -350 0 300 R 50 50 1 1 U
ENDDRAW
ENDDEF
#
```

Picture 30: Symbol name with space character.

Because of the # at the beginning of line 3 i suggest this will be a remark or commentary, and at line 7 the name is between quote signs. But the „DEV“ at the beginning of line 5 suggests, that this is a device definition. At the end of the line is something written only separated by blanks. If the blank is used to recogniced the end of the name, here will occur an error. Now we remove the blank at the name and replace it by a „_“, so the new name is „TRANSISTOR-P-MOSFET_ENHANCEMENT2“ . For consistency i do this change at the name also at line 3 and line 7. Perhaps no catastropical event will occur, but be better carefull...... You can see the result at picture 31. Adding the library to the library list at Eeschema will now not create an error message. Both symbols, „TRANSISTOR-P-MOSFET-ENHANCEMENT“ and „TRANSISTOR-P-MOSFET_ENHANCEMENT2“, are now visible and normal useable.



```
#
# TRANSISTOR-P-MOSFET_ENHANCEMENT2
#
DEF TRANSISTOR-P-MOSFET_ENHANCEMENT2 T 0 0 N Y 1 F N
F0 "T" 500 200 60 H V C C
F1 "TRANSISTOR-P-MOSFET_ENHANCEMENT2" 50 -250 60 H V C C
DRAW
P 4 0 1 0  95 130  50 145  50 115  50 115 F
P 5 0 1 0  65 130  10 130  10 130  10 130  10 130 F
C 145 -100 10 0 1 0 F
P 4 0 1 0  145 130  145 -100  145 -100  145 -100 F
P 3 0 1 0  65 130  145 130  145 130 F
P 4 0 1 0  -40 0  -50 0  -50 0  -50 0 F
P 3 0 1 0  50 -100  10 -100  10 -100 F
P 3 0 1 0  50 350  10 350  10 350 F
P 3 0 1 0  -40 0  -40 245  -40 245 F
S -10 195 10 60 0 1 0 F
S -10 265 10 225 0 1 0 F
```


Picture 31: Space character replaced by underline character.

At the Pin-Tool, where you also have to choose a pin-name, this bug cannot happen in this way, because in this case a space or blank charakter will be replaced automtically by an underline character.

Tipp: Adding author and license remark to a symbol library.

Because i guess, that a # at a library file means a remark or a comment, i suggest to use this as a author and lizenz remark.
Experimental, i addet as second last line
„#GNU General Public Licensed by Author uvw xyz dd.mm.yyyy“
to the ~~.lib file by using a text editor. uvw xyz means the name of the author and dd.mm.yyyy the actual date.
Up to now, i did not notice any problems.

This document is published under the General Public License by
Author Bernd Wiebus – dl1eic- at 31th August 2009. Uedem/Germany

GNU

Dipl. Ing. Bernd Wiebus
Weezer Str. 5
47589 Uedem
Germany

Tel. +49-2825-9399977
Tel. +162-6157950 (mob.)

e-mail: bernd.wiebus@gmx.de
dl1eic@darc.de