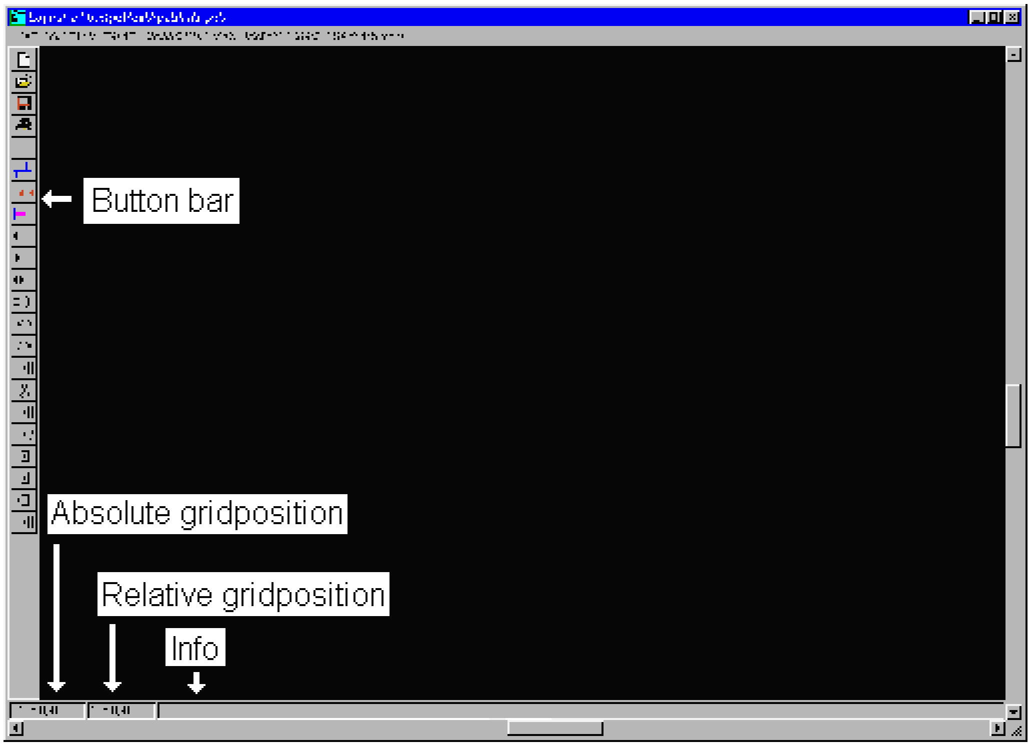
Getting started

Schematic editor PCB elegance

The schematic tool of PCB elegance is a tool to symbols and sheets.

Viewable window



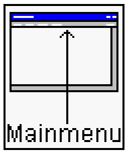
In the viewable window there are four bars visible who have a function. Those four bars are the **Button bar**, **Absolute grid position (0.1 inch)**, **Relative gridposition (0.1 inch) (mils/mm)** and **Info**.

Zero relative cursor

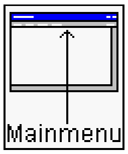
Backup

Backup

New sheet



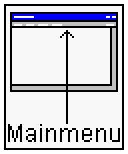
Sub menu **File** menu item **New sheet**



Sub menu **File** menu item **New sheet in new window**

Creates a new sheet file (.sch) in the **sch** subdirectory of the current design.

New symbol



Sub menu **File** menu item **New symbol**

Sub menu **File** menu item **New symbol in new window**

Creates a new symbol file (.sym) in the **sym** subdirectory of the current design.

New sheetsymbol

Sub menu **File** menu item **New sheetsymbol**

Sub menu **File** menu item **New sheetsymbol in new window**

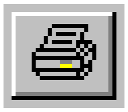
Creates a new sheetsymbol file (.sym) in the **sym** subdirectory of the current design.

Print

Sub menu **File** menu item **Print** menu item

Print in black

Print in color



Press **Print** button

Prints the current file to the printer.

Initialisation file sch.ini

The initialisation file **sch.ini** is used to save some designs parameters. The file will be stored in the designs directory, if there is a design active.

If the schematic editor has been started as stand alone, the search rules for **sch.ini** will be as follows. When the last subdirectory name of the full edit path is **sch** or **sym**, the schematic editor will use the one level up directory. An example:

Symbols pull path: c:\design\test\sym\testsymbol.sym

The schematic editor is looking for the **sch.ini** in the c:\design\test directory. This situation is the case when a symbol or schematic from a design will be opened.

When there is no **sch** or **sym** directory, or the **sch.ini** does not exist, the schematic editor tries to find the **sch.ini** in the same directory as the symbol or sheet.

The following paramaters in the **sch.ini** are used:

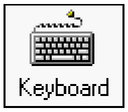
[Settings]

WindowWidth	The width of the windows
WindowHeight	The height of the windows
WindowStartX	Origin X of the windows (0,0 = left top)
WindowStartY	Origin Y of the windows
SymbolDialogStartX	Origin X of the add symbol dialog window
SymbolDialogStartY	Origin Y of the add symbol dialog window
Repeat mode	(0 = off,1 = on)
LineSelectMode	(0 = normal, 1 = extended)
SelectMode	(0 = Appending,1 = Replacement)
BackGroundColor	24 bit RGB color (Stored as 32 bit)
WireColor	24 bit RGB color (Stored as 32 bit)
BusColor	24 bit RGB color (Stored as 32 bit)

BusConnectionColor	24 bit RGB color (Stored as 32 bit)
GlobalConnectionColor	24 bit RGB color (Stored as 32 bit)
JunctionColor	24 bit RGB color (Stored as 32 bit)
NetLabelColor	24 bit RGB color (Stored as 32 bit)
InstanceRefTextColor	24 bit RGB color (Stored as 32 bit)
InstanceValueTextColor	24 bit RGB color (Stored as 32 bit)
SymbolPinColor	24 bit RGB color (Stored as 32 bit)
SymbolPinBusColor	24 bit RGB color (Stored as 32 bit)
SymbolPinTextColor	24 bit RGB color (Stored as 32 bit)
SymbolPowerPinTextColor	24 bit RGB color (Stored as 32 bit)
SymbolPinBusTextColor	24 bit RGB color (Stored as 32 bit)
SymbolLineColor	24 bit RGB color (Stored as 32 bit)
SymbolRectColor	24 bit RGB color (Stored as 32 bit)
SymbolTextColor	24 bit RGB color (Stored as 32 bit)
SymbolArcColor	24 bit RGB color (Stored as 32 bit)
SymbolCircleColor	24 bit RGB color (Stored as 32 bit)
ObjectLineColor	24 bit RGB color (Stored as 32 bit)
ObjectRectColor	24 bit RGB color (Stored as 32 bit)
ObjectCircleColor	24 bit RGB color (Stored as 32 bit)
ObjectArcColor	24 bit RGB color (Stored as 32 bit)
ObjectTextColor	24 bit RGB color (Stored as 32 bit)
ButtonInfoColor	24 bit RGB color (Stored as 32 bit)
GridColor	24 bit RGB color (Stored as 32 bit)

Zero relative cursor

Sub menu **Edit** menu item **Zero relative cursor**

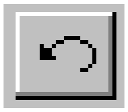


Press **Ctrl z**

The relative grid position will be set to zero.

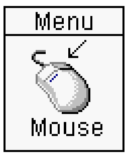
Viewable window

Undo



Press **Undo** button

Press **u**



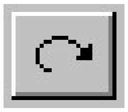
Undo

This function will undo almost all previous actions. Modifying the numbers of parts per package, and modifying pinnumbers package parts can not be undone.

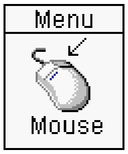
Edit pinnumbers package parts

Edit number of parts per package

Redo



Press **Redo** button



Redo

This function will redo previous undo actions.

View topics

Zoom in

Zoom out

Window based Zooming

Pan window

Window based panning

Return to previous view window

Repaint

View whole design

Change grid

Zoom in

Press **z**

Sub menu **View** menu item **Zoom in**

The **zoom in** function can be used in every possible drawing/moving function.

Window based zooming

Zoom out

Press **Z**

Sub menu **View** menu item **Zoom out**

The **zoom out** function can be used in every possible drawing/moving function.

Window based zooming

Window based Zooming

To zoom in on a window, place the mouse cursor to the left top place of the window. Hold down the **Ctrl** key, than press and hold down the left mouse button. Move the mouse cursor in the right bottom direction of your window. After releasing the **Ctrl** key and the left mouse button zooming in will take place.

To zoom out, use the previous function, but now move the mouse cursor in the left top direction. The non changing rectangle visible is the border of your design. The changing rectangle is the zoom out window. After releasing the **Ctrl** key and the left mouse button, zooming out will take place.

The **window based zooming** function can be used in every possible drawing/moving function.

Pan window

	Press ⇐,⇒,⇑,⇓
	Press **x**
	Press **Shift** and move the mouse the window border
Window	Use the scrollbars

When pressing the **x** key, the window will be panned around the current mouse position, and the mouse position will be moved to the window center.

The **pan window** function can be used in every possible drawing/moving function.

Window based panning

Window based panning

There is a function available to view a different part of your design(special window for panning). To enter this function, hold down the **Ctrl** key, than press and hold down the **right** mouse button. The non changing rectangle visible is the border of your design. The changing rectangle is the viewable window. After releasing the **Ctrl** key and the **right** mouse button panning will take place.

The **Window based panning** function can be used in every possible drawing/moving function.

Return to previous view window

Press **v**

Previous view

Return to a previous view.

The **Return to previous view window** function can be used in every possible drawing/moving function.

Repaint

Press **F5**

Repaint

The whole window will be repainted.

The **Repaint** function can be used in every possible drawing/moving function.

View whole design

Press **Shift F8**

View whole design

The window view will be scaled that the whole design will fit.

The **View whole design** function can be used in every possible drawing/moving function.

Change colors

Sub menu **View** menu item
Change colors

The color settings can be modified in the next dialogbox. The color settings will be copied into the **sch.ini** initialisation file. This file is stored into the directory of the project.

To use those sch colors for new designs, copy this **sch.ini** file to main directory. Whenever a new design is created this **sch.ini** file in the main directory will be copied to the directory of the new design.

Initialisation file sch.ini

Load default colors (Black background)

Load default colors (Gray background)

Load default colors (Black background)

Sub menu **View** menu item **Load default colors (Black background)**

The default color settings with a black background will be loaded.

Initialisation file sch.ini

Load default colors (Gray background)

Sub menu **View** menu item **Load default colors (Gray background)**

The default color settings with a gray background will be loaded.

Initialisation file sch.ini

Short cuts

The most important functions of the schematic/symbol editor have a short cut key (Accelerator). Those keys can be modified by editing the **sch.ini** file, section **[Keys]**.

Change grid

Press **Ctrl g**

Sub menu **Grid** menu item
(**0.1,0.2,1.0)**

Pressing **Ctrl g** will switch the grid **0.1** and **1.0**.

Changing the grid is possible in every drawing/moving function.

View/hide grid

Press **g** (View/hide grid)

View/hide grid

Selection/deselection objects

To select an object, place the mouse cursor above the object, and press and hold the left mouse button. A rectangle will mark the selection window. There are two selection modes available. The first and default selection mode is the **Replacement**, and the second selection mode is the **Appending**.

The **Replacement** selection mode means, every time a new selection rectangle is drawed the previous objects selected will be unselected. When pressing down the left shift key together with the left mouse button it is possible to use more than one selection at a time.

The other selection mode is the **Appending selection mode**. In this mode every object which is selected stays selected, until the **Deselect all** function is executed. To deselect an object press the left mouse and place the selection rectangle around this object again.

To change the selection mode use the **Replacement** or **Appending** in the **Selection mode** section of the menu.

Deselect all

Line select mode

Line select mode

Sub menu **Edit** menu item

Line select mode (normal)

Line select mode (extended)

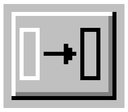
Line select mode can be switched between **normal** and **extended** mode. In **normal** mode a line (wire/bus/line) will be selected if one or two line endpoints are in the selection window.

In **Extended** mode a line (wire/bus/line) will be selected if whatever piece of the line is in the selection window.

Select components

Deselect all

Deselect all



Press **Deselect all** button

Press **F2**

Deselect all

Deselect all function.

Edit component parameters

Press **e**

Edit component parameters

In the next dialogbox the parameters of the **selected** component can be edited.

The following parameters can be changed:

Reference

Value

Part nr

Geometry

Part description

Package part nr

Placing option

Add component

Reference

The reference of the component.

Value

The value of the component.

Part nr

The part nr of the component

Geometry

With the **Select geometry** button a geometry can be selected. In the top window the geometry directories and libraries are visible, and in the bottom window the geometries stored in that directory or library. By clicking on a directory/library item geometries in the directory/library will be listed in the bottom window. By clicking on a geometry the geometry will be visible in a new window to the right. By clicking on the **OK** button the geometry will be selected.

»

Initialisation file sch ini

Part description

The part description is copied from the symbol description.

Package part nr

The package partnumber can be selected.

»

Edit numbers of parts per package

Placing option

When the **Placing option** checkbox has been marked, the component will be marked as a placing option. Behind the component value a string **(*)** will added to indicate the placing option.

Protect symbols

Sub menu **Edit** menu item **Protect symbols**

Selected symbols will be protected. Protected symbols can not be selected.

Unprotect symbols

Edit symbolnames

Edit gate/pin swap

Sub menu **Edit** menu item **Edit gate/pin swap**

Edit gate/pin swap

In the next dialogbox the gate/pin swap parameters of the symbol can be edited. Editing gate swap info for devices like 7400 (Four nand gates) is not necessary.

In the first small editbox a group code (1 to 15) should be edited. In the right large editbox the pinnames of a gate should be edited. Every pinname will be separated by a komma, pin(s)/pinbus(ses) which are swappable should be enclosed by parentheses.

A example:

(3,4,5),(A[0:7]) : Pins 3,4 and 5 can be swapped against each other, and the eight pins inside the pinbus A[0:7] can be swapped against each other.

Edit pinbus reorder

A few examples:

Pin swap example

Gate swap example 1

Gate swap example 2

Pin swap example

7400 TTL device with two pins: IN1 and IN2 which should be swappable:

The gate/pin swap dialog window should be filled like:

	Code	Pins
Line 1	1	(IN1,IN2)

Gate swap example 1

74245 TTL device with eight inputs and outputs. Every Input/Output combination can be swapped against any other seven Input/Outputs. This means there are eight gates, each with two pins.

The gate/pin swap dialog window should be filled like:

	Code	Pins
Line 1	1	2,18
Line 2	1	3,17
Line 3	1	4,16
Line 4	1	5,15
Line 5	1	6,14
Line 6	1	7,13
Line 7	1	8,12
Line 8	1	9,11

All the pins in lines with the same **code** numbers can be swapped. This means the number of pins in each line should be the same, and the numbers of pins enclosed by parentheses should be the same.

Gate swap example 2

74244 TTL device with two sets of four inputs and outputs. Every Input/Output combination of a set can be swapped against any other three Input/Output. This means there are two sets of four gates, each with two pins.

The gate/pin swap dialog window should be filled like:

	Code	Pins
Line 1	1	2,18
Line 2	1	4,16
Line 3	1	6,14
Line 4	1	8,12
Line 5	2	11,9
Line 6	2	13,7
Line 7	2	15,5
Line 8	2	17,3

The four gates in lines 1 to 4 have **code** number one, which means they are swappable. Also the four gates in lines 5 to 8 with **code** two are swappable.

Edit pinbus reorder

Add pinbus reorder -> **Pinbus**

Delete pinbus reorder -> **Pinbus**

Edit pinbus reorder -> **Pinbus**

In the next dialogbox the pinbus reorder parameters of a **selected** components pinbus can be edited.

Pinbus reorder means the sequence of pins will be reordered. Such a pinbus reordering of pins is necessary when pins in this pinbus are swapped. The numbers in a pinbus reorder are index numbers.

For example:

Pinbus: 3,9,45,12,41,89,23,63 with pinbus reorder 0,1,2,3,4,5,6,7

Pins 12 and 23 should be swapped.

The pinbus reorder would be 0,1,2,6,4,5,3,7 (Index 3 and 6 are swapped)

Add pinbus

Gate/pin swap

Unprotect symbols

Sub menu **Edit** menu item **Unprotect symbols**

All protected symbols will be unprotected. Protected symbols can not be selected.

Protect symbols

Edit symbolnames

Export text

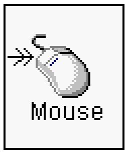
Sub menu **Edit** menu item **Export text**

The text of **selected** pins, powerpins, pinbusses or standard text, will be made visible in a dialogbox. This text can be used to import into a wordprocessor.

Edit any text

Press **e**

Edit any text



Double click on text

In the next dialogbox **selected** text can be edited.

The **selected** text can be:

- Reference name
- Value name (Only for editing sheets)
- Powerpin
- External connection
- Netlabel
- Normal text

Search for any text

Sub menu **Edit** menu item **Search for any text**

Press **s**

---

0 In the next dialogbox the search text can be edited. After pressing the **OK** button the schematic editor tries to find this text, and will pan the view window arround this text. The text itself will be selected. The text can be:

- Symbol name
- Value name (Only for editing sheets)
- Reference name
- Pinnumber
- Any pinnumber in a pinbus
- Netname powerpin
- Any pinnumber in a powerpin
- External connection
- Netlabel
- Normal text

Edit number of parts per package

Sub menu **Edit** menu item **parts** -> **count**

Edit the number of symbols in a device.

Edit pinnumbers package parts

Edit pinnumbers package parts

Sub menu **Edit** menu item **parts** -> **pins**

In the next dialogbox there is a listbox and editbox. The listbox shows the pinnames of the symbol. In the right editbox the pin numbers for each part in the device should be edited, starting with the first part. The pin numbers should be separated by comma's. Spaces are not allowed.

Edit numbers of parts per package

Edit symbolnames

Sub menu **Edit** menu item **Symbolnames**

When editing a symbol, the symbol properties can be changed with this function. The top editbox represents the symbolname. This symbolname can not be changed, because the symbolname and the filename of the symbol are the same. If the symbolname should be changed, simple save the symbol under another name. The checkbox to the right of the editbox defines if the symbolname is visible when the symbol is used in a sheet.

The second editbox contains the interface name. Normally the interface name is the same as the symbolname. When the symbol is a part of a bigger symbol (see Multiple symbols), the interface name will be different. The checkbox to the right of the editbox defines if the symbol is part of a multiple symbol.

The third editbox defines the initial reference name. This initial reference name should always end with a quotation mark **?**. This quotation mark is necessary for annotation. The checkbox to the right of the editbox defines if the initial reference name is visible when the symbol is used in a sheet. In the fourth editbox a description (help) cab be edited.

Annotation

The checkbox at the bottom defines if the symbol is protected in a sheet. Protected symbols in a sheet can not be selected.

Protect symbols

Unprotect symbols

Multiple symbols

When the pincount of a symbol is very high, there is a possibility to split the symbol into two or more separate symbols.

Every symbol contains a certain amount of pins. For every symbol some properties must be edited. To edit the properties of symbol use the **Symbol names** of the **Edit** menu. The **Interface name** should be the same for all the symbols, and the **multiple symbols** checkbox should be marked.

Clear references

Sub menu **Edit** menu item **Clear references**

All the **selected** symbols with numbered references will be reset to xxx?. For example reference **ABC234** will be reset to **ABC?**. References names who have a quotation mark as last character will not be changed.

Annotation

Check sheet

Sub menu **Edit** menu item **Check**

The current sheet will be checked for errors. Possible errors are :

- Not connected busconnections.
- Not connected external connections.
- External connections.must be unique.
- External connections directy connected to a busconnection.
- Wires directly connected to a bus.
- Busconnections not connected properly to a wire/bus.
- In a sheet the pins of a symbol are not connected to a wire/bus endpoint.
- Netlabel is not connected to a wire or bus.

Check symbol

Sub menu **Edit** menu item **Check**

The current symbol will be checked for errors. Possible errors are :

- Double pinnumbers

Annotation

Edit pin normal symbol

Press **e**

Edit text

In the next dialogbox parameters of **selected** pins can be edited. Connection, type, pinname,labelname and visibility can be modified. After clicking the **OK** button the pin will be changed.

Edit sheetsymbol pin

Press **e**

Edit text

In the next dialogbox parameters of **selected** pins can be edited. Connection type and labelname can be modified. After clicking the **OK** button the pin will be changed.

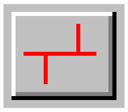
Edit pinbus

Press **e**

Edit text

In the next dialogbox parameters of **selected** pinbusses can be edited. Connection, type, pinname,nr pins and labelname can be modified. After clicking the **OK** button the pinbus will be changed.

Add wire



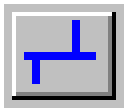
Press **Add wire** button

Press **w**

Add wire

Add wire.

Add bus



Press **Add bus** button

Press **b**

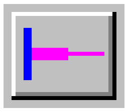
Add bus

A bus is a collection of a number of signals. The amount of signals in a bus depends on how the bus is named. For instance if the bus is a databus with the signals AD0,AD1,AD2,AD3,AD4,AD5,AD6,AD7 the name of the bus would be AD[0:7]. When a bus has such a name the amount is signals in a bus is limited to eight is this case. A bus can also contain an unspecified number of signals, for example a possible name could be SYSCON.

Via busconnections signals are connected to a bus. The signal names that are connected to a bus must be unique. When a bus is directly connected to a pinbus, the pinbus should have a name with a number range. The amount of signals within this number range should be the same as the pinbus signal count.

It is not allowed to connect a signal to a bus once (unused signal). For example if the signal MEMR is connected via a busconnection to the bus MEMCON, the same signal should be connected to this bus on another place. The reason for this is to avoid some mistakes, when connecting signals to a bus. In the previous example signal MEMR is used to connect to a bus. Suppose you have typed MEMR wrong (MEM), for the first connection, and the second signal connection name is typed with MEMR. The result of this is, that the two signals are not connected to each other. As a result of this check an error will be displayed.

Add busconnection



Press **Add busconnection** button

Press **B**

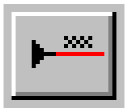
Add busconnection

A busconnection is a small symbol that is used to connect a bus with a wire or pinbus.

After the function is activated the busconnection can be placed. The thick part of the busconnection should be placed against the bus, and the smallest part to the wire or pinbus end point.

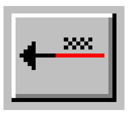
By pressing the right mouse button the busconnection can be mirrored.

Add external connection



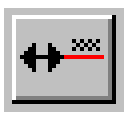
Press **Add external input connection** button

Add external connections -> **Input**



Press **Add external output connection** button

Add external connections -> **Output**



Press **Add external I/O connection** button

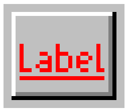
Add external connections -> **I/O**

An External connection is used to connect signals or busses from different sheet in a hierachical design. An external connection is a Input, Output or I/O. The name used for the External connection should be the same, as the pin name in the sheetsymbol in the above sheet.

After the function is activated a dialogbox is visible. You have to fill in a name for the external connection. After the name has been typed the external connection can be placed. The external connection should be placed at the endpoint of a wire or bus.

By pressing the right mouse button the external connection can be mirrored.

Add netlabel



Press **Add label to wire/bus** button

Press **n**

Add label to wire/bus

A netlabel is a text string that identifying the wire or bus. When a wire is connected to a bus via a busconnection, the wire needs a netlabel. A netlabel is always required for a bus. When wires or busses are drawn with two or more lines, only one line of them needs this netlabel.

To add a netlabel to a wire or bus, **select** the wire or bus first. After the wire or busses has been selected use this function. After the function is activated a dialogbox is visible. You have to fill in a name for the netlabel. After pressing the **OK** button, the netlabel can be placed. There is a help line visible, to show at which endpoint of the wire or bus the netlabel is connected.

Add incremental netlabels to wires

Add incremental netlabels to wires

Press **N**

When incremental netlabels are required for a number of wires, this function can be used. First select the wires, and then use this function.

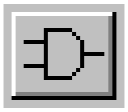
In the dialogbox the name of the netlabel should be filled in.

Add netlabel + wire

Add netlabel + wire

In the next dialogbox the names of the netlabels should be filled in. For every line in the dialogbox a wire and netlabel will be created.

Add symbol



Press **Add symbol** button

Press i

Add symbol

A symbol is a graphical representation of a component (Resistor, capacitor, IC).

A symbol can be imported from a library or a symbol directory.

In the top window the symbol directories and libraries are visible, and in the bottom window the symbols stored in that directory or library. By clicking on a directory/library item symbols in the directory/library will be listed in the bottom window. In the edit box at the bottom of the dialogbox a symbol description is shown for the selected symbol. By clicking on a symbol the symbol can be placed on the schematic. By pressing the **right mouse button** a number of timer the symbol will rotate/mirror. By pressing the **left mouse button** the symbol will be placed. The dialogbox remains visible, and will disappear by clicking on the **Cancel** button.

Initialisation file sch ini

Add component

Add database component -> **Select component**

Adding components can be a lot quicker than adding symbols, because after a symbol has been added some parameters have to be changed. Those parameters are the **geometry** and **value**.

The standard components like resistors, capacitors, 74xx range of IC and some other components can be added directly.

In the series of pulldown menus the component family can be selected. In the next dialogbox the component can be selected and by pressing the **OK** button the symbol can be placed. In this dialogbox there are two listboxes. The top listbox shows the system components, and the bottom listbox the components added by the user.

The information for this component database is put into two files in the base directory of PCB elegance. The names for those files are **compmenu.txt** and **comp.txt**.

Compmenu.txt file

Comp.txt file

Compmenu.txt file

This is a text file containing menu items. The first character of each line will be used for decoding.

First character:

;   Line will be ignored (Also empty lines)

$ The number directly after **$** (Code1 in the **comp.txt** file) is the main index in the component pulldown menu.

The next string will be visible in the menu.

The next number (Code2 in the **comp.txt** file) will not be used if there are further pull down menus. If there are no further pulldown menus the number will be used.

The next string will be visible in the second pulldown menu.

^   The next number is the Code2 in the **comp.txt** file

The next string will be visible in the second pulldown menu.

There are two options to edit this file. The first option is to add at the bottom of the file the new components. The second option is insert lines between the standard menu items. The **Code1** numbers 1 to 8 are reserved, and the **Code2** numbers used by the PCB elegance are ending on a zero. The user can insert other lines, but the **code2** numbers must end on 1 to 9. <u>Do not enter new **Code2** members, because there are reserved for future implementations.</u>

Comp.txt file

The comp.txt is a text database file. Empty lines, or lines starting with a **;** are ignored. Every line contains seven items.

Item1	Symbolname
Item2	Code1 definition
Item3	Code2 definition
Item4	Partnumber (Optional)
Item5	Value
Item6	Geometry
Item7	Description

When pulldown menu item has been selected with corresponding Code1 and Code2 numbers, the program makes a subselection from the comp.txt file. This subselection will be displayed into a dialogbox, and the component can be selected and included into the schematic.

Add other objects

Add line

Add rectangle

Add rectangle (4 lines)

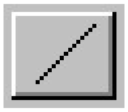
Add circle

Add arc

Add text

Add numbers incremental

Add line



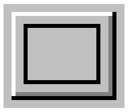
Press **Add line** button (Only when editing symbols)

Press **l**

Add special -> **Line**

Add a line.

Add rect (4 lines)



Press **Add rect** button (Only when editing symbols)

Press **r**

Add special -> **Rect (4 lines)**

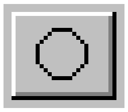
Add a rectangle. This rectangle is build up with four lines. This can be useful when the rectangle should be changed, because lines can be dragged.

Add rect

Add special -> **Rect**

Add rectangle.

Add circle



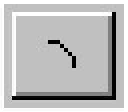
Press **Add circle** button (Only when editing symbols)

Press **C**

Add special -> **circle** -> **Select circle**

Add a (partial) circle.

Add arc



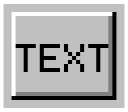
Press **Add arc** button (Only when editing symbols)

Press **a**

Add special -> **Arc**

Add an arc.

Add text



Press **Add text** button (Only when editing symbols)

Press **t**

Add special -> **Text**

Add text

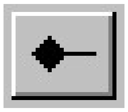
If the textstring equals **$DATE** the textstring will be replaced by the current date.

Add numbers incremental

Add special -> **Numbers incremental**

Add a range of numbers.

Add pin



Press **Add pin** button

Press **a**

Add pin

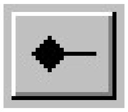
After the function is activated a dialogbox is visible. In this dialogbox one or more pins can be added to the symbol. The dialogbox consists of three edit boxes and two series of radiobuttons. The first editbox is the edit box for the pin numbers (pinnames). The second editbox is the editbox for the pin text. This pintext will be added as normal text. The third editbox is the edit box for the pin labels. This editbox is only relevant when adding pins for a sheet symbol. When adding pins for a normal symbol you do not have to fill this editbox. With the two series radiobuttons, the electrical direction and electrical type can be selected, for all pins that will be added.

To add a powerpin click the ‘Pinname is powernet’ checkbox. The pinname edited will be the net in the schematic. (These powerpins can be used for GND symbols).

After filling the dialogbox click **OK**, and the pin(s) can be placed. To mirror or rotate the pin press the **right mouse button**.

Add pinbus

Add sheetsymbol pin



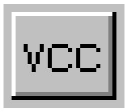
Press **Add pin** button

Press **a**

Add pin

After the function is activated a dialogbox is visible. In this dialogbox one or more pins can be added to the sheetsymbol. The dialogbox consists of three edit boxes and a serie of radiobuttons. The first editbox is the edit box for the labelnames. The second and third editbox is the edit box the incremental placement of the pins. With the serie of radiobuttons the electrical direction can be selected, for all pins that will be added. After filling the dialogbox click **OK**, and the pin(s) can be placed. To mirror or rotate the pin press the **right mouse button**.

Add powerpin



Press **Add powerpin** button

Press **p**

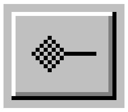
Add powerpin

When adding pins to a symbol who are connected to power (+5V,GND), there is another option adding these pins, the so called powerpins (power pin text). For example for a 74LS00 device pin 7 should be connected to ground, and pin 14 to the VCC. Instead of adding two pins, two powerpins can be added.

When adding powerpins two items should edited, the netname and the powerpin number(s). The netname specified will be used as a standard net for the whole design. The powerpin number(s) specified consists of one or more pinnumbers separated by commas. If necessary two or more powerpins with same nets and different pinnumbers can be added.

To add a powerpin use one of the above three actions. After the function is activated a dialogbox is visible. In this dialogbox two editboxes are visible. In the first editbox the netname has to be specified, and in the second editbox the pinnumbers separated by comma’s. After filling the dialogbox click **OK**, and the powerpin can be placed. To rotate the powerpin text press the **right mouse button**.

Add pinbus



Press **Add pinbus** button

Press **P**

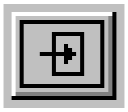
Add pinbus

A pinbus is special pin definition, to replace a series of standard pins. For example if a numbers of pins for a databus should added to the symbol, a pinbus can be used instead of adding every pin of the databus separately. In digital designs with a CPU and memory devices pinbusses are very useful.

To add a pinbus use one of the above three actions. After the function is activated a dialogbox is visible. The dialogbox consists of four edit boxes, and two series of radiobuttons. The first editbox is the edit box for the pin numbers (pinnames) separated by commas. The second editbox is the edit box for the pin label. The third editbox is the editbox for the pin text. This pintext will be added as normal text. In the fourth editbox the amount of pins has to be filled in. With the two series radiobuttons the electrical direction and electrical type can be selected, for the pinbus that will be added. After clicking **OK**, the pinbus can be placed. To mirror or rotate the pinbus press the **right mouse button**.

The amount of pinnumbers separated by commas and the value in the pin count editbox has to be the same. There is a maximum of **64** pins for a pinbus.

Open subsheet



Press **Open subsheet** button

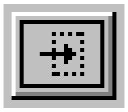
Edit sheet selected symbol

When a sheetsymbol is **selected** and this function is executed, the current design will be left, and the sheet related to this sheetsymbol will be opened.

Goto higher sheet

Hierarchical designs

Open sheetsymbol



Press **Open sheetsymbol** button

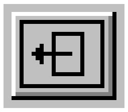
Edit selected sheetsymbol

When a sheetsymbol is **selected** and this function is executed, the current design will be left, and the sheetsymbol will be opened.

Goto higher sheet

Hierarchical designs

Goto higher sheet



Press **Goto higher sheet** button

Goto higher sheet

Go back to a higher sheet. When the **Open subsheet** or **Open sheetsymbol** function was used to open the current sheet or sheetsymbol, **Goto higher sheet** will return to the previous sheet

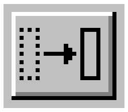
Hierarchical designs

Hierarchical designs

A hierarchical design is a design with more that one sheet. To be able to use more than one sheet an interface has to been used to connect the different sheets with each other. The interface used, consists of sheet symbols in one sheet, and external connections on other sheets. To make this more understandable an example will be used.

In a hierarchical design there is always a start sheet, the so called top sheet. In this top sheet, sheet symbols are included. Every sheetsymbol represents an other (sub)sheet. The pins defined in the sheetsymbol represents the external connections of the subsheet. The label name of a pin in the sheet symbol must be the same, as the name of the external connection in the subsheet. Also the pin type (Input,Output,I/O) must be the same. Every subsheet can also contain sheetsymbols which represents further subsheets.

Move objects



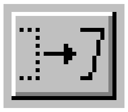
Press **Move** button

Press **m**

Move

Move **selected** objects. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

Drag objects



Press **Drag** button

Press **d**

Drag

Drag **selected** objects. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

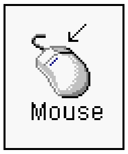
Rotate objects

Press **R**

Rotate

Rotate **selected** objects 90 degrees counter clock wise.

Mirror objects

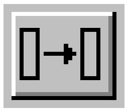


Mirror X

Mirror Y

Mirror **selected** objects in X or Y direction.

Copy objects



Press **Copy** button

Press **c**

Copy

Copy multiple -> **Select number (2..10)**

With this function **selected** objects can be copied once, or 2..10 times. By pressing and keep down the **shift** key and moving the mouse cursor, the moving center will change.

Copy objects to clipboard

Sub menu **Edit** menu item **Copy objects to clipboard**

Press **Ctrl ins**

Selected objects will be copied to the clipboard. This function will work when editing symbols and sheets.

Paste objects from clipboard

Paste objects from clipboard

Sub menu **Edit** menu item **Paste objects from clipboard**

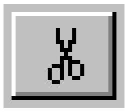
Press **Shift ins**

Paste objects from clipboard

Objects that previously had been copied to the clipboard will be pasted in the current design. This function will work when editing symbols and sheets.

Copy objects to clipboard

Delete objects



Press **Delete** button

Press **Del**

Del

Delete **selected** objects.

Netlabels connected to selected wire(s) or bus(ses), will also be deleted.

Undo

Unselect objects

Unselect -> **Objects**

Unselect objects.

Select only

Select only -> **Objects**

Select only one object type.

Edit symbol

Edit <symbol>

The symbol of the **selected** component can be edited with this function. After closing the symbol editor, the Schematic editor will reload the symbols.