

Layout CAD



**Eagle[®] commands and
how to do it in TARGET 3001!**

Chapter 1: Eagle (R) vs. TARGET 3001!

Eagle (R) vs. TARGET 3001!

This chapter gives an overview of all Eagle (R) commands and its corresponding functions in TARGET 3001! as far as they are needed/exist. This overview is made for an experienced Eagle-user who is migrating to TARGET 3001! or is using both systems parallel. If there remain questions or if there arise problems, please call us from 8.00 am - 4.00 pm CET or send us a mail: target@ibfriedrich.com. Your existing Eagle projects can easily be converted to TARGET 3001!, your Eagle component libraries also. Please have a look at http://www.ibfriedrich.com/english/engl_convert.htm.

* Eagle is a trademark of CADSoft in Pleiskirchen, Germany.

ADD

Insert a component into a drawing.

In TARGET:



Press keyboard key [Ins] or click on this Icon:



Find the appropriate component within the library browser and insert it to the desired place. In The TARGET library browser you can easily browse through components visibly. additional methods of search can be used. You can call for data sheets or call for prices. In TARGET 3001! a component after the insertion of its symbol or package can be edited independently from the library. It is no problem e.g. to move the controlling head of bus driver 74244 downwards, if it seems necessary for better display. Or quickly make Pin 1 of a ICs a square form or bend a pin/pad of a transistor to lead a signal track through.

ARC

Drawing of toruses.

In TARGET:



Click on the icon:  and  and draw a "torus". A doubleclick on the drawn torus-line opens the "Change Torusses" dialogue in which you can edit the torus.

ASSIGN

Setting of Function Keys (keyboard)

In TARGET:

The Function keys are already set to the most useful commands and can not be individually set (programmed):

- [F1] Help
- [F2] Library Browser
- [F3] Cross Probe, switch between schematic and layout
- [F4] Set the Coordinate-unit (mil, inch, mm, um)

TARGET 3001!

Chapter 1: Eagle (R) vs. TARGET 3001!

- [F5] Switch the visibility of the grid (invisible, dots, lines)
- [F6] Switch "snap on grid"
- [F7] Zoom image full size (win fit)
- [F8] Switch the crosshair (invisible, 90°, 90°+45°)
- [F9] Start Simulator (Schematic) or start Autorouter (PCB layout)
- [F10] Menu mode



Some other buttons are programmable: sets of layers visible within the layer-tool or different kinds of vias within the via options dialog or different types of signal tracks in "track-options" dialog.

AUTO

Activate the Autorouter .

In TARGET:





Click upon those Icons:  and  or press in PCB view [F9]. Follow the instructions in the dialog. The TARGET Autorouter is working shape based and gridless. Error messages like "Unreachable pad" do not appear. Tracks are placed at most narrow space as far it is possible at all. A "Routing Grid" is not needed and the speed of the router does not decrease at square to the grid-points.

BOARD

Generate Board-File.

In TARGET:

Not needed, when you are creating a "New PCB with Schematic". TARGET keeps schematic and PCB layout

within one file *.T3001. Switch between schematic and PCB, use the following icons:  and .



If you highlight a package in layout view, press [F3] to see the symbol highlighted in schematic and vice versa. The same with highlighted signals (Cross-Probe).

BUS

Drawing of Busses in the schematic.

In TARGET:



Please click on these icons:  and .

In TARGET you can divide busses by different graphical placement and you also can give different labels to them but every bus in TARGET contains all (!) signals.

CHANGE

Change of parameters.

In TARGET:

Desired element click with a single mouseclick, press key [e] for "edit". The corresponding dialogue opens. Or doubleclick on any element you wish to edit. If there are several elements over each other, please press key [s] for "select" as long as the desired element flashes. Now press enter. The desired element remains highlighted. With [+] and [-] you can search in the closer area around the element also. Now press [e] for edition.

CIRCLE

Insert a circle into a drawing:

In TARGET:

Click on the icons  and  or .

CLOSE

Closes an Edition-window.

In TARGET:

Just do like common in WINDOWS. In TARGET you can open several views at the same time (schematic pages, PCB-views). Also in one Window several projects can be opened e.g. for copying modules by the use of the clipboard.

CONNECT

Assignment of pins and pads in the library

In TARGET:

There are different libraries for schematic symbols and for component packages. Any schematic symbol keeps an assignment of a package proposal. The assignment of pin and pad is organized in TARGET by the pin number. You can create additional pin numbers in the schematic and additional pad names in the layout.

You are allowed to use a different package than the one proposed e.g. in the layout use a SMD14 instead of a proposed DIL14 for a component used in the schematic.

COPY

Copy objects

In TARGET:

Highlight the parts you wish to copy and press the icon  or use [Ctrl] + [Ins]. TARGET copies all to the clipboard, not only for paste into other TARGET- projects but also for carrying them into MS-Word, OpenOffice

Chapter 1: Eagle (R) vs. TARGET 3001!


und Corel-Draw using a Vector-Graphic-Format (WMF), ideal for documentation purposes. The images can be scaled without loss and without getting pixel-like.

Using the clipboard it is possible to copy identical channels, which appear multiple. First draw the channel once in the schematic and once in the layout. Now copy the channel once in the schematic and (soon) in the layout. The 7th channel in the layout knows, that he belongs to the 7th channel in the schematic. Please note: Signals being highlighted completely when copied, are taken as local internal signals and get new artificial name (e.g. Sig\$99). Signals, which haven't been caught completely, (from which minimum one segment remained unmarked, are taken as global external signals and keep their names when being copied (e.g. GND).

CUT

Load a group into the paste-buffer.

In TARGET:

Cut into the clipboard. highlight the desired parts of a drawing and click on icon  or press [Shift] + [Del].

To bring Elements just on another layer, please use key [e] and change the layer.

DELETE

Deletion of objects and elements.

In TARGET:

After highlighting press key [Del]. TARGET has a three step delete-function: If there are autorouted tracks highlighted, only those are deleted at first. If you press [Del] again, the manually routed tracks are deleted too. At least the third press of [Del] also removes the components. If you deleted too quick, please use the undo function in menu "Actions".

DIR

Display of Contents


In TARGET:

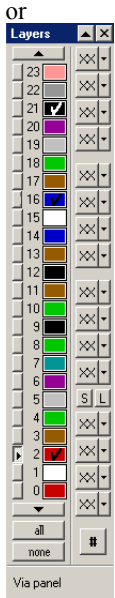
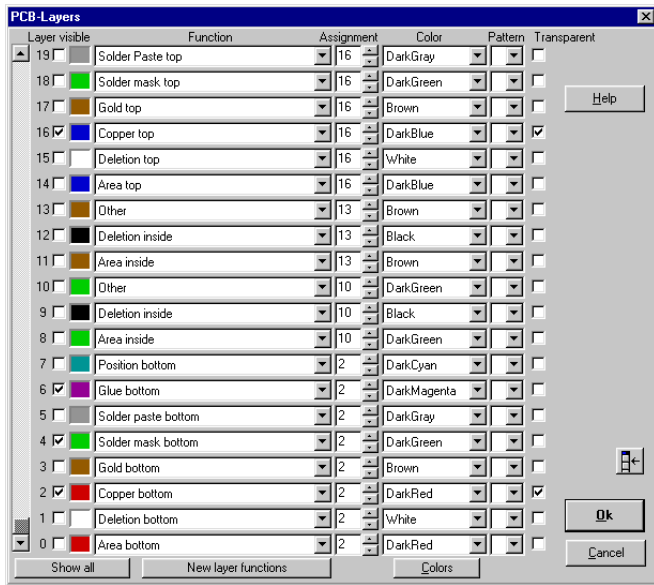
Any action in TARGET follows Windows-compatible dialogues or is controlled by Windows-compatible elements.

DISPLAY

Selection of visible layers.

In TARGET:

Press  or menu "View/Layer-Tool" to get one of the following dialogues:





Ticking or unticking a box in front of the layer rows switches the visibility of layers. In the dialogue on the right buttons can be programmed with sets of layers visible. This helps to toggle quickly between different layer views.

DESIGN RULE CHECK (DRC)
Design Rule Check (Check PCB)

In TARGET:



Click on  and . This function checks your layout (eventually schematic and PCB) against those parameters which are to be defined in the dialogue.

TARGET 3001!

Chapter 1: Eagle (R) vs. TARGET 3001!

EDIT

Load command.



In TARGET:

Open project. Done. A multitude of windows of each project and a multitude of projects in TARGET can be displayed simultaneously. TARGET does not have separate files for schematic and layout like *.SCH + *.BRD in Eagle. All information is gathered within one single file *.T3001. All information for simulation and EMC are in this file also. So there are no internal conversion processes.

ELECTRICAL RULE CHECK (ERC)

Electrical Rule Check (check schematic)

In TARGET:

Click on  and . This function checks your project against those parameters, you have defined within the dialogue.

<hr>

ERRORS

Shows errors, found by the DRC.

In TARGET:

Alle errors found are displayed in a list after end of the check. In the layout you will see those locations, where the DRC announces errors marked by a square with a cross. A double click upon each of those little boxes opens a dialogue which explains what is wrong.

EXPORT

Generate ASCII files



In TARGET:

Please use in menu "File/Input-/Output- Formats" the option "ASCII Interface" for a ASCII export of the complete file.

GATESWAP

Swap equivalent Gates.

In TARGET:

Use this Icon  and then this  to swap identical gates. A swap of gates beyond component-boarders is possible. Two gates are only allowed to be swapped if they are identical and if their swap numbers are equal and greater zero.

GRID

Set display and unit of grid.

In TARGET:

Click on  and  Here do all settings.

GROUP

Selection of several components/elements simultaneously.

In TARGET:

Select an object by single mouseclick. Add elements by holding [Shift] key. Or drag a square (left mouse button held) over the area you wish to group. Such a "Catchwindow" works also in addition to the [Shift] key to add elements highlighted. Please also use the selection assistant in menu "Edit" to e.g. highlight all drillhole pads of diameter 0,6mm to widen them to 0,8mm...

HELP

Display Help for current command

In TARGET:

Press key [F1] to start the interactive Help in any situation.

HOLE

Insert a drillhole to pcb or package.

In TARGET:

Drillholes, if needed, are always defined in unity with a pad. Doubleclick on the click opens dialogue for edition. A pure mounting hole you define by a pad with copper less than the drilling diameter. Also set the pin number zero.

INFO

Display properties of objects.

In TARGET:

Doubleclick upon the object opens a dialogue, which shows the main aspects to be set. If an object can not be touched because lots of objects are lying over each other, please move the cursor over the object and press key [s] for select as long as the desired object flashes. Now press [Enter] to highlight it. Now press [E] for edit to open the corresponding dialogue. Many dialogues offer additional "Properties" buttons which supply further information and allow further settings.

Chapter 1: Eagle (R) vs. TARGET 3001!

INVOKE

Get special gates from components.

In TARGET:

Get special gates from components to the schematic by pressing menu "Components/Import Rests of Components" Any gate is assigned to the property "insert as next" or "Insert as Rest". Normal gates are mostly imported "as next" , supply-symbols mostly "as Rest". Usually first interest belongs to the logic of the circuit and secondly the question of power supply. Supply-symbols are always (!) to be fetched into the circuit and need to be connected. There is no secret connecting of GND signals to the same signal. Also the information which pins are connected to + and - , belongs to the schematic.

JUNCTION

Set a mark for nets which are connected to each other.

In TARGET:

Nodes (Junctions) in the schematic are normally set automatically in TARGET. This happens at a locations where more than two track ends meet. Press key [.] for manual placement of a node in the schematic . A small filled circle is placed, a bit wider than the signal line. If TARGET has set a node too much, just delete it.

LABEL

Label busses and nets.


In TARGET:

For labelling of busses and nets in the schematic, please press key [n], if the cursor exactly is located on a signal element. If now the signal is renamed, its labelling changes too. Because several busses in TARGET are not separated logically, you can just label them using the text-function ["] .

LAYER

Change and definition of layers.

In TARGET:

Use the layer-tool in layout view  to define layers. For visibility please use also the layer toolbar in menu "View".


MARK

Define a mark in the drawing area.

In TARGET:

Set a relative zero by pressing key [Home]. The status line on the bottom of the screen switches the coordinate-property from "ABS" =absolute to "REL" =relative. Relative zero now is the location where the cursor had been placed. A second press of this key gets back the absolute view. For measurements press key [u] to

receive straight distance between two points.

Insert measurement labels by hovering the mouse over the starting point, press key [u], hover the mouse to the end point and press [u] again. If you wish to highlight the measurement label, have a look at this button (under the pointer tool): . If it is set, the measurement display is always highlighted completely. If you release it, single parts can be highlighted.

MENU

Change menu board

In TARGET:

TARGET works with a dynamic menu board. Depending on the view (schematic or layout) the menu board and the tool bar is adjusted.

MIRROR

Mirror Objects



In TARGET:

Highlight the desired elements. the cursor position becomes vertical axis if you now press key [m] for **mirror**. Shall the cursor position become horizontal axis please use key [Shift] + [m]. Please note: SMD components are easily flipped to the opposite copper layer by using this function.

MOVE

Moving of objects and Elements

In TARGET:



Highlight the elements you wish to move. (s.a. GROUP). Press in selection  upon button , to make sure that at click on any element of a desired object either the complete object is highlighted or only the touched part of it. Now click with left mouse button held upon the object and move it towards the desired location. A phantom of the object remains fixed to the cursor, all signal tracks are kept and dragged like rubber. Place the Object as desired.

NAME

Display and change of names

In TARGET:

Doubleclick on a component which carries a name and edit it directly within the opening dialogbox. Also

doubleclick a name for edition. Release button  under button  to separately highlight the fonts. Names can have any number of fonts.


Chapter 1: Eagle (R) vs. TARGET 3001!

NET

Drawing of nets in the schematic.

In TARGET:




Use button , to connect signals between pins. The statusline shows the signalname used during placement. A doubleclick on the signal opens the dialogue for setting the signal properties e.g track width... for later autorouting (button [Properties]). A single click and use of the key [e] for edit does the same.

OPEN

Opens a library

In TARGET:



Either you press function-key [F2] or click on icon  to open the library browser. The same does key [Ins], to insert a component into the schematic. For setting the standard-path to the libraries, please use menu "Window/Settings/Options..."

OPTIMIZE

Putting together of wire segments lying on one line.

In TARGET:

TARGET puts track segments automatically together, when using "Reorganisation" s. menu "File". If you wish to manage this manually, drag [g] a wrong bending to the end of a leg. So the segment gets the length of zero and is deleted immediately. Make all segments/bendings visible by using the quick-draw mode [#] or just click it once to get little squares at any end of a segment. Zoom in, if needed.

PACKAGE

Assignment of a package to a device.

In TARGET:

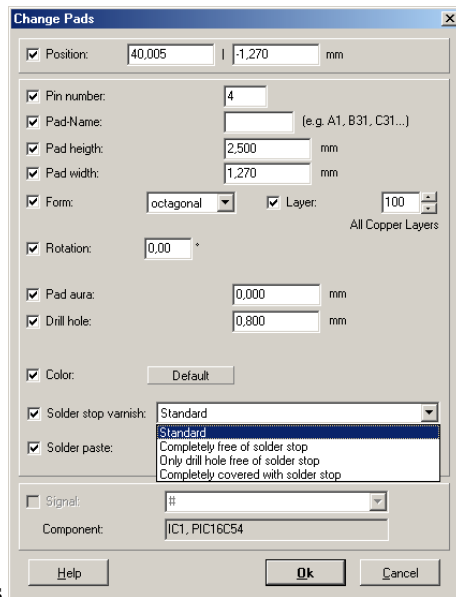
There are two different kinds of libraries in TARGET: symbol-libraries and package libraries. Any component in TARGET is made out of a symbol and a package proposal. The assignment happens when a symbol is drawn and saved to a library. Prior to the storage of a symbol the dialog opens, which requests for the package proposal. It is allowed to vary from this proposal (choose a free package e.g. SMD14 instead of DIL14).

PAD

Placement of pads in packages.

In TARGET:

Pads are needed in layout view, when drawing packages. In layout view press key [1]. A pad is fixed to the cursor. If you now press [o] for options, set them for future pads. Place the pad(s) by mouseclick or by pressing key [1]. For exact placement also use the cursor-keys (arrows). A double click on a pad opens the edit - dialogue



for setting of properties.

PASTE

Insert the contents of the "Past" buffer.

In TARGET:



Insert from clipboard. . Press also [Shift] + [Ins].

To shift an element upon another layer, please use dialog which opens by pressing key [e] and simply change the layer number.

Using the clipboard it is possible to copy identical channels, which appear multiple. First draw the channel once in the schematic and once in the layout. Now copy the channel once in the schematic and (soon) in the layout. The 7th channel in the layout knows, that he belongs to the 7th channel in the schematic. Please note: Signals being highlighted completely when copied, are taken as local internal signals and get new artificial name (e.g. Sig\$99). Signals, which haven't been caught completely, (from which minimum one segment remained unmarked, are taken as global external signals and keep their names when being copied (e.g. GND).

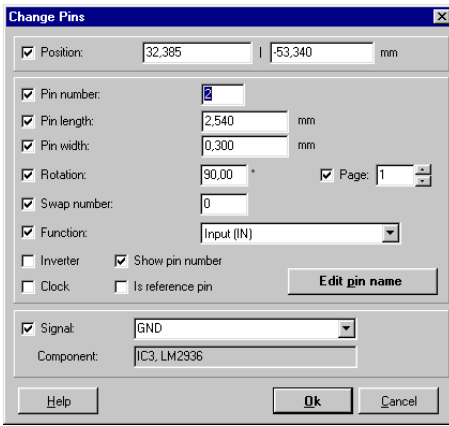
PIN

Define Pins in device.

In TARGET:

Pins are used in the schematic view for definition of connections at symbols. Press key [1]. A Pin is fixed to the cursor. If you now press [o] for options, set them for future pins. Place the pin(s) by mouseclick or by pressing key [1]. For exact placement also use the cursor-keys (arrows). A double click on a pin opens the edit - dialogue for setting its properties.

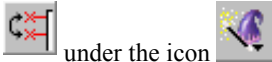
Chapter 1: Eagle (R) vs. TARGET 3001!





PINSWAP

Swap equivalent Pins/Pads, which have an identical swaplevel unequal zero.

In TARGET:

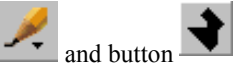




Use button  under the icon  and further a click on both Pins. They must be within the same symbol and show the same swapnumber unequal zero.

POLYGON


Drawing of polygon areas

In TARGET:



Click button  and button . Define by mouseclick the legs of the polygon. Highlight the closed polygon as a whole and assign a signal or a layer to it by pressing key [e] for edit.

In TARGET a polygon is made out of several combined triangles. When highlighting it, look for the position of

the button  (find it under the extension of the pointer tool). Is it pressed, the polygon is always highlighted as a whole. If you release it, only parts of it are highlighted.

PREFIX

Define prefixes for circuit-symbols.

In TARGET:

The prefix corresponding to a component name is to be defined when exporting a symbol. See dialog.

QUIT

Quits working with EAGLE.

In TARGET:

Act like in Windows.

RATSNEST

Computation of air wires and polygons




In TARGET:

Choose in layout view menu "Actions / Compute AirWires (Ratsnest)" or easily press [Ctrl]+[w]. Air wires are dynamic in TARGET. After any modification air wires are computed new. A movement of a component makes all other air wires fade. Only those are kept, who are part of the component. Any movement makes the air wires being computed new and displayed. TARGET searches for closer pads for the signals. This dynamic ratsnest gives really honest help for placement.

RECT

Insert a rectangle into a figure

In TARGET:

Use  and  or  whether you search for a filled or a unfilled rectangle. Choose key [o] for options for the future rectangle. A doubleclick on it or a press on key [e] for edit allows the edition of existing rectangles.

REDO

Repeat command.

In TARGET:

Use , 50 steps possible.


REMOVE

Deletion of files, devices, symbols, packages and sheets.

In TARGET:

Files can be deletet in Windows-Explorer. In TARGET itself all elements highlighted can be deleted using key [Del]. A TARGET circuit always has 100 pages, which can not be deleted. The names of the pages can be set in the hierarchy-browser on the left edge of the screen (Project/Schematic Pages/right mouseclick/Properties).

RENAME

Rename symbols, devices or packages in a library. Release button  if needed.

In TARGET:

TARGET 3001!

Chapter 1: Eagle (R) vs. TARGET 3001!

Doubleclick on the desired text and enter the new name...

REPLACE

Replace an element in PCB layout.

In TARGET:

Delete the desired part and insert a new. if you needed a SMD14 instead of a DIL14, please do the same. If you wish do replace multiple, please use:


In schematic view: Menu "Components", "Search and replace component shape",

in layout view: Menu "Packages", "Search and replace a kind of packages".

RIPUP

"Unwire"already wired signals.

In TARGET:

In layout view: highlight a complete signal and press [Del]. Also search a signal using  and press [Del]. The air wire is visible immediately.

If you are not convinced by a result the router generated, please highlight the complete layout by dragging a square over the layout with left mouse button held. TARGET has a three step delete-function: If there are autorouted tracks highlighted, only those are deleted at first. If you press [Del] again, the manually routed tracks are deleted too. At least the third press of [Del] also removes the components. If you deleted too quick, please use the undo function in menu "Actions".

ROTATE

Rotate objects and elements.

In TARGET:

Highlight object and press key [t]. Press [Shift]+[t] for the setting of any rotation angle. the center of rotation is the cursor position!!!

ROUTE

Wire unwired signals.

In TARGET:

Function key F9 opens the autorouter. In the opening dialog please select the signal(s) to be routed.

RUN

Execute a User Language Program.

In TARGET:

There is no Batch-Script-Language. All functions are already inserted in TARGET (search component, BOM, DXF etc...). If you wish to take influence to you project data later, you can use the open ASCII interface. Those data can be manipulated by the use of your favourite programming language. Read in those data later again.

SCRIPT

Execute an Eagle script file.

In TARGET:

If you wish to take influence to you project data, you can use the open ASCII interface. Those data can be manipulated by the use of your favourite programming language. Read in those data later again.

SET

Modify parameters


In TARGET:

In menu "Window/Settings/Options" many properties can be set. In the hierarchy browser (left edge of the screen) set the parameters within the project properties (right mouseclick). also use the individual dialogs.

SHOW

Display names, elements, and objects.


In TARGET:

Search  signal and toggle by pressing of key [F3] to the schematic or layout view (cross probe). The highlighted element is shown zoomed central on screen.

SIGNAL

Define signals.

In TARGET:

After getting a symbol (this is the schematic related part of a device/component) out of the library browser please wire it by the use of . An acoustic signal confirms the connection. A doubleclick on the signal opens the Edit-dialog. With key [o] for options set properties for future signalel ements.

SMASH

Release of Name and Value Texts.

In TARGET:

TARGET 3001!

Chapter 1: Eagle (R) vs. TARGET 3001!





In the extended pointer mode please release button. This gives opportunity to select any single element, texts also. Doubleclick the text or choose key [e] for edit and follow the dialog.

SMD

Placement of SMD components into the circuit.

In TARGET:

There is no difference between pads in general and SMDs. SMDs are Pads with layer =16 or 2 instead of 100 and drillhole =0 instead of a different value. You are allowed to assign padstacks and so e.g. generate pads with "blind


drillings". Press  and  or press key [1]. Define options of future Pads/SMDs using key [o]. Place SMD's by normal mouseclick or with a further press on [1]. Set layer 16="Copper top". If you later wish to have it on layer 2="Copper bottom" just flip it (use the mirror-function [m]).

SPLIT

Insert bendings to wires.

In TARGET:

Move cursor to the desired wire and click on it (highlight) it. Now see little square boxes at the ends of the wire segment. Touch them with the mouse and drag them as desired. use the right mouse button after highlighting for setting a further square box (a splitting mark). Move this one also by mousetouch. Also use key [g] for drag. First

set the bending modes in the extended pointer tool:  and



(yellow).

Generate a new kind of bending (a bow or a spline curve). If you want to get rid of a bending drag e' with [g] a leg of the bending to zero. Use also a reorganisation for sweeping out superfluous bendings. Make bendings visible by the use of "quick draw [#]" and an appropriate zoom in.

TEXT

Placement of text

In TARGET:


Press key ["] (sign for speech) upon this place, where text shall appear. In the layout please use TARGET-font because the gerber driver currently uses this font.

UNDO

Delete previous command.

In TARGET:



Press button , 50 steps are possible.

USE

Load libraries before placement of elements.

In TARGET:

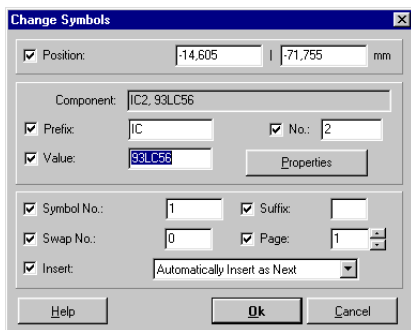
Not needed. Set the path to your libraries (if different from default) in menu "Window / Settings/Options". Open the library browser using key [Ins]. It opens several opportunities for finding components. Use the search functions or the tree view technique. Search for values or other properties. Search over all libraries...

VALUE

Enter or edit values of elements

In TARGET:

Doubleclick on the cross handle of a symbol/package opens the corresponding dialogue for setting the element value. The change happens in schematic view as well as in layout view. For a symbol e.g.



VIA

Placement of vias

In TARGET:

During the placement of tracks please press key [.,]. TARGET now places a via on this spot. By the change of the

TARGET 3001!

Chapter 1: Eagle (R) vs. TARGET 3001!


color please notice to be working upon a different layer. If you wish to predefine future layers, please press [.] when not placing tracks. Now key [o] sets the options for future vias. In the via-dialog on the right please save those vias you often use by programming a button. Those do also appear within the layer tool for an easy use in multilayer technique.

WINDOW

Select view or refresh view.

In TARGET:






Press key [n] for new to refresh the the picture. Press , to display the figure fit to the window. Zoom in and out in 10% -steps by the use of the scrolling wheel . Cursor-position is centered. Using the right mousekey a zoom area can be defined. If you you drag the zoom area backwards you will see your drawing win fit again.

WIRE


Draw wires (Lines)

In TARGET:



For signal tracks in schematic use , for layout tracks in PCB view use . Draw lines by  and



. Pressing key [o] allows the setting of future options of drawn segments. A later change of segment properties is made by the use of the key [e] after highlighting them. A single segment can be edited by a doubleclick upon it.

WRITE

Save a drawing or a library

In Target:

As common in Windows using "Save" etc...