Layout CAD

Schematic

Simulation

Printed Circuit Board

ASIC

Contour Router

EMC Analysis

Contents:

1 Basics 1
1.1 Introduction 1
1.2 Copyrights and Trademarks 2
1.3 Disclaimer of Liability 2
1.4 Package Checklist 2
1.5 System Requirements 3
1.6 Copy Protection 3
1.7 TARGET3001! Installation 3
1.8 What is TARGET 3001! design station? 4
1.9 Conventions 4
1.10 What is what? 5
1.11 Services offered by IBF 7
1.12 PCB-Service 8
2 Operation of TARGET 3001! 9
2.1 Start TARGET 9
2.2 Graphical user interface 9
2.3 Toolbar 9
2.4 Status line 15
2.5 Windows 15
2.6 Mouse Clicks 15
2.7 Key combinations 18
2.8 Working Modes 20
2.9 Help for Dialogs 36
3 Menu System 65
3.1 File Menu 65
3.2 Edit Menu 80
3.3 View Menu 85
3.4 Elements Menu 91
3.5 Components / Package Menu 99
3.6 Actions Menu 104
3.7 Window Menu 110
3.8 Help Menu 111
4 Create Components 113
4.1 Create a simple schematic symbol 113
4.2 Create a schematic symbol with multiple gates 118
4.3 Change an existing schematic symbol 121
4.4 Create a reference symbol 122
4.5 Create a package 123
4.6 Change an existing package symbol 125
5 Draw and edit signals 127
5.1 Draw signals in the schematic 127
5.2 Edit a signal 129
5.3 Place a bus 132
5.4 Place tracks 133
5.5 Draw air wires (ratsnest) 134
6 Automatic functions 135
6.1 Symbol-, Gate-Exchange 135

6.2 Pin exchange 136
6.3 Ripup & Retry Autorouter 136
6.4 Contour router 138
6.5 Autoplacer 142
6.6 Electrical-Rule-Check, Design-Rule-Check 142
6.7 Automatic Groundplane-Assistant 144
7 Crashcourse 145
7.1 Create a simple schematic 145
7.2 Create a PCB 147
8 TARGET 3001! Simulation 152
9 EMC-Analysis 152
9.1 Introduction 153
9.2 Handling of the EMC-Analysis tool 154
9.3 Prevention of EMC disturbance 158
9.4 Improvement of EMC properties 163
10 Data Interfaces and I/O Device Drivers 166
10.1 Gerber Output (Photoplot) 166
10.2 Read in Gerber data 170
10.3 Drilling 171
10.4 Gerber and drill output 171
10.5 Post-Script driver 173
10.6 DXF-driver 174
10.7 Engraving insulation channels 175
10.8 ASCII- Interface 175
10.9 Converter TARGET 2.1 to TARGET V3 181
10.10 Converter TARGET V3 to TARGET 3001! 181
10.11 Read in EAGLE files and EAGLE libraries 181
10.12 XGerber-driver 182

(c) 1989...3001 Ing.-Buero FRIEDRICH, http://www.ibfriedrich.com

Chapter 1: Basics

This Chapter describes general perspectives. It provides you with all the information you need on how to install TARGET, how it
1.9 and 1.10.

We hope you enjoy your work with TARGET 3001! design station!

Contents:

1.1 Introduction
1.2 Copyrights and Trademarks
1.3 Disclaimer of Liability
1.4 Package Checklist
1.5 System Requirements
1.6 Copy Protection
1.7 TARGET3001! Installation
1.8 What is TARGET 3001! design station?
1.9 Conventions
1.10 What is what?
1.11 Services offered by IBF
1.12 PCB-Service

1.1 Introduction

Congratulations, you are using TARGET3001! design station a professional CAD/CAE-software.

With TARGET 3001! design station, a new generation of CAD-/CAE-software programs for circuit design is born. The integration of the schematic-, simulation- and PCB- data into a combined operating interface, and the integration of these data into a combined database file is revolutioning the work of the design engineer during the development process to keep the time-to-market as short as possible.

This manual makes you familiar with TARGET3001! design station and assists you to get started immeaditely. It shows you how to deal with common tasks, gives you helpful information and points to new important features. This manual describes the complete TARGET3001! design station software system.

TARGET3001! design station consists of the following programs:

> Schematic > Mixed Mode Simulation > Shape based / Contour Autorouter > PCB > EMC-Check

For a description of the features supported by different versions of TARGET3001! design station, please refer to the product information sheet. In TARGET 3001! design station, we have considered ideas and recommendations from our customers as well as our own to extend the user-friendliness and performance of previous versions.

1.2 Copyrights and Trademarks

All rights on this manual and the TARGET3001! design station software for Windows, particularly the Copyright, are reserved by

Ing. Buero FRIEDRICH
Harald Friedrich, Dipl. Wirtsch. Ing. (TU)
Street: Am Schwarzen Rain 1
City: Eichenzell
Zip Code: D-36124
Country: Germany
Tel.: ++ 49 (0) 6659 - 919 444
Fax: ++ 49 (0) 6659 - 919 445
Internet: http://www.ibfriedrich.com/english
Email: target@ibfriedrich.com

Microsoft Windows 95/98/ME/NT/2000/XP are registered trademarks of Microsoft Corporation. Gerber is a registered trademark of Gerber Inc. PostScript is a registered trademark of Adobe Systems Inc. DXF is a registered trademark of Autodesk Inc.

All other program- and font names additionally used in this manual or in the TARGET3001! design station software are registered trademarks of their respective holders.

All rights reserved. No part of this manual or the program TARGET3001! design station may be reproduced in any form or means whatsoever. The buyer has the right to use TARGET3001! design station on one computer at his work place but has no ownership of it.

Permission is given to make one safety copy of the program TARGET3001! design station for your own use. This copy is under no circumstances to be forwarded to a third person or to be changed in any way. Each program can be identified individually, therefore each illegal copy can be traced. If the buyer does not follow the outlined conditions, he will be held responsible to pay a compensation to the Ing. Büro FRIEDRICH. A penalty of five times the purchasing price is agreed.

A purchased and unchanged "TARGET 3001! discover" evaluation- or shareware-version can (should) be forwarded to others. However it is reserved for private and educational use. It's forbidden to use it for commercial purposes.

1.3 Disclaimer of Liability

The program TARGET3001! design station and this manual have been constructed carefully. However, we accept no liability or what so ever, that this program TARGET3001! design station is without errors or its use for a special purpose. All legal responsibility or liability in case of a consequential damage is impossible.

Information furnished by **Ing. Büro FRIEDRICH** is believed to be accurate and reliable. We appreciate any feedback from our customers on the contents of the manual and the TARGET3001! design station program.

1.4 Package Checklist

The TARGET3001! design station package contains this manual in HTML format and possibly one or more 3,5" diskettes as well as a CDROM. These media contain the following files:

- TARGET 3001! program (Tar3001.exe)
- Simulation program (sim.exe)
- Component libraries (*.CMP3001)
- Package libraries (*.PCK3001)
- Output drivers for : - Postscript (Post.exe) - GERBER RS274D (Gerber.exe) - Excellon- and Sieb & Meyer circuit board plotter

(Drill.exe) - DXF format (Dxf.exe)
- ASCII- I/O Data interface (Textio.exe)
- Converter for older versions of TARGET and RULE (R2t.exe and V3from21.exe)
- A readme file with the latest information and additions for the TARGET3001! design station program (Readme.txt)
- Several demonstration files (*.T3001)
- This helpfile in HTML format and possibly as a printed manual

1.5 System Requirements

To run the TARGET 3001! design station program, an IBM-AT 486- or compatible computer is necessary. (The processor type should be at least a 80486DX). The software has also been succesfully tested on x86- and Athlon- type computers from AMD. Also, x86- type computers from Cyrix and IDT will work fine.

TARGET3001! design station requires at least 16MB of RAM. A colour monitor and a Super-VGA Video Card with a resolution of 800x600 pixels and more than 256 colours are required. A MS-Windows compatible mouse is also necessary.

TARGET3001! design station requires MS-Windows 95/98/ME or Windows NT4/2000/XP as a operating system. A local CD ROM-drive must be installed. TARGET 3001! design station must be installed on your local hard drive (Exception: the discover version of TARGET3001! design station can be started from the CD).

A fast hard drive, a fast graphics processor and more RAM (32MB or more) will have positive effects on the operation of this program.

1.6 Copy Protection

TARGET 3001! design station for Windows 95/98/ME/NT4/2000/XP is not shipped with a hardware copy protection device (dongle) This has a positive effect on the purchasing price. However, your name is registered in the program. Any manipulations on the Target3001.exe file can lead to a damage of your system (This is not possible with an unmanipulated Tar3001.exe file). Each program can be individually identified, therefore allows us to track the origin of each illegal copy. In case of violations against these agreements will the buyer be helt responsible to pay a compensation to the **Ing. Büro FRIEDRICH**.

A penalty of **five times the purchasing price** is agreed.

1.7 TARGET3001! Installation

Before you start with the installation of TARGET 3001! design station, please check the contents of the software package. Please make a safety copy of the original media and use the copies for the installation. You must have Windows 95/98/ME/NT4/2000/XP installed prior to start the TARGET3001! design station installation. Please check that the Windows system date and time settings are correct.

The following describes the installation procedure. This procedure is independent of the version you have purchased. The procedure is identical for the **light-**, **economy-**, or the **professional** version or for the complete software packet.

Installation of TARGET 3001! design station:

- Insert the diskette (if needed) into your local diskette drive of your computer.
- Insert the CDROM into the local CDROM-drive of your computer.
- In case that the CDROM does not start automatically, please click on **Start** and select the **Run** command. Please enter the following command: **x:\setup** (x = drive character of your CDROM). Press **Ok**.
- Now follow the instructions as they appear on the screen during the installation process.

As long as the main version number (**9**.x.x.xx) stays the same, it is possible for you to download all updates for your TARGET3001! design station software for free from the Internet. On request, we give you the internet address for this service. This allows us to offer a quick support for customers who purchased a version of the TARGET3001! design station in an early stage.

1.8 What is TARGET 3001! design station?

TARGET3001! design station is an object-oriented, 32-bit schematic- simulation- and printed circuit board development system for the PC. It´s a partly automatic tool, that assists you with the drawing of a schematic, the mixed-mode simulation of a circiut and the PCB design with an integrated EMC checking tool. The data of the schematic, simulation, PCB and EMC check are collected and editable in one file. Any changes in one of the program parts will automatically update the remaining view involved in the data file. A inconsistent data basis is therefore prohibitet and a complicated translation or communication via netlists is inapplicable. TARGET3001! design station is the only software system worldwide with the mentioned features in combination with one common user interface.

TARGET3001! design station has extendable symbol-, model- and package libraries and is capable of creating a bill of material (B.O.M) as well as netlists in several formats. Components can not only be placed by hand or automatically, they can also be edited at any time. You can route your PCB´s on the screen interactively and/or fully automatically. The finished PCB file can be translated to several data formats with the help of special output drivers, allowing you to create data which can be used for a selection of different CAM processes.

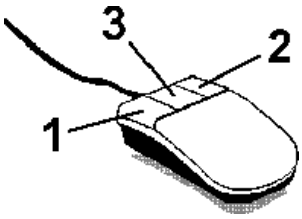
The following capabilities are very interesting:

- Hierarchial design and use of variants of one PCB
- True scale representation on the display, 1 inch on the screen equals 1 inch on the printout (World coordinates)
- Program and manual have been developed and used in Germany since 1989
- Schematic, simulation, PCB and EMC-analysis data in one file (no version conflicts!).
- Schematic, simulation, PCB-Autorouter and EMC-Analysis tools are easy to use (short learning time)
- Direct forward- and backannotation (realtime data integration)
- Schematic, simulation, PCB, EMC-analysis at the same time in several windows
- Automatic generation of ground planes, changes online possible (realtime ground planes)
- Component Wizard for easy edition of components
- Package Wizard for easy edition of packages
- Outline Wizard for easy edition of outlines with outbreaks
- Flexibel use (All components can be edited even if they are already in use)
- All Windows-fonts can be used (not for Gerber output)
- Exchange of modules between different projects through the clipboard
- Easy export of Graphics in Metafile-format through the clipboard for documenatation purposes (Exampel: Export into MS-Word allows flexibel use without loosing the resolution)
- Independent from video cards and printer drivers

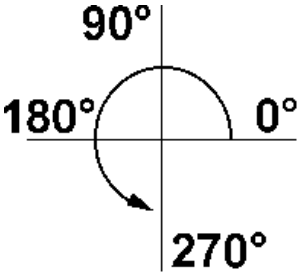
1.9 Conventions

The following conventions are valid for the spelling in this manual:

[X] Expressions in rectangular brackets are pressed keys: **[Pos1]** or **[Home]** means: press the **[Pos1]** key or the **[Home]** key (depending if you use a german or english keyboard). The shift key button will be given in front of the key name.
The following shift key buttons are available: **[Shift]**, **[Alt]**, **[Ctrl]** (In german: Umschalt, Alt, Strg).
[Alt]+[P] means: press and hold the **[Alt]** key and press at the same time the **[P]** key. **[Shift]+[INSERT]** means: press and hold the **[Shift]** or **[Umschalten]** key and press the **[INSERT]** or **[Einfg]** key.
Mx - Expressions starting with a big M are mouseclicks. TARGET recognizes 3 mouse buttons: left (1), middle (3) and right (2). The following combination of mouse clicks are possible:



M1 = click and release the left mouse button immeaditely .
M2 = click and release the right mouse button immeaditely.
M3 = click and release the middle mouse button immeaditely.
M11 = double click the left mouse button.
M22 = double click the right mouse button.
M12 = press the left and the right mouse button at the same time.
M1H = press and hold the left mouse button, move the mouse.
M2H = press and hold the right mouse button, move the mouse.
You can adjust the sensitivity and the double-click speed of the mouse (for **M11** and **M22**) as well as settings for left-handed use in the Windows control panel.



Orientation information in TARGET are given in degree (°). This refers to the right pointing axis of the coordinate system. The counting is done in a positive mathematical way (counter clockwise; positive X-axis = 0°, positive Y-axis = 90° etc.).

1.10 What is what?

To fully understand the manual and the operation of the program, the following definition of terms used should be read:

Circuit or Project: A generic term for the schematic and PCB. Both will be saved in one file (*.T3001). It is possible to have a couple of projects open in TARGET with one or more open windows of each project. It is possible to have a window of the schematic- and a window of the PCB- file open on the screen. To edit both, you simply toggle between the schematic- and the PCB-window with a single key **[F3].**

Schematic: A diagram that shows the electrical perspectives between the connections of components through inter connections done with signal lines and busses. A schematic can contain up to 100 pages.

PCB: Printed circuit board, presentation of the electrical connections of the pins of component packages with solder pads and signal tracks. A PCB can be made out of one or more of the following: ground planes, deletion layers, solder resist layers, insertion legend print layer, solder paste layer, gold coating layer, adhesive coating layer and dimensioning layer, outline layer, routing data layer and a drill hole layer. Up to 100 layers are possible. All layers are viewed from the top (top view). Layers with a lower layer number can be found below layers with a higher layer number. Therefore the solder side of the PCB will be viewed from the top, imagine the PCB is made of glas. An insertion legend print on this layer must be visible mirrored on the computer screen.

Component: For a schematic, a component is made of one or more symbols. To simulate a circuit, only the components simulation model is of interest. A PCB designer is interested in the package of a component. A CD4000 standard CMOS component contains 2 x NOR-symbols, 1 x NOT-Symbol and 1 x POWER-Supply Symbol as well as a GND symbol. To convert a schematic into a PCB, each symbol has to be assigned to an available package. (DIL14 or SMD14). While importing from a library, each component will be automatically assigned with an identification number. A component can have a name (IC17;R8) and a value (CD4000;4K7).

Symbol: A symbol is a schematic representation of a component or parts of it, for example a gate. It is made of a handle (small

cross), its drawing (with a symbol name or a component name and a component value) and the according pins. It belongs to a component.

Package: A package is a physical representation of a component, necessary for the printed circuit board designer. It is made of a handle (small cross), its drawing (outline, maybe with a component name and a component value) and the according pads as footprints (DIL- or SMD-type). It is part of a component.

Pin: A pin in TARGET is a schematic representation of a connection of a electronic component. It has always a pin number and sometimes a pin name. A pin belongs to a symbol and therefore to the component. If this pin is connected in the schematic, it is also part of a signal. Pins not connected, are marked with a small cross at its end. The cross disappears as soon as this pin gets connected.

(Solder) Pad: A solder pad is a physical representation of a connection on the printed circuit board. Such a pad has always a pin number. A pad belongs to a package and therefore to the component. It is part of a signal, if the corresponding pin is connected in the schematic. The association between a pin in schematic and its pad on PCB is exclusively dependent on an identical pin number. A pad with a drill hole appears usually on all copper layers (special layer 100 = all copper layers, or beginning with layer 101 = pad stack). The drill hole diameter for SMD (surface-mounted-devices) pads is zero and it has a normal copper layer (e.g. 16=copper top).

Signal: A signal in TARGET is an assigned characteristic. For a schematic, it can contain pins, signal traces, signal tori, signal curves, and junctions (knots) whereas it contains pads, tracks, tori and track curves, bridges, through-holes and vias on the PCB. To see if two pins or signals are connected, this can be checked by verifying their afiliation to the same signal.

Drawing: A drawing can be made of the following graphical elements: line, triangle, rectangle, slice, tori, curve, orientation mark and names.

Junction (Knot): In the schematic, a junction is used to connect crossing signal traces or to show their connection. A junction is made of a slice which is bigger than the crossing signal tracks.

Node: All signals are assigned to nodes in the simulation program. Because the simulation program sometimes adds internal nodes, their final amount can be higher as the number of signals in the schematic.

Bus: A bus in the schematic is a collection of signal names. All signals can be connected to a bus. It is possible to distinguish graphically between seperate busses (databus, addressbus ...), but TARGET allows to get any signal out of any bus (for example address line A7 from the databus).

Via: A via is a plated through-hole of all copper layers of a printed circuit board (special layer = 100). For the production, it is important to have a sufficient residual ring around the plated through-hole. Please refer to "padstacks" too.

Bridge: A bridge is a wire connection between two points on the PCB. The solder pads of the bridge are made of the current vias. The drawing of the wire should be made on the position layer.

Element: An element is a database assignment for all elements of a drawing. It is the smallest object that TARGET can handle. Elements are: Pad, junction, bus, bridge, track, triangle, rectangle, slice, torus, curve, orientation mark, name, pin, symbol or package. Signals and components are container-elements in the background, they are not visible themself.

Page: TARGET is capable to handle up to 100 schematic pages. Connections between different pages can be made by using reference symbols. The gnd-symbol is a well known reference symbol. These connections can also be reached through bus-islands on different pages. Basically, the same signal name = the same signal = connected.

Layer:TARGET is capable to handle up to 100 printed circuit board layers. (0..99). Each layer can be switched visible and invisible. Each layer can be assigned to a special function, for example: copper layer, position layer, solder-resist layer... All layers (even copper bottom) will be displayed as a view from top. Imagine the PCB was made of glas. Special layers are layer 100 for all copper layers and layers 101...250 for padstacks. Please refer to "padstacks" too.

Simulation: The schematic can be tested and evaluated by the use of the mixed-mode simulator. The simulator in TARGET is

PSPice-and Spice 3f5 compatible. The amount of nodes handled by the simulator is different according to the version purchased. Analog- and digital nodes are counted seperately.

EMC-Analysis: A integrated tool to check the routet PCB regarding all design rules recommended for electromagnetical compatibility (EMC). The inductive, capacitive and galvanic coupling of the tracks as well as the radiation value of individual signals are displayed.

Padstack: A padstack is a plated through-hole of several (not all) copper layers. Padstacks are distinguishable between types that will be drilled only from one side (blind via) and types that connect only the inner layers of a PCB. These types are invisible from the outside (buried via).

Drillhole: You will not find a simple element that represents a simple drillhole in TARGET. To drill a hole, you must place a solder pad and set its pad size dimensions smaller than the drillhole size. Also, you should set it's pin number to zero.

Mode: Depending on the project status, TARGET can be operating in different modes. The status line (bottom of screen) shows the current mode. The status line shows also the dependencies of several keys with their individual meaning in the according mode.

Options: With the options feature in the according mode, it is possible to define the settings for elements to be placed in the **future**. (key **[O]** for options). For example: While you work in the mode "Draw Tracks" it is possible to set the properties for tracks to be placed next (layer, width...).

Change (Edit): The currently selected elements can be changed by pressing the **[E]** key or by selecting the menu Edit / Change command. This means it concerns elements that were created in the **past**.

1.11 Services offered by IBF

The **Ing.-Buero FRIEDRICH** (IBF) and its distributors offer a professional, free of charge technical support.

Telephone Hotline: Tel.: ++ 49 (0) 66 59 / 919 444
For questions regarding **Sales**, please call Monday to Friday between **8:00 AM and 4:00 PM** German time or send an email to or contact our distributor.

For question regarding **technical Support**, please call Monday to Friday between **10:00 AM and 4:00 PM** German time or contact our distributor. We expect you to have the problem file loaded on your screen and that you have an exact definition of your problem.

eMail: You can also transmit your problem file with a description by email to us.

Internet, World Wide Web-Site:Our homepage will announce the latest news about our company and the products. You can download documents, file updates and component libraries. Further you will find a "FAQ" which might answer some of your technical questions.
Please make sure that your TARGET version is the current one. Maybe your problem is currently already solved.

1.12 PCB-Service

Contents:

1.12.1 Printed cirquit board manufacturers
1.12.2 Design Rules

1.12.1 Printed cirquit board manufacturers

The following companies manufacture PCB´s directly from your TARGET-file (*.T3001):

Beta Layout GmbH
Feldstraße 2
65326 Aarbergen / Germany
Tel.: + 49 (0) 6120-907010
Fax: + 49 (0) 6120-907015
www.pcb-pool.com
beta-layout@pcb-pool.com

or

Hofmann Leiterplatten GmbH
Vilshofener Straße 12
93055 Regensburg / Germany
Tel.: + 49 (0) 941-60490-0
Fax: + 49 (0) 941/60490-20
www.hofmannlp.de
info@hofmannlp.de

Our homepage http://www.ibfriedrich.com/Services.htm has a listing of companies who are capable to manufacture PCB´s directly from your TARGET-file (*.T3001).
If you are a PCB manufacturer yourself an advertisement there is free of charge.

1.12.2 Design Rules

When you create your PCB´s, the following design rules have to be followed to prevent additional costs for the PCB manufacturing:

- The minimum track width as well as the track spacing should be 0,22 mm. (smaller is possible, but costs more). The minimum trackwidth after consultation is 0,12 mm. (Please use only in emergency!)
- Drillholes should not be smaller than 0,6 mm. (smaller sizes are possible after consultation). All drillhole diameters starting from 0,3 mm are possible. The sizes are selectable in 0,1 mm steps. Through holes will be drilled 0,2 mm bigger as listed in the drill list, to cover the tolerances after the metallization. Drill holes which are not metallized will be drilled 0,1 mm bigger.
- Please check that the residual ring is at least 0.2 mm on all sides. This equals a through hole diameter of 1,0 mm while the drill hole is 0,6 mm.
- The distance from the tracks to the PCB edge should be at least 0,5 mm.
- Texts in copper should be at least 2 mm high (font).
- In the insertion legend print, the line width of the fonts should be at least 0,3 mm. The height of the texts (font) should be at least 3 mm.
- The outline of the PCB must be drawn on an extra layer (standard: layer 23). The distance between the middle of the lines define the PCB size.

Chapter 2: Operation of TARGET 3001!

How to use TARGET? Please read this chapter carefully before you start working with TARGET to get a quick overview of its procedures and operation. It is very
important to read Chapter 6- "Mouse clicks". Please note also the different functions of the individual tools (marked with a small "arrow" or "three points").

Contents:

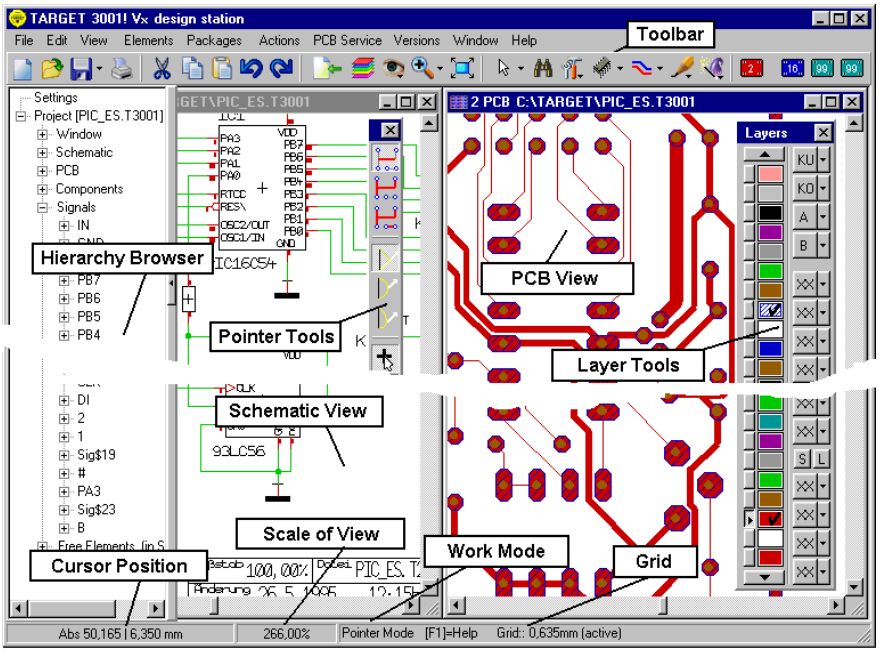
2.1 Start TARGET
2.2 Graphical user interface
2.3 Toolbar
2.4 Status line
2.5 Windows
2.6 Mouse Clicks
2.7 Key combinations
2.8 Working Modes
2.9 Help for Dialogs

2.1 Start TARGET

Please start Windows. Click on "Start", go to "Programs" and double click on the Program TARGET 3001! in the Program group "TARGET 3001!". A window with the TARGET logo, information on your version (license) and information on your version number appears. After reading and entering further dialogs, you will reach the TARGET Start Assistant by clicking M1 on the TARGET logo. Depending to your selection it will guide you to the graphical user interface.

2.2 Graphical user interface

Please activate one of the two buttons shown above. TARGET starts with its TARGET main window showing the following elements: Headerline, Menue line, Toolbar, Working window, Status line, Pointer toolbar, Layer toolbar, and the Hierarchy browser (edge of screen left). Please consult your MS-Windows manual for a description on how to work with windows if you are unfamiliar with the MS-Windows operating system.



2.3 Toolbar

Toolbar in schematic view:

Toolbar in PCB view:

The toolbar shows the most important TARGET commands as buttons with images. If you are insecure about the meaning of a button, please place the mouse pointer for a short while on the button without clicking it. TARGET shows than a short description of its purpose in a small popup window. The following is a description of all tools:

Please note: The toolbar can only be reached with the mouse, and not each of the buttons can be replaced by a menue command. It is not possible, to configure the toolbars individually.

The meaning of the following tools:

Standard tool: Short click with the left mouse button **(=M1)**: direct function

Double-tool with an arrow: The standard function of this button can be reached by clicking on the big field left. A more detailed function of this button is available by clicking on the small arrow on the right side of the button.

This tool is made of several different functions for the following mouse clicks:

- short click with the left mouse button **(=M1)**: function 1
- long click with the felt mouse button **(=M1H)**: function 2

The description of the function in the small popup window starts also with 3 points (function1/... function 2).

The tool functions

The following table describes the tools in more detail:

Create a new project. TARGET opens the Sheet.T3001-file and calls it New.T3001. (SheetP.T3001 for a PCB without schematic)

Open an existing project.

Left = Save project / Right = Save project as...

Print the current project view

Cut and copy marked elements to the clipboard. The data will be saved in TARGET-format as well as the Windows-Metafile-Format (WMF). A easy transfer of your schematic- or PCB view without any quality loss to your favourite word processing software (fro example: MS-Word) is possible.

Copy marked elements to the clipboard

Paste elements from the clipboard

Undo / Redo last action

Change to PCB-view. Has a signal or component been marked in the schematic-view, it will be searched, marked and zoomed in the PCB-view (=cross probe).

Change to schematic-view. Has a signal or component been marked in the PCB view, it will be searched, marked and zoomed in the schematic-view (=cross probe).

Select any schematic page.

Adjust PCB-layers

View: The following tools appear::

- Define a scaling factor
- Define the grid
- Define the colors
- Create an additional schematic window
- Create an additional PCB window

Left = Enter zoom-mode / Right = the following zoom tools appear:

- Zoom out
- Zoom in
- Update and redraw the actual window
- Center the clicked point in the working window (Pan)
- Define a scaling factor

Format-filled scaling of the circuit

Left = Enter pointer-mode / Right = opens the following tools:

After a selection of the following tools, a selected (clicked) signal in pointer- mode will...

- ... mark only the catched signal-segment alone
- ... mark only the combined signal-island

- ... mark the complete signal of the shown view

After a selection of the following tools, a move-function with **[g]** or with **[Ctrl] + M1H** in pointer-mode, a line catched in its center will...

- ... be splitted in two segments
- ... be changed into an arc of a circle
- ... be changed into a **bezier-curve**

- If this tool is activated, a selected handle of a component always marks the whole symbol or package.
- If this tool is activated, each selected element marks the whole symbol or package.
- The tools activates the pointer-toolbar (left) permanently.

Search and mark components

Editing-mode: The following tools appear:

- Measure distances
- Select and mark a single element
- Delete marked elements
- Move marked elements
-
- Mirror marked elements horizontally (on the vertical axis)
- Mirror marked elements vertically (on the vertical axis)
- Change the marked elements
- Select and rename a signal name

Left = Insert a component symbol to the schematic / Right = The following tools appear:

- Inserts the remainders (symbols) of a component to the schematic
- Insert reference-symbols to the schematic
- Combine symbols
- Export the selected symbols as one component to the library
- Starts the library browser
- Delete a component from the library

Left = Insert a package-symbol to the PCB / Right = The following tools appear:

- Export selected elements as a package to a package-library
- Starts the library browser
- Delete a package from the library

Left = Draw signals (schematic) / Right = The following tools appear:

- Draw a signal directly
- Draw a signal (first in horizonta direction)
- Draw a signal (first in vertical direction)
- Draw a signal (first in orthogonal direction)
- Draw a signal (first in diagonal direction)
- Four individually programmable buttons with a presetting for the drawing of signals. A click on the signal with M1: Call saved signal options. A click on M1H: program the button with the actual options
- Edit further signal options
- Place a junction
- Place a bus

Left = Draw track (PCB) / Right = The following tools appear:

- Draw a track directly
- Draw a track (first in horizontal direction)
- Draw a track (first in vertical direction)
- Draw a track (first in orthogonal direction)
- Draw a track (first in diagonal direction)
- Four individully programmable buttons with presettings for the drawing of tracks. A click on the track with M1: Call saved track options. A click on M1H: program the button with the actual options
- Edit further track options
- Place a via
- Place a bridge
- Change solder pad into teardrop
- Draw a spiral, edit spiral track options
- Draw a wire (ratsnest), only for PCB´s without a schematic

TARGET drawing palette = The following tools appear:

- Draw a line, triangle, rectangle, circle and tori

- Draw a filled triangle, rectangle and slice
- Only in schematic-mode: Place a pin, a junction and a bus
- Only in PCB-mode: Place solder pads, vias and bridges
- Only in PCB-mode without schematic: Place ratsnest
- Enter text
- Place orientation marks
- Logic gates according to DIN / IEEE: AND, OR, NOT, EXOR, Schmitt-trigger and buffer
- . Symbols: block-description-fields to create components according to DIN / IEEE
- . External description-fields to create components according to DIN / IEEE
- Internal description-fields to create components according to DIN / IEEE
- . Other drawing elements

Automatic functions:

- Measure distances:
- Pin swap (only in schematic-mode)
- Solder pad swap (only in PCB-mode)
- Gate swap (symbol swap, only in scheamtic-mode)
- Starts the autoplacer (only in PCB-mode)
- Starts the autorouter (only in PCB-mode)
- Opens a dialog to recalculate the ratsnest (only in PCB-mode)
- Starts the groundplane assistent (only in PCB-mode)
- Check project (schematic and PCB) on errors
- Starts the mixed- mode simulation of the schematic
- Starts the EMC-Analysis of the PCB
- Reorganizes the whole project (optimizes the internal database)

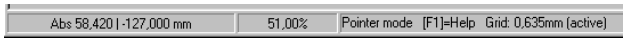
The three last used schematic pages are shown in this tool. After a selection of a tool button with M1, the schematic page will be opened.

The actual used layer is displayed here. A click with M1 on this tool button allows you to select any other layer as the current working layer. For example: It is possiblæe to select the copper-layer in the Place track mode.

The three last used PCB-layers are shown in this tool. After a selection of the tool button with M1, the layer will be activated as the current working layer. For example: It is possible to select the copper-layer in the "Place track" mode.

2.4 Status line

The status line (status bar) can be found at the bottom of the working area of TARGET and shows important information.



The status line shows information regarding the active window: The first column shows the actual coordinates of the crosshair cursor in the active window. The "Abs" indicator means that the coordinates shown are **absolute**. The reference point (0|0) therefore is the exact middle of the usable square meter. The "Rel" indicator means that the coordinates shown are **relative**. The reference point (0|0) has been selected with the **[Pos 1]** **[Pos 1]** key twice at the same point shows again absolute coordinates.

The middle column of the status line shows the actual scaling factor in percentage of the active window. 100% means a display in world coordinates. 10cm in the PCB represent exactly 10cm on the screen. To measure distances on the screen, you could basically use a ruler. There is of course a special function to measure distances. To do a dimension check on components, you can hold components directly to the screen. To have an exact scaling of the window, the **size of the current monitor** has to be entered in menue "window/options".

The third column of the status line shows information which depend on the operation mode. Here you can read about the the mouse button settings for the individual modes. After a click with **M1**
components name and the related signal name to which this element belongs. Also, information about the grid and its settings can be found here.

2.5 Windows

The working area is used to show one or more windows, containing schematic- and/or PCB-views of your project. It is also possible to open several windows from different projects at the same time. The active window is always marked with a dark-blue header field.

The windows are representing the drawing areas of TARGET. The drawing area inside these windows is 1m x 1m = 1m² of size. Because of the limitation of the screen size of the monitor, only a part of the drawing area is displayed. The window acts therefore as a "peek-hole" over the drawing area. The position of this peek-hole is recognizable and adjustable with the two scrollbars at the bottom- and at the right side of the working window. The physical resolution of all TARGET-drawings is 1nm = 1/1.000.000mm = 1/25.400.000 inch. It is possible to insert a freely definable grid to the drawing area. The big cursor (cross hairs) **[F8]** is movable with the arrow-keys or with the mouse. It is possible to increase the moving speed of the cursor to a factor of 2 by **[SHIFT]**, to a factor of 4 by **[Ctrl]** or to a factor of 10 by both **[SHIFT]+[Ctrl]**
project.

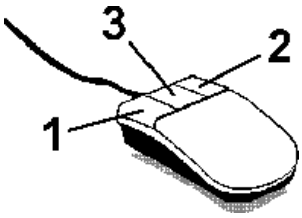
2.6 Mouse Clicks

Contents:

2.6.1 Mouse button settings
2.6.2 Left mouse button
2.6.3 Right mouse button
2.6.4 Middle mouse button
2.6.5 Left and right mouse button

2.6.1 Mouse button settings

The basics chapter explained already some of the mouse clicks with the help of M-expressions. At this point, the possibilities of the mouse clicks will be explained again to cover all instructions given in this manual.



- **M1** = click and release left mouse button
- **M2** = click and release right mouse button
- **M3** = click and release middle mouse button
- **M11** = double-click left mouse button
- **M22** = double-click right mouse button
- **M12** = press left and right mouse button at the same time
- **M1H** = press and hold left mouse button, then move the mouse
- **M2H** = press and hold right mouse button, then move the mouse

All three mouse buttons are used in TARGET. The mouse starts several tasks with a single click **Mx**, with a double-click **Mxx** or with a long click (hold) **MxH** on the according mouse button.

2.6.2 Left mouse button

It is possible to mark all elements (elements are lines, signal tracks etc.. or components and packages) with the left mouse button. Therefore they can be moved or dragged. The following actions are of interest in the pointer-mode:

You want to...

... mark an element, signal or component:

Click with **M1** on the according element or handle. A click with **M1** at a free spot in the project unmarks all marked elements. **[Ctrl]** allows to mark a handle alone. While marking signals, these tools have the following meaning:

- mark a single element
- mark the related signal island
- mark the complete signal

While marking components, these tools have the following meaning:

- the complete symbol/package is marked by hitting the handle
- the according symbol/package is marked by hitting any element of the component

... mark or unmark additional elements:

Click with **[Shift]**+**M1** on the according element or handle.

... mark and directly edit an element:

Click with **M11** on the according element. After this double click, a dialog box appears. This function can also be executed with the key combination **[s]** / **[e]** **[Shift]** and **[Ctrl]** together with **M11** to select elements on a lower location.

... mark more different elements in an area:

Starting at a free spot, draw a rectangular window with the mouse. The elements catched by that window will be marked.

... mark additional elements in an area:

Draw another window with **[Shift]**+**M1H**. The before marked elements remain marked.

... move elements:

Click the middle of an element with **M1H** and move it with the mouse (Drag & Drop). All other marked elements will be moved also.

... bend or drag an element:

Hold the **[Ctrl]** key and move with **M1H**. If you hit a corner: The corner of the rectangle /triangle or the bend in the line will be moved to a new position. If you hit the middle of a line: The line will be splitted into two segments or be changed into an arc or a beziér-curve. Please note the following tools in pointer mode:

2.6.3 Right mouse button

The right mouse button is dedicated to "**Zoom-functions**". All important actions as zoom-in or zoom-out of the actual project are covered by simple mouse clicks.

M2: Zoom-in the area around the current cursor position.

M22: Zoom-out the area around the current cursor position.

M2H, open a zoom window from left to right: Zoom the area of the catched window fitting to the working window. This action is identically to the function "Zoom area" in the view menue.

M2H, open a zoom window from right to left (=backwards): **[F7]**
.

[Shift] M2: The contents of the window will be displayed in a maximum, format fitting size **[F7]**.

[Ctrl] M2: The actual window of the current project will be displayed in a maximum size (maximized). This function equals: on the right upper side in Windows.

[Shift] + [Ctrl] + M2: Display the window at its maximum size and zoom the contents format fitting at the same time.

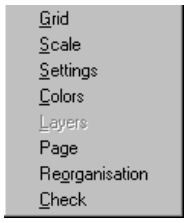
Please regard, that some mouse drivers can redefine the mouse button functions. Therefore you should always set the function to "normal click".

2.6.4 Middle mouse button

The middle mouse button is exclusively used with a single **M3**
button. It has two different functions:

Function 1: If you click **M3** in the "Route Track" mode, a options dialog window appears. After you have done your settings and pressing ok, it will switch back to the current mode. The **[O]** key for "Options" offers the same function alternatively.

Function 2: If you click **M3**



It can be used for direct access to the mentioned functions. These functions are also available in the main menue. As an option, you can use the **[RETURN]** key instead of **M3**.

Please regard, that some mouse drivers can redefine the mouse button functions. Therefore you should always set the function to "normal click".

2.6.5 Left and right mouse button

Pressing both mouse buttons at once (**M12**) will abort the current action (equals the **[ESC]** key). Please regard, that some mouse drivers can redefine the function of mouse keys.

2.7 Key combinations

Function keys

Call Help **[F1]**
Library Browser **[F2]**
Schematic <- PCB (with cross probe) **[F3]**
Schematic <- PCB (without cross probe) **[Shift]** + **[F3]**
Switch Units (mm, µm, Inch and mil) **[F4]**
Switch Grid visible/invisible **[F5]**
Snap on Grid / No Snap on Grid **[F6]**
Zoom image full size **[F7]**
Switch Cursor (Crosshair) **[F8]**
Start Autorouter / Simulator **[F9]**
Menue Mode **[F10]**

Projekt management

New Projekt **[Ctrl]** + **[n]**
Open Project **[Ctrl]** + **[o]**
Save Projekt **[Ctrl]** + **[s]**
Print Projekt **[Ctrl]** + **[p]**
Exit TARGET **[Alt]** + **[F4]**

Edit mode

Undo **[Ctrl]** + **[z]**, **[Backspace]**
Redo **[Ctrl]** + **[y]**, **[Alt]** + **[Backspace]**
Cut **[Ctrl]** + **[x]**, **[Ctrl]** + **[Delete]**
Copy **[Ctrl]** + **[c]**, **[Ctrl]** + **[Insert]**
Paste / Insert **[Ctrl]** + **[v]**, **[Shift]** + **[Insert]**
Select single element **[s]**
Select additional element **[Shift]** + **[S]**
....choose while select **[+]** and **[-]**
Capture window **[c]**
Additional capture window **[Shift]** + **[C]**
Delete selected elements **[Delete]**
Edit selected elements **[E]**, **[@]**
Find and select component/signal **[f]**
Rotate (turn) selecteded elements around the cursor **[t]**
Set rotation (turn) angle and rotate **[Shift]** + **[T]**
Mirror selected elements horizontally **[m]**
Mirror selected elements vertically **[Shift]** + **[M]**
Move/Displace selected elements **[d]**
Drag (sharp bend, arc or curve) **[g]**
....change layer while dragging: **[+]**
Measure distances **[u]**
Rename signals **[v]**

Elements and modes

Place pin / Solder pad **[1]**
Place signals / traces **[2]**
Draw lines **[Ctrl]** + **[2]**
Place bus / Wire-bridge **[b]**
Place junction / Via **[.]** (point)
Draw a filled triangle **[3]**
Draw an open triangle **[Ctrl]** + **[3]**
Draw a filled rectangle **[4]**
Draw an open rectangle **[Ctrl]** + **[4]**
Draw a filled circle= slice **[0]** (zero)
Draw an open circle **[Ctrl]** + **[0]** (zero)
Draw arc/torus **[a]**
Draw air wire in PCB-without-schematic **[w]**
Enter text **["]**
Edit options in the respective mode **[o]**
Exit mode (back to pointer mode) **[ESC]**

Libraries and others

Insert symbols / packages Insert, **[i]**
Insert remaining symbols of components (rests) **[Ctrl]** + **[r]**
Insert reference symbol to the schematic **[r]**
Concat selected elements in the schematic as a symbol **[y]**
Export symbols / packages as a component to a library **[x]**
Start Autorouter (in PCB mode) **[F9]**, **[r]**
Next schematic page **[q]**

Previous schematic page **[Shift] + [Q]**
Edit PCB layers **[Ctrl] + [L]**
Edit colors **[Ctrl] + [k]**
Edit scaling factor **[%]**
Repaint window (new) **[n]**
Attach signal name **[n]** (click exactly onto signal segment)
Attach wire properties **[w]** (exactly on signal segment)
Change the bend mode while placing signals **[Space key]**
Change layer while placing tracks (with via) **[.]**
Change layer while placing tracks (without via) **[:]**

Cursor-keys

Move cursor step-by-step to the left* **[Arrow left]**
Move cursor step-by-step to the right* **[Arrow right]**
Move cursor step-by-step upwards* **[Arrow Up]**
Move cursor step-by-step downwards* **[Arrow down]**
* The movement speed increases with the help of [Shift] and [Ctrl].
Zoom in **[Page down]**
Zoom out **[Page up]**
Maximum zoom **[Strg] + [Page down]**
Zoom format filling **[Strg] + [Page up]**
Relative/Absolute Coordinates **[Pos1]** / 2x **[Pos1]**
Move this point to the center (Panorama, Pan) **[p]**

2.8 Working Modes

Contents:

2.8.1 Pointer Mode
2.8.2 Measure Mode
2.8.3 Catch-Window Mode
2.8.4 Additional Catch Window
2.8.5 Place a Bridge (in PCB only)
2.8.6 Place Pad (in PCB only)
2.8.7 Draw Line
2.8.8 Draw Signal Wires (in schematic only)
2.8.9 Draw Busses (in schematic only)
2.8.10 Place Centering Marks
2.8.11 Draw Filled Triangles
2.8.12 Draw Filled Rectangle
2.8.13 Draw Filled Slices
2.8.14 Draw Torusses
2.8.15 Insert Text
2.8.16 Place Pin
2.8.17 Move Selected Elements
2.8.18 Place Junction (in schematic only)
2.8.19 Place Vias (in PCB only)
2.8.20 Pin Swap (in schematic) and Pad Swap (in PCB)
2.8.21 Symbol/Gate Swap (in schematic only)
2.8.22 Dragging of Corners, Arcs and Spline Curves
2.8.23 Export Package (in PCB only)
2.8.24 Compose Symbol and Place Handle

2.8.25 Draw Tracks (in PCB only)
2.8.26 Draw Air Wires (Ratsnest)
2.8.27 Zoom In
2.8.28 Zoom Out
2.8.29 Select Elements
2.8.30 Place Drill Hole
2.8.31 Turn/Rotate
2.8.32 Mirror
2.8.33 Zoom Area
2.8.34 Panorama (Pan)
2.8.35 Draw Spirals
2.8.36 Teardrops
2.8.37 Draw Open Triangles
2.8.38 Draw Open Rectangles
2.8.39 Draw Open Rings

2.8.1 Pointer Mode

You are in "pointer mode", after loading a TARGET project. Every action in TARGET can be launched from this cental "pointer mode".The status line at the bottom of the window

shows, what mode you're in. You can reach this mode by pressing **[ESC]**.

Every other mode can be left by using the tool . Click right on the small arrow down, to get a toolbar with the settings for the pointer mode:

If one of the following tools is pressed down then clicking a signal segment in "pointer mode" results in ...

- ... the selection of that single segment.
- ... the selection of the connected signal island.
- ... the selection of the whole signal in that view.

If one of the following tools is pressed down then a "dragged" line segment hit in the middle with **[z]** or **M1H** will be ...

- ... split into two segments (= a new corner).
- ... changed into an arc.
- ... changed into a Bezier-curve (spline) .

- If this tool is pressed, a clicked component handle will select the whole symbol or package.
- If this tool is pressed, every clicked component element will select the whole symbol or package.

2.8.2 Measure Mode

You can reach this mode...
- with **[m]**,
- with the point Measure in menu Actions or
- with the tool .

What you can do...
The status line shows the the x- and y-portions of the distance (dX and dY) and the distance itself (L). Eventually you could shortly switch off the grid with **[F6]** to measure very exactly.

Leave this mode...
- with **[ESC]** or **M12**

2.8.3 Catch-Window Mode

You can reach this mode...
- with **[F]** or
- with the point Catch Window in menu Edit or
- with **M1H** in "Pointer Mode" on a free position.

What you can do...
To select some elements together, you can drag a Catch Window over them. Start this Window with **M1H** on a free position or with the key **[F]**. When the window is hanging on your mouse, you can press **[O]** like Options to determine whether only the elements completely inside the Catch Window are selected or their associated elements too.

Leave this mode...
- with **[ESC]** or **M12**

2.8.4 Additional Catch Window

You can reach this mode...
- with **[Shift]+[F]** or
- with the point Additional Catch Window in menu Edit or
- with **[Shift]+M1H** in "Pointer Mode" on a free position.

What you can do...
Same like normal Catch Window, except that the already selected elements stay selected.

Leave this mode...
- with **[ESC]** or **M12**

2.8.5 Place a Bridge (in PCB only)

You can reach this mode...
- with **[B]** in "Pointer Mode" or when drawing tracks or
- with the "Place a bridge" in menu menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options for bridge settings. Place the bridge and press **M1** or **[Return]** at the ending position. TARGET switches into the Draw Tracks mode. A bridge lies normally on layer Copper Top. The ends are normal vias. To set the options for these vias, please go to Place Vias mode and press **[O]** there.

Leave this mode...
- with **[ESC]** or **M12**

2.8.6 Place Pad (in PCB only)

You can reach this mode...
- with **[1]** or
- with the point Place Pad in menu Elements or
- with the tool .

What you can do...
Place pads by pressing **M1** or **[Return]** at the corresponding position. These free pads are normally used only for the definition of new packages for the libraries. The pad numbers are automatically incremented. Press **[O]** like Options to do pad settings.

Leave this mode...
- with **[ESC]** or **M12**

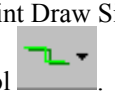
2.8.7 Draw Line

You can reach this mode...
- with **[Ctrl]+[2]** (do not use the "2" in the number block of the keyboard) or
- with the point Draw Line in menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options to do line settings. A **M1** click creates the first segment. Use further **M1** clicks to draw a polyline. These lines cannot be used for signals. They are used for PCB outlines, for inner lines of symbols and packages. They also can be used for the force lines between corresponding potentiometers and between inductivity and contact of a relay. Choose a hatched line style when pressing **[O]** for Options.

Leave this mode...
- with **[ESC]** or **M12**

2.8.8 Draw Signal Wires (in schematic only)

You can reach this mode...
- with **[2]** (do not use the "2" in the number block of the keyboard) or
- with the point Draw Signal Wires in menu Elements or
- with the tool  .

What you can do...
Press **[O]** like Options to do signal settings. The connections of the components in your schematics are done by signals. If you start a signal on another signal wire, the corresponding signal name is continued (e.g. GND). If you start from a free pin, the signal name is derived from the pinname (e.g. A1). If you hit nothing special when starting the signal, a complete artificial signal name is used (e.g. Sig$17). You can see the signal name in the status line.

If you touch another signal with your new signal wire, TARGET asks, whether you want to concatenate these two different signals. If you hit the middle of a signal wire, the wire is automatically split into two segments and a junction node is generated. In Draw Signal Wires mode, you can manually place such a junction node too, by pressing **[.]**.

Snap-on-Pin: If you come near to a pin, TAGRET shows a little Snap-on-Pin box, which is like a magnet to attract the cursor. You can avoid this magnetism by pressing **[Ctrl]**. TARGET generates a beep, if you hit another signal or pin. The x-cross on a pins head disappears, if the pin gets connected.

Backspace Function: To delete the last drawn segments of a signal, you can use the **[backspace]** key. This works like undo and the start position of the cursor gets adapted too.

With an **M2** click while drawing, you can adjust the bending mode of the signal. An existing signal can be edited by moving of segments or by dragging its corners.

Leave this mode...
- with **[ESC]** or **M12**

2.8.9 Draw Busses (in schematic only)

You can reach this mode...
- with **[B]** or
- with the point Draw Busses in menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options to do bus settings. A **M1** click creates the first segment. Use further **M1** clicks to draw a polyline. A bus in TARGET is a big pool, that contains all signals together. Optically you can distinguish between different busses (e.g. Adresses [A0..A15] Data [D0..D7] Control [Clk, Res, etc]), however every signal can be connected to any bus or be fetched out of any bus segment.

When starting a signal with **[2]** exactly on a bus, you get a list of all signal to choose from. Also a button for the **Bus Assistant** is shown.

Tip: Draw a new unknown signal from a component pin in direction to the bus. Fetch a known signal out of the bus and draw it in driection to the component pin.

Leave this mode...
- with **[ESC]** or **M12**

2.8.10 Place Centering Marks

You can reach this mode...
- with the point Place Centering Marks in menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options to do centering marks settings. Every **M1**
films and masks in production.

Leave this mode...
- with **[ESC]** or **M12**

2.8.11 Draw Filled Triangles

You can reach this mode...
- with **[3]** or
- with the point Draw Filled Triangle in menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options to do triangle settings. Set two further points with **M1** clicks to define the triangle. Existing triangles can be dragged on their corners with the dragging function **[z]**.

Leave this mode...
- with **[ESC]** or **M12**

2.8.12 Draw Filled Rectangle

You can reach this mode...
- with **[4]** or
- with the point Draw Filled Rectangle in menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options to do Rectangle settings. When the rectangle is started with **M1**, move the mouse diagonal to open a rectangle. With a **M1** clicks you can finish the rectangle. Existing Rectangles, that are not rotated by angles other than 90° can be dragged on their corners with the dragging function **[z]**.

Leave this mode...
- with **[ESC]** or **M12**

2.8.13 Draw Filled Slices

You can reach this mode...
- with **[0]** (=Zero) or
- with the point Draw Filled Slices in menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options to do circle settings. Determine the radius of the slice with the second M1 click. Open circles or circle segments are drawn with the Draw Torusses command.

Leave this mode...
- with **[ESC]** or **M12**

2.8.14 Draw Torusses

You can reach this mode...
- with **[T]** or
- with the point Draw Torusses in menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options to do torus settings. The secound point you are clicking with **M1** is the center of the torus, the third one is the endpoint. These arcs are always drawn in mathematical positive sense (counter clockwise).

Leave this mode...
- with **[ESC]** or **M12**

2.8.15 Insert Text

You can reach this mode...
- with **["]** or
- with the point Insert Text in menu Elements or
- with the tool .

What you can do...
Click the point with M1, where the text shall appear. Enter the text an its options in the following dialog. Close the dialog with Ok to place the text. Caution: Only the TARGET font can be mirrored in Win 95/98/ME. The Windows fonts can only be mirrored by the 32-bit graphic of Win NT4/2000/XP. Win 95/98/ME have an old 16-bit graphic, wich is not able to mirror Windows fonts.

The TARGET Gerber driver is not able to output Windows fonts. They are replaced by the TARGET font. Allways check the result of this replacement!

Leave this mode...
- with **[ESC]** or **M12**

2.8.16 Place Pin

You can reach this mode...
- with **[1]** or
- with the point Place Pin in menu Elements or
- with the tool .

What you can do...
Now you can place free pins with **M1** clicks. Those free pins are normally only useful to define symbols that shall be saved as a component into a library. The pin numbers are incremented automatically. Press **[O]** for pin Options.

Leave this mode...
- with **[ESC]** or **M12**

2.8.17 Move Selected Elements

You can reach this mode...
- with **[V]** or
- with the point displace/move [d] in menu Edit or
- directly start moving by pressing **M1H** onto an element or
- with the tool .

What you can do...
Click with **M1** on the selected elements. They disappear and are replaced by their phantom picture. Move the mouse and place them where you like with a next **M1** click. If unselected wires were connected to the selected elements, then they are dragged like rubber wires.

When moving packages with air wires, these air wires are recomputed after every mouse move.

When moving, you can press **[D]** or **M2** to rotate the selected elements without leaving the move mode.

Leave this mode...
- with **[ESC]** or **M12**

2.8.18 Place Junction (in schematic only)

You can reach this mode...
- with **[.]** (=full stop) or
- with the point Place Junctions in menu Elements or
- with the tool .

What you can do...
Press **[O]** for junction node Options. If you place a junction node by a **M1** click, TARGET automatically switches to the mode Draw Signal.

Leave this mode...
- with **[ESC]** or **M12**

2.8.19 Place Vias (in PCB only)

You can reach this mode...

- with **[.]** or
- with the point Place Via in menu Elements or
- with the tool .

What you can do...

Press **[O]** for via Options. If you place a via by a **M1**
connection between signal tracks arises, you are asked wheter this is wanted and wich new signal name shall be used. Vias are visible upon all copper layers (special layer 100).

Leave this mode...

- with **[ESC]** or **M12**

2.8.20 Pin Swap (in schematic) and Pad Swap (in PCB)

You can reach this mode...

- with the point Pin Swap or Pad Swap in menu Actions or
- with the tools oder .

What you can do...

Click the two pins or pads of the same symbol subsequently with **M1**. Both pins have to have the same non-zero swap number. In the schematic only the two pin numbers are exchanged. In the PCB the connected tracks are dragged along like rubber tracks. Caution: Short cuts are possible.

Leave this mode...

- with **[ESC]** or **M12**

2.8.21 Symbol/Gate Swap (in schematic only)

You can reach this mode...

- with the point Symbol Swap in menu Actions or
- with the tool .

What you can do...

Click the two handles of the symbols subsequently with **M1**
schematic only the pin numbers are exchanged, therfore the pin names have to be identical. In the PCB the connected tracks are dragged along like rubber tracks. Caution: Short cuts are possible.

The two symbols may also belong to different components.

Leave this mode...

- with **[ESC]** or **M12**

2.8.22 Dragging of Corners, Arcs and Spline Curves

You can reach this mode...
- with **[Z]** or
- with the point Drag in menu Edit or
- directly start dragging by pressing **[Ctrl]+M1H** onto a corner, line, arc or spline curve.

What you can do...
By dragging, you can change corners, existing tracks, signal wires, lines, arcs, spline curves, triangles and rectangles. If you hit a corner, dragging moves this corner. If you hit a straight line segment, it is ...
- split into two segments (=create new corner)
- changed into an arc or
- into a Bezier/spline curve.

If there are more than one layers used on the dragging start point, the active drawing layer is preferred . However with the **[+]** key you can temporarily change the active layer.

Rectangles can only be dragged, if they are not rotated by angles other than 90°.

Leave this mode...
- with **[ESC]** or **M12**

2.8.23 Export Package (in PCB only)

You can reach this mode...
- with **[X]** onto the position the package handle shall appear or
- with the point Export Package in menu Packages and **M1** onto this position or
- with the tool and **M1** onto this position.

What you can do...
Use this export funtion to add a package into a library. First select the self-drawn or changed package, then start this function.

TARGET doesn't store the layer numbers of the elements but the function of the layers. So later at re-import time the package drawing will always land on the "Position Top" layer, independently how the layers are configured in a different TARGET project. User defined layers are distinguished by layer name.

Leave this mode...
- with **[ESC]** or **M12**

2.8.24 Compose Symbol and Place Handle

You can reach this mode...
- with **[Y]** onto the position the symbol handle shall appear or
- with the point Compose Symbol in menu Components and **M1** onto this position or
- with the tool and **M1** onto this position.

What you can do...
A component in schematic always consists of one or more symbols e.g. gates or OpAmps. A symbol itself is made of drawing elements (lines, arcs, pins etc.). Use this funtion to concatenate some elements to one symbol. First select the self-drawn or changed symbol, then start this function. You can press **[O]** now for symbol options. Click now with **M1** onto that position, where the symbol handle shall appear (the little cross).

If you made a mistake and fetched the wrong elements together as a symbol, then you can repeat this function. If a handle is selected, this will be replaced by the new one.

If you want to export the whole component now then press **[X]**
have to be inserted, reconcatenated with **[Y]** and then exported again. The Edit Component function of the Library Browser prepares every symbol this way, so that it can be reexported directly.

Leave this mode...
- with **[ESC]** or **M12**

2.8.25 Draw Tracks (in PCB only)

You can reach this mode...
- with **[2]** or
- with the point Draw Tracks in menu Elements or
- with the tool .

What you can do...
The connections in your PCBs are done with signal tracks and they behave similar to signal wires in the schematic part.

Press **[O]** like Options to do track settings. If you start a signal on another signal wire or a defined pad, the corresponding signal name is continued (e.g. GND). If you start from a free pad, the signal name is derived from the pin name (e.g. A1). If you hit nothing special when starting the signal, a complete artificial signal name is used (e.g. Sig$17). You can see the signal name in the status line.

If you touch another signal with your new signal wire, TARGET reacts in two ways:
- PCB with schematic: TARGET tells you, that new connections can only be made in the schematic and rejects the track.
- PCB without schematic: TARGET asks, whether you want to concatenate these two different signals. If you hit the middle of a signal wire, the wire is automatically split into two segments.

In Draw Tracks mode, you can manually place a via, by pressing **[.]**
excange tracks layers by **[:]** whitout generating a via. The working layer can also be selected by the layer toolbar and in the top toolbar .

You can insert bridges by pressing the **[B]** key. The actual segement is then still positioned as a track, the next segment gets a bridge.

Snap-on-Pad: If you come near to a pad, TAGRET shows a little Snap-on-Pad box, which is like a magnet to attract the cursor. You

can avoid this magnetism by pressing **[Ctrl]**. TARGET generates a beep, if you hit another track or pad.

Backspace Function: To delete the last drawn segments of a signal, you can use the **[backspace]** key. This works like undo and the start position of the cursor gets adapted too.

With an **M2** click while drawing, you can adjust the "drawing corner" mode of the signal. An existing track can be edited by moving of segments or by dragging its corners.

Leave this mode...
- with **[ESC]** or **M12**

2.8.26 Draw Air Wires (Ratsnest)

You can reach this mode...
- with **[L]** or
- with the point Draw Air Wires in menu Elements or
- with the tool .

What you can do...
In a PCB without schematic, air wires document the concatenation of pads to belong to one signal. To "hang off" one pad from a signal, simply click the pad with M11 and delete its signal entry. Eventually, you have to select the pad with [s] and change it with [e], see 2.7

Leave this mode...
- with **[ESC]** or **M12**

2.8.27 Zoom In

You can reach this mode...
- with **[Pg Dn]** or
- with the point Zoom In in menu View or
- with the tool .

What you can do...
The position that is clicked next with **M1** is enlarged and panned into the middle of the screen. That means more details, less overview. **M2** does the opposite. **[Pg Dn]** has the same function, but it can also be pressed in any other mode, without leavig this mode. So if you are drawing tracks, you can use [PgDn] without loosing your track. The stepping of zooming shall be like 2-5-10-20-50-100-200-500-1000. The status line shows the zooming.

In Windows NT4/2000/XP with their real 32bit graphic there is no limitation in zooming in, Win 95/98/ME with their old Win 3.0 16bit graphic drivers can create **system crashes**, when zooming too far. So if you recognize that the display doesn't work correct in a certain zoom step, then don't zoom in any further! (better zoom out one or two steps)

Leave this mode...
- with **[ESC]** or **M12**

2.8.28 Zoom Out

You can reach this mode...
- with **[Pg Up]** or
- with the point Zoom Out in menu View or
- with the tool .

What you can do...
The position that is clicked next with **M1** is scaled down and panned into the middle of the screen. That means less details, more overview. **M2** does the opposite. **[Pg Up]** has the same function, but it can also be pressed in any other mode, without leavig this mode. So if you are drawing tracks, you can use [PgDn] without loosing your track. The stepping of zooming shall be like 2-5-10-20-50-100-200-500-1000. The status line shows the zooming.

In Windows NT4/2000/XP with its real 32bit graphic there is no limitation in zooming in, Win 95/98/ME with their old Win 3.0 16bit graphic drivers can create **system crashes**, when zooming too far. So if you recognize that the display doesn't work correct in a certain zoom step, then better zoom out one or two steps!

Leave this mode...
- with **[ESC]** or **M12**

2.8.29 Select Elements

You can reach this mode...
- with **[s]** or
- with the point select in menu Edit or
- with the tool .

What you can do...
With select you can select elements, signals or components for further actions and to highlight them optically. If you want to change the width of a signal segment, choose the select command. The status bar shows: Choose elements. Click **M1** on the desired segment. Normally the element starts blinking. If the wrong element blinks, press **[+]** or **[-]** to select another. Eventually press **[O]** like Options, to determine whether only the blinking element is selected or its related elements too (whole component or whole signal). Now press **[Return]** to select the element(s).

The advantage of the select command compared to just clicking **M1**
select single elements even in regions where there are many elements on top of each other on different layers or if you have a zoom step, with little details.

Leave this mode...
- with **[ESC]** or **M12**

2.8.30 Place Drill Hole

You can reach this mode...
- At the moment you have to place a free via for a drill hole.

What you can do...
First place a free via. Double click it with M11, set the desired drill hole, set copper height and width smaller than the hole and set its signal name to zero. See s
Leave this mode...
- with **[ESC]** or **M12**

2.8.31 Turn/Rotate

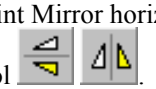
You can reach this mode...
- with **[t]** or **[Shift]**+**[T]** or
- with the point Turn/Rotate in menu Edit or
- with the tool .

What you can do...
Position the cursor onto the rotation centre and click **M1**
angle. TARGET sets this angle to 90° by default. If you want to have another than the current angle, please press **M2** instead of **M1** . A dialog follows, that lets you enter another angle.

You can also directly start rotation with **[t]** or with **[Shift]**+**[T]**, if you want to adjust the angle too.

Leave this mode...
- with **[ESC]** or **M12**

2.8.32 Mirror

You can reach this mode...
- with **[m]** or **[Shift]**+**[M]** or
- with the point Mirror horizontally in menu Edit or
- with the tool  .

What you can do...
Position the cursor onto the desired vertical mirror axis and click **M1**. If you want a horizontal mirror axis then press **M2**. All selected elements are mirrored around this clicked centre.

You can also directly start mirroring with **[m]** for horizontally mirroring using the vertical axis or with **[Shift]**+**[M]**, if you want vertical mirroring using the horizontal axis.

Leave this mode...
- with **[ESC]** or **M12**

2.8.33 Zoom Area

You can reach this mode...
- with **M2h** or
- with the point Zoom Area in menu View or
- with the tool , then using **M1h**.

What you can do...
Choose a rectangle of your circuit to be shown in max scale on the screen.

Tip: If you choose the area **backwards**, that means moving the mouse from right to left, then the whole circuit is shown in max. scale (Win Fit like **[F7]**).
You can also press **M22** for zoom out step by step.
In Windows NT4/2000/XP with their real 32bit graphic there is no limitation in zooming in, Win 95/98/ME with their old Win 3.0 16bit graphic drivers can create **system crashes**, when zooming too far. So if you recognize that the display doesn't work correct in a certain zoom step, then don't zoom in any further! (better zoom out one or two steps)

Leave this mode...
- with **[ESC]** or **M12**

2.8.34 Panorama (Pan)

You can reach this mode...

- with the point Panorama (Pan) in menu View or
- with the tool .

What you can do...
The point next clicked with **M1** will be moved to the center of the screen, the scale remains. This function can also be started with **[p]**. It is useful e.g. when drawing signals, if the aimed point is not on the visible screen any more. Move the mouse to the border of the screen and press **[p]** to advance the paper.

Leave this mode...
- with **[ESC]** or **M12**

2.8.35 Draw Spirals

You can reach this mode...
- with the point Draw Spirals in menu Elements.

What you can do...
First click the center of the spiral with **M1**. If you now move the mouse, you already see the "phantom" picture of the spiral. Now press **[O]** for spiral Options such as number of windings, and direction of rotation. Whit a further click with **M1** the spiral is completed.

Leave this mode...
- with **[ESC]** or **M12**

2.8.36 Teardrops

You can reach this mode...
- with the point Teardrops in menu Elements or
- with the tool .

What you can do...
With a M1 click onto a connected pad, this is converted into a teardrop.

Leave this mode...
- with **[ESC]** or **M12**

2.8.37 Draw Open Triangles

You can reach this mode...
- with **[Ctrl]**+**[3]** or
- with the point Draw Open Triangle in menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options to do line settings. Set two further points with **M1**
of three single lines. Existing triangles can be draged on their corners with the dragging function **[z]**.

Leave this mode...
- with **[ESC]** or **M12**

2.8.38 Draw Open Rectangles

You can reach this mode...
- with **[Ctrl]**+**[4]** or
- with the point Draw Open Rectangle in menu Elements or
- with the tool .

What you can do...
Press **[O]** like Options to do line settings. When the rectangle is started with **M1**, move the mouse diagonal to open a rectangle.
With a **M1** clicks you can finish the rectangle. Existing open rectangles can be draged on their corners with the dragging function **[z]**
.

Leave this mode...
- with **[ESC]** or **M12**

2.8.39 Draw Open Rings

You can reach this mode...

- with **[Ctrl]**+**[0]** (=zero) or
- with the point Draw Open Rings in menu Elements or
- with the tool .

What you can do...

Press **[O]** like Options to do line settings. The circle is started with **M1** which defines the center. The final **M1** click defines the radius. Existing open rings can be draged on their outline with the dragging function **[z]**.

Leave this mode...

- with **[ESC]** or **M12**

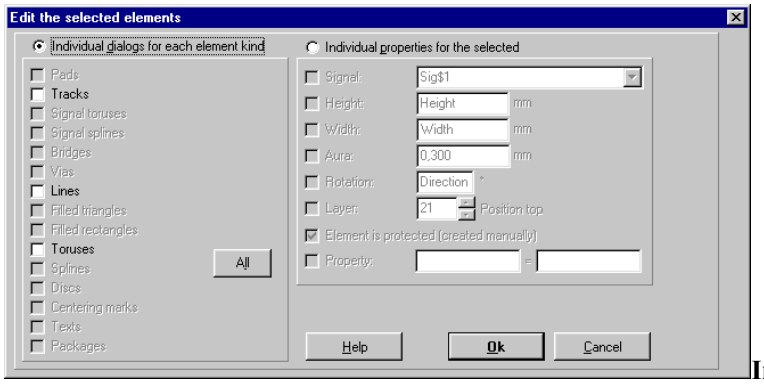
2.9 Help for Dialogs

Contents:

2.9.1 Edit selected elements
2.9.2 Auto Placer
2.9.3 Auto Router
2.9.4 Push Onto the Grid
2.9.5 Search and Replace Component
2.9.6 Reload Component Properties
2.9.7 Export Component
2.9.8 Import Symbols
2.9.9 Delete Library Entry
2.9.10 Check Project (DRC)
2.9.11 Set Rotation Angle
2.9.12 Printing
2.9.13 Personal Layer Functions
2.9.14 Settings / Options...
2.9.15 Catch Options
2.9.16 Color Settings
2.9.17 Package Options
2.9.18 Reload and Replace Packages
2.9.19 Export Package
2.9.20 Import Package
2.9.21 System Info
2.9.22 Air Wires
2.9.23 Set Zoom Factor
2.9.24 New Project
2.9.25 PCB Layers
2.9.26 Check Project
2.9.27 Grid
2.9.28 Rectangle Options
2.9.29 Import Reference Symbol
2.9.30 Reorganisation
2.9.31 Import Rests of Components
2.9.32 Slice Options
2.9.33 Shape Based Autorouter
2.9.34 Rename Signal
2.9.35 Mirror Axis

2.9.36 Spiral Options
2.9.37 Output Bill of Material
2.9.38 Search and Select
2.9.39 Symbol Options
2.9.40 Which Pin is...? (Load Netlist)
2.9.41 Which Component is...? (Load Netlist)
2.9.42 Start Signal from Bus
2.9.43 Drawing Layer
2.9.44 Track Options
2.9.45 Pad Options
2.9.46 Centering Marks Options
2.9.47 Signal Wire Options
2.9.48 Line Options
2.9.49 Text Options
2.9.50 Torus Options
2.9.51 Concatenate Signals
2.9.52 Pin Options
2.9.53 Library Browser
2.9.54 Property Editor
2.9.55 Ground Plane Assistant
2.9.56 Engraving Isolation Channels
2.9.57 Spline Options
2.9.58 Select Schematic Page
2.9.59 Bus Assistant
2.9.60 Read Gerber / Excellon
2.9.61 Contour Auto Router

2.9.1 Edit selected elements



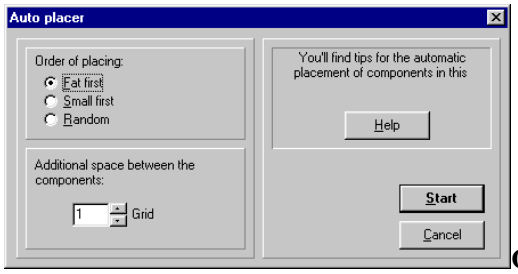
Individual dialogs for each element kind:
Left side of the dialog: You will get successively single change-dialogs for every of the selected element types.

Individual properties of the selected:
Right side of the dialog: Here properties that are common to different element types can be set together in one dialog. **Caution:** If you change to this right side, all element types on the left will get checked, even if you manually unchecked some previously!

Example: You want to move a complete schematic from page 1 to page 7:

Select the complete page 1.
Click point **Change** in menu **Edit**.
Click **Individual properties of the selected**.
Check **Layer** and enter the 7 as **Schematic page**.
Click **Ok**.
Click point **Go to Page** in menu **View** and select page 7.

2.9.2 Auto Placer



Order of placing: Define the ranking of components for auto placing. "Fat" is equivalent to "has many pads".

Additional space between the components: If you choose zero the components are put as close together as possibel. This leaves little space for routing. Probably you should use a value greater than zero.

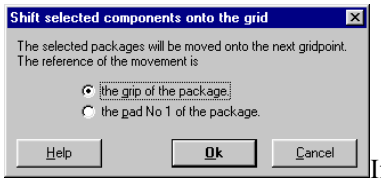
The TARGET autoplacer places every component according to shortest air wires. Please take into account:

- Before launching the placer, you must define a board outline (normally an open rectangle on layer 23).
- The placer uses the library and package that is proposed in the component properties in schematic. Change this value first to use different packages.
- Place connectors and the biggest ICs first manually and spread them over the PBC before launching the auto placer.
- Also blocking capacitors should be placed manually near the power pins of the corresponding ICs.
- TARGET tries to rotate the packages in 90° staps. You can restrict the rotation possibilities to "No rotation" or "180° only". Enter the following line into the component properties: "AUTOPLACER_ROTATION=360" or "AUTOPLACER_ROTATION=180".

2.9.3 Auto Router

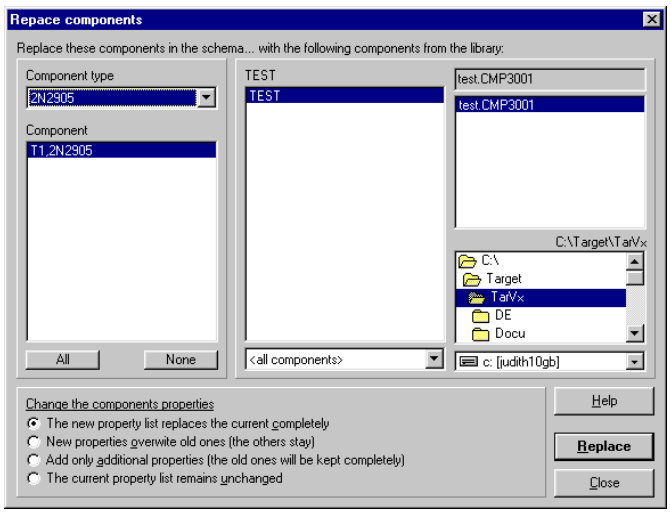
See chapter 6.4

2.9.4 Push Onto the Grid



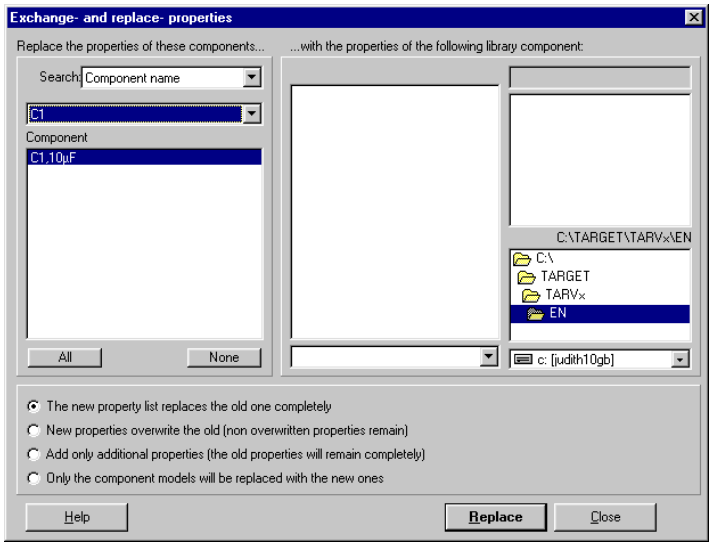
If a component has been moved off the grid and it shall be moved back onto the grid, you can use this function. You can choose, whether the handle or handle of the component or if pad No 1 is the reference of that movement. Anyway the component is moved less than on grid step in X- and Y-direction.

2.9.5 Search and Replace Component



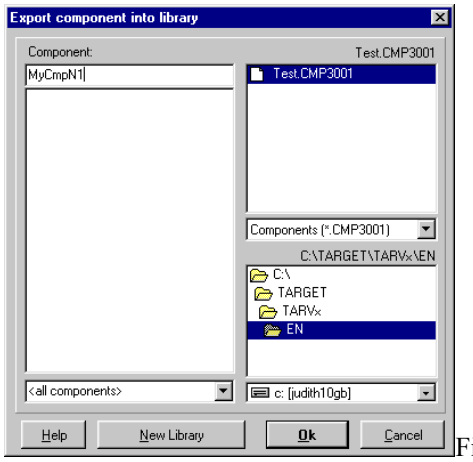
dialog. On the right side you can choose the component to replace with. On the bottom you can select how the component properties are treated.

2.9.6 Reload Component Properties



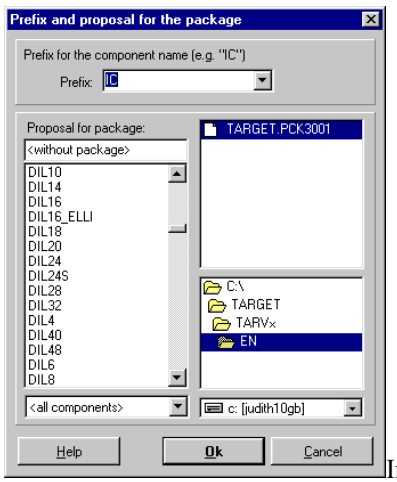
dialog. On the right side you can choose the component to take the new properties from. On the bottom you can select how the component properties are treated.

2.9.7 Export Component



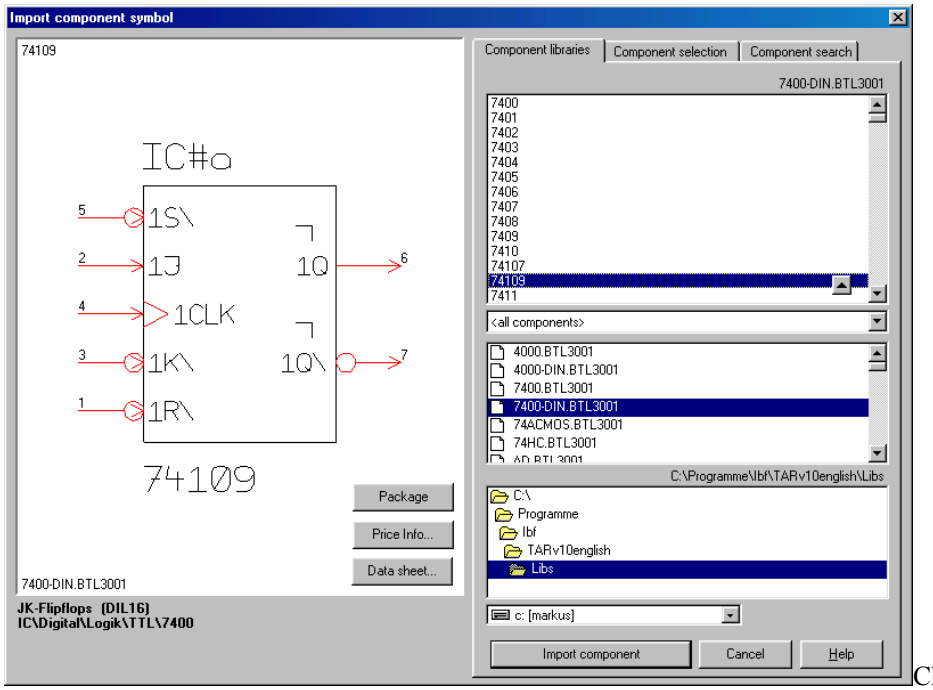
First select a library and enter a matching component name. You can create a new library

too with the corresponding button.



In this second dialog you have to enter a matching prefix such as 'IC' or 'R' and select the corresponding package. If you enter no package or select **<without package>**, the component will not need and will not be able to have a package at all. (e.g. schematic page frames etc.)

2.9.8 Import Symbols



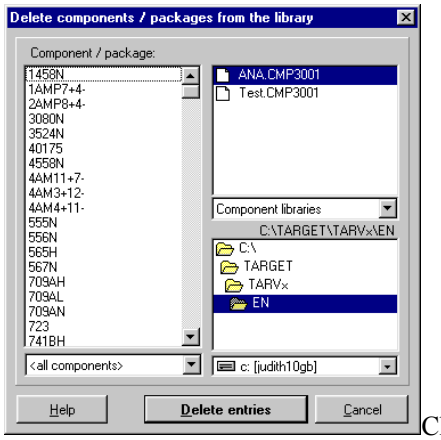
Choose a library on the right side and select a component in the middle.

When opening a library the very first time, an index file like "ANA.CMP3001IND" is automatically created. This index file speeds up the access to that library. If any errors occurr concerning libraries like "read unknown object" then try to delete all index files. These are restored and updated by TARGET whenever needed.

You can also use links to the libraries: ANA.CMP3001LNK. These link files must be normal text files (e.g. from NOTEPAD.EXE) containing one line with the correct path and name of the file:

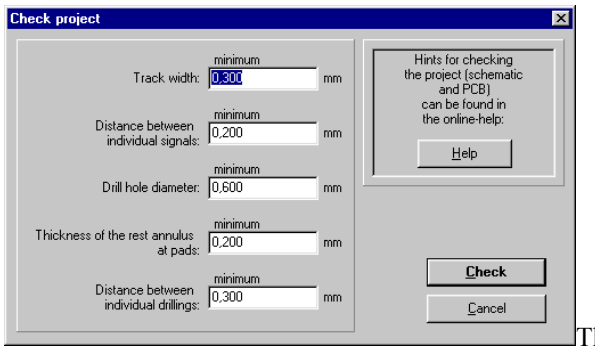
\\server\d\libraries\ANA.CMP3001

2.9.9 Delete Library Entry



Choose a library on the right side and select one or more components on the left. Press "Delete entries" to extinguish the selected components.

2.9.10 Check Project (DRC)



The design rule check (DRC) finds spacing and minimal measures errors in the PCB. The errors of the electrical rule check are found too. All errors are listed and may be saved into a file. Some statistics for the PCB are shown at the end of the list. The errors that correspond with a certain position in PCB are marked by an error sign. A double click with M11 onto such an error flag shows the exact error text.

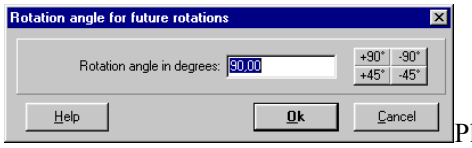
You can find the description of all error messages in chapter 6.6.

Thickness of the rest annulus of pads: Let the pad diameter be 1mm and the drill hole diameter be 0.6mm then the thickness of the annulus is 0.2mm.

Distance between individual drillings: Let the distance between two holes be 1mm and the diameters are 0.8mm and 0.6mm then the distance is 0.3mm.

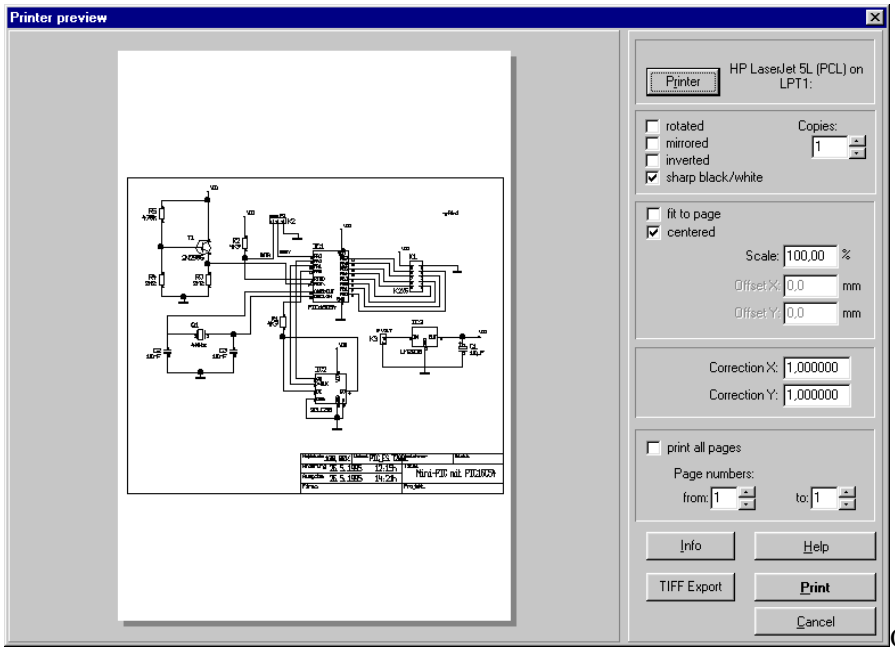
CAUTION: The DRC does not check the ground planes at the moment. So if you used the Ground Plane Assistant and insert packages afterwards, remember to give the newly inserted pads an extra aura!

2.9.11 Set Rotation Angle



Please enter the relative rotation angle for future rotations. All real number angles are valid.

2.9.12 Printing



On the left side you see your project in preview. You see exactly these layers or sheets that were visible in the active window (WYSIWYG what you see is what you get).

The printed area is computed from all visible parts, so that you probably should switch the outline layer visible to have everything on the same position during multiple sheets.

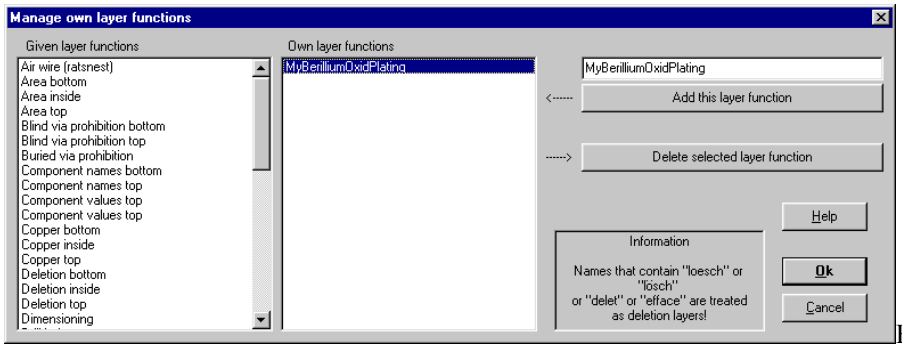
The **Scale** can be entered in percent. Additional you can enter separate **Correction** factors for both X and Y axis, if the output devive causes anisotropic distortions. With the **Offset** parameters you can shift the drawing on the sheet manually. **Centered** brings the drawing into the middle of the sheet and **fit to page** shows it as big as possible.

All colors may be converted to black using **sharp black/white**.

CAUTION: If you don't see no white drill holes in your black pads, probably the parameter **Drill holes as pictograms** in the TARGET menu Window/Settings is checked.

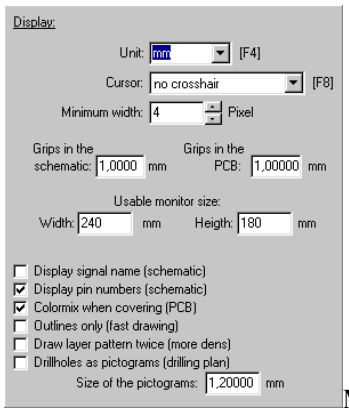
2.9.13 Personal Layer Functions

Pressing the button New layer functions in the big layer dialog, TARGET shows the following editor for personal layer functions:



Enter the desired layer function name into the edit field top right and press **Add this layer function**. The new layer function is added in the middle. Also the packages in libraries can work with these new functions. Regard, that the correspondance is not built via the layer number but by the **exact identity of the layer name**! So if you imagine MyBerilliumOxydPlating, please remember that name well!

2.9.14 Settings / Options...



Minimum width: This entry determines up to how many pixels a line is drawn as a one pixel line instead of a filed area. Due to rounding errors it can occurr that a filled area gets a width of zero pixels. The result would be: From a certain scale step some lines are omitted from drawing. To prevent this, set Minimum width to e.g. 4.

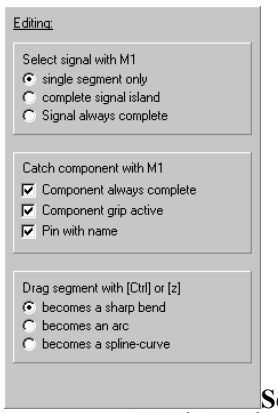
Handles in the schematic / PCB:

Usable monitor size: Enter here the size of your whole visible screen in mm. TARGET computes all screen outputs in that way that at scale=100% 10cm on the PCB are exactly 10cm on the screen (world-coordinates). You get a very real impression of how your PCB will be even though it doesn't exist jet.

Colormix when covering: If two layers overlap in your PCB normally the top layer is visible, the other layer is hidden. Using this option will mix the colors of the layers so that one can recognize where and how the tracks overlap.

Outlines only: This option tells TARGET not to fill lines and polygons but only to draw their outline onto the screen. This results in faster drawing and in a kind of transparency of the layers.

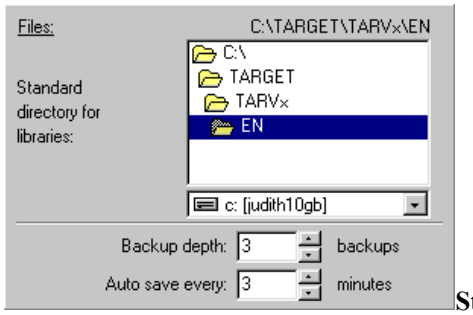
Drillholes as pictograms: This option is to create a drilling plan for manual drilling. Instead of holes, TARGET draws little symbols as points, triangles, asterixes and so on to flag the drill diameters at the different positions. This is the main cause for the **printing problem**: I cannot see the white holes in my pads any more!



Select signal with M1: What will get selected if you click onto a signal wire or track with M1. Corresponds to the tools , and .

Catch component with M1: What will get selected, if you click elements of a component with M1. Corresponds to the tools and .

Drag segment with [Ctrl]+M1h or with [z]: What happens, if a segment is dragged in the middle. Corresponds to the tools , and .



Standard directory for libraries: Choose the path, where all your libraries reside. Normally C:\Program Files\ibf\target\

Backup depth: When you save your project manually, TARGET always keeps the elder version as a backup file. How many backups are saved, is determined by that entry. If the number is 2, the backups will be as follows:

- TEST.T3001
- TEST.BK1
- TEST.BK2

Files with higher numbers are the older ones.

Auto save every: Here you can enter a period between 1 and 99 minutes for automatic saves of your project. Zero means no auto saves. The following files will be created:

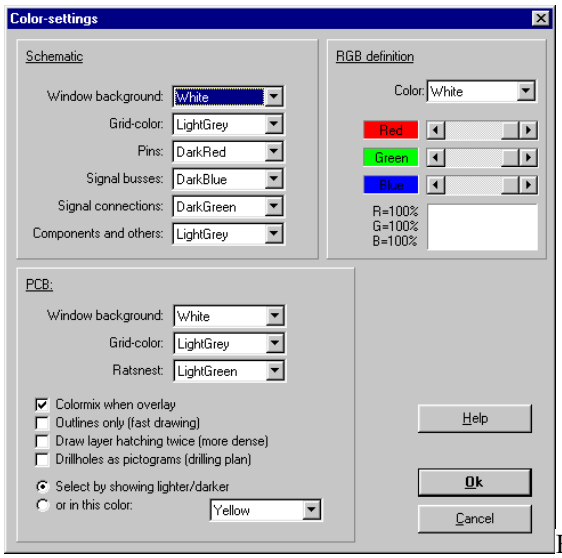
- TEST.T3001
- TEST.T3001a
- TEST.T3001b

The "b" file is the oldest one.

2.9.15 Catch Options

- **Catch additionally to the already selected:** The elements already selected stay selected
- **Catch pins with the pin name:** The pin name gets always selected too, when the corresponding pin gets selected.
- **The handle selects the symbol/package completely:** Corresponds to .
- **Symbols and packages will always be selected completely:** Corresponds to .
- **Select an island of a signal completely:** Corresponds to .
- **Select signals complete:** Corresponds to .

2.9.16 Color Settings



Here you can assign colors to all things that are not dependant on a layer color. You can select one out of 16 colors. Even the colors themselves can be adjusted.

Colormix when overlay: If two layers overlap in your PCB normally the top layer is visible, the other layer is hidden. Using this option will mix the colors of the layers so that one can recognize where and how the tracks overlap.

Outlines only: This option tells TARGET not to fill lines and polygons but only to draw their outline onto the screen. This results in faster drawing and in a kind of transparency of the layers.

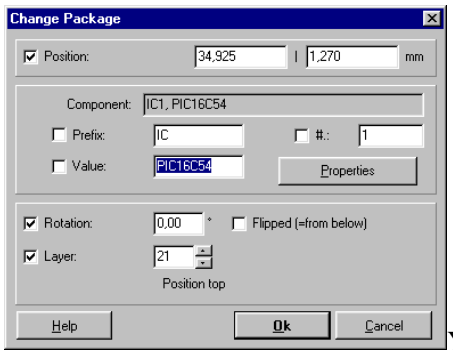
Drillholes as pictograms: This option is to create a drilling plan for manual drilling. Instead of holes, TARGET draws little symbols as points, triangles, asterixes and so on to flag the drill diameters at the different positions. This is the main cause for the **printing problem**: I cannot see the white holes in my pads any more!

Select by showing lighter/darker: If this option is checked, elements in the PCB are selected by highlighting. So a dark color turns into a light color and vice versa.

Select by showing in this color: If this option is clicked, the selected objects turn into the specified color.

- **Caution:** When playing with colors, please always think of selection by highlighting too! So if you select a dark green signal wire in the schematic it turns into light green. If you e.g. change the background to black, then a light gray IC symbol may be

2.9.17 Package Options



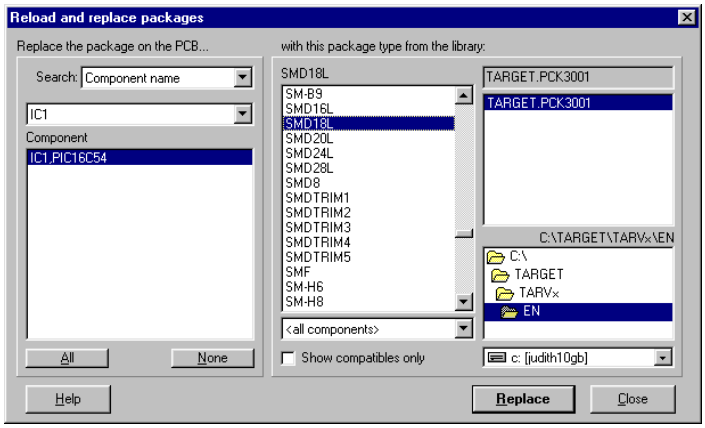
You can adjust the **position** of the whole package, by changing this value.

The component name consists of **prefix** (IC) and **number** (1): IC1.

The **value** of a component is for a resistor e.g. 4k7 and for an IC e.g. CD4040.

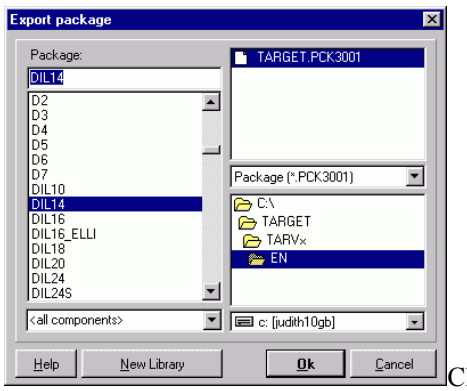
Flipped means the component is mounted from down under, e.g. a SMD component. This field is set automatically and used in the bill of material for automatic mounting.

2.9.18 Reload and Replace Packages



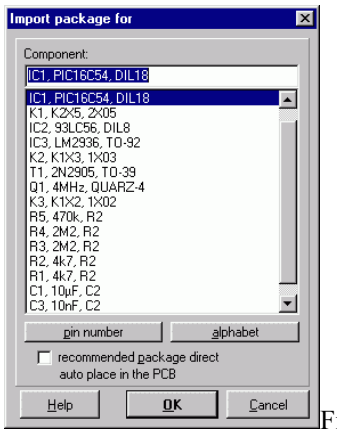
of the dialog, then choose a library and a package to replace them.

2.9.19 Export Package

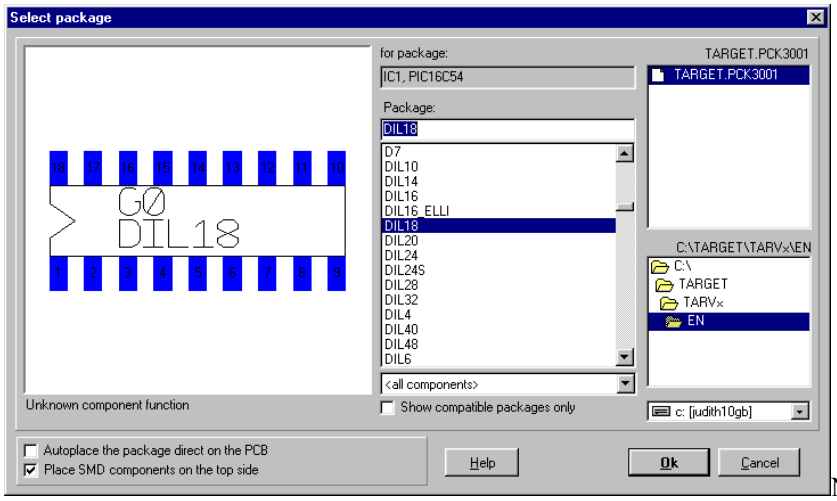


Choose a destination library and enter a meaningful package name. You can also create a **New Library**.

2.9.20 Import Package



First select the component to be placed next. The components can be sorted according to the alphabet or to the respective pin number.



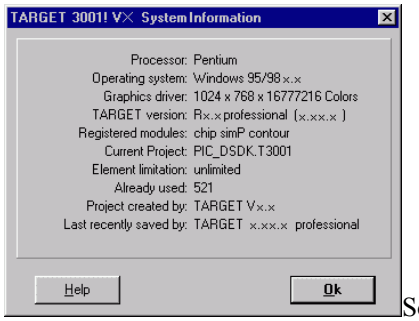
Next you have to choose the corresponding package. Normally TARGET proposes a useful package and shows it directly.

When opening a library the very first time, an index file like "TARGET.PCK3001IND" is automatically created. This index file speeds up the access to that library. If any errors occurr concerning libraries like "read unknown object" then try to delete all index files. These are restored and updated by TARGET whenever needed.

One can also use links to the libraries: TARGET.PCK3001LNK. These link files must be normal text files (e.g. from NOTEPAD.EXE) containing one line with the correct path and name of the file:

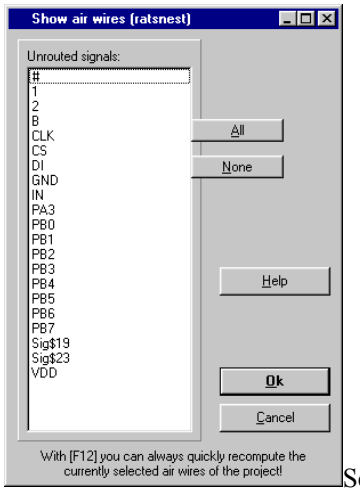
\\server\d\libraries\TARGET.PCK3001

2.9.21 System Info



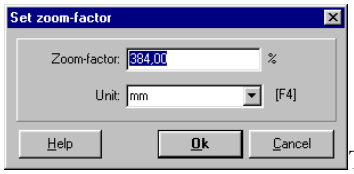
Some information on your system are listed here. One Info is the the usage of the TARGET element database. This is very important for versions with a limited number of elements.

2.9.22 Air Wires



Select the air wires you want to see next. Due to the fact, that TARGET sometimes recomputes all air wires, you can just press **[F12]** to get the old view.

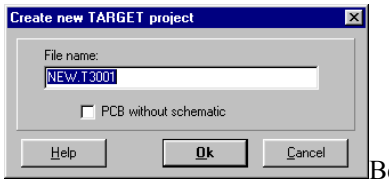
2.9.23 Set Zoom Factor



This is the scale for the circuit in the current window. 100% means world coordinates see also Chapter 2.9.14.

Don't enter too big values for the scale, as errors in Win 95/98/ME can occur.

2.9.24 New Project

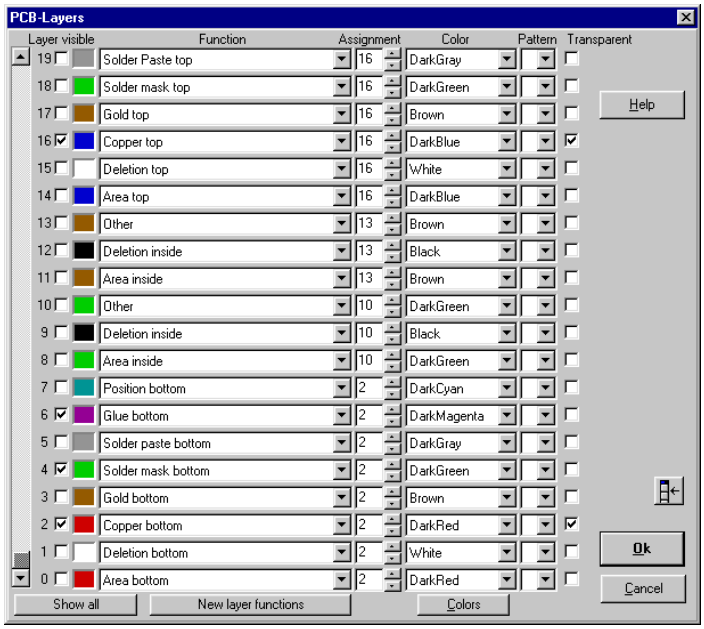


Before creating a new project, please give it a meaningful name. If you want a **PCB without schematic**

TARGET now opens the file DEFAULT.T3001 *) and gives it the name you entered. If you want to change the defaults, so please edit that file.

*) DEFAULTP.T3001 for PCB without schematic.

2.9.25 PCB Layers



Every layer can be switched **visible** and invisible.

The following predefined layer **Function**s are possible:

- **Copper (bottom, inside, top):** For pads, tracks etc.
- **Position (bottom, top):** For packages and other paintings and texts.
- **Deletion (bottom, inside, top):** For deletions in areas, especially for automatic ground planes.
- **Area (bottom, inside, top):** For automatic ground planes.
- **Solder mask (bottom, top):** For solder stop varnish.
- **Drill holes:** For drill holes in pads and vias etc. Not galvanized drill holes can be output separate by the drill driver.
- **Solder paste (bottom, top):** For SMD components when mounted and solderd in "reflow" techinique. Caution, the solder paste is not shown in TARGET directly like the solder stop mask. The solder paste mask is generated in the Gerber driver of TARGET!
- **Gold (bottom, top):** For a partial gold plating.
- **Glue (bottom, top):** For SMD components when mounted and solderd in "wave" technique.
- **Dimensioning:** For documentation purposes.
- **Outline:** For the drawing of the PCB outline. The middle of the lines count. The outline doesn't have to be rectangular, you can use curves, break-outs or non-round holes inside.
- **Milling:** For the exact way of the milling heads. Normally: Do not use!

- **Pad numbers:** Shows the pin numbers inside the pads.
- **Air wires (Ratsnest):** Shows the unfinish signal nets as optimized air wires.
- **Errors:** Used by the Design-Rule-Check.
- **Signal names:** Shows the signal name in the middle of every track segment.

- **Route prohibition:** The auto router is not allowed to place his tracks here.
- **Route permission:** The auto router is allowed to place his tracks here.
- **Place prohibition:** The auto placer is not allowed to place components here.
- **Place permission:** The auto placer is allowed to place components here.
- **Via prohibition:** The auto router is not allowed to place vias here.
- **Via permission:** The auto router is allowed to place vias here.

- **Other:** Layer without a specific function. Set copper layers that you don't need to function Other.
- **Free defined layer functions:** chapter 2.9.13.

The entry **Assignment** is important for automatic ground planes (area and deletion), for solder stop masks and for solder paste masks. These layers all need a copper layer to refer to.

There are 16 **Color**s to choose from. For colors see Chapter 2.9.16.

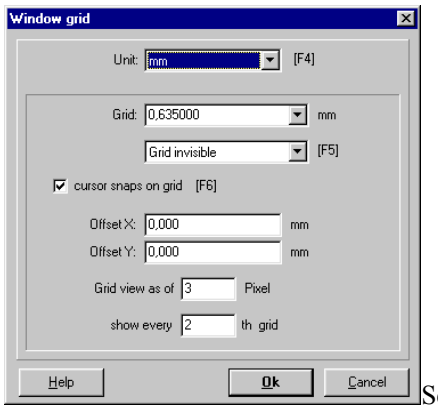
A **Pattern** can be assigned to every layer. Normally the Layers are all solid. Hatching layers brings a kind of transparency.

Transparent says, whether the layer is able to interfer with other layers when overlapping. See Color mix in Chapter 2.9.16.

2.9.26 Check Project

See Chapter 6.6.

2.9.27 Grid



Select one of the following display **Unit**s: Inch, mil, µm and mm.

Enter a **Grid** value bigger than zero and select how the grid shall be displayed. An invisible grid can be active too, dependent on **cursor snaps on grid**!

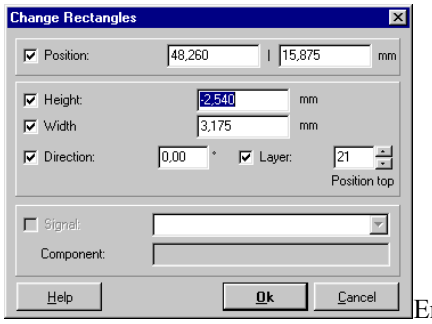
Normally you should work with a grid value of 0.635mm, for our complete library is tuned to that value. You should have good arguments to use another value.

With **Offset X/Y** you can enter a displacement of the grid relative to the point (abs 0/0).

Grid view as of n pixel says that the grid is not displayed, if it is more dense then 3 pixels. This saves TARGET from drawing 1 million grid lines or grid pixels.

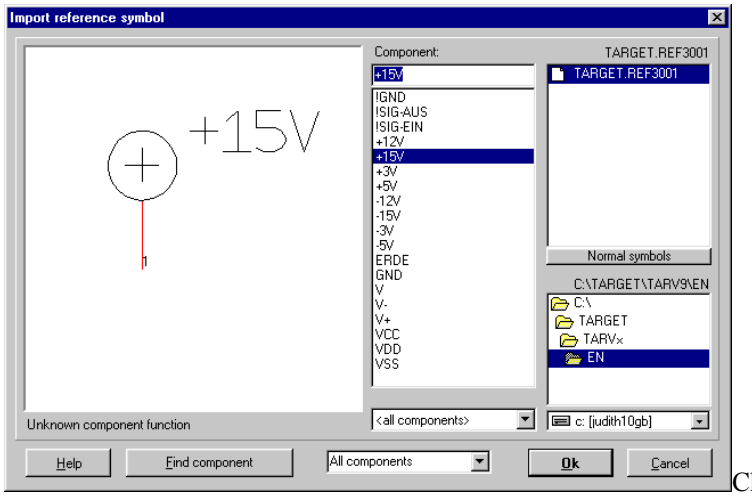
Show every n th grid determins that not every line or dot has to be drawn.

2.9.28 Rectangle Options



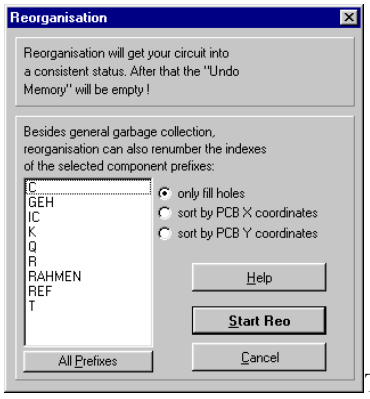
Enter the properties of the rectangle. Direction means rotation.

2.9.29 Import Reference Symbol



Choose one of the reference symbols. Symbols with an exclamation mark at the beginning care for the fact that TARGET directly asks for a signal name, when imported. The others keep the names first, but can be changed afterwards too.

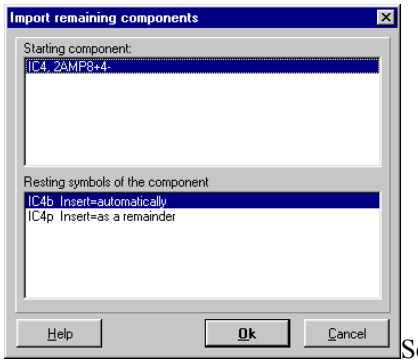
2.9.30 Reorganisation



The TARGET reorganisation is able to renumber the different indexes of the different prefixes. Regard, that the automatic numbering works with PCB coordinates only. So if you have a schematic without a PCB, renumbering by coordinates will not work.

The undo buffer is cleared by reo too!

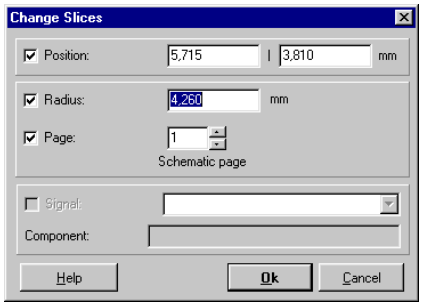
2.9.31 Import Rests of Components



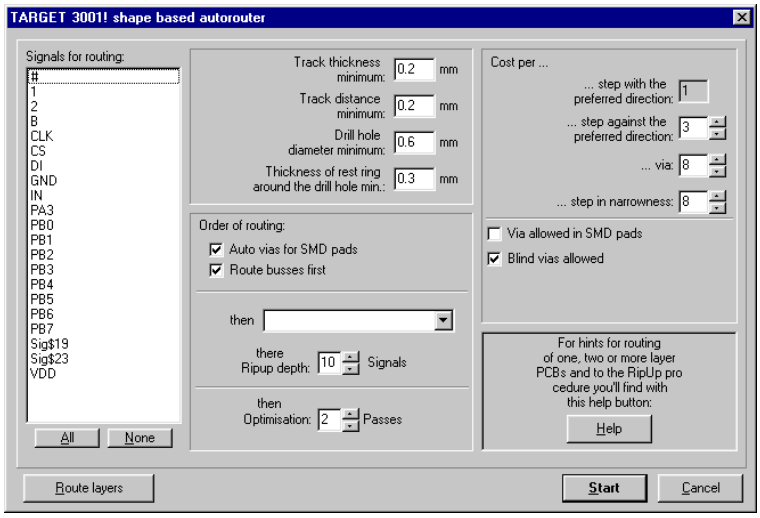
Select in the list above, which component and in the list below which resting symbol of that component shall be imported. TARGET naturally shows only components that are not completely used up (rests of components do exist) .

Usually the power symbols are drawn together with their respective blocking capacitors somwhere in a corner of the schematic or on a page far behind. By drawing this way, the "logic" of the scheme is not disturbed by the power wires.

2.9.32 Slice Options



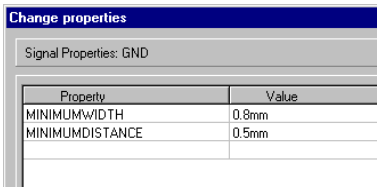
2.9.33 Shape Based Autorouter



the design rules are defined.

If you want another track width for several signals, you have to define this in the schematic before. The signal properties may

contain the following entries:



The **Order of routing** is as follows:

- **Auto Vias for SMD pads:** Places fanout vias for critical SMD pads. The router connects a via to every pad so that the pad can be accessed from every layer.
- **Route busses first:** Routes parallel tracks first. Interesting for RAM / ROM routing.
- **Long / Short signal first / Random selection:** The tracks are now selected and routed by a euristic method.
- **Ripup depth:** The router now places as many tracks as are possibible whithout violating rules. If it finds a track to be unroutable, it selects other tracks to ripup them and tries to route the previous signal now. If this works, TARGET retries to route the ripup signals again. Ripup depth defines how many signals may be ripped up just to route one other signal.

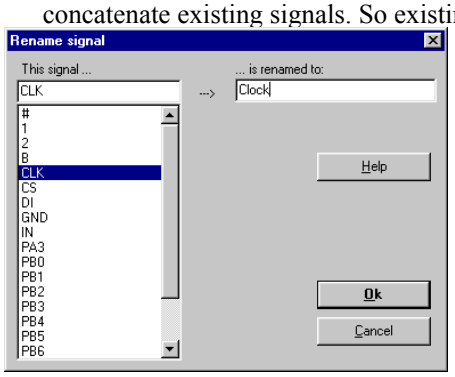
- **Optimisation:** TARGET now has some optimisation runs, just to tighten the tracks and to reduce vias. During optimisation the preferred directions of the layers are not effective.

- **Cost per step...:** If the router works on more than one layer, those get different orthogonal preferred directions. You can define what price a step against such a preferred direction costs, compared to a step with a correct direction. Also the prices for vias and extra costs for steps in extreme narrowness can be determined.

2.9.34 Rename Signal

There are two different possibilities to rename a signal:

- **Rename a signal island:** Click **M11** onto a segment of the island. Enter a new signal name or select an existing one. The whole island of that signal (the optically directly connected part) is moved to another or to a new signal.
- **Rename Signals completely** **:** This function renames existing signals with new signal names. It cannot be used to concatenate existing signals. So existing names are rejected. The following dialog is used:

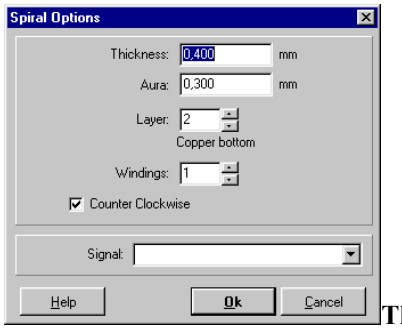


Caution: Changes of signals in the schematic are only transported to the corresponding pads! The tracks remain the old signals. So you can reanimate some tracks by clicking the islands in PCB with **M11** and giving them the new signal too. Errors are reported by the design rule check or electrical rule check.

2.9.35 Mirror Axis

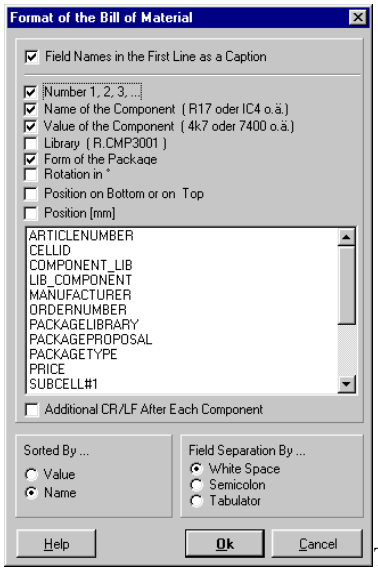
Please enter the mirror axis for the next mirroring.

2.9.36 Spiral Options



Thickness: Track width.
Aura: Width of the distance to an automatic ground plane.
Layer: Which copper layer shall the spiral be drawn on.
Windings: How many windings shall the spiral have?
Counter Clockwise: Which direction (inside-out) shall the spiral spin?

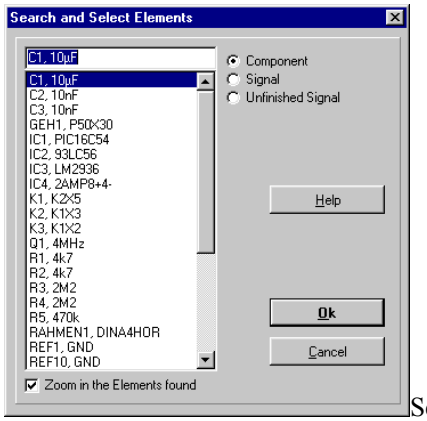
2.9.37 Output Bill of Material



The bill of material is created as an ASCII text file, that contains line oriented entries. Bills of material are useful for technical documentations and for economic purposes as well. You can select some predefined properties to be output and all of your self defined component properties too!

The output is normally importable for database programs and also for automatic component mounting machines.

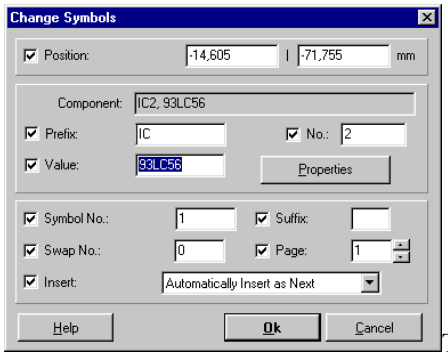
2.9.38 Search and Select



Select whether you want to search for **Component**s, **Signal**s or **Unfinished Signal**s. Then choose the corresponding element from the list.

If **Zoom in the elements found** is checked then the respective page is activated and the desired objects are zoomed in. If the objects lie on more than one schematic pages, TARGET opens the first and reminds you of the others.

2.9.39 Symbol Options



The **Symbol No** should start with 1 for every component. The **Suffix** is normally a non capital letter starting at 'a'. It names the symbols within the component: e.g. IC1a. A power symbol often gets the suffix letter 'p'.

The **Swap No** is for gate swapping. If the number is greater than zero and equal to the number of another symbol, the symbols may be exchanged (gate swapping).

The **Insert mode** has the following meaning: **Automatically Insert as Next** means that this symbol/gate is always imported in the normal order when the user starts an "Import Symbol" function. **Extra Insert as Rest** is for symbols that shall only be inserted by "force". The user has to "Import Rests of Components" to get these extra symbols. This option is often used for power symbols.

Caution: At least on symbol must have **Automatically Insert as Next**. Otherwise the whole component is unusable!

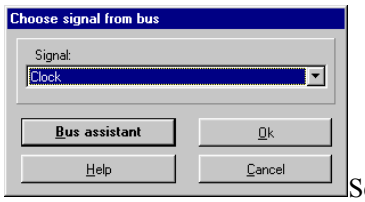
2.9.40 Which Pin is...? (Load Netlist)

Select one of the pins listed. This pin is set equivalent to the pin needed. This question can occur due to different pin naming in your project and the netlist read: e.g. Reset=RES=Clear=CLR...

2.9.41 Which Component is...? (Load Netlist)

Select one of the components listed. This component is set equivalent to the component needed. This question can occur due to different compunent naming in your project and the netlist read: e.g. IC1, U1, X1 ...

2.9.42 Start Signal from Bus

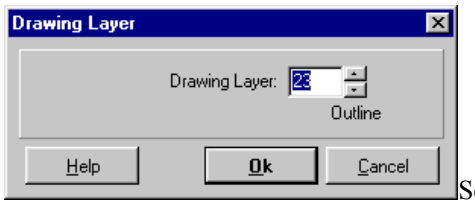


Select signal to fetch out of the bus and press Ok.

A bus in TARGET is a big pool, that contains all signals together. Optically you can distinguish between different busses (e.g. Adresses [A0..A15] Data [D0..D7] Control [Clk, Res, etc]), however every signal can be connected to any bus or be fetched out of any bus segment.

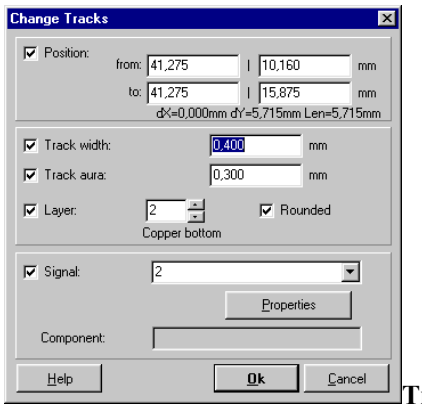
You can also run the **Bus assistant** from this dialog. See Chapter 2.9.59.

2.9.43 Drawing Layer

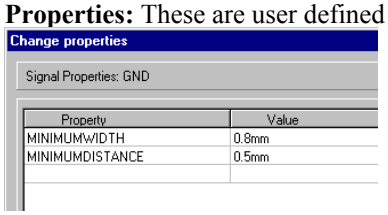


Select the layer to draw on.

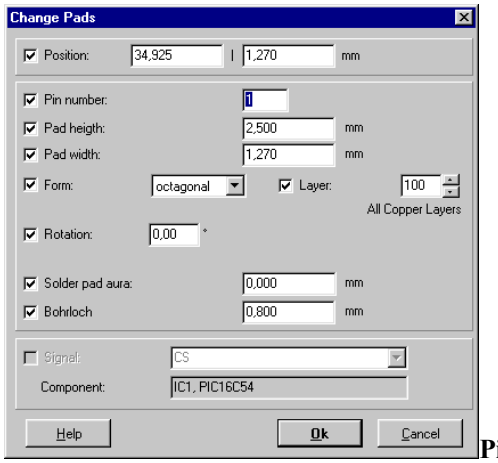
2.9.44 Track Options



Track aura: Width of the distance to an automatic ground plane.
Rounded: The ends of the track segment are with round caps. This causes smart connections of segments with sharp bends.
Properties: These are user defined properties for the whole signal. Two properties are very important for the auto router:

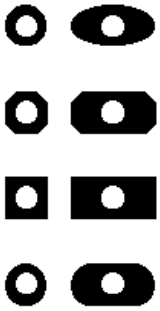


2.9.45 Pad Options



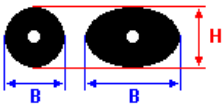
Pin number: created. If the pin number is missing, TARGET estimates this pad as a via.

Form: The following pad forms are possible:



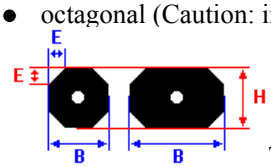
H = Height and B = Witdth of the Pad. The Coordinate X|Y gives the middle of the Pad. An optional rotation is executed after height and width are determined. Is height > width then these two values are exchanged and the rotation is enlarged by 90°.

- round (for round and elliptic pads)



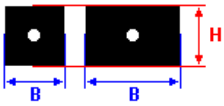
B = H results in a circular pad (=ROUND). Elliptic pads are rather unusual. Therfore in Gerber export elliptic pads are often changed into oval ones (=OBLONG). But unfortunately this can result in small spacing violations.

- octagonal (Caution: in TARGET height and width may differ!)



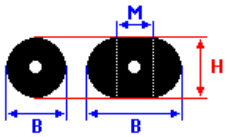
The cut corner has a value of E = Min (B, H) / 4. Therefore in the special case of B = H a uniform octagon (=OCTAGON) is **not** the result. But in Gerber output this can safely be exchanged by a uniform octagon, for the substitute is smaller than the original. Long octagons are seldom too in the Gerber world so these are also often replaced by ovals (=OBLONG). Oblongs are smaller than TARGET's long octagons so there is no danger of spacing violation too.

- rectangular (for squares and rectangles)



Naturally the corresponding Gerber apertures are squares and rectangles (=SQUARE or RECTANGLE).

- oval (for oblong pads)



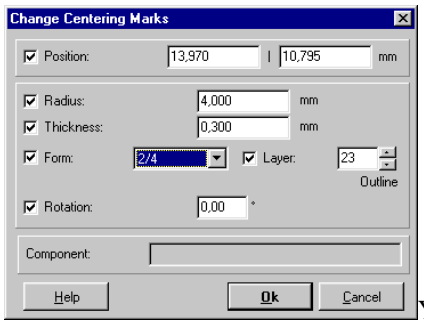
For B = H also a circle pad results (=ROUND). In every other case there is a rectangle nescessary in the middle. It has two half-circle caps right and left. The value of the rectangle is M = Max (B, H) - Min (B, H). For Gerber-output a long round aperture is used (=OBLONG).

Pad aura: Width of the distance to an automatic ground plane.

Drill hole: Diameter of the drill hole. SMD pads have a drill hole of zero.

Layer: For SMD pads a single layer like 16 is entered. For through hole pins, the special layer 100 is used, which tells the pad to appear on all copper layers. Numbers bigger than 100 are extra pad stacks for blind and buried vias.

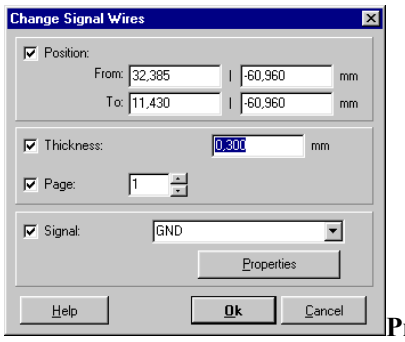
2.9.46 Centering Marks Options



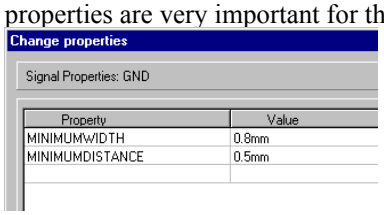
You can enter the **Radius** and select from the following **Form**s of centering marks (fiducials):

- "Target": "Arrow": "2/4":

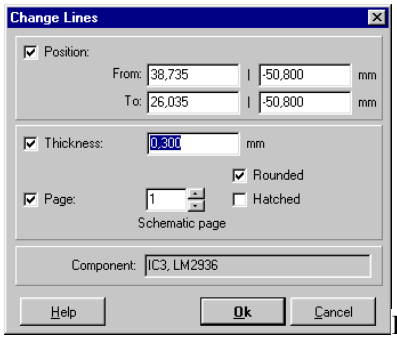
2.9.47 Signal Wire Options



Properties: These are user defined properties of signal wires for the whole signal. Two properties are very important for the auto router:

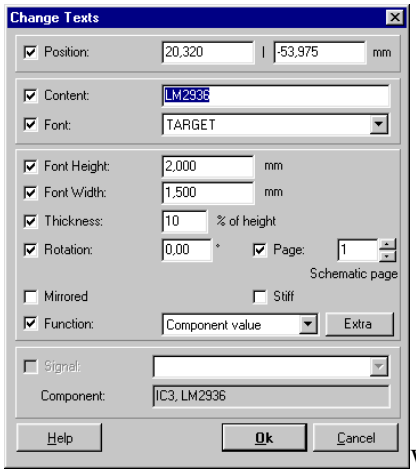


2.9.48 Line Options



Rounded: The ends of the line segment are with round caps. This causes smart connections of segments when sharp bending.
Hatched: Hatched lines can be used for the force lines between corresponding potentiometers and between inductivity and contact of a relay (line options).

2.9.49 Text Options



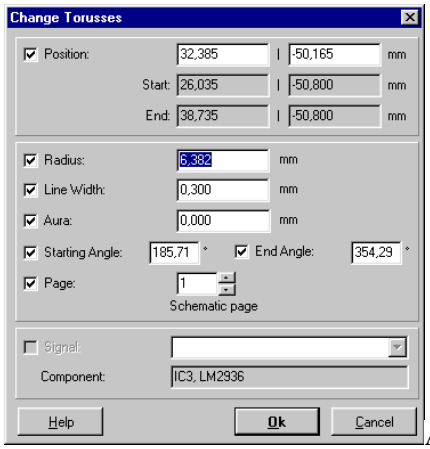
When selecting a **Font**
Windows NT is capable of mirroring Windows fonts. So in schematic you normally can use any font you like, however be cautious to use Windows fonts in PCB.

Only the TARGET font uses **Thickness** for the character lines directly. The Windows fonts switch from normal to bold between 10% and 20%.

When rotating texts by any angle, they are adjusted in the way that one can allways read then in normal view or in view from the right side. If you check **Stiff,** then this rule is switched off. The text rotates stiff, according to the angle. Eventually someone can better read it from the top or from the left side.

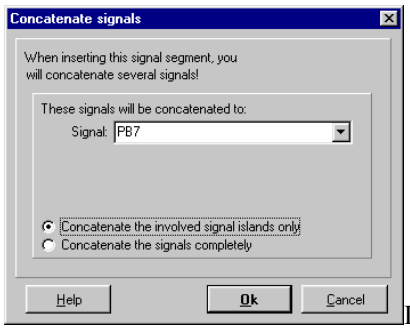
You can also choose a special text **Function**. When selecting somthing else than **Normal text**, then the **Content** is adjusted too. These text functions are e.g. for components or for sheet frames.

2.9.50 Torus Options



Aura: Width of the distance of the torus to an automatic ground plane.

2.9.51 Concatenate Signals

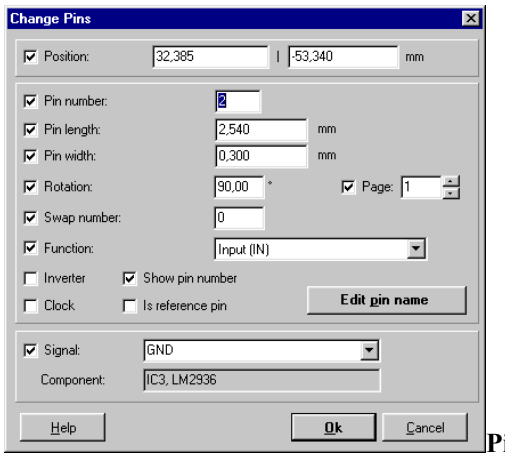


Do you want to connect the different signals? No? Then quick press Cancel.

If you want to connect them, you have two possibilities. First select the name for the resulting signal. Then you can

- **Concatenate the involved signal islands only**.
- **Concatenate the signals completely**.

2.9.52 Pin Options



Pin number:
package is this number. Pin names are not regarded. Pad names like A1-A20 and B1-B20 and C1-C20 are not regarded.

Inverter: The pin gets an inverter point.

Function: Defines the function of the pin. **Power pin** means that this is a pin that needs power. **Power (supply)** means that this pin

gives power. Normally a connector pin is set to Power (supply). Power comes through this pin onto the PCB. Also pin 2 of a 7805 is a power supply pin. It supplies the other ICs with power. Their Power pins are consumer pins.

Clock: A little arrow shows that this pin is a clock input.

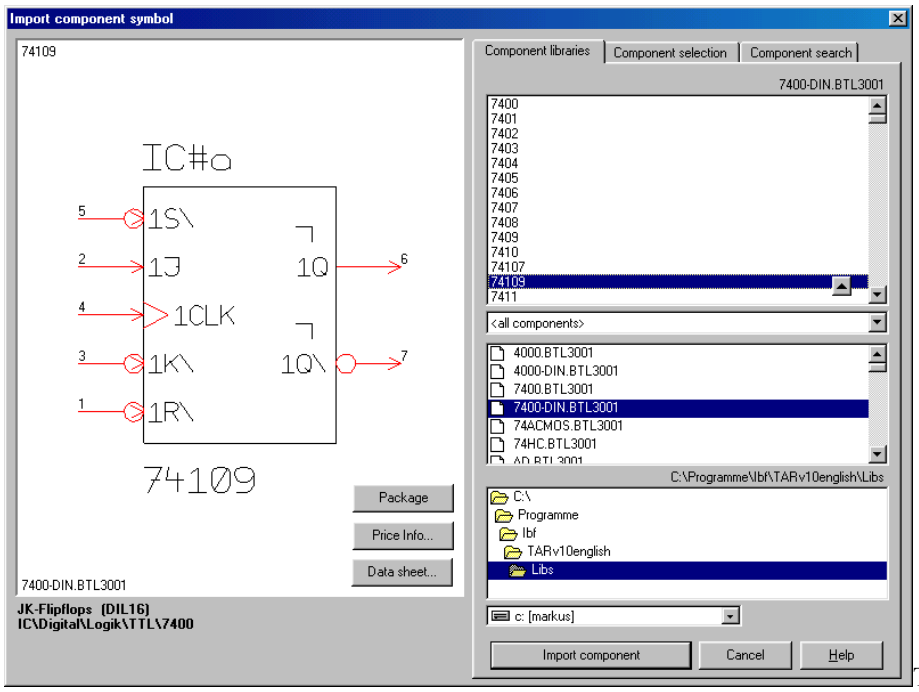
Swap number: Pins within the same symbol or gate and with the same Swap number greater than zero can be swapped with this pin.

Show pin number:

Is reference pin: If this box is checked, the pin is not representing a pad in PCB but is a signal reference pin such as used for ground symbols. This box shall only be used, when creating reference symbols for the reference symbol library.

Edit pin name: The dialog for the pin name will be opened.

2.9.53 Library Browser

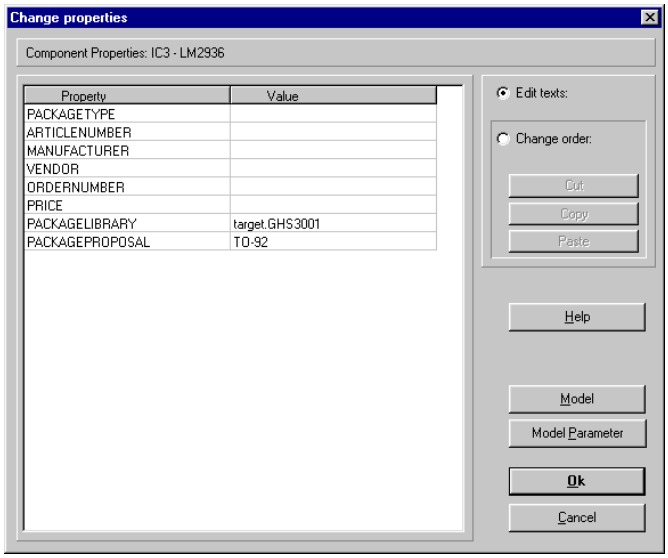


The library browser is very useful when operating with components and libraries. It is not the component editor, for in TARGET any component can be created or changed and exported at any free place of the schematic or PCB.

You can move and copy library components just by **dragging and dropping** them. The destination can be another library or the dust bin. Normally TARGET copies components from one library to anotherwith **M1h**. If you use the **[Shift]** key with **M1h**, the components are moved. That means they are rubbed out in the source library.

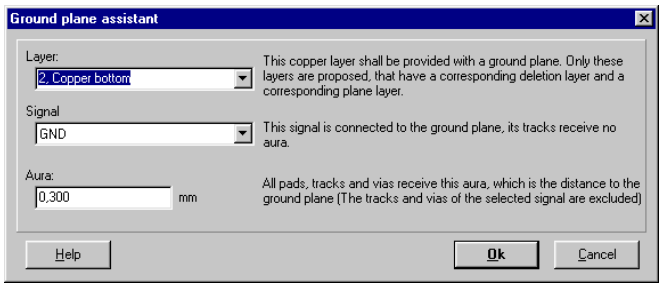
You can also use the **[Del]** key to delete the selected components.

2.9.54 Property Editor



can add a new entry by clicking on the right side of the last entry.
You can enter anything you like. If you want to delete entries or something like that, press **Change order**.
For components, you can also enter simulation model properties and edit their models.

2.9.55 Ground Plane Assistant



plane area, the deletion and the normal copper layer with tracks and pads. TARGET uses an optical method to display these ground planes: First the solid area is drawn, next the deletion layer is subtracted and last the tracks and pads are added.

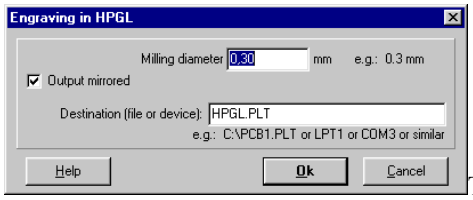
This method has one big advantage: It costs no time to compute the ground plane. Every change can directly be seen on the screen.

This automatic ground plane assistant is only capable to create groundplanes that cover the whole PCB. Manually you can create partial ground planes and different planes on one layer (e.g. AGND and DGND).

Select one **Layer**, select a **Signal** and enter the value for the **Aura**. Press Ok and the ground plane is ready.

If you have a layer completely reserved for one signal (e.g. ground layer), you must route this signal in this layer anyway. The thermal pads would be forgotten in the other case. The arising thermal pads don't look very good at the moment, for they have only one or two narrow stripes to connect them with the plane. Electrically, everything is ok. The stripes are thick enough for the connection.

2.9.56 Engraving Isolation Channels



The required insulation between the different signals is created by engraving isolation channels into the copper of the PCB material. TARGET computes the outlines of the signals and outputs them as HPGL files. CNC machines can use these data for protptyping.

Before starting this dialog, you have to display exactly that layer in the topmost window you want to engrave.

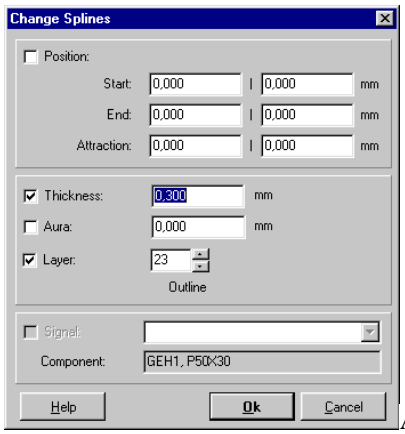
TARGET uses the following HPGL pens:

- Pen No 1: PCB outline
- Pen No 2: Signal outlines
- Pen No 3: Drill holes

Caution:

- Ground planes are ignored!
- Only signal carrying elements are regarded!
- Only the TARGET font is supported!

2.9.57 Spline Options

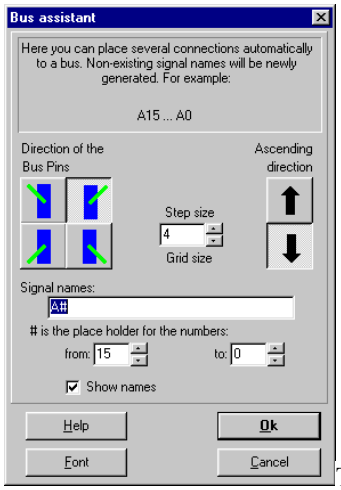


Aura:

2.9.58 Select Schematic Page

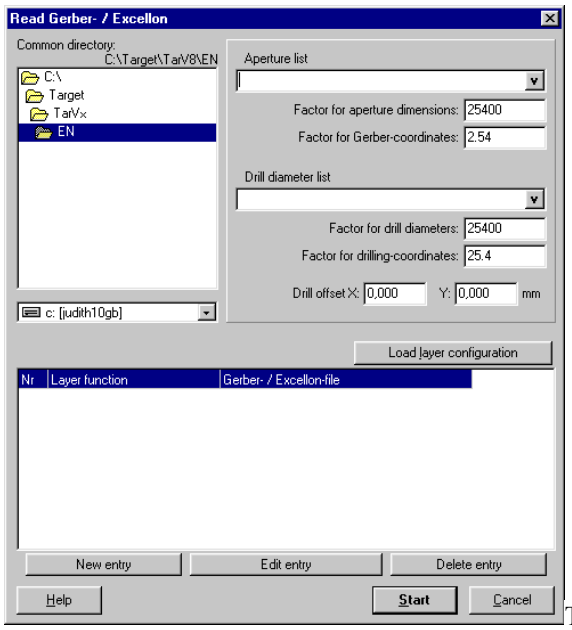
Enter the desired schematic page number. TARGET works with up to 100 schematic pages.

2.9.59 Bus Assistant



The bus assistant automatically connects several signal wires to a bus. Enter the desired directions, the signal names and the font type and start with Ok.

2.9.60 Read Gerber / Excellon



This part of TARGET is on the one hand a kind of Gerber viewer and on the other hand a tool to import foreign data into TARGET.

If you use it as a Gerber viewer, you can exactly check your Gerber data before production.

If you want to import other Gerber files into TARGET, look in the import filters of the file selection tools, whether your program is listed. If it is not listed, please give us a description, where the data comes from, and send us some example files. We try to insert your format and send you a new TARGET!

2.9.61 Contour Auto Router

See chapter 6.4

Chapter 3: Menu System

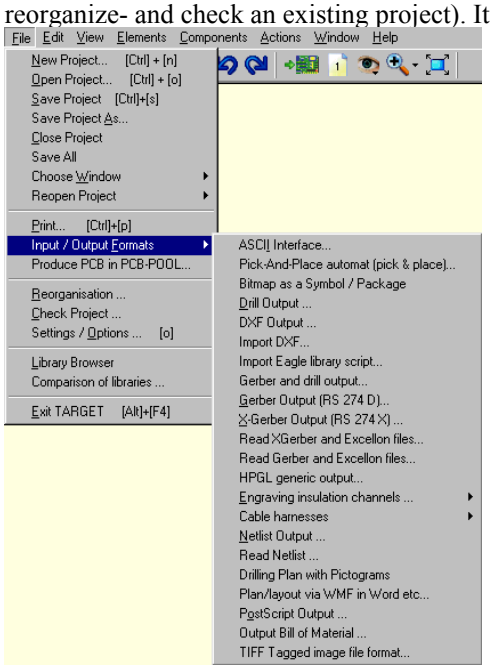
The following chapter describes the usage of the several TARGET menu commands. It is arranged, so that the menus and there corresponding commands appear as shown in this chapter (from left to right and from top to bottom). Depending on the active view (PCB- or schematic mode), only the available menus with there according commands are available.

Contents:

3.1 File Menu
3.2 Edit Menu
3.3 View Menu
3.4 Elements Menu
3.5 Components / Package Menu
3.6 Actions Menu
3.7 Window Menu
3.8 Help Menu

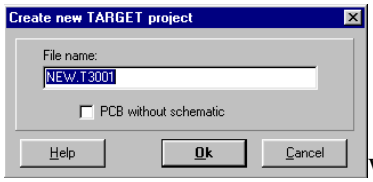
3.1 File Menu

The following chapter describes how to create a new project as well as how to manage existing projects (open-, save-, close-, print-, reorganize- and check an existing project). It will explain also how to generate several different output formats



The project management in TARGET is mostly done through the commands in the File menu. Essential Windows knowledges are neceessary for their usage.

3.1.1 Create a new Project



When you click on the tool TARGET opens a dialog Create new TARGET project. You will be asked to enter a new file name. You can also decide, if the project contains a schematic or if it is a PCB without schematic. Please note, that its not possible to change this status afterwards. This means, that you can´t add a schematic if you started with a PCB without schematic. Also, you can´t ignore the schematic when your project contains a schematic and a PCB. According to

The file **DEFAULT.T3001 (or VORLAGEP.T3001 for a PCB without schematic)** will be opened, renamed and displayed as NEW.T3001 on the screen. These two standard files can be opened and edited. Therefore you can set and safe preferences for future projects.

3.1.2 Open an Existing Project

TARGET displays a dialog box where you can choose a project from a list of existing projects. If the selected project contains a schematic, TARGET will open schematic page 1. Otherwise a PCB view is presented. You have also access to drives on your local network.

Usually, *.T3001 files will be loaded. It is possible to directly load older *.T2001 or *.T3 files from the 16-bit Windows version of TARGET. Also, backup files as *.BK1 ... *.BK9 which are older versions of your project can be loaded directly. The files named *.T3001A and *.T3001B, are created from the last autosave, act as a security copy, in the case that you can´t save the file correctly.

3.1.3 Save Project

The project who belongs to the actual window will be saved. All schematic pages and the complete PCB will be saved that way. TARGET uses the file name displayed in the window.

3.1.4 Save Project As...

Click on the arrow beside the picture.The project who belongs to the actual window will be saved. All schematic pages and the complete PCB will be saved that way. In the dialog, you have the possibilities to enter a new file name as well as a different directory. You can even create here a new directory. You have also access to drives on your local network.

3.1.5 Close Project

The project who belongs to the actual window will be closed. Any other open windows who belong to this project will be closed also. TARGET of course asks you if you want to save the changes. Open windows from other projects remain uneffected.

3.1.6 Save All

All open windows will be checked for changes since the last saving and saved if necessary.

3.1.7 Choose Window

Here you can switch between activ windows.

3.1.8 Reopen Project

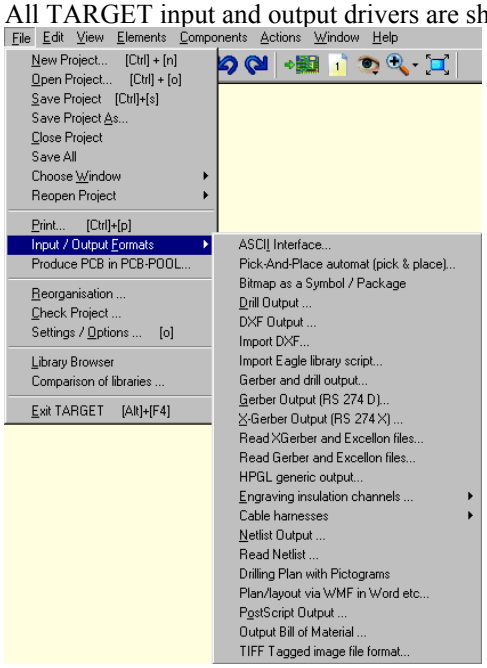
This list shows recently opened projects.

3.1.9 Print

The image in the actual window can be scaled and printed. The "Print Preview" dialog opens. You can see your project and how it is placed on the print page in this preview window. If your printer has problems with the x- or y- direction, correction factors can be added to compensate these errors. The image can be moved away from the left- and top-paper edge with the "Offset-X" and "Offset-Y" values. To prevent an output of gridded PCB-copper planes (greyscales), it is possible to convert these colors/greyscale areas in a sharp black/white areas.

3.1.10 Input/Output Formats

All TARGET input and output drivers are shown here except the Windows compatible print features.



Contents:

3.1.10.1 ASCII Interface
3.1.10.2 Pick-And-Place automat (pick & place)
3.1.10.3 Bitmap as a Symbol/Package
3.1.10.4 Drill Output
3.1.10.5 DXF Output
3.1.10.6 Import DXF
3.1.10.7 import EAGLE library script...

3.1.10.8 Gerber and drill output...
3.1.10.9 Gerber Output (RS 274 D)...
3.1.10.10 XGerber Output (RS 274 X)
3.1.10.11 Read XGerber and Excellon files
3.1.10.12 Read Gerber and Excellon files
3.1.10.13 HPGL generic output
3.1.10.14 Engraving Isolation Channels
3.1.10.15 Cable harnesses
3.1.10.16 Netlist Output
3.1.10.17 Read Netlist
3.1.10.18 Drilling Plan with Pictograms
3.1.10.19 Plan/Layout via WMF in WORD etc...
3.1.10.20 Post Script Output
3.1.10.21 Output Bill of Material ('B.O.M)
3.1.10.22 TIFF Tagged image file format
3.1.10.23 Spice-Circuit-File

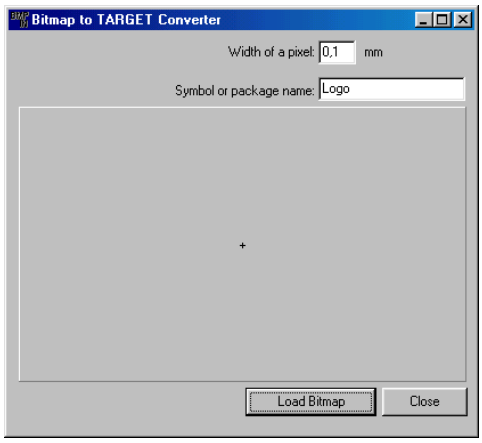
3.1.10.1 ASCII Interface

Save or load your project database as text file. See also chapter 10.8

3.1.10.2 Pick-And-Place automat (pick & place)

3.1.10.3 Bitmap as a Symbol/Package

Existing bitmaps of symbols or packages can be inserted.



Select desired bitmap from wherever you saved it and load it to TARGET. Afterwards export it to a library if you like. Please make sure thet the bitmap isn't too big for your PCB (regarding the size of the file as well as it`s measurements)

3.1.10.4 Drill Output

Here you start the Drill-output driver. See also Chapter 10.3

3.1.10.5 DXF Output

Here you start the DXF-driver. See also Chapter 10.6

3.1.10.6 Import DXF

The DXF-Import-driver can be activates here. See also chapter 10.6

3.1.10.7 import EAGLE library script...

Eagle libraries can easily be transferred into TARGET. See chapter 10.11

3.1.10.8 Gerber and drill output...

The function Gerber and drill output... creates a glance on the Gerber and drill data of yout project for to offer detailed information to your PCB producer showing him what he is expected of. See also chapter 10.4

3.1.10.9 Gerber Output (RS 274 D)...

Start the Gerber output driver here. See also chapter 10.1

3.1.10.10 XGerber Output (RS 274 X)

This dialogue enables you to export your TARGET project in XGerber format. See chapter 10.12

3.1.10.11 Read XGerber and Excellon files

TARGET is able to read files contending XGerber and Excellon data for to import those into a TARGET project.

3.1.10.12 Read Gerber and Excellon files

Start the Gerber-Viewer here. You can check your PCB before production wether all exports are OK. You can also read in and edit Gerber data from another CAD system. See also chapter 10.2

3.1.10.13 HPGL generic output

This output drive generates a *.plt file of your TARGET project in HPGL format. In the main thing it`s useful for to transfer coordinates of your project into other systems. For example if you want to know whether your PCB Outline is fitting into a special package, you have drawn with another CAD System, you may import your TARGET project in HPGL format.

3.1.10.14 Engraving Isolation Channels

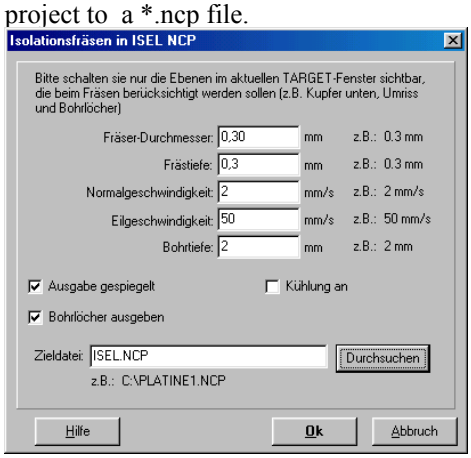
With the help of this tool, the necessary isolation between tracks is made by engraving of isolation channels. TARGET calculates the outline of the tracks and outputs them in HPGL-file format. These outline- and engraving data can be further used by CNC-maschines to manufacture sample- or small series of your PCB. See also Chapter 10.7

3.1.10.14.1 HPGL

Generate your TARGET project for insulation milling purposes in HPGL format. If you want to mill "copper bottom", hook "Output mirrored" and copy the generated *.plt file into a directory of your choice.

3.1.10.14.2 isel NCP

TARGET is able to drive isel and ProMa milling machines. By the help of the following dialog you can convert your TARGET project to a *.ncp file.



3.1.10.15 Cable harnesses

If you use TARGET to design cable connections (harness, switchboard, automotive etc.), the according cabel information and bills of material can be generated here.

3.1.10.16 Netlist Output

Netlists will be written in "ORCAD-Multiwire"-format or in "PROTEL"-format. The output is written to a text file.

3.1.10.17 Read Netlist

Netlists can be read in "ORCAD-Multiwire" format or in PROTEL format. This is basically an ASCII-file. To import the netlist, you must first import all components according to the B.O.M. in your empty project. Then you read the corresponding netlist. The supported read in formats are shown in dialog box Read Multiwire Netlist in menu File/Input-Output Formats/Read Netlist. Open pull down box on the bottom of the mentioned dialog.

3.1.10.18 Drilling Plan with Pictograms

Here you create the pictogram of the drill plan. TARGET uses 20 different pictograms (figures) for 20 different drill hole diameters. Also, an ASCII-text-file with information regarding the assignment of the pictograms to the individual drill hole diameters is created. The creation of graphical pictograms is used to control individual drill hole diameters on a non-automatic drilling (manual drilling). The size of the pictograms can be edited in the "Window" menu under the "Options" command. You should create an output of the drill holes as pictogram shortly after you entered the values. If you have made in the mean time changes in your circuit, the legend information will probably not match with the drill holes.

3.1.10.19 Plan/Layout via WMF in WORD etc...

TARGET enables you to insert parts of your schematic or layout into text documents. Use the clipboard function within this dialogue.

3.1.10.20 Post Script Output

Here you start the PostScript output driver. See also Chapter 10.5

3.1.10.21 Output Bill of Material ('B.O.M)

Outputs of a list in a text file (for example: TEST.TXT), containing all parts used on your PCB. The bill of material (B.O.M) is created as a text file which consists of entries made line by line. The entries from "Position" to "Rotation" are information needed for an automatic population of the PCB.The entries from "Part number" to "Price" are only available, when the according information have been entered in the "Component property"- field . If necessary use "Help" in the corresponding dialog in the program.

3.1.10.22 TIFF Tagged image file format

TARGET offers the opportunity to generate pictures of your project in TIFF format. Choose the view you would like to generate in TIFF format (schematic or layout) and click this function. Now be guided by the dialog.

3.1.10.23 Spice-Circuit-File

With this interface, it is possible to create a SPICE-circuit-file from the schematic and start an external simulator directly. TARGET supports the following formats: Spice3f5, PSpice, MicroCap und ICap.

3.1.11 Reorganisation

Here you reorganise the memory. Because deleted elements are still available because of the undo- and redo function, they take memory space, even if they are not visible.With the "Reorganisation" command, these elements will be deleted from the memory. A undo or redo is therefore not anymore possible. The remainders of components, which are used partly, will hereby not deleted (for example: IC1b is used, IC1a will not deleted, even if it has been deleted).

The "Reorganisation" also cleans unused signal names. The PCB will be resized to 1 m² area. The reorganization also cleans up the prefabricated component-order-name. Select the prefixes, their prefabricated part again should be numbered. The new numbering also can takes place after the X - or Y-Koordinatens of the components in the circuit board (!).

Please Note: If you delete a schematic symbol, even if the corresponding package still exists on the PCB, this schematix symbol will not dissapear from the list after a "Reorganisation"!

3.1.12 Check Project

With the help of this function, the whole project can be checked for logic "Correctness" (Electric-Rules), and the routing of the PCB is checked according to rules set in the "Design-Rule-Check"-dialog. Attention: aura and ground planes are currently not supported. See also Chapter 6.6 Check project.

3.1.13 Settings / Options

Fix general settings/options concerning the current project within this dialodue.

3.1.14 Library Browser

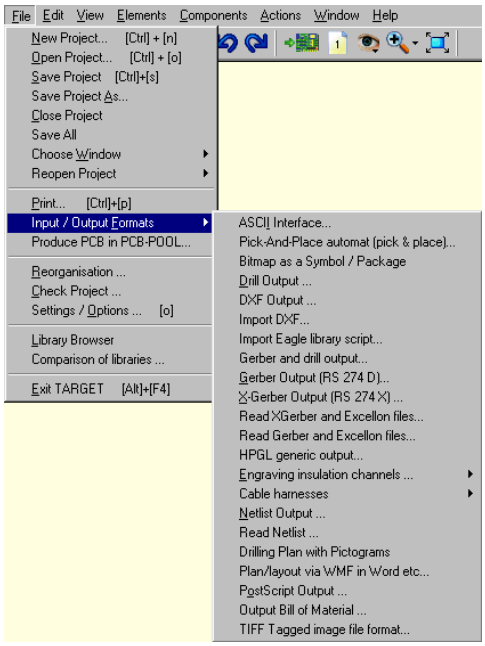
Contents:

3.1.14.1 Administration of libraries
3.1.14.2 Select component
3.1.14.3 Search component

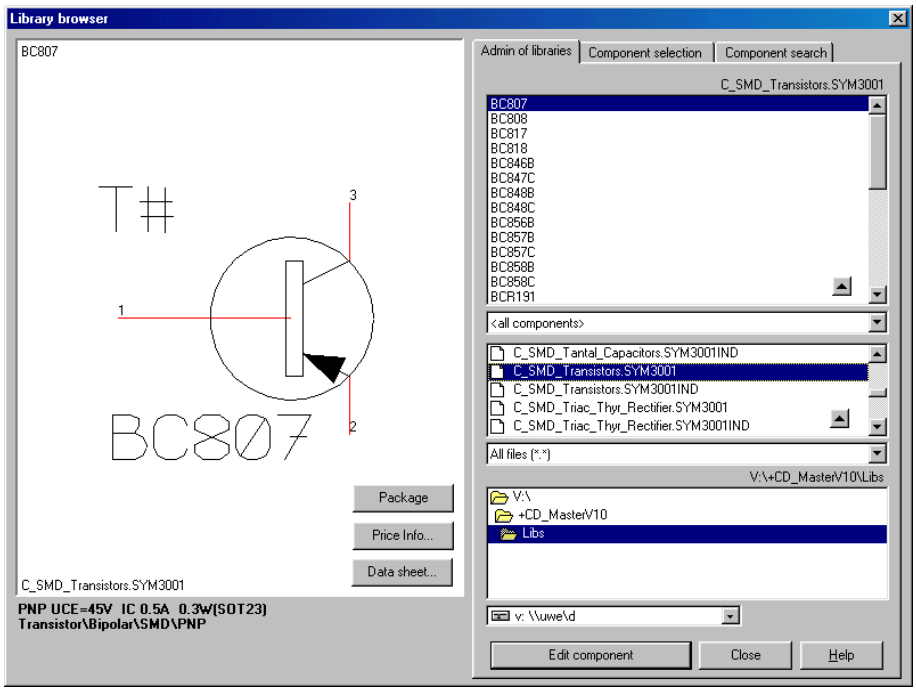
The library browser gives you all opportunities for your component handling (i. e. symbols, packages or reference symbols). You can edit components, search them or add properties to them. Build up groups of components or save them in individual libraries.

3.1.14.1 Administration of libraries

You will find the library browser in menu "File"



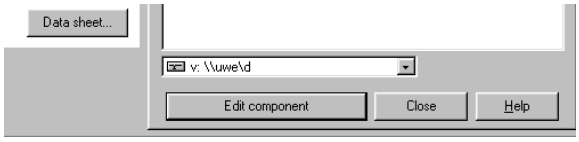
Press F2 or click M1 on "Library-Browser". The following picture appears:



Left on the picture shows the drawing of the symbol or package of a component. With buttons "Package" or " Symbol" you can toggle between each of the images. You will also see the buttons "Price-Info..." and "Datasheet", the library (where to find the component) in white area below left, its name, essential technical data and below the path of the tree view place of the component. Right on the picture you can choose several library functions in a register.

3.1.14.1.1 Button "Symbols" / "Packages"

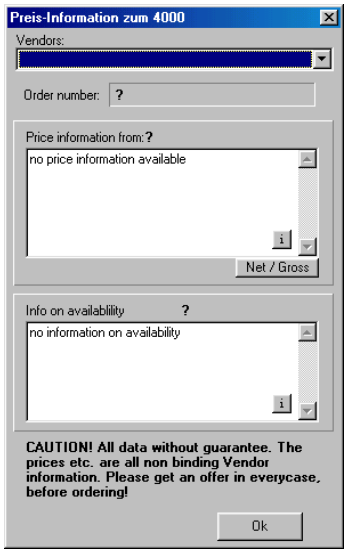
Each component has two kinds of imaging: the symbol and the package. With the buttons "Symbol..." and "Package..." you can activate each of the images. You can also edit the component shown using the drawing functions. Press button "Edit component" with M1.



3.1.14.1.2 Button "Price-Info"

A price info concerning the selected component and the respective distributor you will find here. Stock-informations can also be deposited here. The format of deposition find with the Help of i.

All pricing - and stock information deposited here have been carefully collected. they are not binding informations of the respective distributors. Ingenieurbuero Friedrich cannot give any warranty in terms of correctness and up-to-dateness. Please solicit a binding offer in any case from your desired distributor.

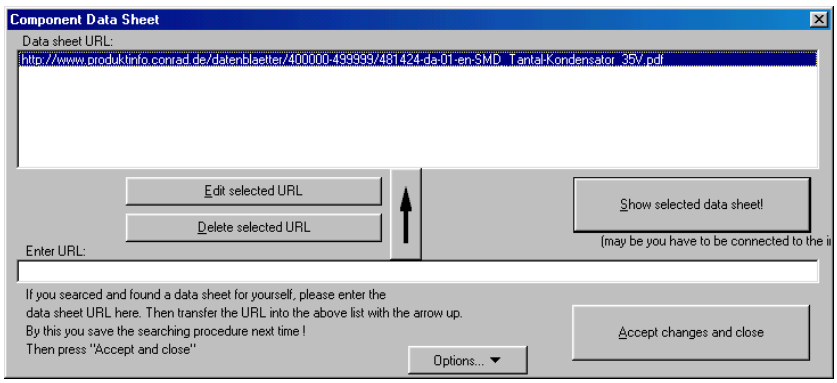


Preisinfo-Dialog.

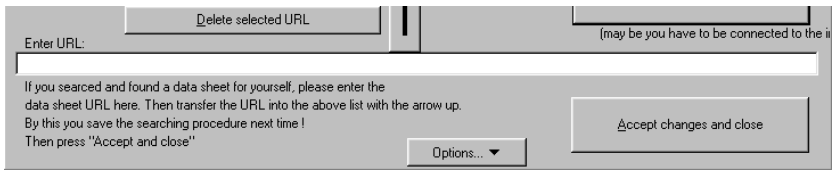
You can see prices and -scales also within the component properties.

3.1.14.1.3 Button "Datasheet"

For displaying a datasheet the upper field of the dialog must show one or more datasheet URL's. When finding an additional URL insert it into the field "Enter URL" and press the "arrow up" button. Editing or deleting an existing URL you act similar.

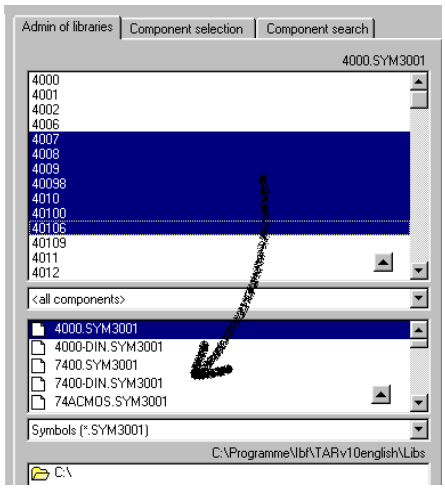


Button "Options" enables you to enter the path of an internet browser of your choice.



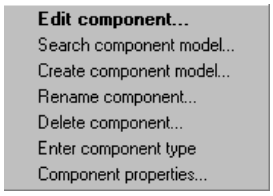
3.1.14.1.4 Component list

Now we talk about the right half of the dialog "Library-Browser". The upper-third window shows a list of those components wich are found in the highlighted library below. (List of components).



List of components

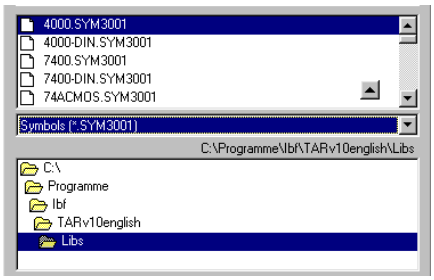
Click M1 on the component and see its symbol or package. Several highlighted components can be dragged and dropped (cut and pasted) into other libraries. Holding key [ctrl] simultaneously copies them. Clicking M2 in the white area or clicking M1 on button opens the following PopUp menu:



According to the option you selected in relation to the highlighted component the respective dialog opens. You can arrange components individually in groups in the PopUp menu under "Component-Properties" for to filter them within the pull down menu below.

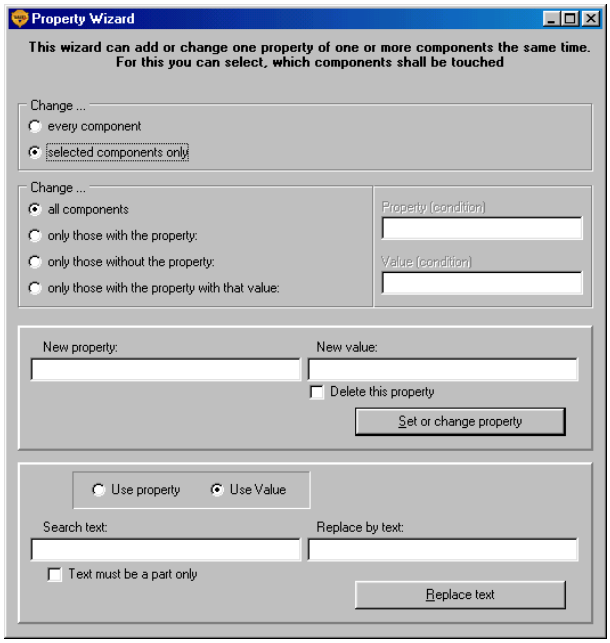
3.1.14.1.5 Component library

The window in the center right shows the libraries wich are to be found in the directory shown in the window below.



Click M2 in the white area or click M1 on button opens the following popup menu:

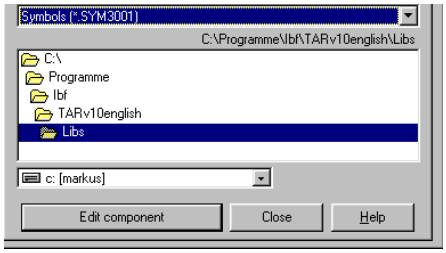
You can add properties to components or to groups of components, change or delete them, and edit text. You may generate new libraries and new listings of components, place passwords or change them, compare libraries or delete them.



Properties Assistant in popup dialog "Libraries"

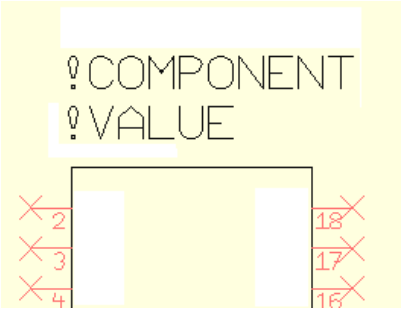
3.1.14.1.6 Library directory

The window below right within dialog "Library Browser" shows the path to the directory of your library. Choose the place of saving within the window below.



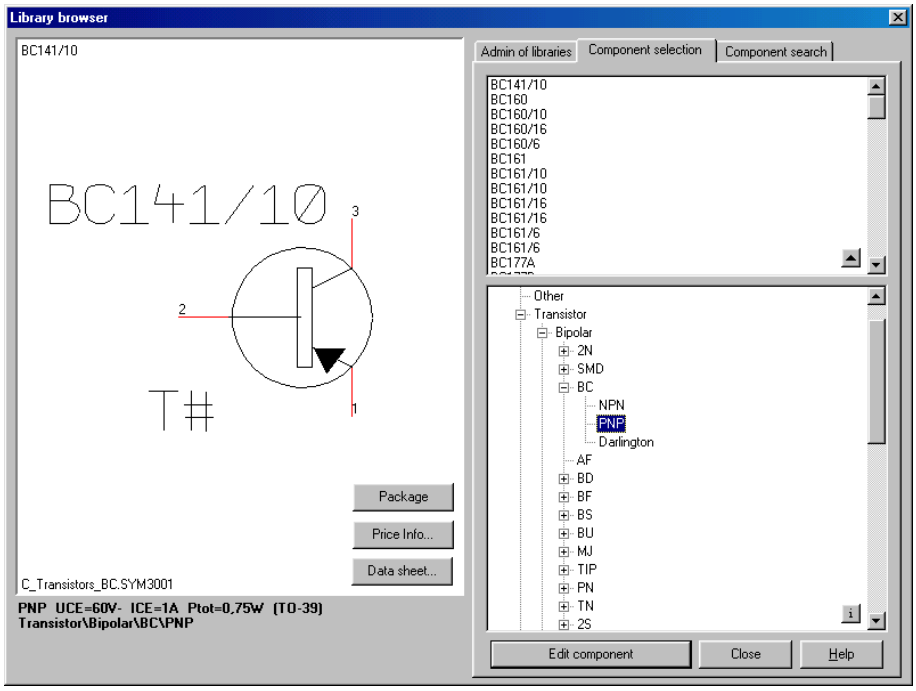
3.1.14.1.7 Edit component

The button "Edit Component" enables you to do so. Use the normal drawing functions and export the component edited by pressing the key [x].



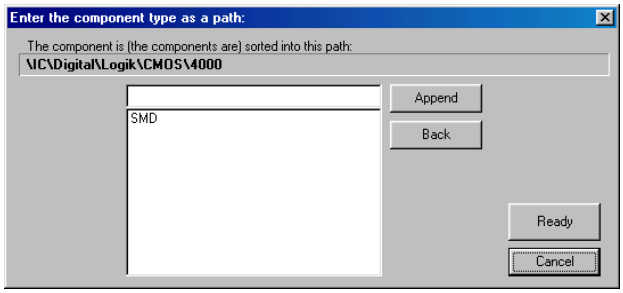
3.1.14.2 Select component

The "Select Component" dialog gives you within a tree view all opportunities for arranging and selecting components.

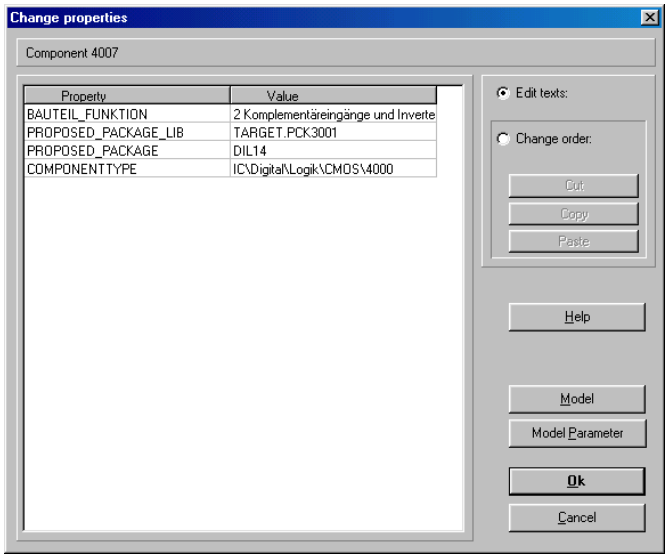


If you intend to store a component within another branch of the tree view, please highlight the component within the component list and press button .

In the now opening popup-menu please select option "Enter ComponentType". The following dialog opens:



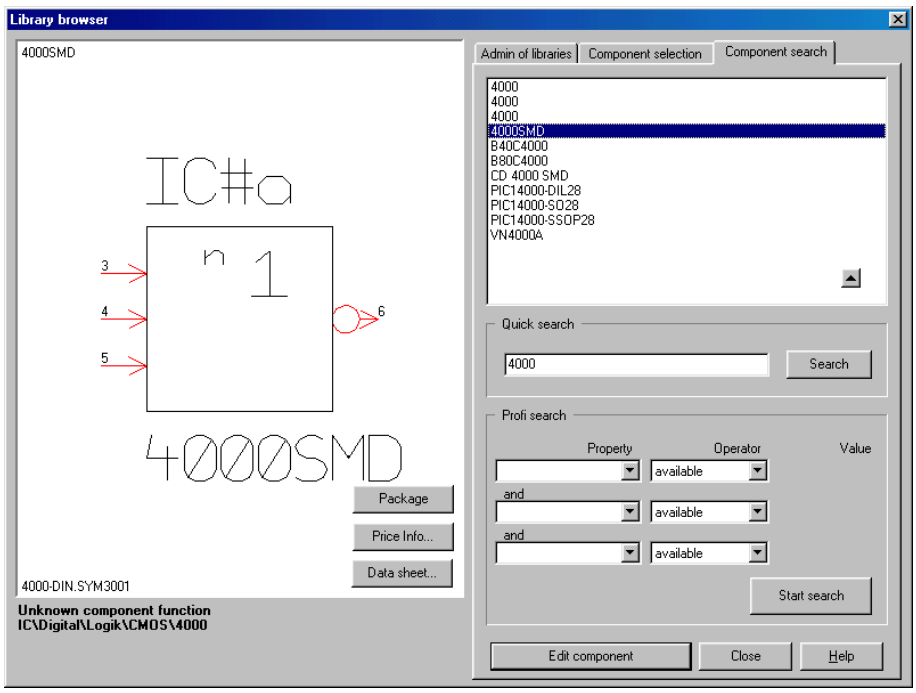
Now the path is shown, the component is placed within the tree view structure. Now you may define additional branches for to store the component or cut off those or define a totally new path. The highlighted component thus will be found in a new branch, or in a higher category. You may rename components or set individual properties. Click on and select option "Component Properties". The following dialog appears:



Vary existing properties, define new properties, edit text or arrange a new order.

3.1.14.3 Search component

TARGET 3001! offers a wide range of opportunities for component search.

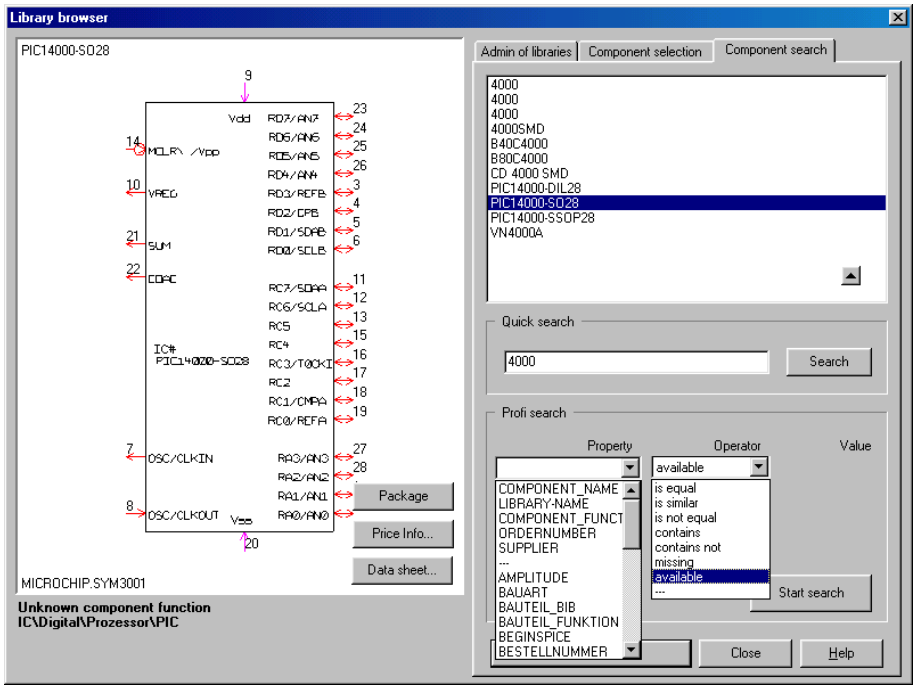


3.1.14.3.1 Quick search

Regarding a desired component enter a component attribute known to you to the "Quick Search" box and click M1 on button "Search". The search function brings you close to the desired component. If you like to enter more significant data, please use the data boxes of the "Profi Search".

3.1.14.3.2 Profi search

The more you know about a desired component, the easier and faster you will find it. In box "Profi Search" you can combine several properties of a component with corresponding operators as search-criteria.



As far as your desired component should equal a significant value or be similar to it or be unequal to it, you can define it before search by setting the corresponding operator related to the corresponding value..

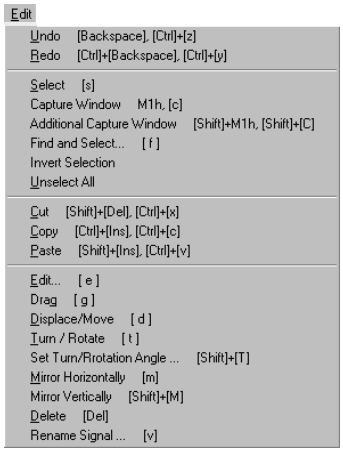
3.1.15 Comparison of libraries

Components (symbols or packages) of a target directory are compared to those of a source directory. Differences can be saved within a .txt file.

3.1.16 Exit

Exit your TARGET work. All windows will be closed. TARGET asks to save all projects. TARGET writes the base configuration into the "Registry".

3.2 Edit Menu



Contents:

3.2.1 Undo
3.2.2 Redo
3.2.3 Select
3.2.4 Catch Window
3.2.5 Catch additional Window
3.2.6 Search and Select
3.2.7 Invert Selection
3.2.8 Unselect All
3.2.9 Cut
3.2.10 Copy
3.2.11 Insert (Paste)
3.2.12 Edit
3.2.13 Drag
3.2.14 Displace / Move
3.2.15 Turn / Rotate
3.2.16 Set Turn / Rotation Angle
3.2.17 Mirror Horizontally
3.2.18 Mirror Vertically
3.2.19 Delete
3.2.20 Rename Signal

3.2.1 Undo

The command Undo can resolve errors during input or editing, when you select it directly after the last change. For example, if you delete a component or signal by mistake, it is possible to Undo this last step and reload this component or signal again. This command can be selected by clicking on the tool or through the Undo command in the File menu. TARGET can undo the last 50 steps.

3.2.2 Redo

When you press the Undo key by mistake twice, which means you deleted a good action, it is possible to click with M1H on this button. TARGET resolves then the last view before you did the Undo. TARGET allows the last 50 steps to be undone.Redo.

3.2.3 Select

Select is used to mark and highlight an element, signal or component for other actions. For example, if you would like to edit a name, you would use the help of this menu command to enter the Select element mode. TARGET shows this new mode through a different mouse cursor and an with entry in the status line. In Select mode, you can click on any element with **M1**. The handle of the selected element immeaditely starts blinking. In case, that not the handle of the selected element starts blinking, but a neighboured element, you can use the **[+]** and **[-]** keys to make your final selection. To decide if you want to select only the component with the blinking handle or several components at once, you can press the **[o]**
Select element command can be executed also by pressing the **[s]**
simple M1 mouseclick. See also **Chapter** 2.6.2.

3.2.4 Catch Window

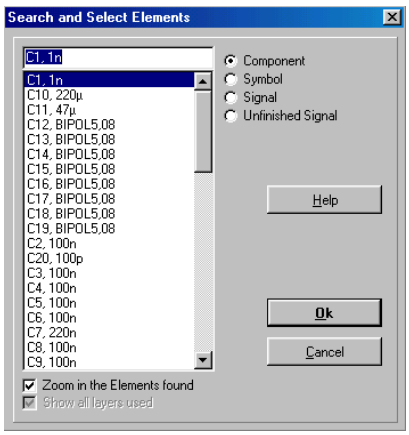
To mark several elements in an area, use the mouse to draw a window over these elements. Start with M1 at a free spot. With the [o] key you can edit the attributes of the Catch Window function. You can decide if the selection should contain all of the elements in the selected area, or additional elements. As pointed out in the menu, the Catch Window function can be executed by pressing the **[c]** key. Catch Window starts at cursor position. This is recommended for layouts of high density, where it is possible to find an empty spot for **M1H**.

3.2.5 Catch additional Window

Like Catch Window. To select new elements together with the already marked elements, use this function or the **[Shift]+M1H** combination in pointer mode.

3.2.6 Search and Select

This function is used to locate components and signals in the schematic- and PCB-view. The according elements will be marked and the active window will be scaled for optimal view. The following dialog appears:



All signals can be listed, which are not completely routet. Selecting the function Zoom found elements in the center places the elements to be searched centered on the screen. The according schematic page will be displayed in schematic mode. Is a signal or component located on several pages, TARGET will display special note. Search and Select.

3.2.7 Invert Selection

All selected elements will be unselected, all visible but not selected elements will be selected. This function can especially be of value in the version of TARGET, who is limited to do a layout of the size 160mm x 100mm. If you have moved the layout ouside the limited area of 160mm x 100mm, it is usually not possible to catch the layout completely, because of the limitation of the active mouse area. Please click now with M1 on a free spot outside the layout area, to make sure that no element has been selected and use

3.2.8 Unselect All

All elements will be unselected.

3.2.9 Cut

The selected elements will be deleted from the circuit and copied into the clipboard. The clipboard of TARGET can export data in the Windows-Metafile format as well as its own format. The data in the clipboard can be exported to external programs for documentation purposes (text programs) as well being used to be imported again as modules. If only a part of an element is selected, the whole element will be transferred to the clipboard. Cut.

3.2.10 Copy

Like Cut, but the selected elements remain on the screen. Copy.

3.2.11 Insert (Paste)

TARGET can import the data contained in the clipboard at any position in the circuit with M1. Only data which have been exportet by TARGET can be importet again. The transfer of schematic data into the PCB and vice versa is not possible. A simple rule is valid when you assign signal names: Has the signal been completely selected before cutting or copying, the signal will be used as a local or internal signal. Then a new signal name is assigned at during the import (for example: Sig$164). When the signal has not been selected completely, it will be handled as a global or external signal and therefore, the signal name remains unchanged (for example: GND). Insert. Paste.

3.2.12 Edit

Here you can edit afterwards all selected elements. Are several kinds of elements marked, TARGET gives an overview of the different kinds of elements. You can individually select here which kind of elements you want to change. Contains a kind of element only one selected element, all data of this element will be shown in the Edit dialog. The access field for all properties contain a hook. If you have selected several elements, only the data of the first selected element will be shown in this dialog. Only the access fields for the properties of identical elements contain a hook. For example, if you want to change the width of all selected solder pads to the same size (all of the selected solder pads are of different size), please add the new size in the according field and place a cross in the access field. For example, selecting Change single properties allows a change of the aura for all selected elements.

3.2.13 Drag

You can relocate already placed tracks, signal tracks and lines with the Drag function, as well as changing triangles and rectangles. Select this function and place the cursor on the bending point. Press now **M1** to start the Drag function. The bend will be placed on the preferred spot with **M1**. This function can also be started on the center of a track or signal track. The track will be (depending on the selected bending mode):

- splitted into two segments (bend)
- changed to an arc of a circle or
- changed into a bezier-curve.

Triangles and rectangles can be dragged at their corners with the help of this function. It is not possible to drag rectangles when they have been turned at an unregular angle.It is better to use the drag feature without this menu command.

By pressing **[g]** or **[Strg] + M1H**, the element can be dragged directly.

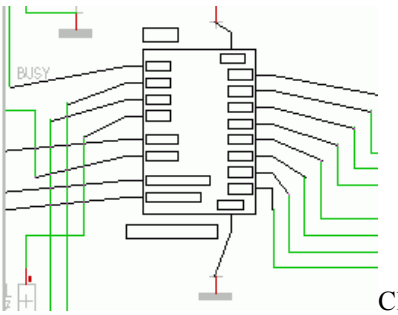
Example: A straight line should be converted into a bezier-curve:

- Click with **M1** on the folowing tool button:
- Click on the tool and change into the Create a bezier-curve mode.
- Place the cursor on the center on the straight line.
- Hold the **[Strg] key +M1H** pressed and move the mouse. The outline at the cursor shows the created bezier-curve.

3.2.14 Displace / Move

For displacement of all selected elements, place the cursor exactly on the handle of one of the selected elements. Press the **[d]** key or select the according menu command (Edit menu - Displace / Move). Move the mouse and press **[Return]** or **M1** to finish the moving. If tracks or signals have been connected to the selected pins, signals or tracks, they will be moved along like an elastic band. If a package is moved, a Ratsnest is created and displayed for signals from all unconnected solder pads to the next solder pads with identical signal name. This help tool for the placement is updated in realtime after each movement. You can start a rotation in move mode with **[d]** without leaving the move mode. The move mode is handled the best by the use of the mouse.

Click with **M1H** in pointer mode exactly on the element or on the handle of the component and move it with the mouse (drag & drop). Was the element/component selected, all selected elements will also be moved. The element´s outline sticks to the cursor (see images below). For components with connected signals, the signals will also be moved. Displace Element. Move Element.



Click with M2 or [d] while you Move, TARGET rotates the according element/component according to the preset rotation angle. The rotation angle can be set with **[Shift]+[D]**.

3.2.15 Turn / Rotate

This function enables the Turn / Rotate mode. Place the cursor into the rotation center and press the mouse button M1. All selected elements rotate now around the cursor according to a preset rotation angle. This rotation angle is set to 90° by default. To edit this value, press **M2** instead of **M1**. Any rotation angle can be entered now in the appearing dialog window. In case you want to change the rotation angle, you can start the Rotate function also with the **[t]** key or with **[Shift]+[D]**. You are automatically in the Move mode when you place components. You can Rotate a component with **M2**.

3.2.16 Set Turn / Rotation Angle

Select this menu command if you want to set a new rotation angle in the appearing dialog window for future rotatings.

3.2.17 Mirror Horizontally

Choose this function to mirror any selected elements horizontally (= along the vertical axis). The spot you click on with **M1**, defines the position of the vertical mirror axis.You can of course execute this command with the **[m]** key. When you mirror elements in the PCB view, elements placed on the top layer will be transferred automatically to the bottom layer. A smd solder pad in tthe copper layer top will therefore be mirrored and at the same time transferred to the copper bottom layer. This equals a process when you move a component from the component side to the solder side.

3.2.18 Mirror Vertically

Choose this function to mirror any selected elements vertically (= along the horizontal axis). The spot you click on with **M1**, defines the position of the horizontal mirror axis. See also 3.2.17

3.2.19 Delete

This function is used to Delete selected elements in the circuit. Delete is a three step procedure:

- If any additional tracks are selected who are created from the autorouter, only these ones will be deleted.
- If manually routet signals are selected, only these ones will be deleted.
- All selected symbols or packages dissapear.

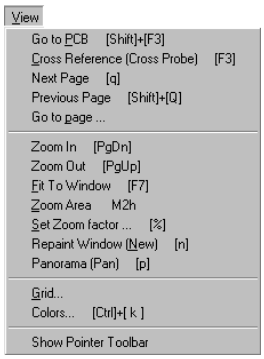
In some cases you have to execute the Delete command (or pressing the [DEL] key) three times, until all selected elements and components dissapear. If you delete the only signal segment connected to a pin, the pin will be released again and is therefore available to be connected again (the pin gets back an X). The last 50 changes (including deletions) can be undone with the Undo function!

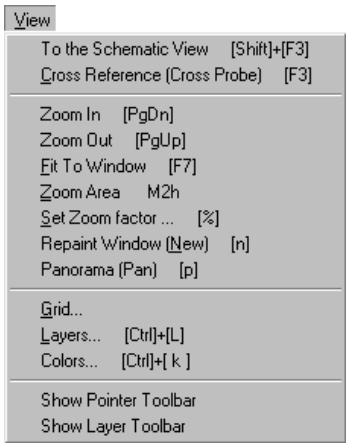
3.2.20 Rename Signal

This function is used to rename a complete signal (not only a signal segment). Already placed tracks on the PCB will be renamed also. You will find this Rename Signal command in the Edit menu, by pressing the **[U]** key or as an according tool. You select the signal to be replaced on the left side in the dialog window and on the right side you select or enter a new signal name. Rename Signal.

If you want to rename only a related signal island but not the complete signal, double click the according signal with M1 and select or enter another signal name. It is not possible to rename signal segements, junctions and pins inside a signal island or within a complete signal! Basically, the complete signal island will be changed completely.

3.3 View Menu





Schematic- View and PCB- View

3.3.1 To the PCB view

Is the current activ window a schematic window, you can use **[Shift] + [F3]** to change to the PCB view. If a PCB window is already open, it will be set as the active window. Is no PCB window open, a new one will be created as the active window. Please use the same key combination to switch between schematic- and PCB view.

3.3.2 Cross Probe

Cross Probe means a logic oriented switching between schematic and PCB. The user can switch signal- or component oriented between the schematic and the PCB very quick. For example to find the according track in the PCB which belongs to a selected signal in the schematic. The related element will be displayed in its maximum size (format filling).

Cross Probe is executed through this memu command, through a click with **M1** on the above tool or by pressing **[F3]**. The according component or signal must be selected before you use this feature. The following tasks are supported:

- Find the track of a selected signal element.
- Find the component on the PCB, which belongs to the selected schematic symbol.
- Find the component on the PCB, when a related element of a schematic symbol (pin, line, text) is selected.
- Find a signal in the schematic, which belongs to the selected track segment on the PCB.
- Find the schematic symbol, which belongs to the selected component package.
- Find the schematic symbol, when a related element of a component package (solder pad, line, text) is selected.

The elements (signals, components) will be searched on all schematic pages. If the result shows several schematic pages, the user will be informed.

3.3.3 Next Page

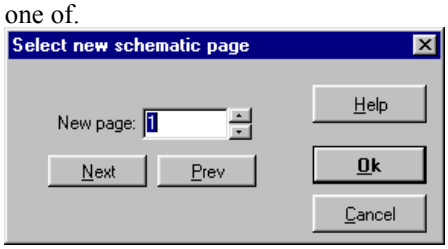
You can change to another schematic page here, if you use more schematic pages.

3.3.4 Previous Page

You can change to the previous page here, if you use more schematic pages.

3.3.5 Go to Page

You can select here another page to be viewed in a schematic window. TARGET supports up to 100 schematic pages in one project (= circuit). Connections between the individual pages are done through reference-symbols or busses. Schematic page, select one of.



3.3.6 Zoom In

M1, will be zoomed in and displayed in the center of the screen. This means you can view more details, but you have no good overview of the whole contents of the window. You can Zoom Out with **M2**. The **[PageDown]** key has the same function, but it can be pressed in any other mode without the neccessarity to leave the mode (for example to move a track). The zoom factor of the Zoom In function equals approximately 10-20-50-100-200-500-1000. The key **[Strg]+[PageDown]**
cursor is placed. While in pointer mode, the right mouse button is automatically assigned with the zoom function. See also **Chapter** 2.6.3.

3.3.7 Zoom out

This function enables the Zoom Out mode. The spot you click on with **M1**, will be zoomed out and displayed in the center of the screen. This means you get a better overview of the whole contents of the window, but you will see less details. You can Zoom In with **M2**. The **[PageUp]** key has the same function, but it can be pressed in any other mode without the neccessarity to leave the mode (for example to move a track). While in pointer mode, the right mouse button is automatically assigned with the zoom function. See also **Chapter** 2.6.3.

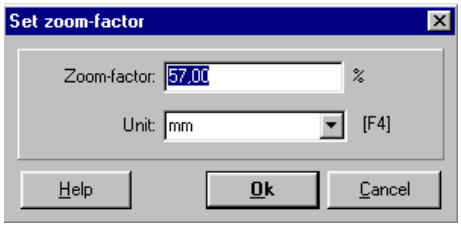
3.3.8 Fit to Window

Use this function to display and center your drawing fitting to the window. The **[F7]** key and the **[Strg]+[PageUp]** have the same function. This function can also be called when you are in a different mode, without the neccessarity to leave that mode (mode-transparent-function). The **[Strg]+[PageDown]**
maximum size. While in pointer mode, the right mouse button is automatically assigned with the zoom function. See also chapter 2.6.3. Fit to Window.

3.3.9 Zoom Area

You can display a selected area of your project to maximum size with this function. A magnifier glass appears at the crosshair cursor. Select now the area of your project which you want to zoom. Press **M1** or **M1H** and select a part of your project with the mouse. The elements to be zoomed will be displayed automatically in there maximum size on the screen. Hereby the mouse must be moved from left to right when you create the window. If you move the mouse backwards, this would equal the function Fit to Window **[F7]** **M2H**.

3.3.10 Scale...



The Scale... function allows to enter a zoom-factor for the window. 100% means that 1 cm PCB = 1 cm on the screen = world coordinates. You can choose between the units inch, mil and mm (1 inch = 1 = 25,4mm; 1 mil = 1/1000 = 0,0254mm). To allow a display of world coordinates, you have to enter the exact usable size of your screen in the menu Window - Options. You will find this button within the extension of

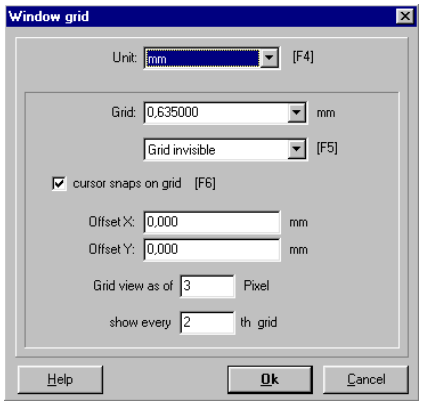
3.3.11 Repaint Window

This function repaints the actual window. This is useful when for eaxmple the crosshair cursor is stuck on the screen or if functions display your drawing somehow wrong. Repaint window. You can also just press the key **[n]**. You will find the above shown button within the extension of

3.3.12 Panorama

You will find this button beneath the extension of This function brings the view at cursor position with **M1** into the center of the screen.

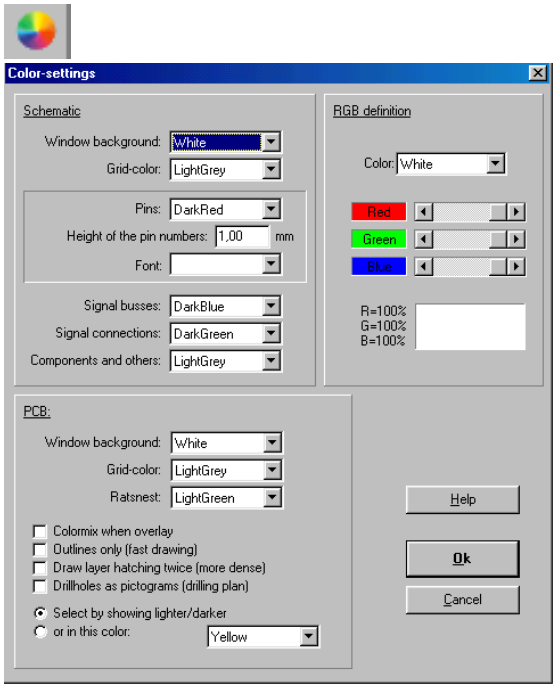
3.3.13 Grid



You will find this button beneath . Each value between 0,001 mm and 10 mm can be entered as a Grid value. The value must not be divideable by 0,001 mm (for example: 0,378512769). The grid can be displayed either as Lines or as Points. An invisible grid is not turned off. Depending on the cursor snaps on grid entry, the 90°crosshair cursor snaps onto the grid. The distance of the grid reference point to the absolute zero point (Abs 0|0) is set by the **Offset X/Y** values. This is an advantage if you need to place a connector on a metric euro-size-PCB (160 x 100mm) exactly at the small side where the grid unit is set to inch.

Grid view as of means that the grid is only displayed, if there is a minimum amount of pixels between the grid points on the screen. This protects the computer to fill the screen with grid lines or points which makes it difficult to be seperated from each other. You can also decide here, if TARGET should draw each grid point or only each second. The following settings have proved to be useable: Grid: 0,635mm = 1/40, invisible, cursor snaps on grid, Offset X/Y = 0/0, Grid view as of 3 pixel, show every 2nd grid.

3.3.14 Colors



You will find this button beneath

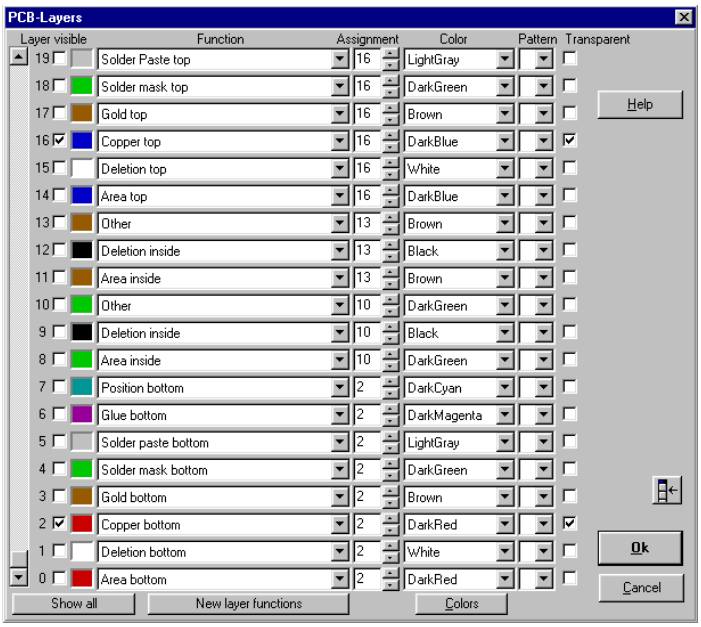
- **Window background:** Background color of the window.
- **Grid-color:** Grid color. Ratsnest: Color of the ratsnest
- **Pins:** Pin color in the schematic windows.
- **Busses:** Color for signal busses in the schematic windows.
- **Signal busses:** Color for signal lines and junctions in the schematic windows.
- **Colormix when overlay:** A red- and a blue line on top of each other result in a violet line. This works the best with clean RGB-colors and a black PCB-background color or with mixed CMY-colors and a white PCB-background color. More then 256 colors should be possible with the graphic card driver.
- **Outlines only:** Only the outline of lines and areas will be displayed, the filling is ignored. The kind of presentation draws the contents of the actual window much faster and it is possible to see structures inside of areas. Draw layer hatching twice: If you assigned a pattern to the layers in the layer dialog, these patterns will be drawn more dense.
- **Drillholes as pictograms:** For a drilling plan, small symbols can be created for each drill hole diameter. This option is the reason why drillholes will not be printed as open drillholes. This has been several times an issue for the technical support. You must call first the function File / Input/Output formats / Drilling plan with pictograms. The call of this function automatically enables this feature (a cross will be set at the according field). After you have printed out the drilling plan, you should disable this feature again. If you forget this, small little triangles and stars will be printed on the black solder pads, therefore an open (white) drillhole will not be seen.
- **Select by showing lighter/darker:** Selected elements in PCB mode can be displayed either by using the same color but in a different intensity or by the use of a total different color. The first option has the advantage that its possible to recognize the original colors. Option two is more clear. For example, it is not always easy to see clearly the difference between a dark light-green and a light dark-green color on the display.

It is possible to adjust the according VGA-colors for each of the 16 colors directly through their RGB-value (red, green, blue). Therefore you can adapt the colors easily to the different display modes of the graphics card and the monitor. As an example, the intensity of individual colors can be influenced.

3.3.15 To the Schematic View

You can switch with **[SHIFT]** + **[F3]** to the schematic view if the actual window dispalyed is a PCB child window. If a schematic window of the project is already open, it will be displayed as the actual window. In case if no schematic window is open, a new one will be displayed as the top window. TARGET can keep a maximum of 10 windows open at the same time. Schematic view, toggle to.

3.3.16 PCB-Layers



This function opens the layer dialog. TARGET works with up to 100 layers (0..99) in PCB mode, each of the layers can be individually assigned to a function. It is recommended to keep the default layer settings (assignment between a layer and its function) if you decide to let a printed circuit board manufacturer produce your PCB. You can individually set each layers function, reference layer, color and pattern. Additionally, each layer can be switched visible or invisible (do not set any empty layer to visible, since valuable time to update the contence of the display will be wasted. Because it is difficult to set the options of each layer in a single dialog, a dialog window which can be scrolled is used for setting the individual parameters. The following layer functions are possible:

- **Copper (bottom, inside, top):** Used for solder pads, tracks etc...
- **Position (bottom, top):** Used for package outline drawings and PCB texts.
- **Deletion (bottom, inside, top):** Used for deletions in areas, used especially for automatically generated ground planes.
- **Area (bottom, inside, top):** Used for automatically generated ground planes.
- **Solder mask (bottom, top):** Used for the solder mask on copper tracks.
- **Drillholes:** Used for drillholes of solder pads, through holes etc...
- **Solder paste (bottom, top):** Used for the placement of solder paste for smd-components in reflow soldering mode.
- **Gold (bottom, top):** Used for a partial surface finishing. This for example is used for a gold plating of connectors on a connector card.
- **Glue (bottom, top):** Used for glue necessary to mount smd-components in wave-soldering mode. All smd component drawings from there individual manufacturers contain a glue-point.
- **Dimensioning:** Marking used for documentation purposes.
- **Outline:** Used for PCB outline- and cutout- drawings.
- **Milling:** Used to give a direct information to milling routers.
- **Pad numbers:** Used to display the name of individual pins.
- **Air wires (ratsnest):** Used to display unfinished connections.

- **Errors:** Used from the design-rule-check to display distance errors.
- **Other:** Used to assign a layer with any other function.

The entry reference layer for area-,deletion- and solder paste layers mean that this layer belongs to the copper layer xx. Any other layer can be used as a reference layer for all of the others. It is recommended to use layer -1 as the reference layer.

The following 16 colors are available for the layers: black, light gray, dark blue, light blue, dark green, light green, dark cyan, light cyan, dark red, light red, dark magenta, light magenta, brown, yellow, dark gray and white.

Suggestions for assigning a color to a layer: layer colors should be of dark colors, because when you select an element, dark colors are dispalyed as light colors and vice versa. It is not convenient to display light colors with more intensity and having the elements- and window background color of the same color. But it´s up to you to find your preferred combination of colors displayed for the individual layers. But don´t be surprised if your color combination shows white, means nothing, when you highlight an element.

The New layer function button allows you to assign your own layer function which is available for any of the 100 layers. Also the libraries can work with elements which contain self defined layers. In this case the matching of the name of the layer is very important. Platin-plating und Platinplating are two different names and will not lead to the expected result.

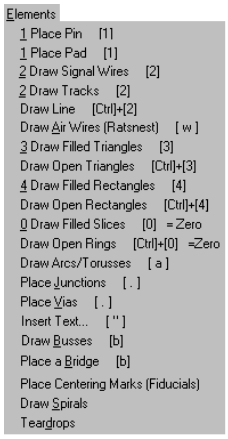
3.3.17 Show Pointer Toolbar

After selecting this feature, the pointer toolbar will always be displayed on the left side of the window.

3.3.18 Show Layer Toolbar

After selecting this feature, the layer toolbar will always be displayed on the right side of the window.

3.4 Elements Menu



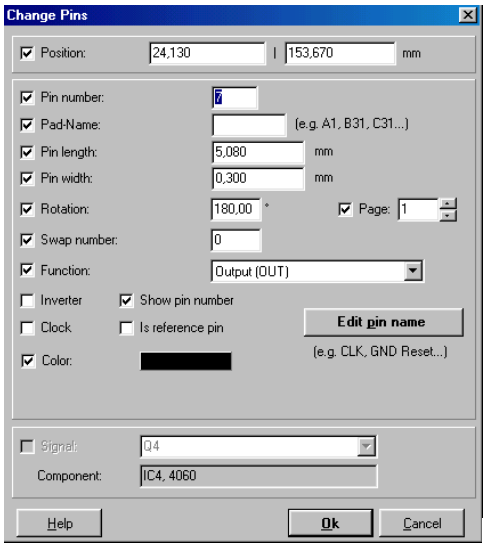
Contents:

3.4.1 Place Pin
3.4.2 Place Pad
3.4.3 Draw Signals
3.4.4 Draw Tracks
3.4.5 Draw Lines

3.4.6 Draw Air Wires (Ratsnest)
3.4.7 Draw Filled Triangles
3.4.8 Draw Open Triangles
3.4.9 Draw Filled Rectangle
3.4.10 Draw Open Rectangles
3.4.11 Draw Filled Slice
3.4.12 Draw Open Slices
3.4.13 Draw Torusses
3.4.14 Place Junctions
3.4.15 Place Vias
3.4.16 Insert Text
3.4.17 Draw Busses
3.4.18 Place a Bridge
3.4.19 Place Centering Marks
3.4.20 Draw Spirals
3.4.21 Teardrops (Tears)

3.4.1 Place Pin

Select this function or press [1] in pointer mode to place a pin. Free pins are only used when you define your own symbol to be exported to a library. You can create new pins who will be cosecutively numbered by pressing **[1]** or **M1**. You can leave this mode with **[Esc]** or **M12** after you have placed the last pin. TARGET assigns the first pin placed with pin number 1. It is possible to change the pin number, the form and the function of the pins by pressing **M11** in connection mode or with **[O]** for options. The following Options dialog shows some infos about the component - and signal assignment. Other entries have the following meaning:



- **Position:** Shows absolute coordinates.
- **Pin number:** Shows the pin number.
- **Pad name:** If the naming of the pins within the package is alphanumerical, (e. g. A1 - A32, B1 - B32, etc.), here you can drop those names. The pad name must be identical to the one of the corresponding pad of the package. The pin number must be identical (e. g. pin name of package A1= pin number 1, A32=32, B1=33, B32=65, etc.).
- **Pin length:** Shows the length of the pin from starting at the cross.
- **Pin width:** Shows the width of the pin.
- **Rotation:**
- **Page:** Shows the actual schematic page.
- **Swap number:** Pins inside the same symbol with the same swap number can be exchanged with this pin (for example: two

inputs of an OR-Gate, the exchange number must be higher than 0).

- **Function:** The following pin functions can be selected:

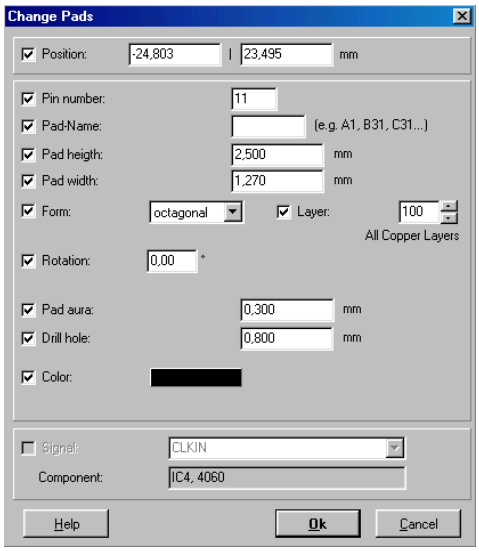
□- Not connected (N/C): pin not used (not connected)
□- Input (In): Simple input Output (Out): Simple output
□- Input/Output (I/O): Pin changes its state
□- Open Collector (OC): open collector (could also be open drain)
□- Power pin (Power input): Power supply input of a component
□- passive: Pin of a passive element, the pin function may not be clear.
□- Tristate: Output, that exhibits three states: High, Low, and Open (disconnected). This open state is called high impedance (or high Z-state).
□- Power (Supply): Power supply input of the circuit or power supply output of a component (for example: 7805).

- **Inverter:** Enable this feature if the pin is an inverter. The symbol shows the inverter sig at its end.
- **Show pin number:** The number of the pin (= Component-Pin-Nr.) is shown in a small size on the side of the pin.
- **Clock:** A small arrow is inserted at the end of this pin, if it has some clock function (clock input/output).
- **Is reference pin:** This feature means that the pin is not representing a solder point, but is used an a signal name reference (for example: for connections used over several schematic pages (reference-symbol)). Can be set first to [Y] during export!
- **Color:** Define color of a pin independently of predefined color. Is color white (R=255, G=255, B=255) chosen, the predefined standard-color is used. If you wish color white although, you need minimal offset from the above mentioned RGB characters (just one point fits). Pin, color of.

A name is generated while creating a pin. This name can be selected seperately to be renamed or moved. If you need for the logic of the circuit only the name of the pin but not the pin number please change the name and delete it with **M1** and **[DELETE]**. It will remain invisible.

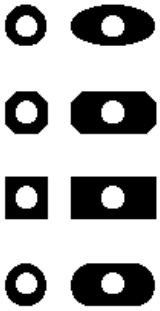
3.4.2 Place Pad

To place free solder pads, press [1] in pointer mode or press the button shown above. These free solder pads are used for the definition of own packages to be exportet into the library. Solder pads with running numbers are created by pressing **[1]** or **M1**. You leave this mode with [ESC] or M12 after you have placed the last solder pad. In Place Pads mode, it is always possible to edit the number and the form of the pins by pressing M11 or [o]:



- **Position:** Shows absolute coordinates
- **Pin number:** Shows the number of the pin, the pad is related to.
- **Pad Name:** If the naming of the pins within the package is alphanumerical, (e. g. A1 - A32, B1 - B32, etc.), here you can drop those names. The pad name must be identical to the one of the corresponding pin of the symbol. The pin number must be identical (e. g. pin name of package A1= pin number 1, A32=32, B1=33, B32=65, etc.).

- **Pad heigth and width:** Shows the dimension of the pad.
- **Form:** Shows the outline form of the pad. The following variations are selectable:



- Round (for round and elliptical solder pads (heigth < width))
- Octagonal (also oblong heigth < width possible)
- Rectangular
- Oval (for oblong solder pads with round endings)

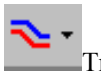
- **Layer:** Defines the layer where the solder pad should appear. For an smd-pad, simply a copper layer number must be entered. For a solder pad with drillhole, layer 100 (special layer) should be used, which means that the drillhole appears on all copper layers. You can define also padsyacks. The according layers herefore are layers 101...254.
- **Rotation:**
- **Pad aura:** The solder pad is also drawn in the corresponding deletion layer. The outline of the solder pad in the deletion layer is drawn bigger with the value given for the aura. This is of importance for generating ground planes.
- **Drill hole:** Sets the diameter of the central drillhole. The drillhole diameter value for smd-pads is 0 (zero).
- **Color:** Define color of a pad independently of predefined color. Is color white (R=255, G=255, B=255) chosen, the predefined standard-color is used. If you wish color white although, you need minimal offset from the above mentioned RGB characters (just one point fits). Pad, color of.

A drillhole for mounting purposes can also be created with the help of a solder pad. First, set the drillhole diameter of the solder pad according to the drillhole diameter needed and secondly, set the pad heigth and width smaller as the drillhole diameter. After you have placed the drillhole, set the pin number to zero with **[w]** and **[:]**.

3.4.3 Draw Signals

Signals represent connections in the schematic. Press **[2]** the menu Elements- Draw Signals to start a signal.

3.4.4 Draw Tracks

 Tracks represent connections on the PCB´s. They behave similar to the signals in the schematic. To have TARGET accept a valid connection, always try to hit the solder pads you want to connect in the center. Press **[2]** in the PCB, select the tool shown above or select the tool from the menu Elements -Draw Tracks to start a track.

3.4.5 Draw Lines

To Draw Lines which are not part of a signal, select the button shown above or press **[Strg]+[2]** (Please note: don´t use the [2]-key from the numeric keypad at the right of your keyboard). These lines are used to draw lines in symbols or packages or an outline of a PCB. Lines can also be used to draw parts for moveable components (potentiometers, relais etc..). For these lines you should select a different kind of line. Select with **M11** or **[o]**
width of the dashed line. If you want to have the ends of the line segments rounded, please select rounded option. A line can be drawn with several **M1**- clicks. You can leave this mode with **M12** or **[Esc]**. Lines can also be changed afterwards with the drag function.

3.4.6 Draw Air Wires (Ratsnest)

To draw connections in a **PCB without schematic**, place them directly pin to pin as air wires (called ratsnest). The autorouter will then take care to route the tracks accordingly. To release a solder point from a signal, double click on the pin with **M11** and delete the signal entry. Pad, release from signal.

3.4.7 Draw Filled Triangles

To Draw Filled Triangles, select the button shown above or press **[3]** in pointer mode. Then you must define two further points in the drawing area with **[Return]** or **M1**. You can call an options dialog with **M11** or **[O]** to select the layer and the page on which this and future filled triangles should appear. Triangles can be dragged at their corners to be resized.

3.4.8 Draw Open Triangles

The same procedure as Draw Filled Triangles, but three seperate lines will be created. This function can also be started with **[Strg]** + **[3]**.

3.4.9 Draw Filled Rectangle

This function is used to Draw Filled Rectangles. Place the cursor on one of the required corner points and press the **[4]** key, click on the button shown above or select the commend from the menu. Then go to the opposite (diagonal) corner point of the rectangle and press **[Return]** or **M1**. An options dialog appears when you press **M11** or **[O]** where you can define the layer and the page on which this and future filled rectangles should appear. Rectangles which have not been rotated, can be dragged [z] at their corners to be resized.

3.4.10 Draw Open Rectangles

The same procedure as Draw Filled Rectangles, but 4 seperate lines will be created. This function can also be started with **[Strg]** + **[4]**. Draw open rectangles.

3.4.11 Draw Filled Slice

Slices are filled circles. Place the cursor in the center of the required slice, click on the button shown above, press the [0] (zero) key or select the command from the menu. Then click with **M1**-or press **[Return]** and enter the radius of the slice. An options dialog appears when you press **M11** or **[O]** where you can define the layer and the page where this and future filled slices should appear. Draw filled slice. Draw filled circle.

3.4.12 Draw Open Slices

The same procedure as Draw Filled Circles, but only one closed torus is created. This function can also be started with **[Strg] + [0]**. Draw open slices

3.4.13 Draw Torusses

You select this feature if you need to draw only a segment of a circle (torus). Place the cursor on the starting point of the torus. The second point required to draw a torus is the center of the circle and can be entered with **[Return]** or **M1**. The third point required is the stopping angle of the torus. Are point one and three identical, a full circle will be drawn. Torusses will be drawn in a mathematical positive direction. This means against clockwise. You can call an options dialog with **M11** or **[O]** and enter the following information:

- **Line width:** Width of the torus line.
- **Aura:** The torus is also drawn in the according deletion layer. Additionally, the aura value is added to added to the line width.
- **Layer/Page:** Here you define on which layer of the PCB or page in the schematic this and future torusses will be drawn. Draw torus.

3.4.14 Place Junctions

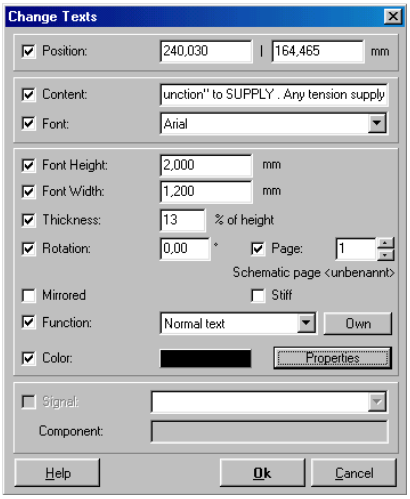
To enter the Place Junctions mode, click on the button shown above, press the **[.]** key or select the commend from the menu. Press again on **[.]** or click **M1** to place a junction on the drawing area. Is a new connection established between two different signals, TARGET asks you, if you accept this connection and which new signal name should be used. After a junction is established, TARGET switches into draw signals mode. Press M11 or [o] in junction mode to select the options of this and future junctions. Node.

3.4.15 Place Vias

To enter the vias mode, click on the button shown above, or press the **[.]** key. Pressing again on **[.]** creates a via on the drawing area. Is a new connection established between two tracks with different signal names, TARGET asks you, if you accept this new connection and which new signal name should be used. TARGET switches immeaditely into place tracks mode. Press **M11** or **[O]** in vias mode to select the options for this and future vias. (see also solder pads). Vias are visible on all copper layers, (special layer 100) or Padstacks (layers 101...254).

3.4.16 Insert Text

To enter the "Insert Text" mode, click on the button shown above or press **["]** or **[\]** directly in the window or select the commend from the menu. The following dialog box with text options appears:



Place your text or any name in the content field. Then select a Windows **font**. Additionally, the font "**TARGET**" appears in the list of fonts. This "**TARGET**" font is used for outputs to external devices (for example: GERBER). Please note that these Windows fonts except "**TARGET**" are currently not supportet from these output drivers! **Font height and width** define the size of the letters. Windows fonts can differ significantly from these values. The thickness option is only used directly from the "**TARGET**" font. When the normal font is between 10% and 20%, TARGET switches to bold font. Texts can be **rotated** and **mirrired** in any direction. When you rotate texts in TARGET, these texts are usually readable from below or from right, according to DIN. The entry "**Stiff**" means, that a text, if rotated against these rules, will be rotated stiff. The **function** field lets you select from the following options:

- "**Component name**" for the designation of the component (for example: IC17, R15);
- "**Component value**" (z.B. 4040, 4k7);
- "**Symbolname**" for the name of a symbol (for example:. IC17A);
- "**Signal name**" für den Namen eines Signales (for example: GND, Sig$13);
- "**Pin name**" für den Namen eines Anschlußes (for example: RES\, Pin8).
- "**Package, Part number, Manufacturer/Vendor, Order number, Price**" is related to the components and is used to generate a bill of material (B.O.M).

Several name exchanges:

The contence of the following names will be exchanged on the screen when displayed:

- **!FILE:** with the file name (for example: TEST.T3001)
- **!SCALING:** with the output scaling factor (for example: 200%)
- **!C-DATE:** Date of the last change of the file (for example: TT.MM.JJJJ)
- **!C-TIME:** Time of the last change of the file (for example: HH:MMh)
- **!P-DATUM:** Actual date of the output (for example: TT.MM.JJJJ)
- **!P-TIME:** Actual time of the output (for example: HH:MMh)
- **!PAGE:** Current schematic page

Please note the correct writing of the letters (Upper case!). These names are mainly placed in the drawing outline. (Frame.cmp3001).

3.4.17 Draw Busses

A bus in TARGET is a collection of all signals who appear in the circuit. You can distinguish several busses in your schematic graphically (databus, address bus...), however you can connect every signal to each bus.When you start a signal from a bus, a list with all signals is displayed to choose from. To start a bus, select this menu command or press **[b]** in the schematic. Place individual bus segments **[b]** or **M1**. You can leave this mode with **[Esc]** or **M12**. A Click with **M11** or **[O]** for options displays the bus options dialog box. Here you can set your parameters. The following entries are of interest:

Width: Width of the bus lines. Place a new, unknown signal in direction **to the bus**. Take a known signal **out of the bus**. See also Chapter 5.

3.4.18 Place a Bridge

To "**Place a Bridge**" in TARGET, select the button shown above, press the **[b]** key or select the command from the menu. Place the end of the bridge with **[Return]** or **M1** on the PCB. A Click with **M11** or **[O]** for options displays the options dialog box. Here you can set your parameters. The following entry is of interest:

Width: Width of the bridge lines.

A contact is automatically created at the beginning and on the end of the bridge. The drawing of the bridge is made on the **position top layer**. A bridge behaves like a track. You are in the track mode after you have placed a bridge.

3.4.19 Place Centering Marks

Centering marks (fiducials)are used later on to center all of the layers used in the project for the production process. Select this modus to place the passermarks with **[Return]** or **M1**-clicks. Click with **M11**or press **[O]** for options to dispaly the options dialog box. Here you can define the form and the radius of the fiducials. You can select from the following forms:

- **Destination:** A fiducials like this is made of concentrical circles and a central cross.
- **Arrow:** A circle with an arrow in the center. This fiducial also informs about the placement direction.
- **Two Quarters:** A circle contains two quarters of a circle. This fiducial can be placed and turned at 90 on several layers and is ideally suited to adjust several foils. When these foils do exactly match, a full circle is displayed.

The line width defines the width of the line that is used to draw the fiducial. The **direction** parameter is used to define the angle on which the fiducial is **rotated** around its center. Also, you define the **layers** on the PCB on which this and future fiducials should appear.

3.4.20 Draw Spirals

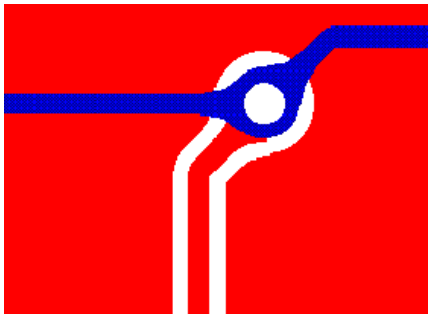
It is possible to place spirals in TARGET when you select this symbol or select this command from the menu. Place the end of the spiral with **[Return]** or **M1** on the PCB. Now you can edit its options with **M11** or **[O]**. A spiral reacts similar to a track and is made of several torusses. You are automatically in the track mode after the placement of a spiral.

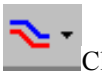
Click with **M1** on the small arrow on the button shown here on the right side: Click with **M1** on the spiral symbol shown above.
Click with **M1**
Pressing the **[O]** key or **M11** shows a dialog box, where it is possible to edit the spirals options.(Amount of windings, Rotation orientation).

Press **OK** (Return key).
Place the spiral with **M1** on the PCB.

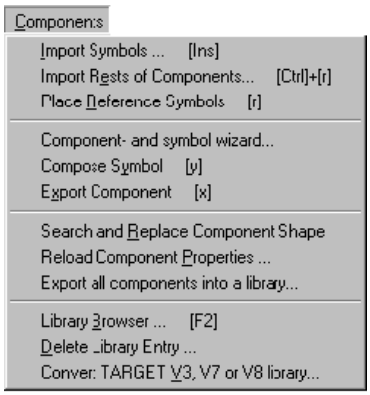
3.4.21 Teardrops (Tears)

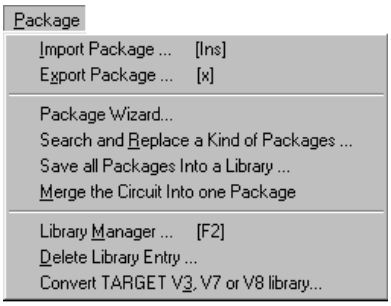
To enter the "Place Teardrop" mode, select the button shown above. A click with **M1** on a solder pad can change the solder pad with a track into a solder pad with teardrop connection.



- Click with **M1** on the small arrow on the button shown here on the right side:  Click with **M1** on the teardrop tool (Water drop). The teardrop is symbolically displayed at the cursor.
- Place the cursor on a solder pad on the PCB.
- Click **M1**. In the picture below, you can see that TARGET has changed automatically all according tracks of a solder pad or a through hole into a teardrop. A already generated ground plane is updated and displayed in realtime.

3.5 Components / Package Menu





Components menu in schematic view and package menu in PCB view

Contents:

3.5.1 Import Symobols
3.5.2 Import Rest of Components
3.5.3 Place Reference Symbols
3.5.4 Compose Symbol
3.5.5 Export Component
3.5.6 Search and Replace Component
3.5.7 Reload Component properties
3.5.8 Import Package
3.5.9 Export Package
3.5.10 Search and Replace a Kind of Package

3.5.11 Save all Packages into a Library
3.5.12 Merge the Circuit into one Package
3.5.13 Library-Browser
3.5.14 Delete a Library Entry
3.5.15 Convert TARGET VX Library...

3.5.1 Import Symobols

Select this function, press **[INSERT]** or **[e]**, select the button shown above or select the command from the menu. This function is used to insert the first symbol or further symbols of a component. A dialog box is displayed where you can select a library on the right side and a component from the according library on the left side. After you confirm the import dialog, the first symbol of the component is inserted to the schematic. The outline of the symbol is attached to the cursor (move mode) and can be once or several times be placed with **M1**. Click **M2** to rotate the symbol if neccessary. Your cursor gets rid of the component by pressing **M12**.The component is automatically numbered consecutivrly (for example: . IC1). If the symbols includes also texts, the following replacements take place: the name **!COMPONENT**
component designator (IC1, R1); the name **!SYMBOL**
designator (IC1a); the name **!VALUE** or a text which is saved as the component value is replaced with a component value (4000, R). The component values 4000 and R should be changed afterwards to CD4000 and 4k7 with a click on **M11** on the symbol handle. With this function, only symbols with the attribut: Import Rest of Components are inserted from the library (for example the two NOR-gates of the CMOS 4000). Symbol, import symbol to schematic.

3.5.2 Import Rest of Components

This function is used to import the remaining symbols of a component into the schematic. For example, the power supply symbol of a component can be importet, which is usually not neccessary in the schematic (attribut: insert mode on request). To use this function, select the button shown above or press **[Strg] + [r]**
Select now the name of the component. TARGET will display a list showing all remaining symbols of this component to choose from. Components not listed in this dialog do not exist or have been importet completely. Components, rests of.

Attention: Deleted schematic symbols are shown in this list. To finally delete a symbol from this list, choose the reorganization command in the File menu. See also the notes in Chapter 3.5.1.

3.5.3 Place Reference Symbols

Visually not corresponding islands of the same signal or connections over several schematic pages should show there correspondence with the help of reference symbols. Often power supply connections are drawn with the help of arrows and ground symbols. To start such a connection, use this menu command, click on the button shown above or press the **[R]** key to select the preferred reference symbol. The visible pin name must match with the signal name. If not, please edit the reference symbol after its placement with **M11** (Change pin name). You can use reference symbols to create connections over several schematic pages. See also Chapter 4.4.

3.5.4 Compose Symbol

A component in the schematic contains at least one or more symbols. A symbol is made of several drawing elements (lines, pins etc..), which have been composed to a complete signal sometime ago with the help of this menu command, or it needs to be composed to a complete signal.(recreation of a component). Select the according drawing elements who should belong to a symbol and call this function. Afterwards, click with **M1** on the spot where the handle of the symbol should appear. You can also use the **[y]** key for this function. The cursor should be placed on the spot where the handle should appear. Call the options dialog with **M11** or **[O]** and set the attributs of the symbol to be created next. The symbol number should start with one in any case. The suffix is the designator of this symbol (for example: a bei IC1a). TARGET automatically increase the the symbol number and the suffix when a new symbol is created. The swap number can only be inserted afterwards by a click with **M11** on the symbol handle. A value other than 0 (zero) means: Symbols in a component with the same swap number can be exchanged (symbol change, gate change). The entries for the Insert mode have the following meaning:

- **Insert automatically as next means** that the symbol will be importet into the schematic, when you select next time the Import symbol command. If all as next- symbols of a component are used, a new component will be started.
- **Insert extra as Rest means** that the symbol will only be importet into the schematic on special request. You must place it directly with the help of the command Import Rest.

Swap Number: Symbols, who have the same swap number in the component, can be exchanged with each other (symbol-/gate exchange). The swap number must be greater than 0 (zero). Page: The page where the symbol is currently placed. See also Chapter 4.1. Compose symbol.

3.5.5 Export Component

You must use the Export Component function to export a component with its schematic symbols to a library. Draw the different symbols of the component on a free spot in the schematic and use the Compose Symbol procedure to collect the symbol. Select now all symbols of the component at the same time (for example: draw a catch window with **M1H**) and call the Export Component function from the menu. You can select now a library and select or create a new component name. Afterwards, you must select a designator prefix for the component (for example: T for transistoren or IC for Integrated circuits) in the center of the dialog box. Also, you have to enter a package name (for example: DIL14). This package name will later be proposed when this package is selected for the component. At this point, you can only select a package who already exists. This means that the package must be contained in the library. In some cases, the package must be drawn first. If you have no recommendation for the package, you have selected **<without package** , no package is later on required for the PCB (for example: the component DINA4HOR (schematic outline frame) will never need a package).

If the component name already exits in the library, TARGET will ask if you want to overwrite it.

3.5.6 Search and Replace Component

Select first a kind of component in the left side, and the components to be replaced from the list. Then select the new component from the library on the right side. Activate the according check-box for the component information of the attribut list in the section below. This function is very helpful if you need to replace global several components in the schematic. Component, search and replace.

3.5.7 Reload Component properties

Only the attributs of the components in the schematic will be replaced here. Select first a kind of component on the left side and the components for the attributs to be replaced from the list. Then select the new component from the library on the right side. Activate the according check-box for the component information of the attribut list in the section below. This function is very helpful if you need to replace global several component attributs in the schematic. Reload component properties.

3.5.8 Import Package

To import a package, select this function, press directly the [INSERT] or [e] key in the PCB or select the button shown above. Now you first have to select the component to whom the importet package should belong (for example: for IC1). In case you need a loose package (a package with no reference to the schematic), select free package. A dialog will appear where you can select a library on the right side and the package of choice on the left side. Usually, a library and a package will be suggested. In most cases, you should confirm this suggestion with Ok. You should only edit this dialog if you are sure that you need a completetly different package (for example: SMD16 instead of DIL16). After you confirmed this import dialog, the package will be placed on the PCB. The package outline is attached on the cursor and can be placed **M1**, or rotated at a preset value with **M2**. Get rid of it by **M12**.

If a text element belongs to a package drawing, the following names will be exchanged: The name **!COMPONENT** or a text saved as the component name, will be exchanged with the component designator (IC1, R1); the name **!VALUE** or a text saved as the component value, will be exchanged with the component value (4000, R). Package import.

3.5.9 Export Package

You must use the Export Package function to export a package with its solder pads and outline drawing to a library. Draw the package outline and the solder pads of the component on a free spot on the PCB. Select all parts of the package and call the Export Package function in the menu. As with the Import Package function, you can select now a library and a name for the package.

If a package already exists in the library with the same name, TARGET asks if you want to overwrite it. As far as it concerns the layer information for graphical elements, the layer function is saved instead of the layer number. A line on the Position Top layer will always be placed on the Position Top layer during an import, independant of the individual assigned layers during an import or export. If you have used your own layer functions, the name of the layer function will be saved also. TARGET checks here for exactly the same name. Package export.

3.5.10 Search and Replace a Kind of Package

Select a kind of component on the left side and the package to be replaced from the list. Select now on the right side the new package from a package library. This function is of great help, if you need to replace global several packages on the PCB. Search and replace kind of package.

3.5.11 Save all Packages into a Library

This menu function saves all packages of the actual project into a selected library. Save package.

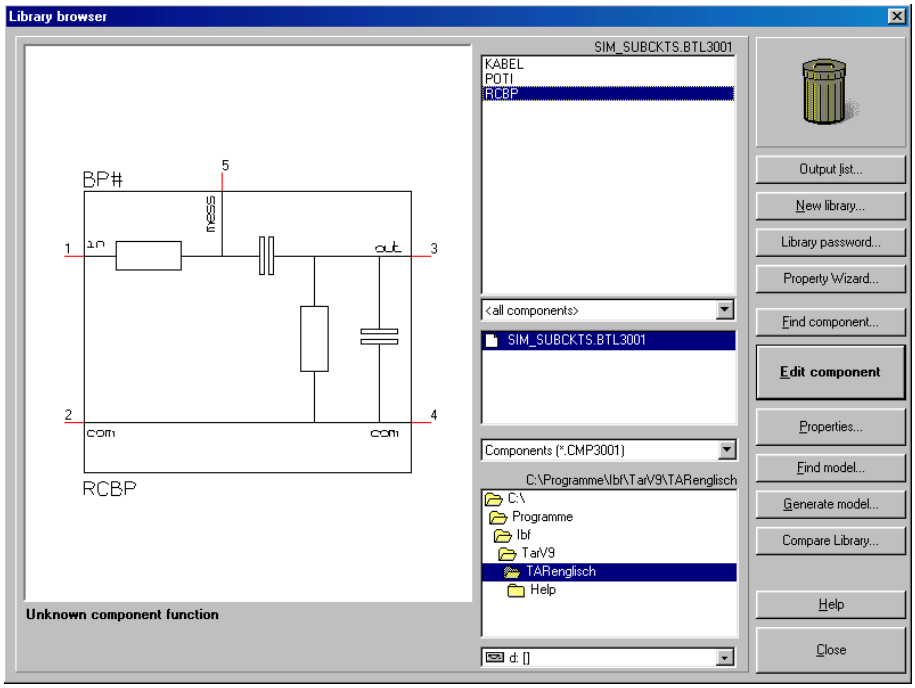
3.5.12 Merge the Circuit into one Package

The whole circuit will be transferred into one package, similar to a modul. The advantage is that signals can already be placed inside the component. Only the amount of external pins of the modul and their signals must be defined. All not assigned signals remain internally. Merge circuit into one package.

This is the dialog:

3.5.13 Library-Browser

The library browser uses drag & drop to manage its components. Additional information can be edited for each component to generate a B.O.M. The library browser can be seen as a standalone program.



First select the library you want to edit. From the right list, select the actual library and from the left list, select the actual component that you want to edit. The component appears in its maximum size in the preview window.

Drag & Drop: It is very easy to manage the libraries with the drag & drop function. You can select on or more components in one list to delete, move or copy them.

1. Selection of Components

Select a Component

- Click with **M1** on the component or press **[TAB]** to go to the component list, then use the **[ArrowKeys]**.

Select entries that folow each other

- Click with **M1** on the first entry, press and hold the **[Shift]** key while you click on the last entry of the group.

- Press the **[Strg]** key while you click on individual entries or press **[Shift]+[F8]** and move the selection with with the **[ArrowKeys]** to the individual entries. Press the [Empty Space key] to select or deselect. Aterwards, press **[Shift]+[F8]** again.

2. Deletion, Moving and Copying of Components

You can delete, move and copy components from on location (source) to the other location (destination) with the drag & drop method. The destination can be another library (for example a special library for your project) or the trash can.

Deletion of Components

- Select the components you want to delete, press the [DELETE] key or drag all selected components with M1 into the trash can.

Moving of Components

- To move one or more components into another library, press and hold the **[Shift]** key, while you drag the object with **M1H** (a component symbol with a minus sign appears at the cursor). The components dissappear in the source library.

Copying of Components

- To copy components into another library, drag the object to the according library (a component symbol with a minus sign appears at the cursor).

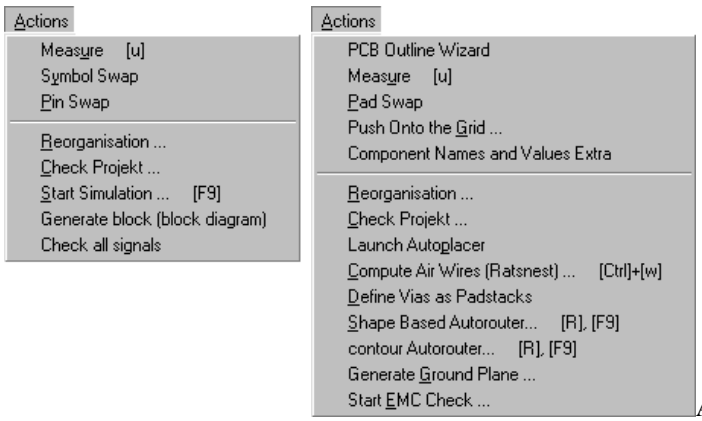
3.5.14 Delete a Library Entry

Select the component to be deleted from the list on the left side from the selected library on the right side. Pressing OK will delete the component.

3.5.15 Convert TARGET VX Library...

Select Convert TARGET VX Library... to convert older libraries from TARGET for 16-bit Windows 3.1 to the modern TARGET3001! format. The libraries will be automatically converted after the according selection in the File-dialog and get the file extensions *.cmp3001 and *.pck3001. The new libraries will be saved in the same directory as the old VX libraries. Library, converting.

3.6 Actions Menu



Actions menu in schematic- and PCB-view.

Contents:

3.6.1 Measure

3.6.2 Symbol Swap
3.6.3 Pin Swap
3.6.4 Reorganisation
3.6.5 Check Projekt
3.6.6 Start Simulation
3.6.7 PCB Outline Wizard
3.6.8 Push Onto the Grid
3.6.9 Component Names and Values Extra
3.6.10 Launch Autoplacer
3.6.11 Compute Air Wires (Ratsnest)
3.6.12 Define Vias as Padstacks
3.6.13 Shape Based Autorouter
3.6.14 Contour Autorouter
3.6.15 Generate Ground Plane
3.6.16 Start EMC-Check...
3.6.17 Generate block (block diagram)
3.6.18 Variants of component insertion

3.6.1 Measure

Use this function from menu or the button shown above to measure distances in the schematic and PCB. Click with **M1** on the starting point. The status line shows the x- and y-values of the distance (dX,dY) as well as the distance itself (L).

3.6.2 Symbol Swap

Also called gate-exchange. Select this menu function or the button shown above if you want to exchange two identical symbols to simplify the routing of a PCB. The symbols must be identical and of the same exchange number (0). Symbol swap.

3.6.3 Pin Swap

To swap (exchange) two identical pins of a symbol, select this menu function or the button shown above. The pins must have identical swap numbers (0). Pin swap.

3.6.4 Reorganisation

This function is used to reorganise the whole project. See also .

3.6.5 Check Projekt

This function is used to check the complete project on electrical connection errors, logical connection errors (Electrical Rule Check = ERC) and on layout errors (Design Rule Check = DRC). See also

3.6.6 Start Simulation

This function starts the TARGET 3001! Mixed mode simulator. See also Chapter 8.

3.6.7 PCB Outline Wizard

The PCB outline wizard assists you step by step in creating the PCB outline.

3.6.8 Push Onto the Grid

Sometimes, the components cant be placed on the presetted grid. With this function, it is possible to move the package onto the next grid point. Pin #1 or the handle of the package is used as a reference point.

3.6.9 Component Names and Values Extra

This menu function defines seperate layers for the component name and - value and moves the **existing** texts to these layers. If the PCB is also populated on the bottom side, 4 new layers are created. Now it is possible to display the texts of the components name and - value seperate and independant of the package drawing. All components placed later on will use these layers, if they have texts with these text functions.

If you want to assign an additional name or value text to a component, so create a text near to the component handle. Set the text function to e.g. "Component name" and the layer entry to "Position top". the text will be assigned to the component and shows the right component name directly.

3.6.10 Launch Autoplacer

This function starts the "**Autoplacer**". The "**Autoplacer**

Chapter 6.5.

3.6.11 Compute Air Wires (Ratsnest)

This function recalculates the display of the air wires (ratsnest). If your project had no "**Air Wires**" layer, a layer is generated and attached to the current layers and displayed. See also **Chapter 5.5**.

3.6.12 Define Vias as Padstacks

This function is used to write special through holes (vias). Standard vias on layer 100 reach all copper layers at this spot and the solder pads of this stack are identical for all copper layers.(Copper top, bottom, inside ...). On multi layer PCBs (for example: 4-layer PCB), it is often desired to connect only the two top layers. This construct should takefore no space on the two bottom layers. To solve this problem, a special plated blind hole is necessary, called a **blind via**. It is also possible to construct vias to who connect only the two inside layers without being visible from the outside. These vias are called "**buried vias**". These two methods are very **expensive and complicated**! Also, it is possible to resize the solder pads of such a solder pad stack (**Padstack**) on each individual layer.

3.6.13 Shape Based Autorouter

Here comes the "**Autorouter**" in place. It can be used to route single tracks or to route the whole PCB. This function starts the "**Ripup & Retry Autorouter**". The type of autorouter works gridless internally, but uses a predefined route-grid. See also **Chapter** 6.3 - 6.4.

3.6.14 Contour Autorouter

The "**Contour-Autorouter**" is a fully gridless and grid-independant router. It places the track in the first run so, as the track is the only one to be placed. Errors caused by crossings and distance violations are normal and almost desired. In the following runs, the errors get expensive for the router and therefore more rare until a 100% resolution is hopefully reached. In the following "**Optimizing**" runs, the preferred directions are "**forgotten**" and the traces will be straightend. See also **Chapter** 6.3 - 6.4.

3.6.15 Generate Ground Plane

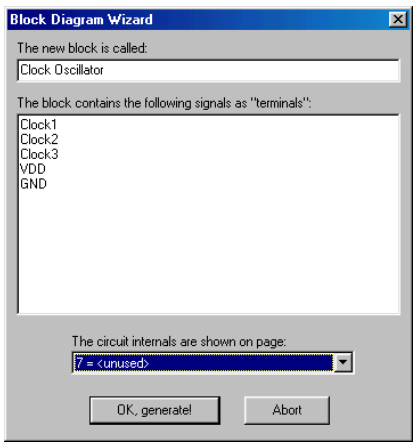
This function allows you to generate ground planes in realtime (no calculation time), by just clicking a button. TARGET uses the "**Automatic Goundplane-Assistant**". TARGET is the **only**
at no calculation time. See also **Chapter** 6.7.

3.6.16 Start EMC-Check...

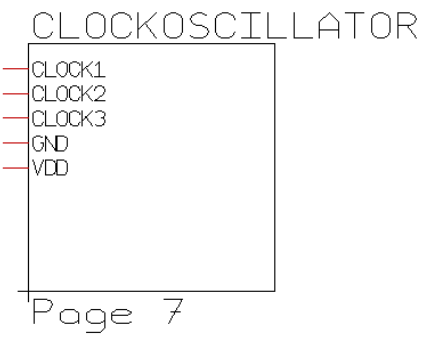
This function starts the "**EMC-Check**". See also **Chapter** 9.

3.6.17 Generate block (block diagram)

TARGET allows to generate block diagrams very easily:

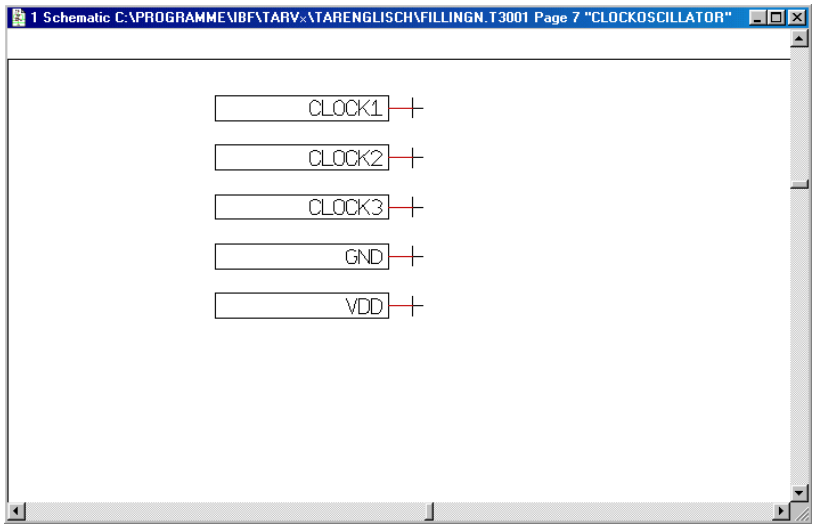


Now you can move and drop the following block:



The reference connections can be displaced within the block by **M1h**. Please note that key is not active. would it be so, the whole block would move .

Simultaneously you will find the corresponding block on page 7

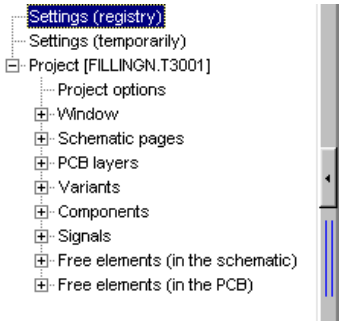


Now you can post the reference symbols to the desired spot.

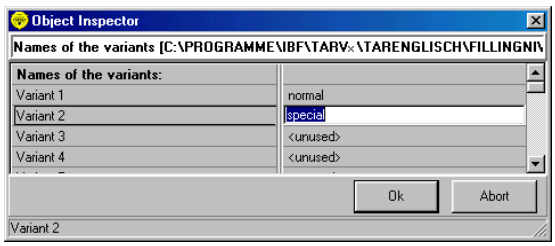
3.6.18 Variants of component insertion

PCBs with its components differently inserted regarding the variants of component insertion can easily be handled.

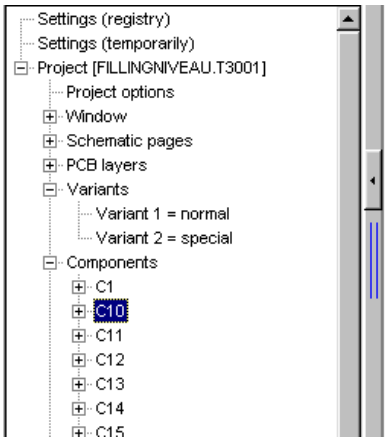
Mark in the hierarchy browser (see right edge of your screen) by M1 "Variants":



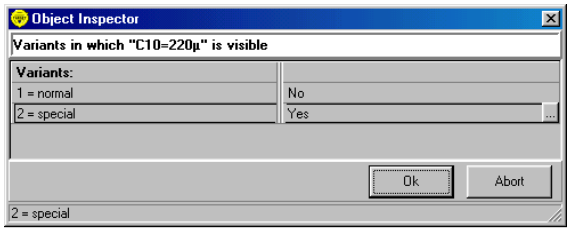
Now click M2 for to select "properties" within the context menu. Within the following dialog you can name variants as desired:



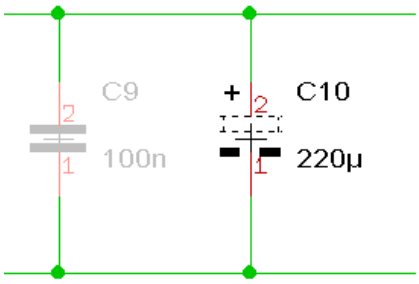
In the hierarchy browser now click M1 within components the varied component and then M2 to reach menu "properties":



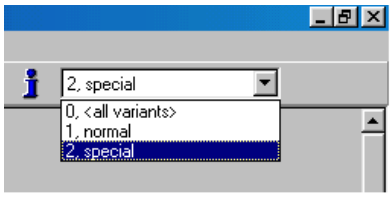
The variants in which the component is proposed to appear can now be edited:



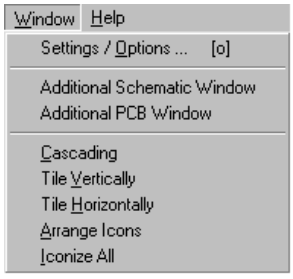
The outline of the component appears hatched, as far as the component is not wanted within all variants.



For toggling between the variants use the pull down info box to the right on the toolbar (appears only if variants really do exist).



3.7 Window Menu



Contents:

3.7.1 Settings/Options
3.7.2 Additional Schematic Window
3.7.3 Additional PCB Window
3.7.4 Cascading
3.7.5 Tile Vertically
3.7.6 Tile Horizontally
3.7.7 Arrange Icons
3.7.8 Iconize All

3.7.1 Settings/Options

This dialog is for to fix main settings/options. Press key [o]. See also chapter 2.9.14.

3.7.2 Additional Schematic Window

Creates an additional schematic view. Those page is shown you lately worked in. Eventually use function "go to page" within menu View for zo choose another page (e.g. page 2).

3.7.3 Additional PCB Window

Creates an additional PCB window. First a standard view is shown. Is a PCB view yet open so it is copied within another window. You eventually must activate further layers (by "Layers"-function) for to toggle within several PCBviews quickly.

3.7.4 Cascading

TARGETshows all its "child-windows" cascading.

3.7.5 Tile Vertically

TARGET shows all its "child-windows" tiled vertically.

3.7.6 Tile Horizontally

TARGETshows all its "child-windows" tiled horizontally.

3.7.7 Arrange Icons

TARGET "child-windows" can be reduced to icons within the lower edge of its "mother-window" .

3.7.8 Iconize All

Reduces all windows and arranges them on the bottom left of the TARGET main window.

3.8 Help Menu

Contents:

3.8.1 Tools
3.8.2 Index

3.8.3 Context help
3.8.4 Info (used pins/pads etc.)
3.8.5 Online Service, approval and criticism
3.8.6 Free update
3.8.7 Recommend TARGET3001!
3.8.8 About TARGET

3.8.1 Tools

The following tools can be used:

Package Color Codes (Resistors, small capacitors, big capacitors)
Compute the Insulation Spacing
Compute the Track Resistance
Compute the max. Track Current Load

3.8.2 Index

If help is needed, several ways of achieving it are possible. Similar to the help in Windows you can search for topics, display an index or switch whithin the help over to next topic using cross refers (hypertext). All files are available in HTML format thus any browser can be used. You may use **Netscape Communicator** or **Microsoft Internet Explorer** for watching the files and additionally use functions as e.g. **printing chapters**, which is not supplied by the browser delivered.

3.8.3 Context help

Activate the online help whithin your current working mode (context sensitive). Press **[F1]**.

3.8.4 Info (used pins/pads etc.)

Gather information on your system with this function. You may be interested in the occupation of Windows or theTARGET database in respect to your license (see "Used Pins").

3.8.5 Online Service, approval and criticism

You can directly contact Ingenieurbuero Friedrich out of the program. You can also add your problem file.Contact to Ingenieurbuero Friedrich.

3.8.6 Free update

This dialog enables you to free update your version as long as the initial version number is identical (e.g. 10.xx.xx.xx)

3.8.7 Recommend TARGET3001!

You can directly open a Text document to recommend TARGET to somebody.

3.8.8 About TARGET

An information-sheet opens about TARGET, Ing.-Buero FRIEDRICH and the **license holder**.

Chapter 4: Create Components

The library contains more than 5000 components. You can easily draw any component within TARGET´s user environment in that case, that the library does not
contain the needed component(s). A special program (component-editor) is not necessary. This chapter explains in detail the steps how to create schematic
symbols, reference symbols and component packages.

Contents:

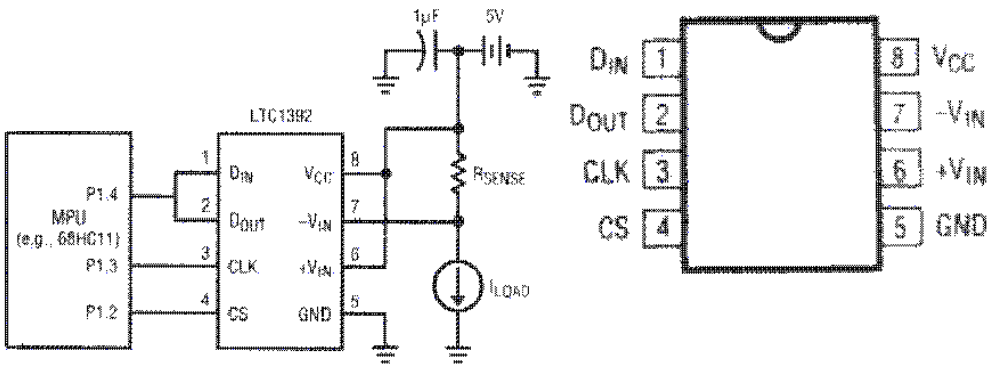
4.1 Create a simple schematic symbol
4.2 Create a schematic symbol with multiple gates
4.3 Change an existing schematic symbol
4.4 Create a reference symbol
4.5 Create a package
4.6 Change an existing package symbol

In case you don´t find a certain component in the libraries, you can easily create it yourself. The component will be drawn directly in the schematic. It is recommended to open an empty project to "play around" with that feature. The grid-size should be set to 0.635mm. The pin-to-pin distance in the
schematic should also equal 2.54mm (= 4 steps).

4.1 Create a simple schematic symbol

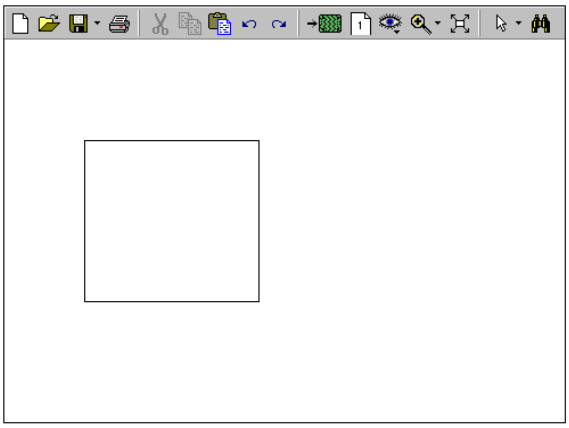
Lets say, you wouldn`t find the LTC1392 from Linear Technology in the library. To create this component, please use the components original datasheet (This datasheeet can be obtained from the Linear Technologies homepage at http://www.linear-tech.com).

View from the top and a typical application:

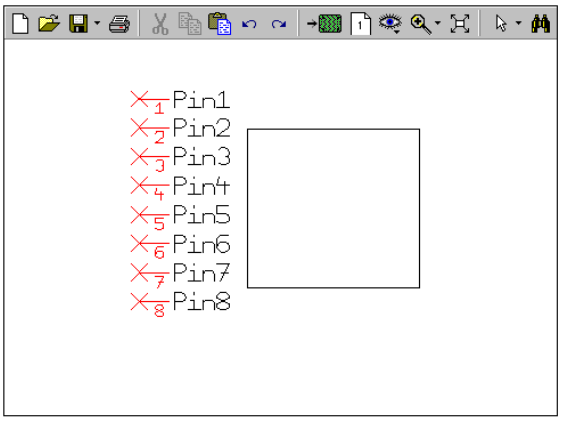


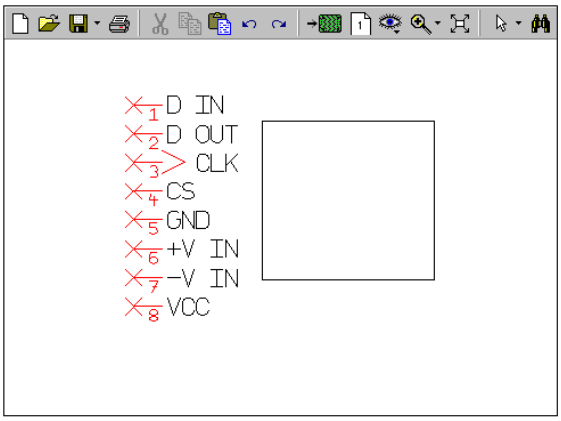
1. Open a new project with an empty schematic page.

2. Either, use the lines **[Ctrl]+[2]** or an open rectangle **[Ctrl]+[4]** to draw the components outline.

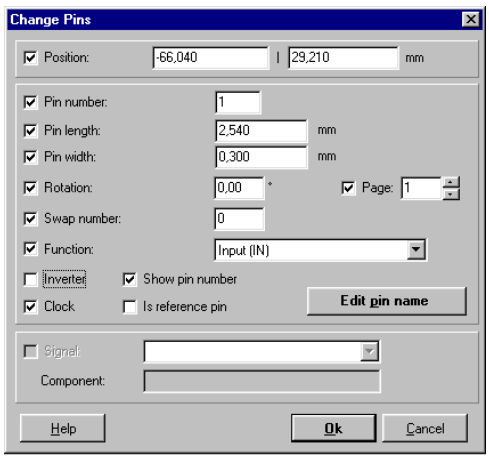


3. Use the key **[1]** and then **M1** to place all necessary pins one beneath the other on a free area in the schematic. TARGET automatically assigns the pin names and the pin numbers.





4. Select the correct function of each pin according the given information in the datasheet. To do that, please click with **M11** on a pin. The dialog" Change Pins" appears.

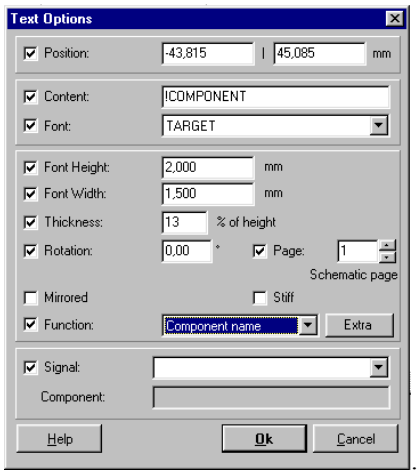


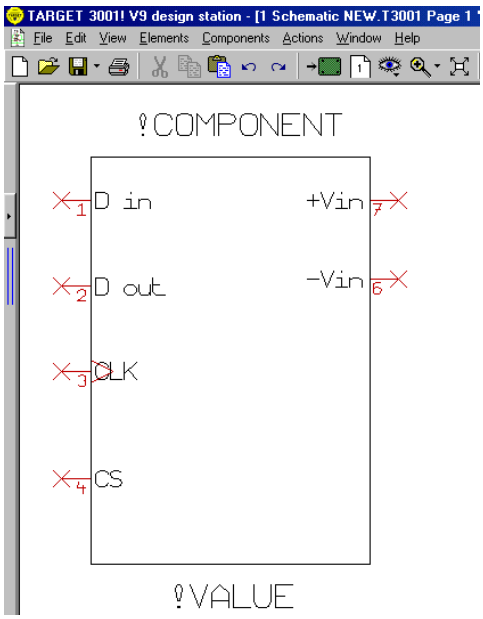
5. Click on the **Function box** and select the according **pin function** of pin #1 Input (IN). You can also edit the function "**Inverter**" and "**Clock**" (e.g. pin #3) in this dialog. The description of the individual pins can be edited by clicking on the "**Edit pin name**" button.

6. Select OK (Return key).

7. Mark all pins with **M1** and move them with **M1H** to the correct location in the symbol. If they need to be rotated around cursot position press **[t]** for rotation .

8.Assign a name and a value as the components name with the **["]** key. Select "**Component name**" in the **function field** in this first text option dialog. A place holder "**!COMPONENT**" appears automatically representing the components name. Later, this place **IC8**. Please set the **function field** to "**Component value**" in the next text option dialog. **!VALUE** appears now automatically as a place holder representing the components value. Again, this place holder will be substituted later by the components real value, for example **LTC1392**.





9. Mark (catch) the symbol (rectangle, pins and names must be selected completely!). Do this by drawing a catch window with **M1H** in "pointer-mode".

Please Note: The power-supply connections (Pin #8 = VCC and Pin #5 = GND) will be drawn and defined seperately, so please do not mark them now !!

10. Place the cursor on the spot where the handle should appear (for example in the middle of the symbol).

11. Press the **[Y]** key. All marked elements will be assembled now to allow an export of a symbol.

12. Creation of the power-supply connections:

Please place the connections as shown in the following picture and assign a component name. The pin names can be deleted with the **[Delete]** key, but they will still exist logically.

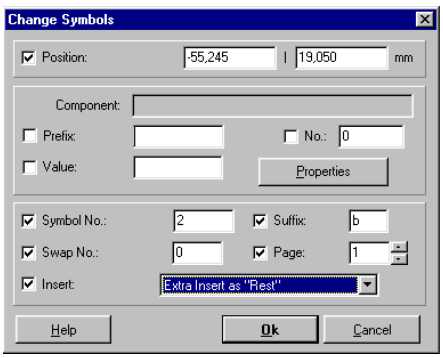
13. Mark (catch) the power-supply connection (name and pins must be selected completely) with **M1H** in "pointer-mode" by drawing a catch window over the symbol.

14. Place the cursor at the spot where the handle should appear (for example in the middle of the power-supply symbol).

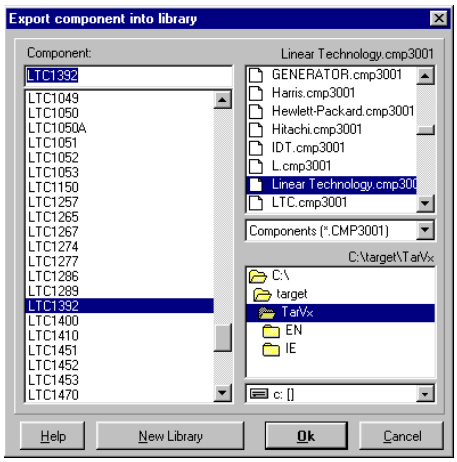
15. Press the **[Y]** key.

16. Click with **M11** on the handle and edit the following modifications in the shown "**Change Symbols**" dialog:

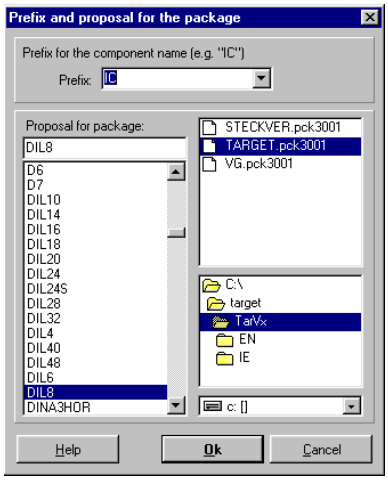
- Symbol number = enter 2
- Insert = "Extra Insert as Rest" (power- supply connections will be placed as a "Remainder of a component" in the schematic)

- Suffix = p for power (power-supply connection) 

17. Now mark the component and its power-supply symbol together by drawing a catch window over the selected symbols. Press the **[X]** key to export the component. The dialog "Export component into library" appears:



- Select the according symbol library
- Enter a component name, for example **LTC1392**
- Select **OK**



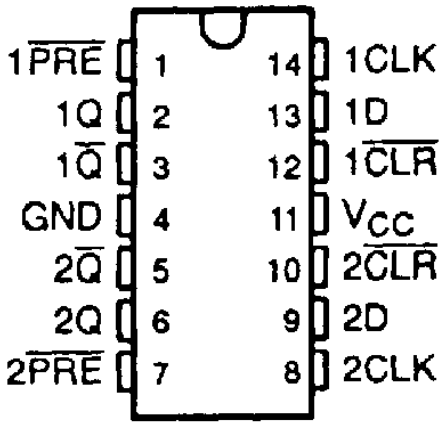
- Enter **IC** as a **prefix.**
- Select the according component library and package (for example **DIL8)**.
- In case you dont want to assign a package to a component (in the case of a reference symbol or a schematic outline frame), you can select <No Package>.
- Select **OK**

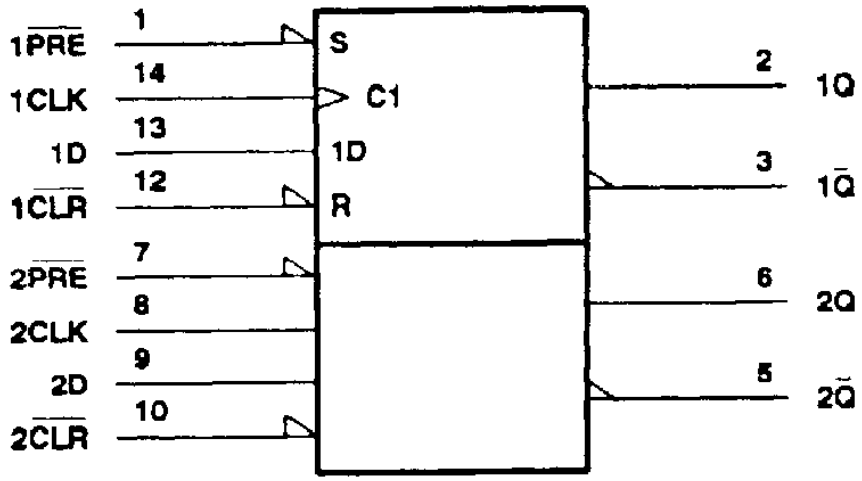
Create symbol

4.2 Create a schematic symbol with multiple gates

Lets say that the 74ACT11074 from Texas Instruments is not in the component library. To create this component, please use the original datasheet as a reference. This datasheet can be obtained from Texas Instruments homepage under and contains the following figures:

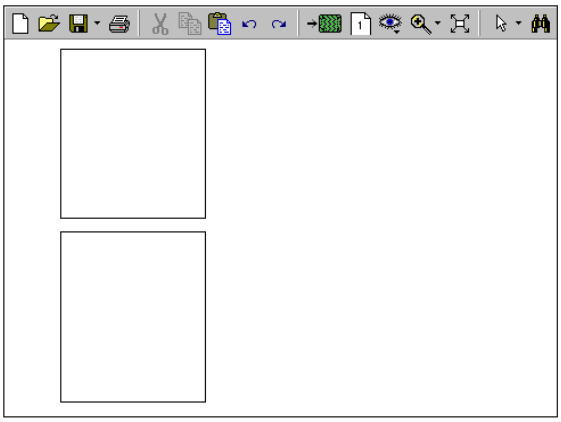
Top view and logic symbol:





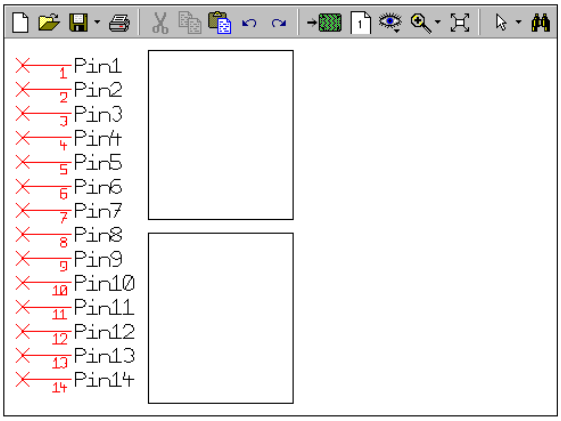
1. Open a new project with an empty schematic page .

2. Draw the symbol with the draw line tool **[Ctrl]+[2]** or with the draw rectangle tool **[Ctrl]+[4]**. You should draw all gates at once for a component with multiple gates (for example: 74ACT11074).



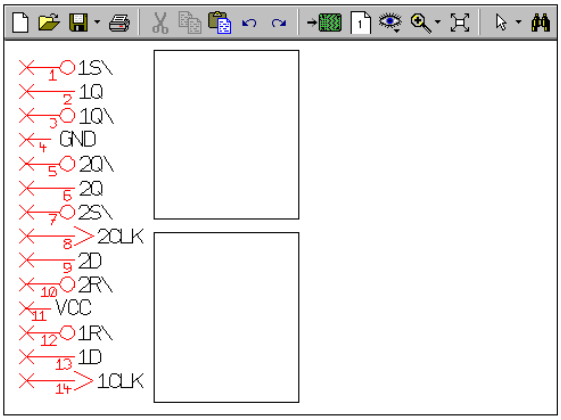
* Draw symbols

3. Place all necessary pins one below each other with the **[1]** automatically.



* Place the pins

4. Enter the functions of the pin names according the information given in the components datasheet. Click with **M11** on a pin (the " **Change Pin**" dialog appears).



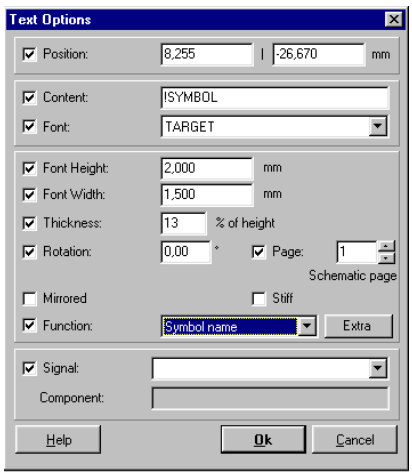
* Set the pin properties

5. Click on the "**Function**" window and select the right pin function (for example: Pin #1: **Input(IN)).** The clock- and/or inverter functions of a pin can be selected here in this dialog (for example see Pin #1: 1S\ = Inverter, Pin8: 2CLK = Clock).

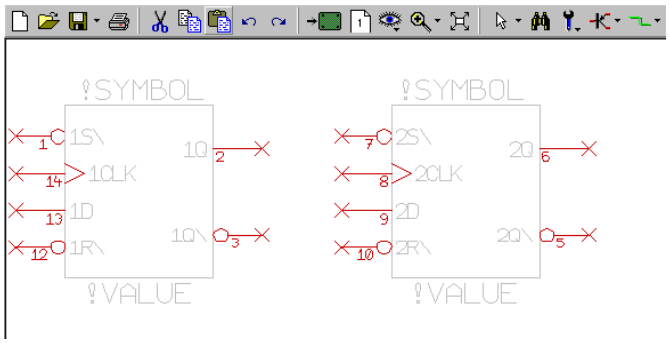
6. Change also the **pin name** here.

7. Mark all pins with **M1** seperately and move them with **M1H** to the correct spot at the two symbols.

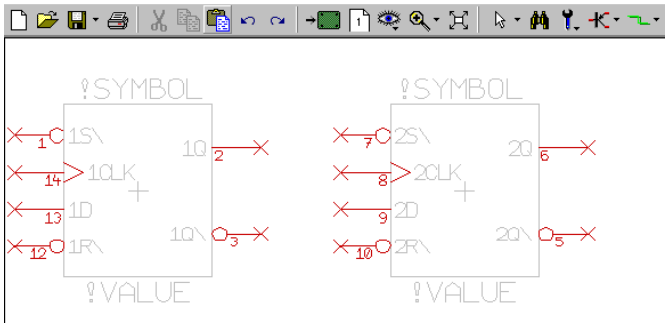
8. Name each symbol with **["]** as following (the "**Text Options**" dialog appears):



- Enter **"!SYMBOL"** for the component name and **" !VALUE"** for the component value in capital letters.
- Click on the according function "**Symbol name**" or "**Component value**".
- **Attention**: Enter the names only in capital letters (!COMPONENT, !VALUE)
- dont enter values like 4K7...



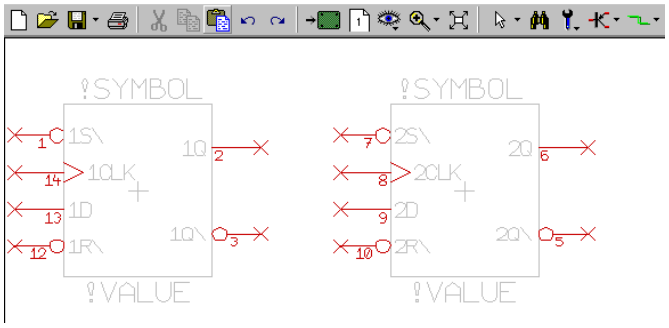
9. Place now a handle to each symbol. To do this, please mark the according symbol (rectangle, pins and names must be selected completely) with **M1H** in "**pointer-mode**".



Please Note: The power-supply connections (Pin #11 = VCC and Pin # 4 = GND) will be drawn and specified seperately, so do **not** mark them now!

10. Place the cursor on the spot where the handle should appear (for example in the center of the symbol). See the picture.

11. Press the **[Y]** key.



12. Create the power-supply symbol now as described in .

13. Click with **M11** on the according handle and do the following modifications:

□□□**symbol 1**□**symbol 2**□**power supply symbol**
symbolnumber□□ 1□□ 2□□ □3
suffix□□□ a□□ b□□ □p
insertion mode□□ next□□ next□□ on demand

14. Mark the component (two symbols with power-supply symbol) completely with the mouse.

15. Press the **[X]** for more details. Symbol with multiple gates.

4.3 Change an existing schematic symbol

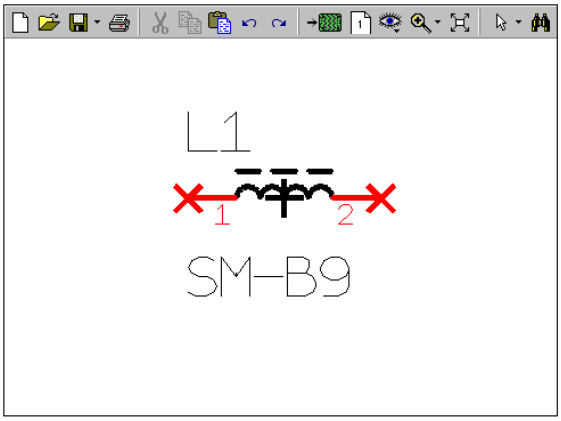
With TARGET it is possible to change all existing components and export them to a library as new components.

Example: You want to change the existing schematic symbol for the smd inductivity SM-B9 from the library L.BTL into a magnetic shielded smd inductivity SMs35.

SM-B9 **SMs35**

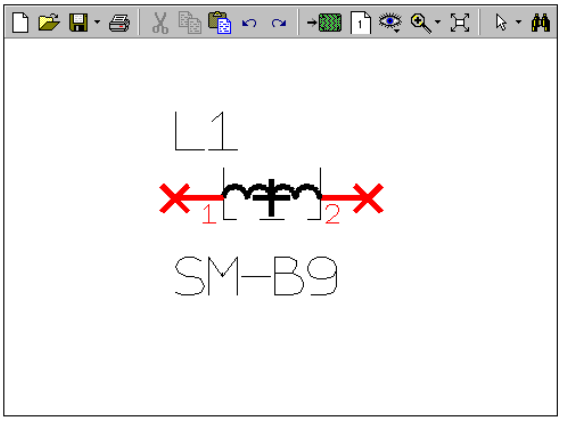
1. Open a new project with an empty schematic page.

2. Import component SM-B9 from the L.BTL library into the schematic page.



3. For better editing, please select the format filling scale factor with the **[F7]** key.

4. Now do all the necessary changes with the help of the drawing functions. Place cursor in pointer mode on an element, press [s] for "select" or **M11**(element is blinking) now press [del]. You can of course add new pins, add text or do whatever changes that need to be done.



5. Mark the new symbol (drawings, pins and names must be **marked completely**!), by dragging **M1H** in "pointer-mode" a catch window over the component.

6. Place the cursor on the spot where the handle should appear (it is recommended to place the cursor on the former location of the handle). **Attention:** It is not necessary to delete the old handle, because TARGET overwrites it automatically.

7. Press the **[Y]** key**.**

8. Press the **[X]** key to export the component.

9. Enter all necessary information in the "Export Component" dialog. Note that it is important to choose the right package for for your component.

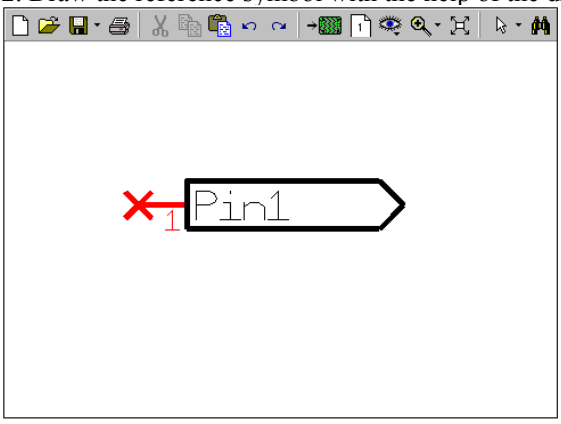
Please Note: Labeling of existing components: Labeling of an existing component from a library with !COMPONENT, !VALUE or !SYMBOL is not necessary, the existing names can be left.(Changes and a new labeling are of course possible, please see also). Change existing schematic symbol.

4.4 Create a reference symbol

Connections over several schematic pages and not visible signal islands of the same signal are connected through reference symbols.

Please pay attention to the following description while creating such symbols:

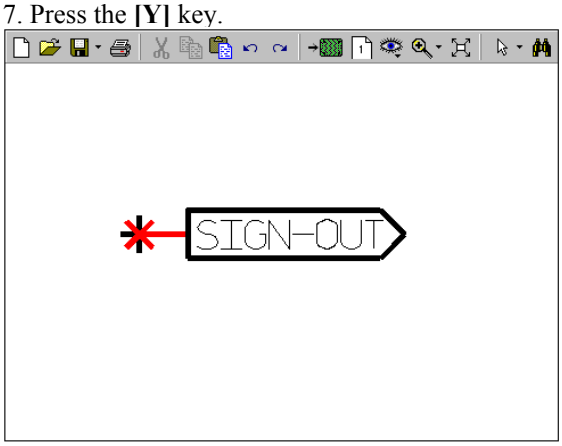
1. Open a new project with an empty schematic page.
2. Draw the reference symbol with the help of the drawing tools from TARGET (lines, pins....).



3. Click with **M11** on the pin name. (the "**Change Texts**" dialog appears).

4. Enter the pin name in the first field (text), for example **Sig-Out**.

6. Set the cursor on the spot where the handle should appear (for example on the pin).
7. Press the **[Y]** key.



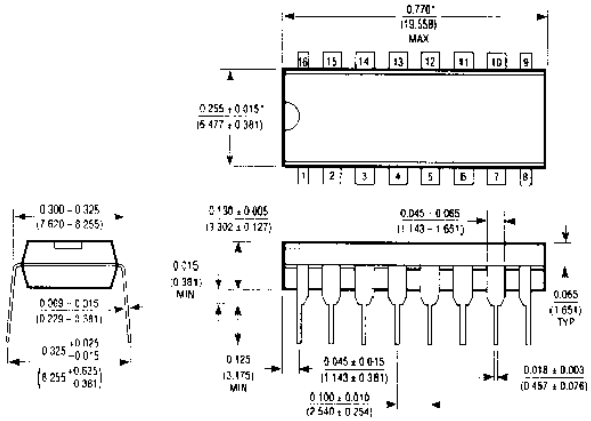
* Dont show pin numbers

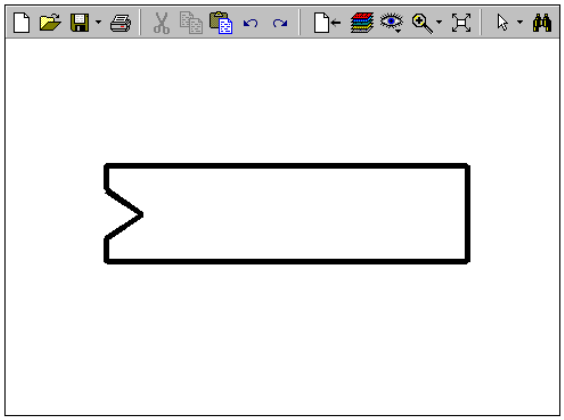
* Clickl here on **reference pin**. Click with **[SHIFT] + M11** (or **M11**) on the pin. The "**Change Pins**" dialog appears as shown above.
9. Click twice on the **Pin number** box (the hook disappears).
10. Click on the "**Is reference pin**" box. The hook is shown and the function is activatet. (This should not be done before you use the **[Y]** key. Otherwise a message "**Pin is already connected with a signal**" appears).
11. Select **OK** (Return key).
12. Mark again the reference symbol (drawing, connection) with the mouse.
13. Press the **[X]** key. The "Export Component" dialog appears.
14. Select **Reference Symbols** from the list (*.REF3001).
15. Enter a name for the reference symbol.

4.5 Create a package

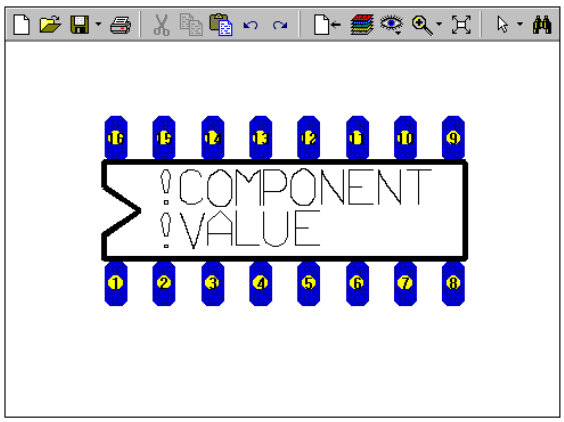
You can esaily create missing packages yourself. Select a grid where the component can be draw the easiest, for example 0,635 mm = 1/40 inch or 0,5 mm for metric components (for smd components). Create Packages.

Example: The package DIL16 should be created from a given datasheet.

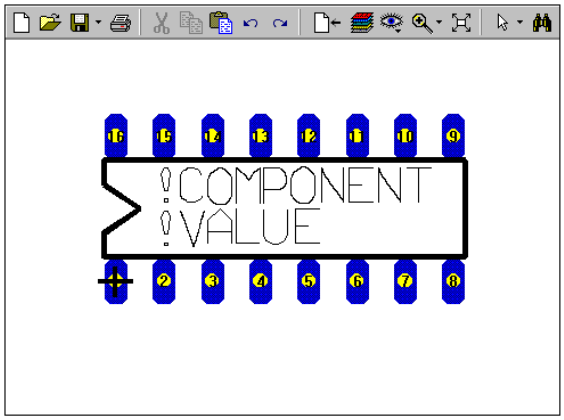




* Draw Package with lines

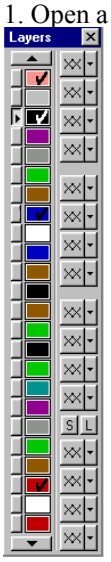


* Place the solder pads and texts



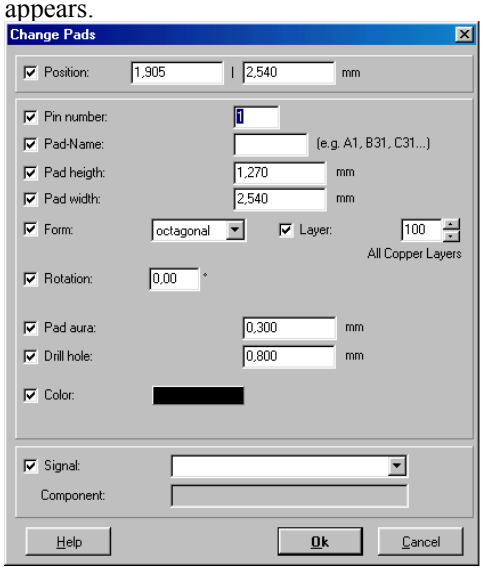
* Place the cursor on a spot and press the **[X]** key for export.

1. Open a new project "PCB without schematic" with an emty PCB page. Select a suitable grid size. Draw upon layer 21 (black).



For making this tool visible, select within the "Layers" tool. Click the button in front of layer 21. See current layer you are working on shown upon main toolbar.

2. Draw the package with lines **[Ctrl]+[2]**
3. Place all necessary solderpads in the right distance to the package (according the datasheet) with the **[1]** key. Select with **[O]** the kind of solder pad before you place the first solder pad in the "Place solder pad mode" to the component. Click the pad with M1 and rotate it with **[t]** if necessary.
4. To change all existing solder pads, catch them with **M1H** in "pointer mode". Press the **[E]** key, the dialog "Edit The Selected Elements" appears. Click "Individual dialogs for each element kind" and then click "pads" and OK. The "Change Pads" dialog appears.



5. Enter the parameters for the pad heigth, pad width, pad form and the drillhole for all 16 solder pads.
6. Select **OK** (Return key).
7. Name the package with !COMPONENT and !VALUE as described in , section 11.
8. After you created the package for the component correctly, catch it completely (drawing, solder pads and texts) with the mouse **M1H**.
9. Place the cursor on the spot where the handle should appear (on solder pad #1 for example).
10. Press the **[X]** key. (the step to collect the package with **[Y]** as shown in the schematic is not necessary).
11. The "Export Package" dialog appears.
12. Select the according package library.
13. Enter a name for the package.
14. Press **OK** (Return key).

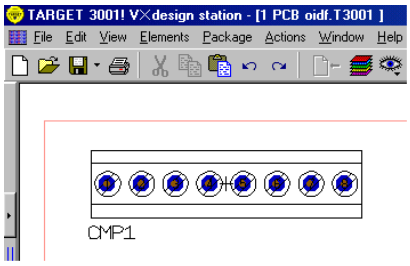
4.6 Change an existing package symbol

With TARGET, you can of course change all package symbols seperately and export them again to a library as a new component.

Example **STECKVER.GHS** into a 10-pin terminal block.

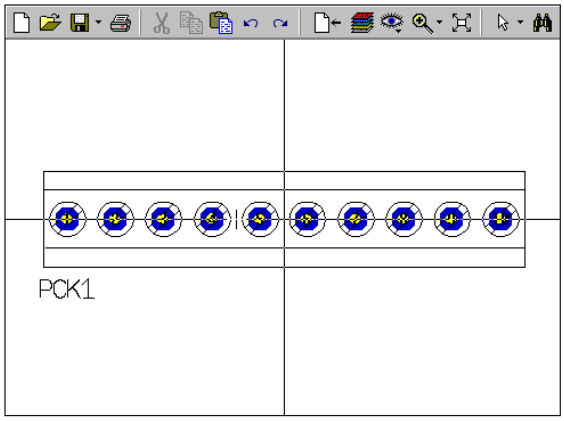
1. Open a new project as "**PCB without schematic**" with an empty PCB page.

2. Import KLEMME-8 from the Connectors_pck.3001 library into the PCB.



3. For a more convenient editing, please choose the format filling scaling factor with **[F7]**.

4. Please do all necessary changes at the component with the help of the drawing functions. You may use the drawing functions or the clipboard functions , and . When copying solder pads, highlight them afterwards by pressing [S] and set (by [E]) the correct sequence of pad numbers.



* crosshair = spot of the old handle
* Place the cursor on the spot, where you later want to have to new handle and press the **[X]** key for export.

5. Mark the new package symbol (drawings, solder pads and names must be selected **completely**) with the mouse.

6. Place the cursor on the spot, where the handle should appear. **Attention:** It is not necessary to delete the old handle, because TARGET overwrites it automatically.

7. Press the **[X]** key.

8. The "Export Package" dialog appears.

9. Select **OK** (Return key).

Chapter 5: Draw and edit signals

An example for signals are signal traces in the schematic and traces on a PCB. In this section you will learn how to use signal traces in a schematic, busses, traces on a PCB and air wires (ratsnest). Helpful tips complete this chapter.

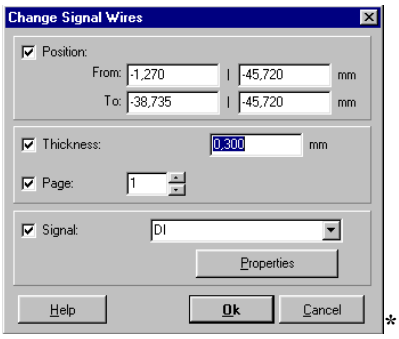
Contents:

5.1 Draw signals in the schematic
5.2 Edit a signal
5.3 Place a bus
5.4 Place tracks
5.5 Draw air wires (ratsnest)

5.1 Draw signals in the schematic

Place wire.

Connections inside a schematic are made of signal traces. Press the **[2]** key in your schematic or click on the tool shown above to start a signal. In case you hit another signal when starting, the signal name of this other signal will be automatically taken over. This means that you will continue this signal (for example: "GND"). When you hit a pin which is not already assigned to a signal, the pin name of that signal will be used as a name for that signal (for example: "A1"). In case you didn`t hit a pin to start the signal, a virtual signal name will be assigned (for example:"SIG$17"). The signal name is displayed in the status line. During the placement of a signal, its attributes can be edited with **M11** or **[O]**.



* **Thickness:** Here you enter the thickness of all signal segments placed in the future.
* **Page:** This shows the current schematic page.

You can set the preferred bending mode while you connect signals by clicking **M2**. After clicking of **M2**, you must move the mouse first to activate the changes. The following tools allow a presetting of a preferred mode. You can use the **[SPACE]**- key to switch between related modes (for example: switching between the diagonally first- or orthogonally first mode).

Draw Signal (diagonally first)

Draw Signal (orthogonally first)

Draw Signal (vertically first)

Draw Signal (horizontally first)

Draw Signal (directly)

If you hit another signal at the end of the signal trace, TARGET will ask if you want to connect both signals. If yes, TARGET asks for a name for the resulting signal. In case that you have hitted the signal segment in its center location, the signal will be divided into two segments and a junction will be created automatically. You can also place a junction at any time in the "**signal mode**" when you press the **[.]** key. You can leave the mode "**Place Wire**" by pressing the **[ESC]** key or by clicking **M12**.

If you get close with your cursor to a pin, than you can place signal directly with **M1**. TARGET helps you to hit a pin exactly with a **pin-catch-box**. TARGET automatically connects the signal with to the according pin and the red cross at the pin dissappears, indicating that a connection has been successfully established. A placed signal can be edited by moving its segments or by the moving of its bending points. The **pin-catch** feature can be disabled by pressing the **[Ctrl]**- key.

The following rules are valid when placing signals:

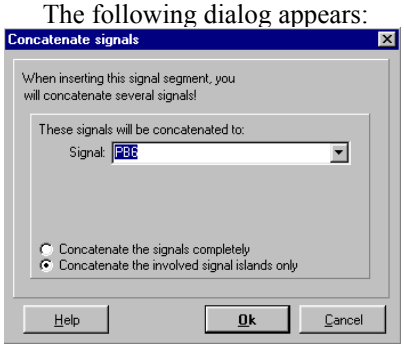
- **Same signal name = same signal!** Pins who are not visible connected can belong to the same signal. Check by highlighting:

  Click in "pointer mode" on this toolbox to select the hitted signal segment completely. Afterwards, select the signal segment with **M1.** The complete signal including all pins will be highlighted. Also, you can use the search function to highlight a signal. Please see for more information..

- If a connection is made successfully, TARGET indicates it with a "beep" and the red crosses at the pins dissappear. If not, the connection has not been made successfully. In this case, delete the last signal segment with **[BACKSPACE]** and zoom in with **M2**. Now place the signal again. The pin-catch-function will be a valuable help to establish a succesfull connection. While bending signal traces or moving signal segments, connections can also be established or deleted. Direct overlaying pins are not connected. They must be connected at least with a small green signal segment. The following tools are responsible for the marking with **M1**: Selects the hitted signal completely Selects the hitted signal island completely Selects only the hitted signal segment

- Signals islands of the same signal which are not connected visually should show there membership through reference symbols. The power-supply connections are a good example. Instead of connecting them to all to the relevant pins, reference symbols are used to show there membership (GND- and Vcc- symbols). Also, connections of signals over several schematic pages can be made this way. To start a connection like that, select the according reference symbol with the **[R]** key.The visible pin name must equal the signal name. You might have to edit the properties of the reference symbol after its import (click with **M11** direct on the name text. Attention: A change of the symbol name by clicking with **M11** on the handle has no effect). For reference symbols whose name starts with an (!), TARGET will ask for a signal name of choice.

- When you try to connect two signals as one , TARGET asks if you would like to do that and asks for a name of this new signal. The following dialog appears:



- Do delete a signal completely, please select a signal segment with **M1** (please first set: "**Selects the Signal Completely**" in " **pointer mode**") , and delete it with **[DELETE]**

- If you want to split a signal, please delete the connected segments with the **[W]** and **[Delete]** keys . Select a segment of the new created signal island with **M11** and assign it a new signal.

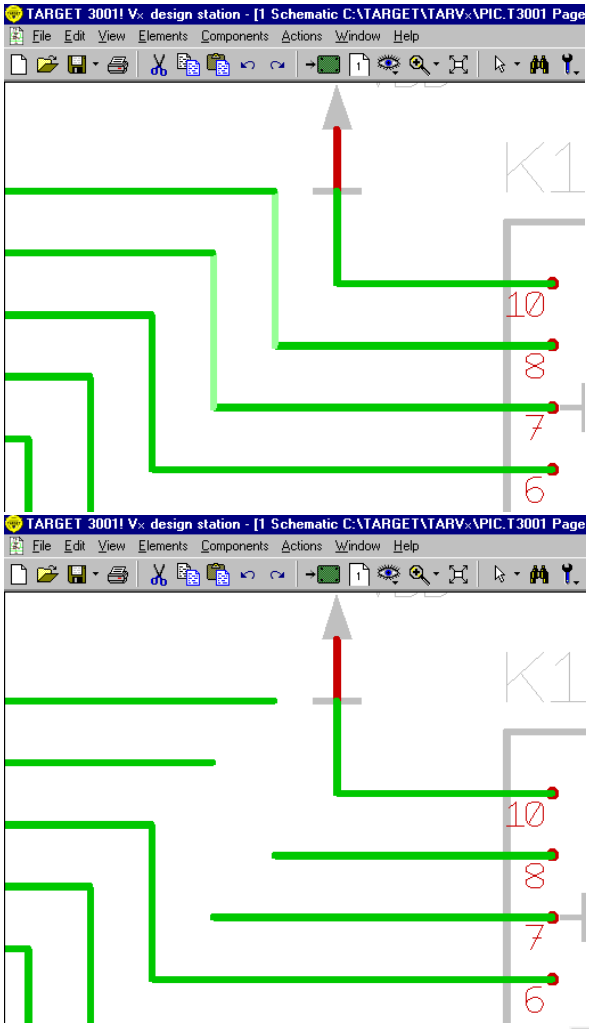
5.2 Edit a signal

Contents:

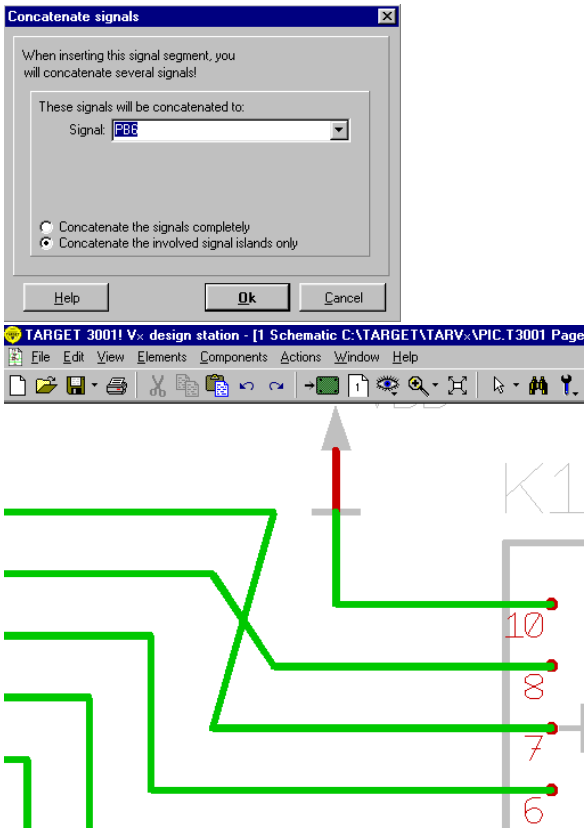
5.2.1 Splitting and reconnection of signals
5.2.2 Rename a signal
5.2.3 Delete a signal
5.2.4 Distribute a signal over several schematic pages

5.2.1 Splitting and reconnection of signals

Example: The signals PB6 and PB7 from the example project PIC.T3001 should be splitted and exchanged.



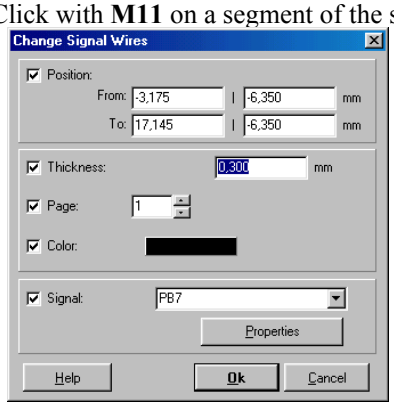
- Place the cursor at the marked spots (see above). Delete with **M1** and **[DELETE]** the two signal segments.
- Connect the created endings over cross. The dialog "Connect Signals" appears:
- Activate the button: "Concatenate the involved signals only "
- Select "PB6" in the combo-box (when connecting PB7 with PB6)
- Press the **OK**-key (Return key).



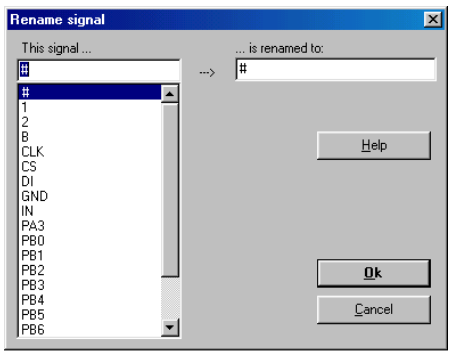
Signals, splitting and reconnection of

5.2.2 Rename a signal

There are two methods to rename a signal:

- A click with **M11** on a signal segment changes the **complete signal island**: Click with **M11** on a segment of the signal you want to change (for example: signals PB7). The dialog "**Change Tracks**" appears: 

Enter a new signal name, for example **TEST**. Select **OK** (Return key). **What happened?** The complete signal island, which has been changed, has been assigned to a new signal name. Hereby, the signal names of the pins, the junctions and the tracks have been assigned to a new name (TEST). The change will only be executed on the selected signal island, even if the original signal inherited several signal islands.
- Rename signals **completely**: With this funnction, a complete signal (not only a signal island) can be renamed. Even if the track has already been placed, it will be renamed. Select "Renname Signal" in the "Edit" menu, the **[V]** key or the according tool.

- 

Select the signal to be replaced in the left side of the dialog. Enter a signal name in the right entry field.

Attention: Modifications on signals will only be transferred to the solder pads in the PCB view. Eventually existing tracks will not be changed or deleted in the background. By checking the project with the "**Check Project**" command in the "File" menu, failures as short circuits will be shown. It is not possible to rename signal track segments, junctions and pins within a signal island or within a complete signal. In general, the according **signal island will always be changed completely.** Signal, rename.

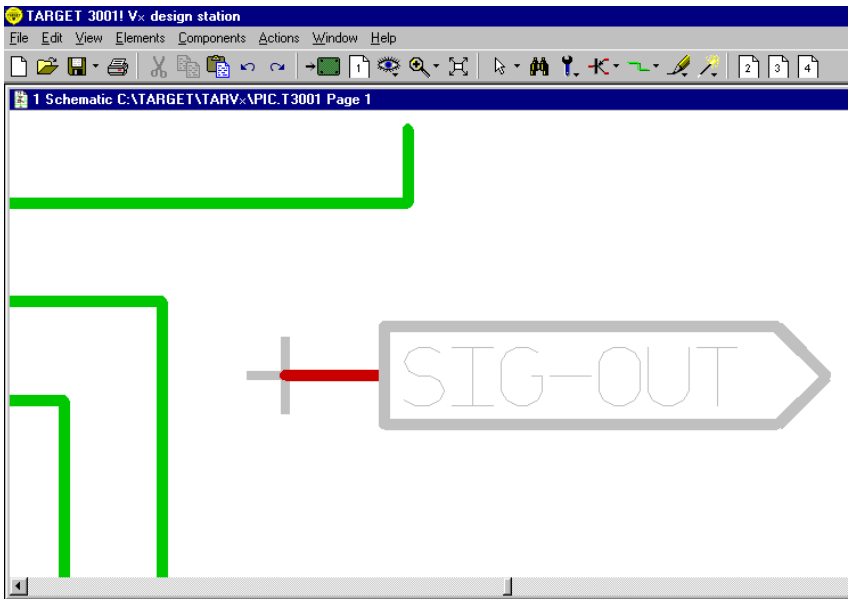
5.2.3 Delete a signal

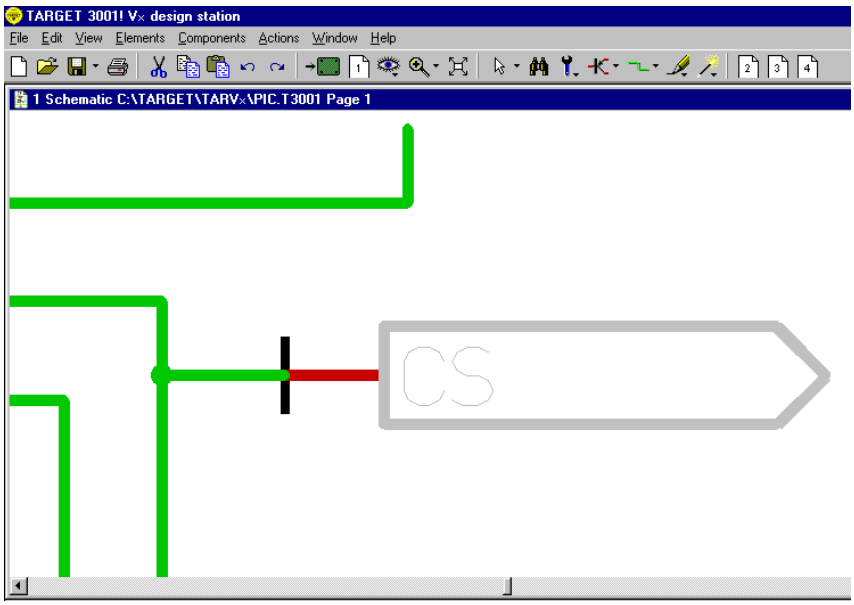
If you delete a signal in the schematic view **completely**
view also. If you delete only **a part of a signal**, no tracks in the PCB view will be deleted. The **checking** of the project finds out these resulting errors and displays them. Signal, delete.

5.2.4 Distribute a signal over several schematic pages

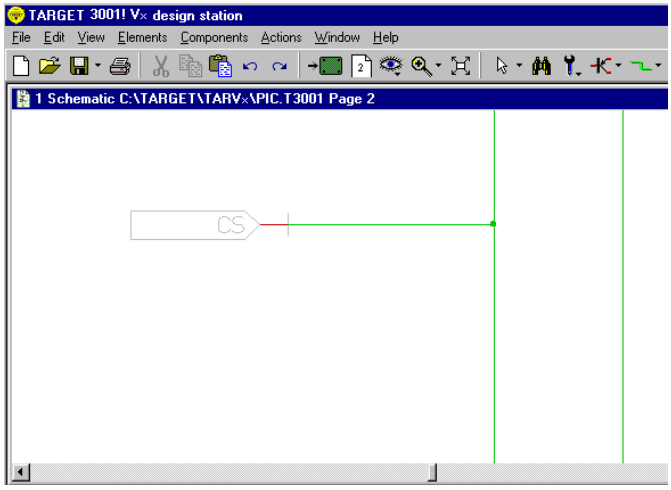
You can seperate your circuit over several schematic pages. The following example shows the logical connection between the first- and the second schematic page:

- By pressing the **[R]** key, import the reference symbol "SIGN-OUT" to schematic page 1. Which reference symbol you choose is only a question of personal taste.
- Connect now the signal you want to have on schaematic page 2 on schematic page 1 with the reference symbol (the name of the reference symbol changes automatically into the name of the signal (for example: CS)).





- Change to schematic page two with this tool:

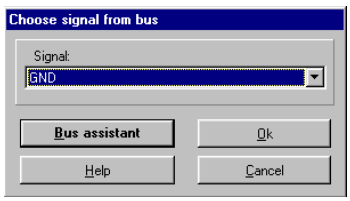


- Import the reference symbol (SIGN-IN) with the **[R]** key. You can choose any reference symbol.
- Click with **M11** on the text "SIGN-IN". The dialog "Change Text" appears.
- Replace the text "SIGN-IN" with the word (Signal) "CS" (A change of the symbol name with **M11** on the handle has no influence).
- Select OK (Return key).
- To place the signal, press the **[2]** key directly on the pin. The following texts appears in the status line:

Signal, distribute over several schematic pages

5.3 Place a bus

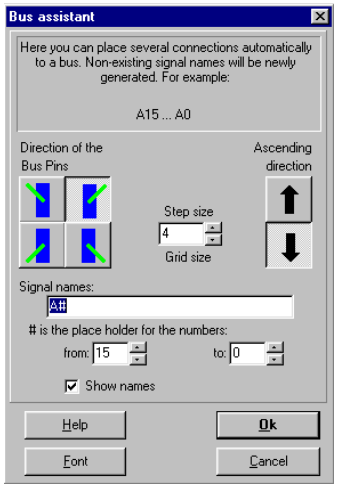
You can use signal busses in the schematic for simplification purposes. You can start a bus with the **[B]** key or by clicking on the tool. A bus **contains in general all existing signals**. You can draw **several different busses** and distinguish them graphically to simplify a schematic (databus, addressbus, controlbus etc...), however for TARGET's logic, all of them are identical. To connect a new signal to the bus which is not contained in the schematic, you have to **start the signal wire from a pin** (for example A0). Place now the signal exactly on to the bus. In case you want to connect an existing signal **from the bus to a pin**, in this case you have to start from the bus. When you start with **[2]** orr **M1**
below). When you connect to a bus, it is recommended to enter the proper signal name.



Please select in the combo box a signal name or start the **bus-assistant**.

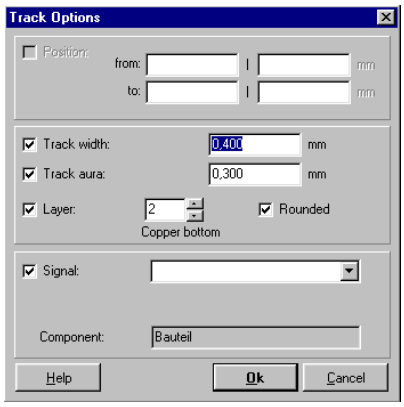
When you start from the bus, TARGET operates directly with the 45°-bend mode. You can of course with **M2** (+ moving the mouse) select any other preferred bend mode.

Bus-Assistant: Several signal traces can be connected automatically to a bus with the help of the bus-assistant. Enter the direction of the bus connections, the running direction, signal names and their text fonts.



5.4 Place tracks

A track represents a connection between the pins of components according to the signals outlined in the schematic. These tracks behave similar to the signals in the schematic. Always try to hit the center of the solder pad when you place a track from one pin to another, so that TARGET can accept it as a new connection. Press either **[2]** in PCB mode or click on this tool to start a new track.When you start a new track from a pin or solder pad who already belongs to a signal, which means that it already has a signal name, this new track will take the existing signal name. When you hit a solder pad which is not assigned to a signal name, the name of the pin will be used instead for assigning a new signal name. In case you do not hit any pin or track, a virtual signal name will be generated. The actual signal name can be seen in the status line. While you bend a track, you can edit its attributes with **M11** or by pressing the **[O]** for options key.



- **Track width:** Please enter the width of the tracks you want to place.
- **Track aura:** The tracks you will place will be added into the according deletion layer. The size of the aura will be added to each

of the tracks size in this layer and be used to calculate the distance to ground planes.

- **Rounded** means that the ends of a track will be rounded.
- **Layer** means the location (layer) on the PCB, on which the tracks will be placed. Usually, copper layers are used to place tracks.

When you create a new project and tell TARGET that you want to work in schematic mode, it is not possible to make any new connections in the PCB mode. When you are in the "**PCB without schematic mode**" and hit with the end of a track another track with a different signal name, TARGET asks if you want to connect both signals. If yes, TARGET asks for a name of this signal. When you hit this track segment in the center, it will be splitted into two segments. When you want to change the copper layer during the placement of a track, please press the **[.]** key. At this spot, a **through hole** will be placed and the actual copper layer will be changed. Have you been working on the copper layer top, TARGET will change to copper layer bottom or vice versa. Pressing the **[:]** key will change the copper layer without placing a through hole.

You can set the route layer also in the toolbar by clicking on one of the following tools: . Also you can place on the layer tool bar with **M1** a small arrow in front of the according layer to indicate a route layer: , see for more information. These two possibilities work also during the placement of tracks.

During the placement of tracks it is possible to insert a wire bridge. Press the **[B]** key (stands for bridge). The last track segment will be placed and the following will be a wire bridge. After that you can continue with the placement of tracks.

When you are close to a solder pad with the cursor, it is possible to use **M1** to place the signal directly. TARGET connects automatically the signal with the according solder pad. The **solder pad-catch-function** helps you with a small **catch-box** to exactly catch a solder pad. This function can be disabled with the **[SHIFT]** key.

A placed track can always be edited by moving of its track segments or by bending it.

5.5 Draw air wires (ratsnest)

With the function "**Draw Air Wires**", logical connections between individual package- components can be made in the "**PCB without schematic**" view. The connections can be drawn pin-to-pin crossover the page. Establishing logical connections using this method is only recommended when the final routing is done from the "Autorouter". Only this tool can route the PCB based on a ratsnest in a short time.

Placed air wires cannot be deleted directly. You can only disconnect individual solder pads from the signal.

- Click with **M11** on a solder pad of a connected signal.
- Click with **M1** in the dialog "Change Solder pads" on the field "**Signal**" and delete by pressing of the **[DELETE]** key the signal name.
- Select **OK**.

Is the air wire visible after you disconnected the signal, than do a "**Reorganization**" first (see **Chapter** 3.1.11 for more details). Then select "**Compute air wires**" in the "**Actions**" menu.

Chapter 6: Automatic functions

Many automatic functions are available to help you with your daily work with TARGET3001! and to significantly reduce the development time of your project. The realtime data integration makes a symbol- and pin exchange a very easy task. You can place package symbols simultaneously to the placement of schematic symbols with the "**Autoplacer**". The "**Ripup & Retry Autorouter**" routes your PCB gridless. The unique "**Groundplane Assistant**

Contents:

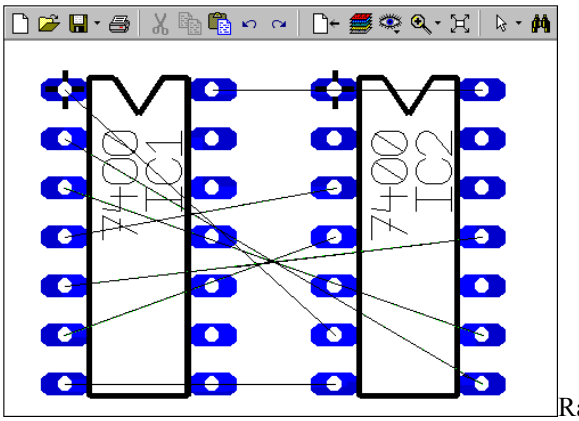
6.1 Symbol-, Gate-Exchange
6.2 Pin exchange
6.3 Ripup & Retry Autorouter
6.4 Contour router
6.5 Autoplacer
6.6 Electrical-Rule-Check, Design-Rule-Check
6.7 Automatic Groundplane-Assistant

6.1 Symbol-, Gate-Exchange

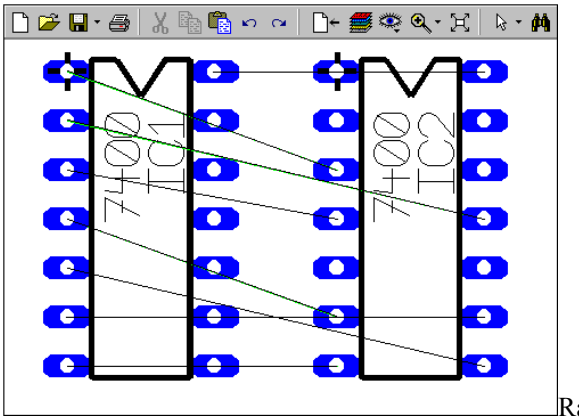
To exchange two equal symbols to allow a simplified routing, select the tool. Afterwards,click on both symbols from the same component, that you want to exchange. If the symbols are not identically or the exchange number mismatches, the exchange procedure will not be executed.

The names and pin numbers in the schematic view will be exchanged, therefore the pin name must match. Already connected tracks on the PCB will be moved to the new pins. **Attention**: **Short circuits** can be created from that procedure!

The result of the symbol exchange and the density of the layout is shown with the display of air wires (ratsnest) between the package components. After the symbol-exchange, the ratsnest is updated immeaditely. Symbol Exchange, Gate Exchange



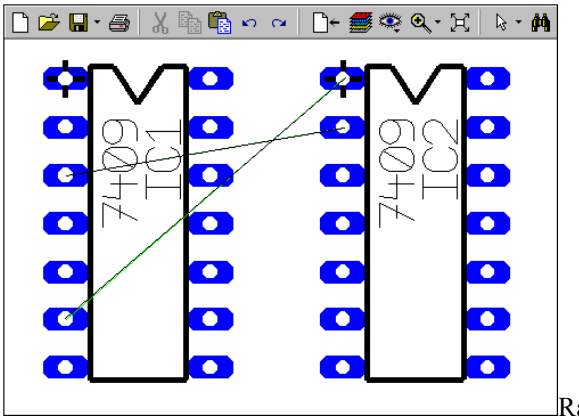
Ratsnest before the gate-exchange



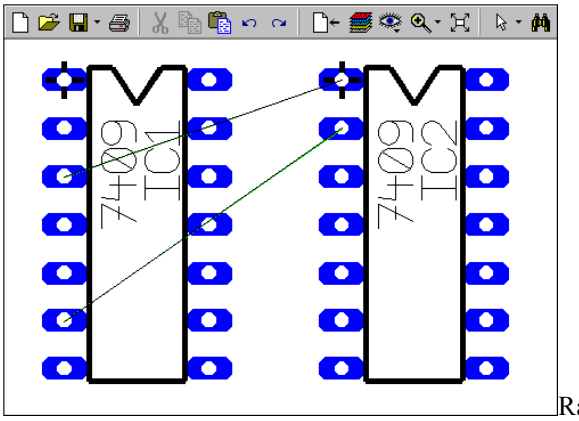
Ratsnest after the gate-exchange

6.2 Pin exchange

To exchange two equal pins of a symbol, select the tool in the schematic view, or select the tool in the PCB view. Afterwards, click on the first and the second pin of the same symbol, that you want to exchange. If the exchange number of the two pins dont match, TARGET will notify you and the exchange will not be executed. If an exchange is possible, the names and pin numbers of the pins in the schematic view will be exchanged. Already placed tracks in the PCB view will be moved to the other solder pads. **Attention: Short circuits** can be created from this procedure!
The result of the pin exchange and the density of the layout is shown on the display with a display of air wires (ratsnest).After the pin-exchange, the ratsnest is updated immeaditely. Pin exchange.



Ratsnest before the pin exchange



Ratsnest after the pin exchange

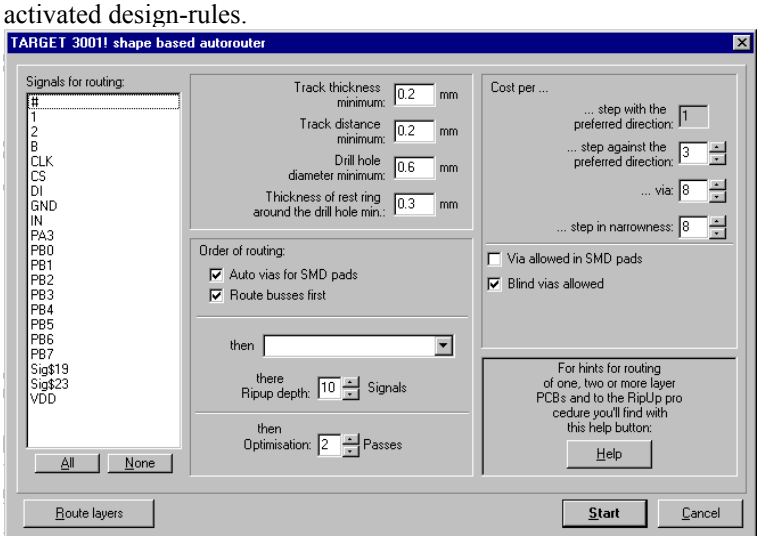
6.3 Ripup & Retry Autorouter

Contents:

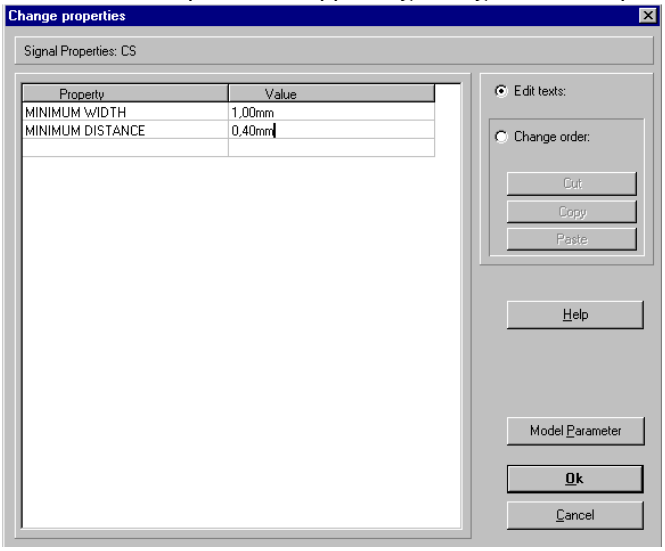
6.3.1 Autorouter with schematic
6.3.2 Autorouter without schematic

6.3.1 Autorouter with schematic

This command starts the Ripup & Retry-Autorouter. This router can route on one or more copper layers at the same time. It can route smd-PCB´s, multilayer PCB´s, double-sided and single sided PCB´s. The Ripup & Retry Autorouter uses a heuristic routing method. This routing method uses knowledge-based algorithms and can be configured individually. He follows automatically all activated design-rules.



All signals are entered on the left side. If you want to route only a single signal, select it from the list. All design-rules are entered in the center field. The signal-track-width refers to all signals. If you want to route e.g. CS signals with a higher track-width, than the following settings must be made in the properties list for the signal wire in the schematic !! Doubleclick in schematic view the desired wire and pres in the appearing dialog button "Properties" :



All GND-signals will now be routet with a minimum width of 1mm and a minimum distance of 0,4 mm The routing is done in the following order:

Auto Vias for SMD Pads: Function inactive.

Route busses first: Parallel signale will be routet (Attention: This function does not necessarily route the signals connected to a bus. In general, all parallel signals will be collected.

Short/long signals, random selection: A predefined, heuristic method is used to place the tracks.

Ripup-depth: The Ripup & Retry Autorouter places tracks as long as the first signal cannot be placed directly anymore. After that, TARGET looks for other already placed tracks (heuristically), to take them back (Ripup). The Autorouter tries no to place this track again (Retry). This action is successful, if all tracks taken back can be placed again. In the opposite case, TARGET reconstructs the stage before the track has been taken back and places the unready track as far as possible. TRhe entry Ripup-depth defines, how many other signals can be taken back for a not placed track. This value is in a range between 0 - 20. 0 means, that the routet does not start any Ripup-actions. High values might cause a long calculation time.

Optimization: The optimization-run will be started immeaditely after a route-run. The amount of runs can be set from 0 to 15. The Autorouter is straightening hereby the signal tracks and reduces the amount of vias, whereas the order of preferred directions will be lost.

Costs per step: If the router should work on more than one layer, different preferred directions (orthogonal) are valid for the individual layers. The value **against** means the costs of a step against the preferred direction or diagonal. The value **via** means the costs of a via. **Space** means the costs, if there is not much free space on the PCB. For each of them, you can select values between 1 and 15, whereas 1 means **cheap** and 15 means **expensive**.
Allow vias in smd-pads: Activate this option if you want to allow the placement of vias inside smd-pads.
Blind vias allowed: Allows TARGET to place blind vias on top of each other.
Route-layers of the **Shape based Autorouter**: Please define the route-layers for the actual PCB first. For a double-sided PCB, set the preferred direction of all signals on the layer copper top as horizontal and the preferred direction of all signals on the layer copper bottom as vertical. Should a special signal only be placed on one copper layer, please select this signal from the list. The signal appears in brackets after the entry of the copper layer in the list on the left side.

Meaning of the upcounting display during the route run in the **status line**:

1) The number which alone is counted up, is an internal counter without further meanings.

2) For example: **AutoRouting: Signal PB4 7/6/21 (2/5):** Ripup-Retry-Route-run on: PB4 is the **seventh** signal, six signals have already been placed successfully, **21** signals must be placed in total, **two** of **three** signal islands of PB4 are **remaining**.

3) For example: **Optimizing #1: Signal PB4 7/6/21 (2/3)** Optimiztion-run: PB4 is the **seventh** signal, **six** signals have already been placed successfully, **21** signals must be placed in total, **two** of **three** signal islands of PB4 are **remaining**.

Attention:

- On single sided PCB´s, the signal tracks will only be straightened!
- The optimisation-run can only be started together with the route-run.

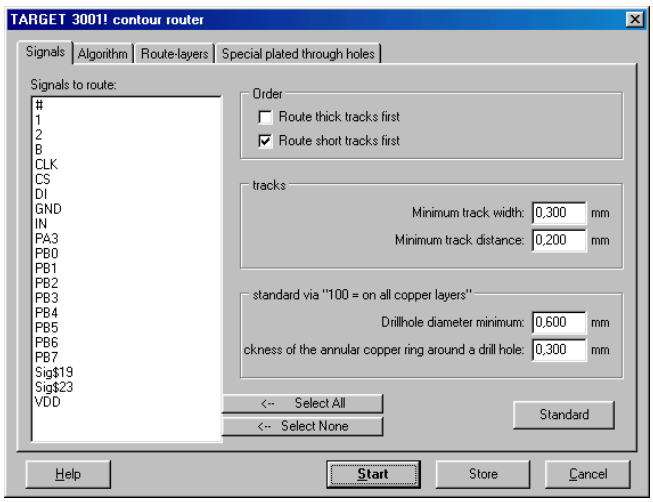
6.3.2 Autorouter without schematic

You would like to create a new PCB without drawing a schematic, and you would prefer to use the Autorouter. Creta first a new project and select **PCB without schematic**. Place all of the packages. Draw all connections as a ratsnest. Please note, that that the solder pads you want to connect, must be hitted exactly in the middle. To check a connection, select a connection with **[2]**. The complete signal including all connected solder pads must be displayed highlighted. If some solder pads remain not highlighted, you must delete the connection and reconnect again. It is really important to hit the solder pads exactly in the middle. If you have drawn all connections, your schematic looks like a wool. Select individual signals to be routet with **[R]** or let all signals be routet at once. Autorouter without schematic.

6.4 Contour router

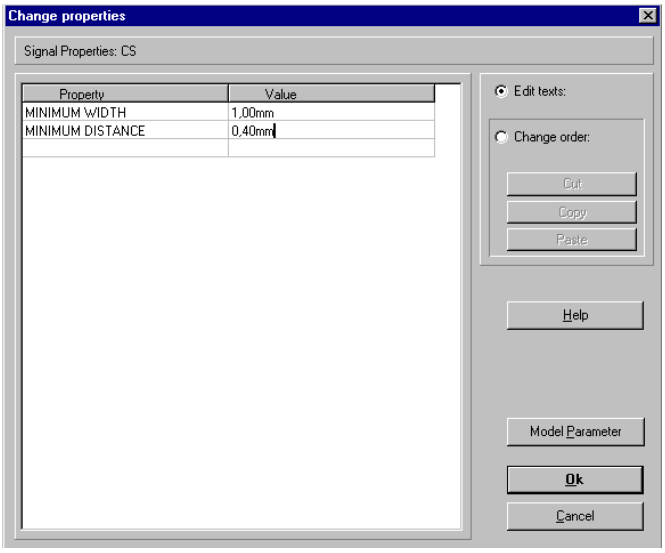
The new contour router works after a new powerful principle. After several preperations begins the router to place the tracks, where as distance- and crossing errors are allowed. The costs of these errors are increasing after several route-runs. At the end, no errors should be left. This logical statement insures that during the route-run of a single track, all other signals are noted. The management of the routing-area internally is completely vector-oriented and therefore independant of the memory from the grid (clearly contour-oriented). The route run can ber stopped by pressing the **[ESC]** key.

The contour router is perfectly suited for larger projects with more subtile structures (for example: smd- or chip design) and layouts with several routing layers.

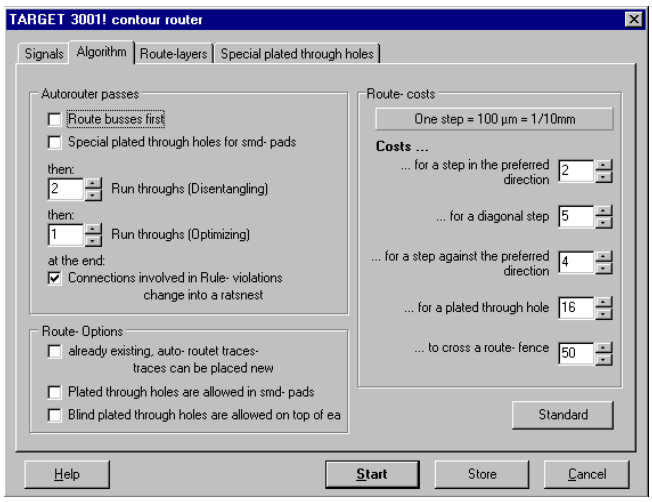


Select the **signals to be routed** on the left side.
You can define the **order** of the signmals to be routed as following:

- **Route large signal tracks first:** Signals with a larger track-width will be routet before signals with a lower track-width.
- **Route short signal tracks first:** Signals with a shorter ratsnest will be routet before signals with a larger ratsnest.
- **Minimum signal track-width:** Please enter the minimum track-width here. The property **Minimum width** of a signal can overwrite this value:

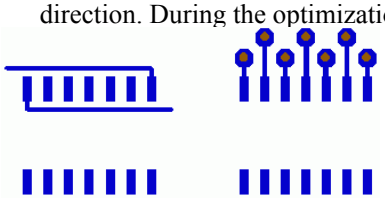


- **Minimum track-distance:** Please enter a value for the minimum width here. The minimum-width property can overwrite this value.
- **Minimum drill-hole diameter:** Please enter a value for the minimum drill-hole diameter of a standard via.
- **Minimum width of the annular restring of a drill-hole:**
  a drill-hole.



Route-Runs:

- **Route busses first:** This route-run is dedicated to bus- and memory structures.
- **Vias for SMD-pads:** Smd-pads located on a preferred direction, will get a via to allow access to layers on an opposite preferred direction. During the optimization stage, only these vias remain which will be used.



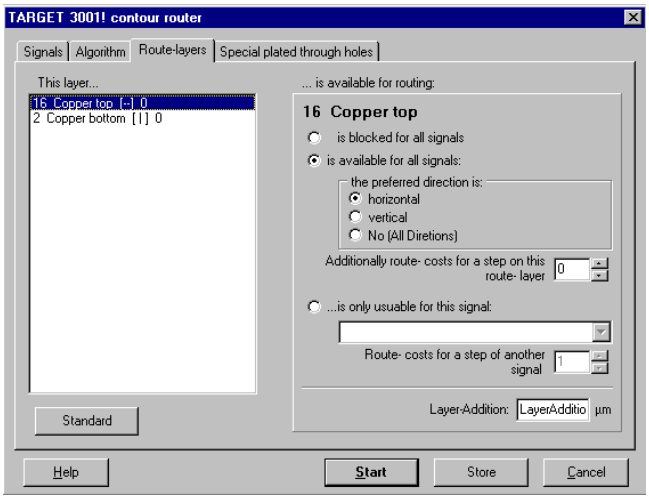
- **Runs (disentangling stage):** During the disentangling stage, all signal tracks will be placed independend of distance violations and crossings. These violations however will get more and more expensive after each run.
- **Runs (optimization stage):** During the optimization stage, no distance violations should appear. The preferred directions are ignored to reduce viases. All signal tracks will be straighend.
- **Change connections involved in rule-violations back to ratsnest:** Signal tracks, who cause distance violations or crossings (short circuits) with other signals, will be cut at the end and displayed as unroutet ratsnest.

Route-Options:

- **Tracks which have already been routet from the Autorouter can be placed again:** Please check, if tracks which have already been placed by the Autorouter can be opened and replaced again. Manually routet tracks remain unchanged.
- **Allow vis in smd-pads:** Activate this check box, if you want to allow viases in smd-pads.
- **Allow blind vias on top of each other:** Several vias of different signals can be placed on top of each other (for example: blind hole; blind via).

Route-Costs:

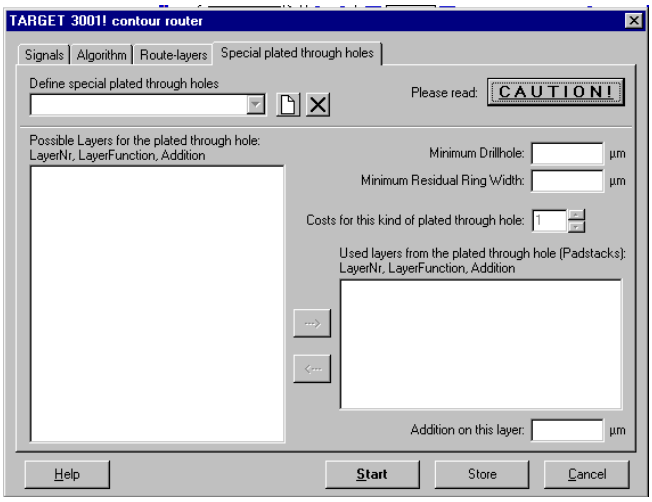
- **... for a step in the preferred direction:** Each layer can have a preferred direction. The value entered here represents the costs for a step in the preferred direction.
- **... for a step diagonal:** These are the costs for a diagonal step. The result is relative to the costs in the preferred direction.
- **... for a step against the preferred direction: These are the costs for a step against the preferred direction:** The result is relative to the costs in the preferred direction.
- **... for a via:** These are the costs for a standard via (=100). The result is relative to the costs in the preferred direction).
- **... to cross a routing-fence:** These costs appear if a signal crosses a routing-fence. The result is relative to the costs in the preferred direction. Routing-fences can be of interest to distinguish between digital- and analog circuit blocks on the PCB or between circuit blocks with different voltage levels. A routing-fence is drawn as a line on the routing-fence layer. A layer with this function must be created.



This layer ...
All layers capable to be routet will be listed here. Please select a layer to be edited here.

Route-layer:

- **... is prohibited for all signals:** This layer is prohibited for routing.
- **... can be used for all signals:** This layer can be used to route signals, but might have a preferred direction (**horizontal, vertikal, none**). **Additional routing-costs for a step on this route-layer:** This is the result of the costs for a step on this layer. With this option, you can assign different costs for the individual layers. The result adds to the costs of the in the preferred direction.
- **... can only be used for one signal:** This layer can only be used for a given signal (without preferred direction)**.** To place another signal on this layer will cause extra costs, as shown in Routing-costs for a step of another signal.
- **Addition on this layer:** Signals on this layer will be assigned to get an additional track width set by this value.



Define special vias (Padstacks): here you can creat or delete a special via as a padstack.

Possible layers for the via: Layers displayed here, will not be used from the via.

Layers used from the via (Padstack): Layers displayed here, will be used from the via.

Minimum drill-hole diameter:

Minimum annular ring: Please set the value for the minimum annular ring for a drill-hole of a via here.

Costs for a via: These are the costs for a via of this kind. The result is relative to the costs in the preferred direction.

Addition on this layer: The output of the pad in the marked layer should be ....larger or smaller (-) as the original pad.

6.5 Autoplacer

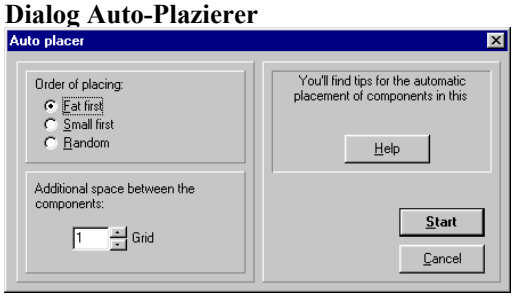
You can let TARGET automatically place the packages of the components. The Autoplacer calculates immeaditely the shortest signal track-path between the packages during the placement. Herefore, you have two possibilities to activate the Autoplacer:

Press the button, after you have finished the schematic and the definition of the PCB outline. After confirmation of the Autoplacer dialog, all packages will be placed according the predefined settings.
The component-packages will be placed now simultaneously to the schematic symbols.

Please Note:

- A PCB-outline must be defined before you start the Autoplacer (see also **Chapter 7.2.1**.).
- The Autoplacer places the packages of these components, which have been predefined as the corresponding schematic symbols. If you want to place a different package, you have to change the entry in the property list of the schematic-symbol.
- Place all connectors and critical packages (for example: large IC´s) manually before you start the Autoplacer.
- Place all blocking capacitors close to the power supply pins of the according IC´s manually,before you start the Autoplacers.

Dialog Auto-Plazierer



* The order of the placement of the components can be set by activating the according checkbox of choice.
* A 0 setting will place all components very close together. A higher value will place all components according to the predefined grid setting.
* At the moment, the Autoplacer can only place components on the component side of the PCB.

6.6 Electrical-Rule-Check, Design-Rule-Check

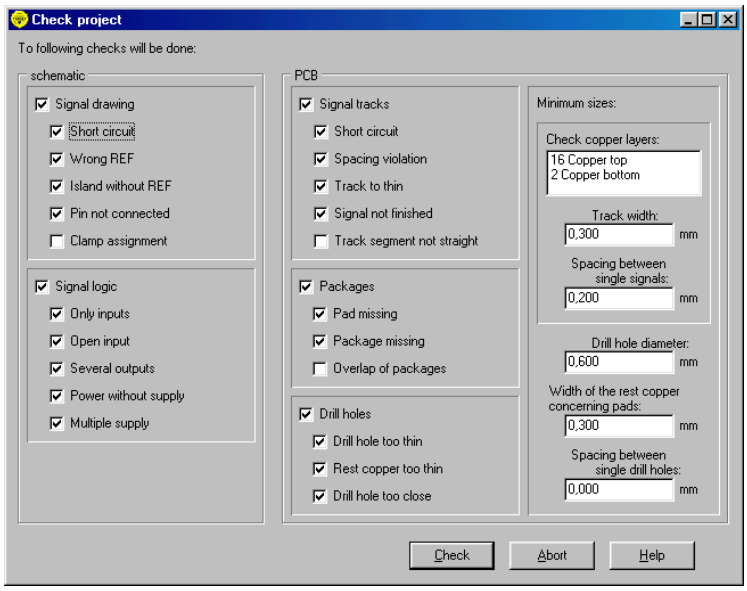
The check-routine will be started automatically after clicking the button. Basically, you can do almost everything in TARGET: components without a package, packages without components, short circuits, open circuits etc... Sometimes these situations are wanted or necessary. For example, if you only want to draw a PCB without having to think about drawing a schematic first. Nevertheless, to have everything proceeding the right way, you can call this function. The first hundred errors are automatically displayed in a popup window. You can also let TARGET output this list in a file called TARGET.ERR into the actual working directory. The following error messages are possible during the execution of the **Electrical-Rule-Check**:

- **Missing package:** The schematic drawing contains a component which requires a package. The package however is missing on the PCB. Please place the missing package on the PCB.
- **Unfinished signal:** The schematic requires connections which are incomplete on the PCB. Either individual solder pads of the signal are not connected or the signal is made of two or more signal islands. For checking, activate the pointer-option Mark signal islands. When you click now with **M1** on a track, you can see exactly where the signal flow stopps.
- **Power without supply:** The power supply connections of components are connected to a signal, but there is no power connected to the PCB. Usually, some pins of a connector are assigned to **Function=Supply Output-error:** Several output pins are connected to a signal at the same time or an output pin and a supply pin.
- **Only inputs:** There is no defined voltage level on a signal, because only inputs are connected (signal is flowing).
- **Open input:** An input is flowing. Input pins of unused symbols must be connected. In case that the pin should really be unconnected, set its function to NC (not connected).
- **Short circuit:** A pin on the PCB is connected to a different pin as required in the schematic. This is not a Design-Rule-Check, short circuits caused by crossing tracks and distance violations, are not recognized here.

- **Pin without a solder pad:**There is a solder pad missing on the PCB for an existing pin in the schematic.Maybe you have deleted a solder pad afterwards from an existing package or you have used the wrong package.
- **Multiple supply:** A signal is connected to several different poer supplies.
- **More signal islands than reference symbols:**
  reference symbols or through a connection to a bus. in this case, at least one signal island is not connected to a reference symbol. Click on each of the signal islands with **M1**. Hereby, the tool Mark signal island completely must be selected. This error could also be named as **double signal name**!

Note: Use these error messages to check if your circuit has any short circuits. If both pins of a resistors are connected to the same signal, a signal island is created. The Autorouter therefore short circuits the two solder pads of the according package symbol.

With the help of the **Design-Rule-Check**, the routet PCB will additionally be checked according the settings in the Design-Rule-Check-dialog. (Please note: This function is only available in the professional version of TARGET3001!l).



- **Design-Rule-Check error messages:** Most of them are spacing violations or tracks that are too thin. TARGET regards the Values entered in the DRC dialog as well as the properties of signals "MinimalWidth" and "MinimumSpacing".
- **Package Overlap:** Two packages are overlapping. If a package shall be excluded from this check (e.g. if the PCB outline itself was imported as a "package", it would have conflicts with all other packages), then this component must get the property "IGNORE_OVERLAP" with the value "YES".

Please set the according parameters to check your project. After pressing the Check button, all Design-Rule-Check-errors and Electrical-Rule-Check-errors will be displayed in an error list and optically highlighted in the PCB. Clicking with **M11** on the according error designator will display a window with a detailed description of the design error. The display of the design errors can be disabled by deactivating of layer # 28. The reorganization command will cause that all error designators will dissappear.

Finally, TARGET shows (for example) the following error list:

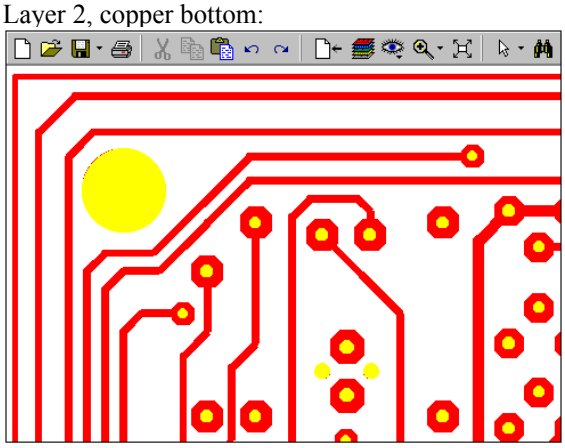
0 Bridges
5 Vias
211 Drill holes
Minimu drill hole diameter: 0,6 mm
Minimum distance between drill holes: 0,3 mm
Minimum annular ring of solder pads: 0,235 mm
Thinnest track width: 0,3 mm
Minimum signal distance: 0,25 mm
PCB size: 160 x 100 mm

6.7 Automatic Groundplane-Assistant

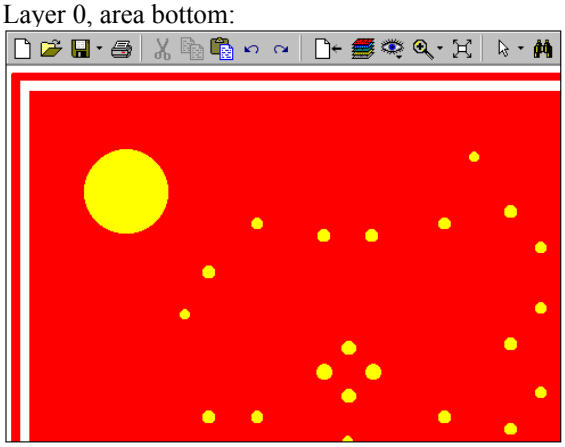
While working with high frequency signals, it is often required to shield them with ground wires or to embedd them in ground planes. In TARGET, such a copper layer which contains a ground plane can be best described if you think of a three layer sandwich. The advantage of a graphical area calculation is that a change of the ground plane area is immeaditely visible on the screen without any time loss.

Example: A groundplane on the layer copper bottom

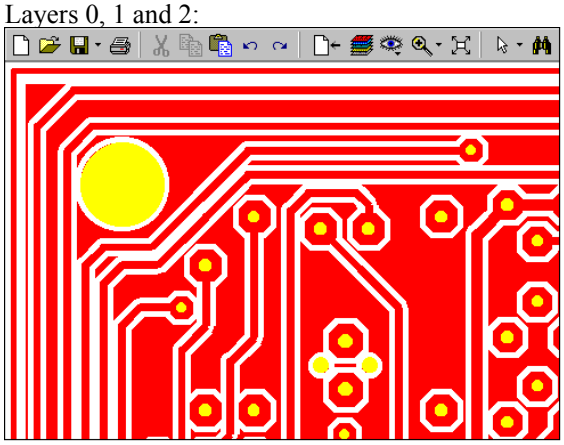
Layer 2, copper bottom:



Layer 0, area bottom:



Layers 0, 1 and 2:



During the routing of a double-sided PCB, the copper layers 2 and 16 are usually used. The groundplane-assistant checks if the corresponding deletion- and area- layers of the copper layers are active. If this is the case, the groundplane-assistant shows the result as in the top window (for example: copper bottom). Also, a second list proposes a signal which can be embedded into the groundplane and therefore does not have any aura (distance between the track and the groundplane). You can set the value of the aura in mm in the bottom field. The aura of each of the elements is displayed as a large shadow in the according deletion layer

(layer #1). You can of course assign any other elements as deletions in this layer (for example: rectangles to limit the size of the ground plane).

Any text on this layer will be displayed inverted, which indicates, that the text is engraved into the copper. Please remeber to activate the according deletion- and area-layers later on when you output the PCB. Groundplanes created this way can not be outputted to all plotters.

Creation of groundplanes of any form:

With the help of the groundplane assistant (Menu/Actions or the magic wand in PCB view)), only groundplanes of rectangle forms are supported. The following example describes how to create a groundplane on the bottom copper layer on a PCB which has the outline of a ring.

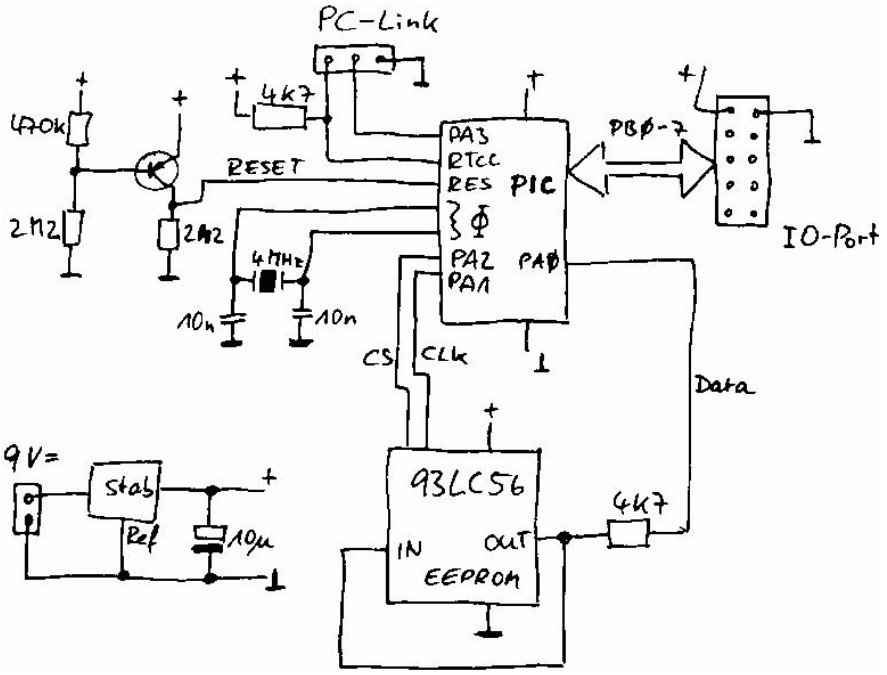
- Draw a filled slice over the whole PCB with the help of the according drawing function on layer #0 (area bottom).
- Draw now all cutouts (inner circle) on layer #1 (deletion bottom) as a filled slice. It is of course possible to integrate texts and triangles into this groundplane.
- Mark the whole PCB (with a large catch window) and press the **[:]** or **[e]** key.
- Click on solder pads, signal tracks and vias as well as individual properties of the marked and aura in the dialog Change of marked elements.
- Set the value of the aura for example to 0,3 mm and press OK.

Chapter 7: Crashcourse

This crashcourse gives a quick introduction to basic tasks in TARGET3001!. Please note that only basic functions are explained in this chapter. Automatic functions and their use are covered in detail in Chapter 6 - Automatic Functions.

7.1 Create a simple schematic

Please read chapter 2 before you start with the following exercise.



The following exercise shows how to create a simple schematic in TARGET and explains the necessary steps to convert the data to

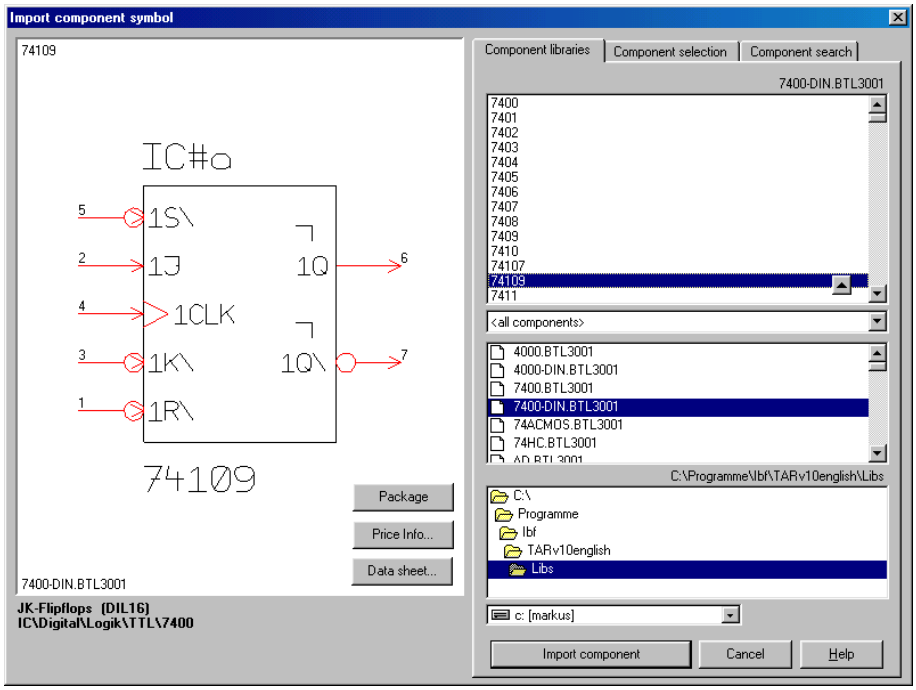
create a PCB :

This example explains how to import symbols and place tracks in the schematic view and shows how to place packages and connect traces in the PCB view. Topics whereas the automatic placement of components (Autoplacer), the mixed-mode simulation and the EMC-analysis are not covered here.

7.1.1 Import Schematic Symbols

First, we must create a new project. After you have started TARGET, select New Project from the File menu. A popup window appears and asks for a file name. Please enter the following name: PICTEST.T3001. TARGET creates now a new file and opens an empty schematic window. A click with **[Shift] + [Strg] + M2**
this schematic page is 1m². It´s origin (0|0) is approximately in the middle of the window. This window can be moved with scrollbars on the right side and the bottom of the window. This window basically acts as a magnifier lens which can be moved over the 1m².

Next, we would like to import a drawing frame DIN A4 horizontal. Click on the button in the menu toolbar or select the Import Symbol command from the Components menu. The following dialog appears:



Below left you will find a short description of the component. Right on the top you will find the information, that frames for schematics are in the file Frame.CMP3001. Therefore, select the according library from the list shown on the right side and then click on the file name FIELDA4HOR in the middle window. Click Ok. The outline of this frame is attached an visible on the cursor. Place the frame with **M1**. Press **[F7]** to display the frame in full size format. The frame must not be adjusted to the 0/0 coordinates. Add now the PIC16C54-DIL18 component from the Microchip.CPM3001 library and palace it in the middle of the drawing area of the window. You can rotate the phantom image of the component with **M2** at the actual preset value. In case you cannot see the phantom image, please move the mouse a little bit.

Place all components according to the following table:

Name:	Type:	Library:
FRAME1	FIELDA4HOR	FRAME.CMP3001
IC1	PIC16C54	CPU.CMP3001
IC2	93LC56	CPU.CMP3001
IC3	LM2936	ANA.CMP.3001 (alias 78LXX)
T1	2N2905	TRANS.CMP3001
Q1	CRYSTAL	EXTRA.CMP3001
C1	ELKO	C.CMP3001
C2, C3	BIPOL	C.CMP3001
R1..R5	R	R.CMP3001
K1	K2X5	CONN.CMP3001
K2	K1X3	CONN.CMP3001
K3	K1X2	CONN.CMP3001

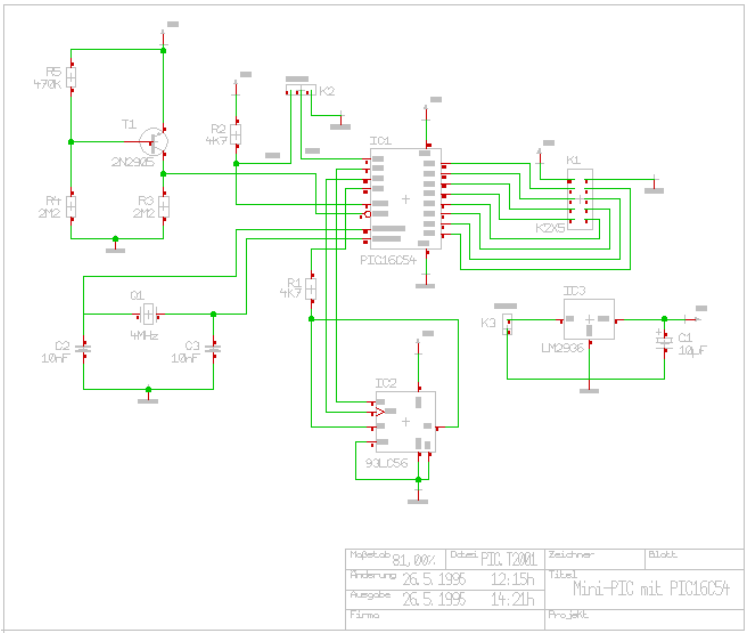
Help, where are the power connections of the gates?
A component in the schematic-view consists of one or more symbols. In the PCB-view, a component consists of one or more symbols and an additional package. The first and all identical symbols of a component are commonly placed with **[Insert]** in the schematic-view. Symbols with special functions as part of the component (for example: power supply...) must be importet or placed with the Import Rest of Components command from the Components menu or with **[Strg]+[R]**.

How do I enter the component value ?
To edit a value of a component afterwards, (for example: Change the value of a resistor from R to 4k7 or change a 7404 gate into 74HCT04), select the handle of the component with **[M11]**
entry.

7.1.2 Connecting the schematic symbols

Connecting components in TARGET is done by placing signal wires. Please press the tool to enter the Place signal wire- mode. You can place now as many signals wires as you need with **M1**
and the connection-cross dissappears. A crosshair is now attached to the cursor. Select with **M2** the required bend modus while placing signal wires. TARGET selects a useful name for the signal (see status line). In case if you do not like the proposed signal name, it can be edited during the placement with **M11** or **[O]**
Place all signal elements with **M1** and leave the mode afterwards with **M12** or **[ESC]**. The last drawn segment can be deleted during placement with **[BACKSPACE]** (Undo).

Your schematic should look similar to the following:



7.2 Create a PCB

If you switch to PCB-view by pressing the **[F3]** key, an empty PCB-square meter is displayed. You should first think about your

- **Which grid settings should be used for the PCB?:**
  Please use different values only in special cases (for example: 0,3175 mm or 1/80 inch). The Ripup-Retry-Autorouter is for example optimized to work best with 1/40 inch, but he works also on all other grid sizes and can connect solder pads which are not on the grid. Within the grid dialog settings, you should activate the cursor snaps on grid option.

- **On which layers should the PCB be displayed?:** TARGET allows you to use all layers from 0 to 99 for the PCB . You should nevertheless only display the really needed layers to improve the refresh rate of the screen. For a standard single-sided PCB, we recommend to use the following layers:

2 for copper bottom,
21 for area top,
23 for the outline drawing,
24 for drillholes and
26 for solder pad numbers

The function of layer #16 should be set to others instead of copper top for single-sided PCBs. For a double-sided PCB, layer #16 should be set to copper top and displayed. For a PCB which is populated on both sides, layer #7 should be set to Position bottom and displayed. The use of the solder mask layers are only useful when you output the PCB. In case you automatically want to create copper areas, the according layers deletion top/deletion bottom as well as the area top/area bottom layers must be activated and displayed. (0 and 1 for layer 2 and 14 and 15 for layer 16). The ceation of automatic copper areas is described in detail in **Chapter 6.7**. The mentioned settings for the layers are the factory settings of TARGET. You can of course change the layer settings to your convenience. You should always be aware of how you want to use the layers before you start the design of the PCB. Also you must inform your PCB manufacturer (who is using the TARGET data) about your special settings of the layers. This might cuse extra costs and can lead to errors.

Which functions should be assigned to indiviual layers?:
sufficient for your design and if you get all necessary functions. Maybe you have to adjust the layer functions according to your requirements. Please note that you should not assign more copper layers as really needed. Therefore, unused copper layers should be set to the others function to prevent that the Autorouter creates a multilayer PCB of your design.

7.2.1 Define the PCB-outline, import and place packages

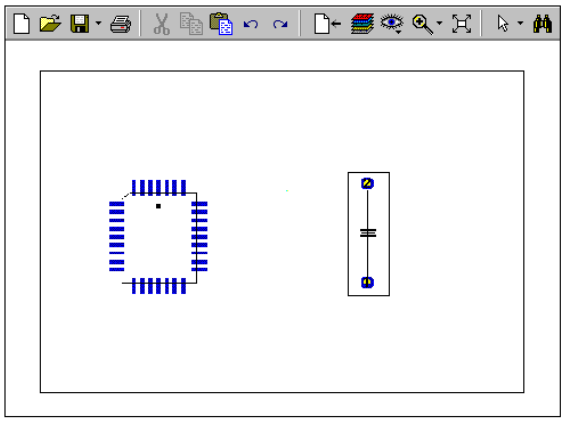
Our next task is to define a PCB area. Included in the component library of TARGET are some predefined PCB´s of different sizes.

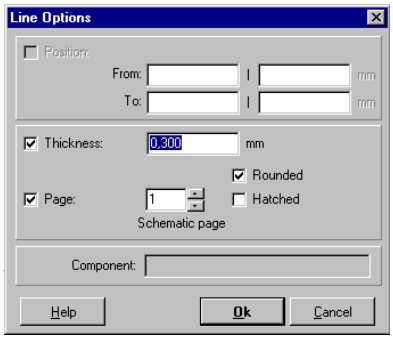
To import a PCB, press the **[INSERT]** key or click on the
because the PCB outline is not a component of the schematic. Select the P50X30 PCB from the library TARGET.CMP3001. Press again **[F7]** to display the empty PCB format filling on the screen and **M1** to place the outline frame of the PCB. Afterwards, press **M12** to leave the Insert-mode. In case you do not want to use any of the predefined PCB outlines, you must draw the outline of the PCB on the outline layer #23. This can be done as following:

Free defined PCB-outline:

- Change to PCB-view. Here it doesn´t matter if you have placed your components already or not.
- Set the outline layer to layer # 23 (Menue View) as visible by pressing the according checkbox with **M1**.
- Select a useful grid value for the PCB-outline (for example: 1 mm)
- Press the **[POS1]** key at about the middle (for example as the below left corner of the PCB) to set the relative coordinates to 0/0. (see also the status line, below left).

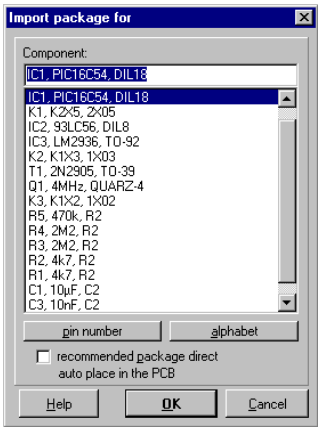
- Click on the drawing functions tool . The drawing palette of TRAGET will be opened.
- Click on the tool Draw open rectangle .
- Place the cursor on the 0/0 coordinates.
- Click on this spot with **M1**.
- While you draw the rectangle, press the **O** key. The dialog window Line options appears. Set the layer number to 23 (Outline layer).

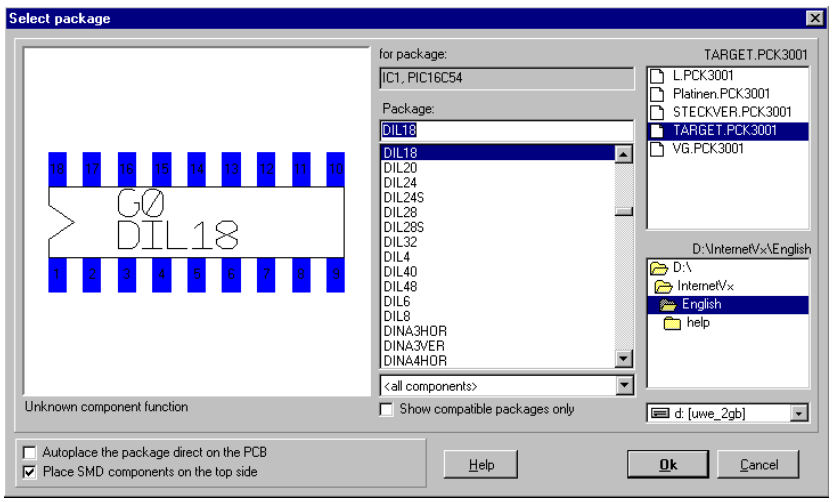




- Select **OK**.
- Draw now a rectangle of the right size. Please note the display of the relative coordinates in the status line.
- Place the rectangle with **M1** on the preferred position.
- Please set the grid parameters back to the old values (for example, 0,635 mm)

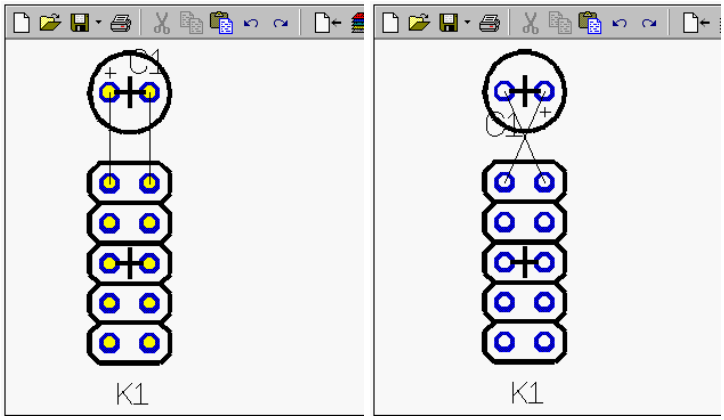
You must place all from the schematic required components inside this area. Call the Import packages function by pressing the **[INSERT]** key. A window appears showing all components where no package has been defined yet. The order of the list can be either <according pin number or <alphabetically.





For each component, a package must be selected. TARGET displays a proposed package, which has been saved during the creation of the component. You can of course select another kind of package (for example, SMD instead of DIL). The component can be rotated with **M2** and be placed with **M1**. Please beware of the following rules while you select and place a kind of package: Place all components first, who must be placed on a fixed location (connectors etc...). Next, all big components should be placed (components with a high amount of pins as well as components with central functions). Finally all small components should be placed to fill the gaps between the big components. TARGET displays a ratsnest while you place components. This indicates to which signal of the same name the component currently placed will be connected to in the PCB. The ratsnest will be calculated and displayed in realtime after each movement. A careful placement of componets is very important. The work of the Autorouter can be much more easier and efficient if the components are placed as good as possible. It is difficult for the Autorouter to route badly prepared PCB´s.

Favourable/unfavourable placement:

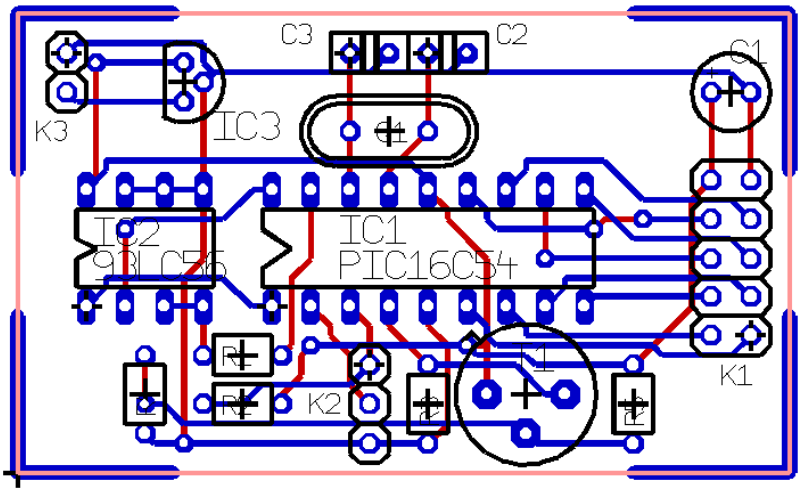


7.2.2 Placemant of signal tracks

Place your signal tracks according to the following rules: Place all signal tracks with special requirements first. As an example, the power supply connections (GND, VCC,VDD etc) do often require a higher signal-track-width than standard signal-tracks. Also

Afterwards, all further signals can be routet with the help of the autorouter or manually. Press the button to enter the place signal-track mode. You can place now with **M1** as many signal-tracks as necessary. You can select the copper layer where you want to place the signal-track from the toolbar. You can select the bend mode with **M2.** When you hit a signal (for example, a solder junction) while you place signal-tracks, TARGET sounds a BEEP. All corresponding soldering points and signal tracks will be highlighted. You can create a through-hole and change the copper layer with the **[.]** key.Always, the last drawn segment can be deleted with the **[BACKSPACE]** key.

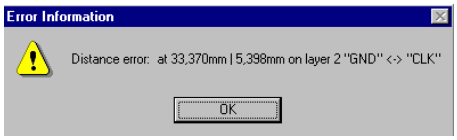
To start the Autorouter, please click on the AUTO key. The Autorouter is described in detail in **Chapter 6.3 - 6.4**. The following is a result of a double-sided PCB, routet with the help of the Autorouter:



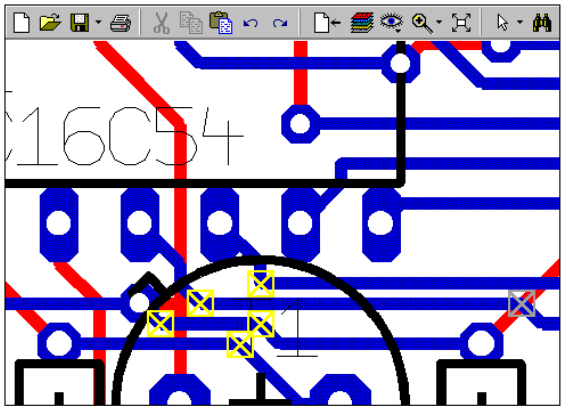
To check the complete project, select Check project under Menue/File. Confirming with OK starts a check of the layout according to predefined design rules:

- minimum track-width: 0,3 mm
- minimum distance between individual signals: 0,3 mm
- minimum drillhole diameter: 0,6 mm
- size of the annular ring of solder pads: 0,2 mm
- minimum distance between individual drillholes: 0,3 mm

All design errors will be marked on the PCB with a checked box (light magenta by default). A double click on a failure mark opens a dialog with a detailed description of the according failure message:



The following design errors are found in the area of IC1/T1:



Design Errors are marked by check boxes.

Additionally, all errors will be written into a text editor to have a fats access to the error list. This list can of course be printed.

Error list of our project:

1: distance error: VDD <- PB0 at 48.895mm | -4.445mm on Layer 16
2: distance error: VDD <- PB0 at 48.895mm | -4.445mm on Layer 16
3: distance error: VDD <- PB0 at 46.990mm | -3.810mm on Layer 16
4: distance error: VDD <- # at 48.260mm | -5.080mm on Layer 16
5: distance error: VDD <- # at 45.720mm | -4.445mm on Layer 16
6: distance error: VDD <- # at 45.720mm | -4.445mm on Layer 16
7: distance error: PB1 <- PB0 at 57.150mm | -3.810mm on Layer 16
8: distance error: PB1 <- PB0 at 48.895mm | -3.175mm on Layer 16
9: distance error: PB1 <- PB0 at 48.895mm | -3.175mm on Layer 16
0 bridges
9 vias
74 drill holes
Smallest drillhole diameter: =0.600mm
Smallest distance between two drill holes: =0.300mm
Thinnest restring: =0.200mm
Thinnest track: =0.300mm
Smallest signal distance: <=0.234mm
Size of PCB: 50.165mm x 29.845mm

Please start again the design rule check and set the smallest distance between signals value to 0,2 mm. (A minimum distance of >= 0, 2mm is usually accepte from printed circuit board manufacturing companies and can be produced without extra costs.).

After a second run of the design rule check, the errors were fixed and the markings in the PCB dissappeared.

Chapter 8: TARGET 3001! Simulation

Chapter 9: EMC-Analysis

The increasing expectations on the immunity and sensibility for disturbance with electronical device thus regarding their PCB is an important topic. Using the TARGET EMC tool, the PCB can be checked in terms of EMC while designing it. Also analizing the auto immunity of a PCB and its influences it can take by emitting diestortion can be analyzed. The chapter is completed with hints and advice for designing an EMC optimized PCB.

Contents:

9.1 Introduction
9.2 Handling of the EMC-Analysis tool
9.3 Prevention of EMC disturbance
9.4 Improvement of EMC properties

9.1 Introduction

Electrotechnical progress has lead to increasing pollution by electromagnetical radiation.
The reasons are:

- permanent increase of electronic device in the homes and offices (mainly telecommunication device
- more and more electronic device standing nearby
- decreasing structures of modern semiconductor technique
- increase of working speed of electronic units

As a consequence we face increasing disturbance and interference levels. The disturbance can be electromagnetical noise, undesirable signals or a modification of the spreading medium itself.

To make sure that electronic device are able to work with their surroundings satisfactory, lots of rules and regulations do exist we have to stick to.
On January 1st 1996 the referring act (EU-law 89/336/EWG), binding in Europe, statutes new conditions regarding electronic sets brought into circulation first after the mentioned date. That means any electronic set on the one hand must work error free in a electromagnetical disturbed surrounding and on the other hand must not disturb other equipments. As a consequence a conventional development of electronic device needs to be expanded by EMC analysis for to avoid later (expensive) corrections. For a developer it would be most comfortable if the design package he uses would guide him to an immune PCB.

The most important construction element of a PCB is the signal track. It connects signal source and signal sink. Its performance has significant influence on the EM properties of the circuit. The purpose of EMC is the generation of an unweakened and disturbance free transmission of the signal from sender to recipient. This aim always can be reached only approximately.

The following calculation methods have been considered for to recognize misleading influences on signal tracks:

- **Calculation of galvanic coupling:** Galvanic couplings happen, if a circuit (source) uses the same return line. It mainly occurs on ground lines or power tracks.

- **Calculation of inductive coupling:** The cause of inductive disturbing coupling is the change of the magnetical flow, which the circuit of the disturbing source inflicts into the circuits area of the disturbance sink. So those signals are interesting on which significant changes of flow in short time happen or those leading flow in high frequency.

- **Calculation of capacitive coupling:** Capacitive couplings occur between two circuits whose signals show different potentials. High frequency or low rise times respectively have influence.

- **Calculation of the coupling factor:**

- **Calculation of the radiation coupling:** In high clocking semiconductor circuits signal loops are the reason for radiation coupling. Because of relatively high flow steps in the conductors dynamic magnetic fields are generated. The individual current loops have an antenna-effect with the characteristic of a magnetic dipole. The amount of the resulting magnetic and electric disturbance field strength depends on the rise time, the clock frequency, the current of the loop and the corresponding loop area.

- **Calculation of the maximum track length:** A signal track may not have any length. The signal frequencies and the rise times only allow a maximum length. Is it passed, disturbances can occur e.g. reflections. The original signal could be modified and an error could be the result.

9.2 Handling of the EMC-Analysis tool

Contents:

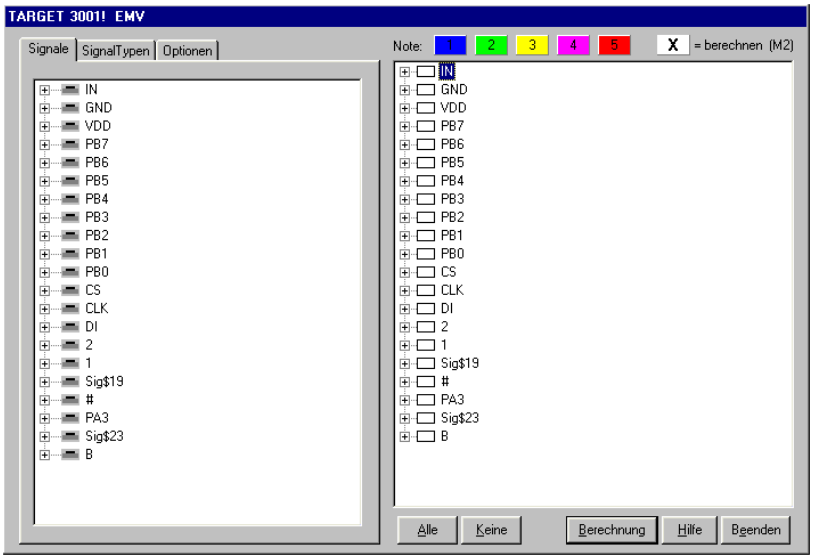
9.2.1 Input dialog
9.2.2 How to proceed
9.2.3 Kinds of coupling

9.2.1 Input dialog

You can start the EMC tool at any time. It makes sense starting it after all signal tracks have properly been routed.

Start this tool in layout view in menu "Actions" and "Start EMC check..." or choose the button shown above under the wizard. The complete handling is managed in one dialogue. The Windows 95-Tree-View-Technique has been used you may know from the Explorer.

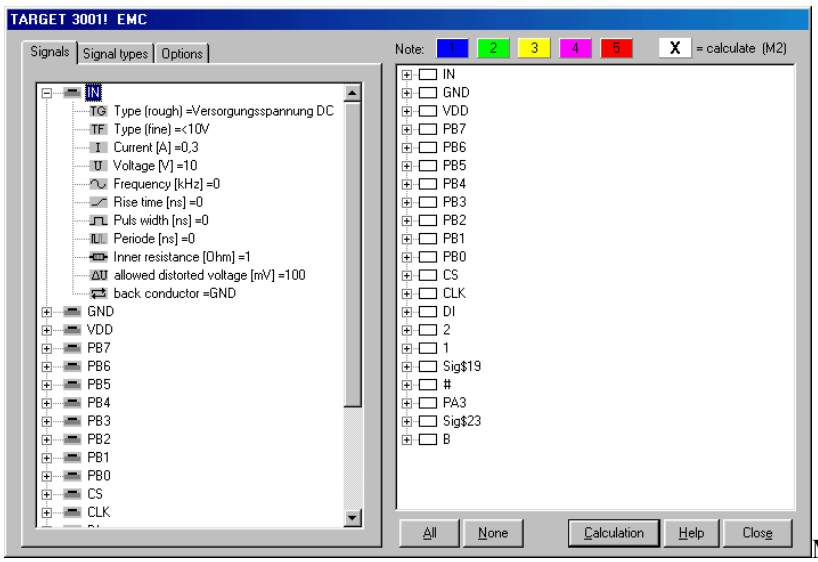
In the dialog on the left you see all signals routed in PCB view listed. Afterwards the signals are provided with different properties. In the dialog on the right you choose the check routine and the signals to be checked. Analysis and evaluation are displayed coloured.



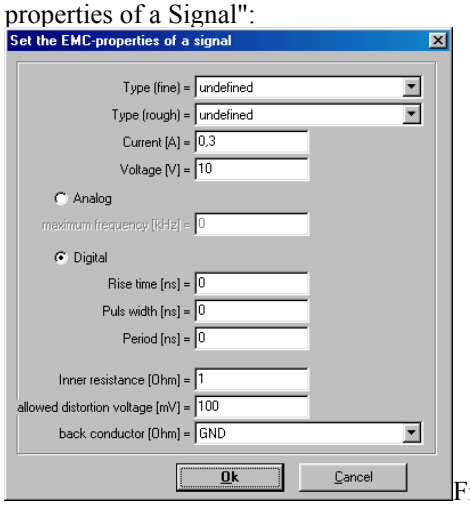
9.2.2 How to proceed

All settings and allocations are now described like a list:

1) Edit signal list
In the left part of the dialog the complete signal list of the **already wired signals** is shown. Click on ⊞ and all possible properties of a signal are listed.

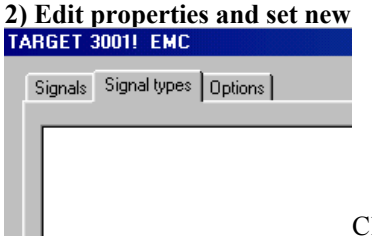


M11 on an entry opens the dialog "Set the EMC properties of a Signal":



First allocate the requested properties to the signals. Type (rough) means classification "rough", Type (fine) means classification "fine". For example rough = digital signal TTL, fine = technology HCT. By the entry "back conductor" (= return line) you can set the reference signal of a disturber. This entry is important (fundamental) for calculation of the coupling factor, the capacitive and the inductive coupling. Without a definition of those properties a sense making EMC analysis is not possible.

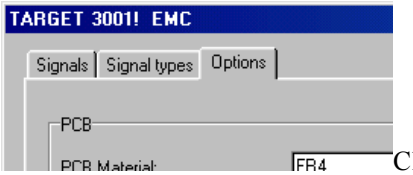
2) Edit properties and set new



Click on register "SignalTypes". First all of the existing properties are listed. By **M11** the properties can be edited. For avoiding the case that a signal can not be provided with standard properties press button "New" for to set individual properties.
First choose a suitable name for the property. Afterwards the signal type has to be defined.
If you highlight "analog", please set the frequency at "technical data". Otherwise (when choosing "digital" type) the risetime and the pulsewidth must be entered. Furthermore the values for current and voltage are important for the EMC calculations later. They need to be set.

This point 2) can be ignored if you don't need to define new signal properties.

3) Set Options



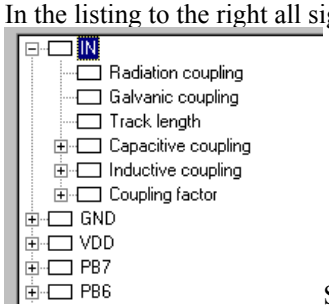
Click on register "Options". Please enter data regarding the board material:

- kind of material (e.g. FR4)
- dielectricity constant (is defined by the kind of material)
- thickness of material (e.g. 1,5 mm = 1500 µm)
- copper coating (e.g. 35 m)

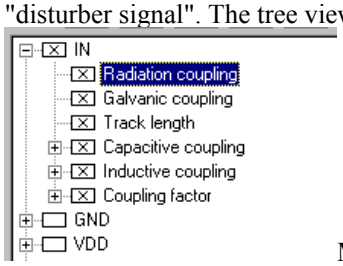
All other geometrical data are calculated by TARGET (track length, track width, number of layers).

4) Select analyzing modes

In the listing to the right all signals are shown. Highlight the signals you wish to check regarding EMC as follows:



Select by **M11** a signal to be checked (in our example its the signal IN). This signal is used as a "disturber signal". The tree view suggests automatically all opportunities of calculation.



M2 on signal **IN** prepares all analyzing modes for the calculation later on. You can individually select by M2 only those which are of interest for the check.

The **return line** can not be choosen as a **disturbed** because it had been defined with the signal list.

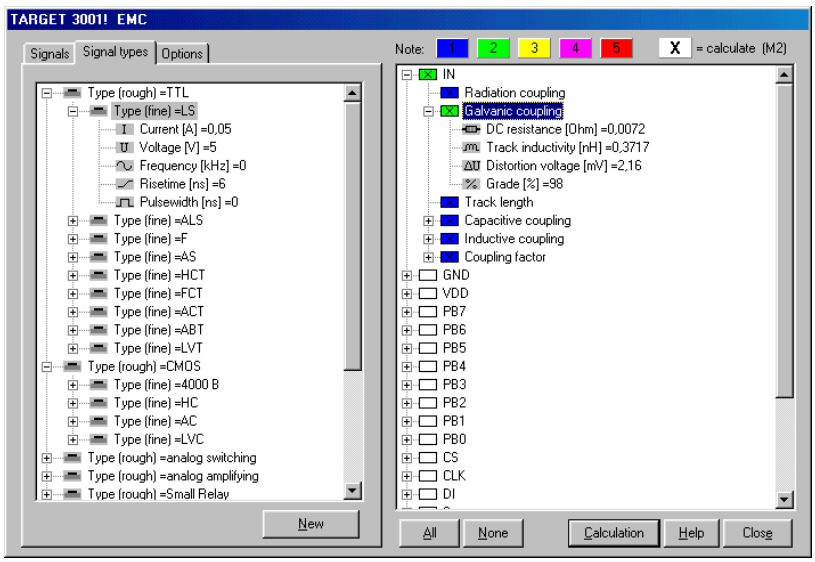
5) Start calculation

Click on button "Calculation". As a graphical analysis the signals are shown in a colored rating ("Note") from 1 to 5.

Meaning of the colors:

- blue: Note 1: (rating: 100%)
- green: Note 2: (rating: 90% - 99%)
- yellow: Note 3: (rating: 50% - 89%)
- purple: Note 4: (rating: 1% - 49%)
- red: Note 5: (rating: 0%)

The calculation results are shown in a list:

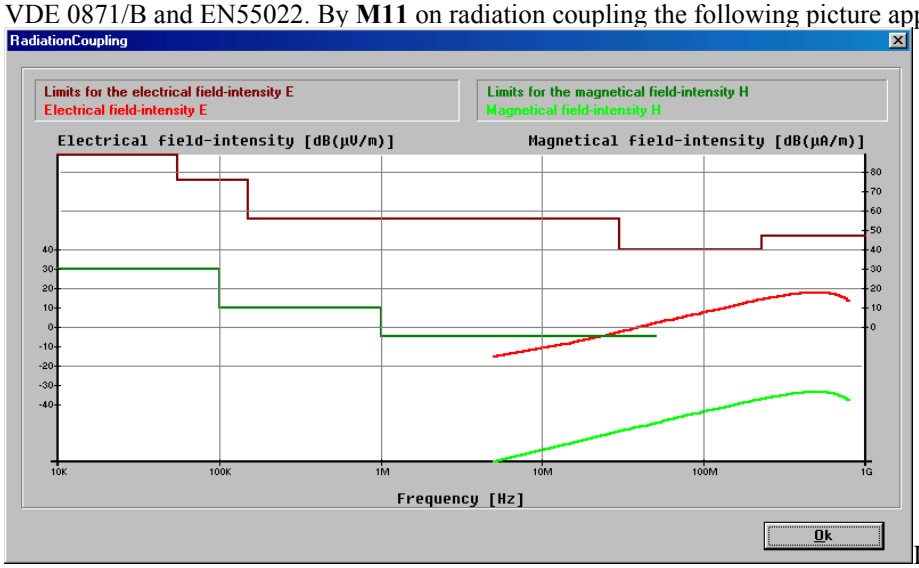


Using the following formula the analysis of the percentual rating is explained:

Regarding the length of the track is: Percentage= $100\% - \left(\frac{100\% \cdot l_{signal}}{l_{zul}}\right)$ Regarding all other calculations is: Percentage= $100\% - \left(\frac{100\% \cdot U_{Stör}}{U_{zul}}\right)$

lsignal = length of signal (track)
lzul = allowed length of signal (track)
Ustör = disturbance voltage
Uzul = allowed voltage of the signal

The result of the radiation coupling is shown in a graphic. The calculated results are compared with the limits following VDE 0871/B and EN55022. By **M11** on radiation coupling the following picture appears.



If the calculated values grow beyond the limits, the critical area is shown yellow (in the listing the box is shown red).

The abscissa shows the frequencies in logarithmic scale, the ordinates on the right and on the left refer to the electrical resp. magnetical field strength.
The brown line shows the defined limit line regarding the electrical field strength. The calculated curve of the electrichal disturbing field strength is red.
The green line shows the defined limit line regarding the magnetical field strength. The calculated curve of the magnetical disturbing field strength is green.
In an ideal case the calculated curves show below or upon the limit line. In other cases countermeasures need to be done - e.g. reduction of the loop area.

9.2.3 Kinds of coupling

Calculation of the coupling factor
In the tree view the calculated results are shown after the button "Calculation" was clicked. Calculated are the coupling factor between the disturbing and the disturbed sitnal (taken from the inductive and capacitive track stressings) and the wave resistance of the arrangement. Moreover an EMC rating is made. TARGET calculates a percentage which informs the user whether a critical value is reached.

Calculation of the inductive coupling
The results regarding the inductive coupling between the disturbing and the disturbed signal are shown. The following results are displayed:
Counter inductivity of the coupling arrangement and the disturbance voltage, which is transferred from the disturbing signal to the disturbed signal by inductive coupling. Also an EMC rating is made, showing whether both signals perform in an EMC critical range.

Calculation of the capacitive coupling
The following results are calculated and shown: coupling capacity between disturber and disturbed, disturbed signal's current, caused by capacitive coupling of the disturbed signal and the disturbing voltage over the inner resistance of the disturbed signal, caused by the cross coupled disturbing current. Also an EMC rating is made, showing wheather both signals perform in an EMC critical range.

Calculation of the galvanic coupling
After pressing button "Calculation" the following results appear: DC-resistance of the signal, line inductivity of the signal and the disturbing voltage upon the signal, caused by galvanic coupling on this

Calculation of the allowed track length
The maximum length is compared to the real length. The user can see whether reflexions to the track (line) could occur.

Calculation of the radiation coupling
Please see chapter 9.1.

9.3 Prevention of EMC disturbance

The electromagnetical compatibility of a device needs to be made sure in three aspects:
- no external disturbances
- no internal disturbances
- it must not act as a source of disturbance

For reaching those aims a developer has to follow several rules. The most important points and the corresponding explanations are shown in the following chapters. If you follow those advice you won't face severe EMC-problems.

9.3.1 Wiring of reference potential (Ground)

It is obvious to minimize galvanic coupling by avoiding a shared use of signal tracks. This could be reached by placing completely separated return lines (star shaped ground). Because of the increasing amount of tracks and their density the capacitive and inductive couplings will increase. The separatin of the shared tracks and the reduction of the coupling impedances is a practicable measurement.

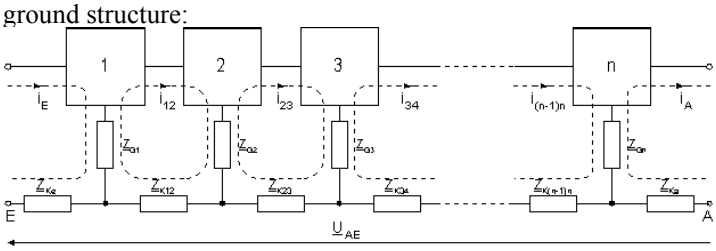
The lower the track impedance and the generating current resp. change in current is, the lower is the disturbance voltage regarding galvanic coupling. The current is defined by the need of it thus not a free disposable value. As a measurement for opressing a disturbance coupling we only can choose the reduction of the track impedance. For its decrease the following aspects may be concsiderable:
- short tracks (lowers the ohm- and inductive proportions as well as the skineffekt; having effect on the entire frequency range)
- big track diameter (lowers the ohm- proportion, is especially effectful at DC and low frequencies)

- big track surface (lowers the skin effect, effective at high frequencies resp. quick current changes e.g. several thin tracks besides each other)
- small area of current loop (lowers auto inductivity radiation surface)

The highest disturbance voltages in galvanic coupling arise because of voltage falls when quick changing current. The ohm-proportion has influence because of the skin effect. Also the auto inductivity of the track is effective. For the auto inductivity changes only few regarding to the track diameter and as it can be modified only few on the board , we concentrate on the most effectful measurements: shortening the track lengthes and arrange them in respect of their use.

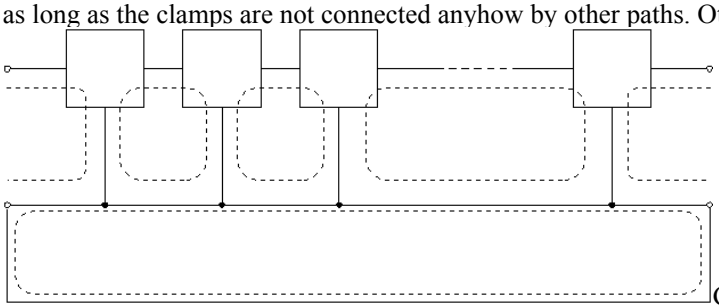
9.3.1.1 Star-shaped structure of reference conductors

The impedance may not be ignored when wiring the reference potential. In the worst case the related potential can show different values on different points. This case for instance is made clear in the following picture showing a substitute with an elementary row ground structure:



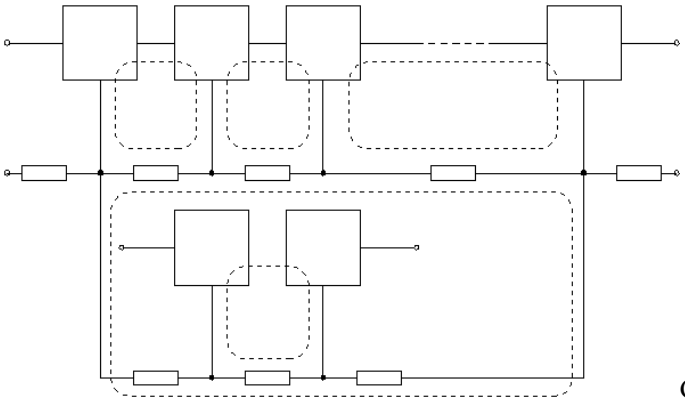
In a circuit of this structure every step controls only the following step and is controled itself only by his predecessing step.The impedance Zgi of the track from the steps ground star point to the systems potential compensation (point A to E with impedances Zki,(i+1)) couples entry- and exit circuit of this step. Wheather this effects disturbing can be defined without recognizing the rest of the circuit because a disturbance is caused only within this step. On the related potential resp. ground between the steps i and i+1 (impedance Zki,(i+1)) only the current between both step flows. Also the ground input and exit clamps of the circuit are flown through by the related input and exit current.

At the impedances Zki,(i+1) of the ground parts between the steps decreases of voltage arise caused by each of the signal currents.They add up to the voltage UAE between entry and exit ground clamps. This fact remains without distubing consequences as long as the clamps are not connected anyhow by other paths. Otherwise a ground loop is built up.



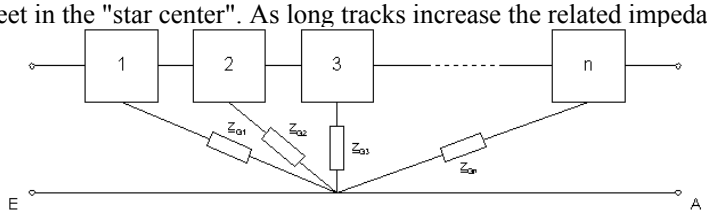
Ground loop on an "elementary row ground structure"

The signal currents not only flow back to the source by the related impedances Zki,(i+1), but also by the grounds of the other paths. But so all steps touched by this paths are coupled as the current analysis shows. The next picture shows the coupling of all steps in a parallel connection of ground whithin two circuits with "row ground structure".



Ground loop out of two circuits with "row ground structure"

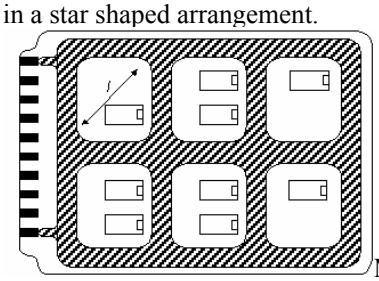
If the circuit is optimized the way that all Zki grow zero, a star structure of ground emerges. The ground connections of all steps meet in the "star center". As long tracks increase the related impedances Zgi, short tracks should be choosen.



compensation system between each of the steps.

9.3.1.2 Meshed tracking

If a star shaped structure is not effective because the signal tracks grow too long meshed tracking can be used. Also using this method a decrease of galvanic coupling can be reached. Here several parallel reference conductors are connected to each other like honey combs. They make up several closed track loops having shorter track lengthes and lower impedances for signals flowing than in a star shaped arrangement.



Meshed tracking

Have a look at the maximum width of the meshes. A current flow in them causes a radiation. It remains as low as its largest measurement **l** is. With decreasing **l** the radiation decreases quadratic so that in normal current flows no radiations of note arise.

9.3.1.3 Separation of signal and current referring conductors

A further option to avoid galvanic coupling is the separation of the reference flow conductors of signal leading circuits from those which have bigger needs of current. It can be devided in:

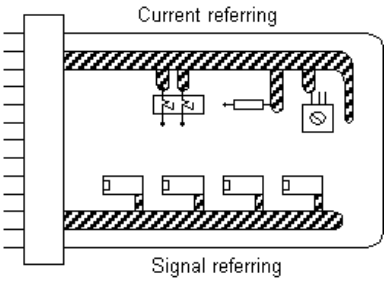
- signal reference flow conductors
- current reference flow conductors

Signal referring conductors are tracks not having stressed the reference potential regarding current (switch inputs). I. e. the sensible circuit areas as well as the reference potentials and the input supply of pre amplifiers. The current flow through those tracks amounts up to some mA.

Current referring conductors are tracks which often are stressed by higher (return) flows. Those target the power supply (OV) as well as the return flow of the initial current signal.

This division is arbitrary. Its aim is to divide between rather disturbance generating and rather disturbance sensible tracks. If the reference tracks can be divided to both groups one will have a system which is nearly insensible against galvanic coupling. This is

When wiring the reference potential (star structure) we can see, that the reference potential of the analog system had been separated from the digital one.



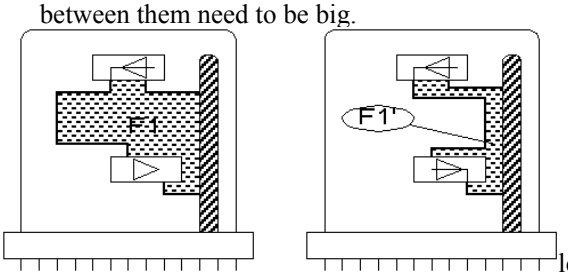
9.3.2 Wiring of signal tracks

For judging the EMC properties of signal transferring the whole circuit, i. e. the signal track as well as the return line has to be recognized. Humble structures show the current flow returning by the reference potential so we face an asymetrical signal coupling. Most of the signal tranferrances have those structures. Asymetrical arrangements have the currents returning by the same tracks and so inherit the danger of galvanic coupling. If immunity should be maximized the following aspects need to be recognized:

- a nearly non disturbed reference potential must be offered to the signals. (Using a star-structure the galvanic coupling is reduced.)
- for reaching a low inductive coupling the whole current loop (signal track and referance potential) must show a minimal area. This can be reached by:
- a short track (in general)
- a closel arrangement of signal track and return line (see picture).

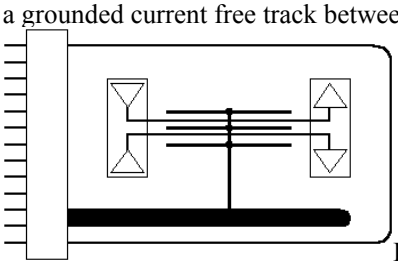
Following those rules also the disturbances going out from this circuit as a source are reduced.

- for minimization of capacitive coupling the tracks and especially their parallel placement need to be short and the distances between them need to be big.



left: subopimal because of big current loop area
right: favourable because of lowering the disturbance by reducing the current loop area.

A common transferrance of several signals a low coupling of signals should be considered. It can be reached for instance by placing a grounded current free track between signal leading tracks.



Lowering the the capacitive disturbances by insertion of shielding tracks.

This kind of tracking is suitable when routing multilayer PCB.

9.3.3 Wiring of supply tracks

Supply tracks can be divided into:
- DC supply (VCC) and
- return line (0V; common reference potential)

The problems placing power tracks are similar to those of the reference potential: The power supply of a component generates an decline of voltage within the power supply track which means a decline of power voltage. The galvanic disturbances influence the voltage leading tracks as well as the 0-Volt track (return line).

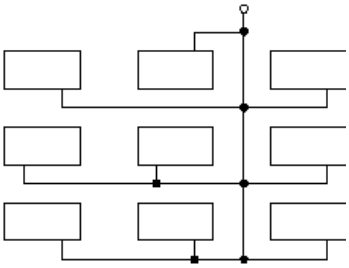
The 0V- supply track is the reference potential (GND). Take care of a low Ohm performance. Tracks should be short and broad.

9.3.3.1 Wiring of voltage leading tracks

Because most of the circuits are more immune against variations of the supply voltage rather than against variations of the referring potential slight disturbung voltages effect less dramatic here than on the reference potential. Nevertheless big disturbing voltage can be the reason for disturbances.

For the reason of disturbing voltages is to be seen in voltage declines whithin the track impedances especially quick changes in current flow are harmful.

For lowering those disturbance voltage the same aspects can be recognized as already shown in chapter 9.3.1 (see picture).
().



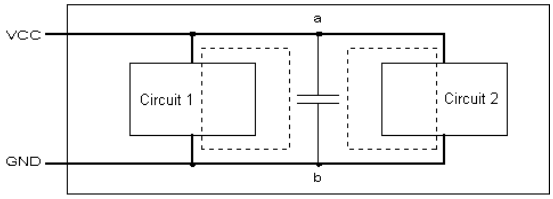
Arrangement of the power track with low coupling effects.

Components with big current needs should be placed close to the power source and the supply track to each of the component is short. Placing VCC and GND on the outer edge of the PCB also supports the decline of disturbance. Measurements regarding the placements of tracks don't need to be done following the same consequence as regarding the reference potential. Here we have the opportunity to buffer the required current using blocking capacitors - taking support by the reference potential.

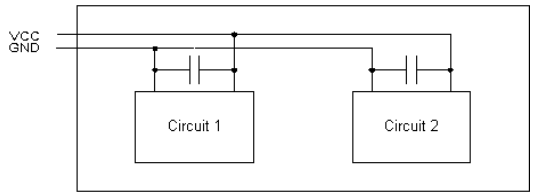
9.3.3.2 Use of blocking capacitors

Between power voltage and the reference potential at any circuit (place of consumption) a capacity for buffering reasons is fixed (blocking capacitor). This capacity rises up to the supply voltage. If needed current (potential) can be delivered. So the power voltage can be kept whithin a calculable range if sudden potential demand occurs.

The consuming loop nedes to show a low impedance for to let the potential flow from the capacitor quick enough (i. e. the buffering effect). The consuming loop consists of:
- the length of all connecting wires of the blocking capacitor
- the inner ways whithin the capacitors (including capacitance per unit length)
- the tracks from capacitor connection to consuming component
- the inner tracks whithin the shielded component



Elimination of disturbing influence by shifting the connecting star points

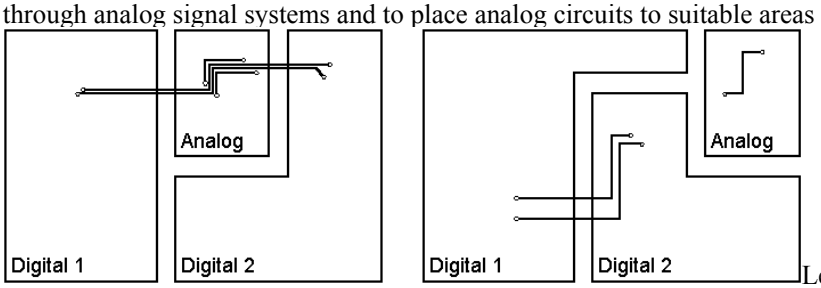


Favourable arrangement of blocking capacitors

The faster the signals are that means the faster the current nees rises the closer need the blocking capacitors be fixed to the component connections of VCC and GND. At high frequencies it depends on every mil of track length. The value of capacity should not grow too big because beyond the optimun value its effect declines. Blocking capacitors are also suitable for easing of Ohm caused voltage losses which arise at increased current needs of medium duration. Here higher capacity values need to be used wich do not have to be fixed as close to the component.

9.3.4 Separation of digital and analog units

Unwelcome electromagnetical influences can appear in a circuit between analog and digital structures . It makes sense separating and decoupling analog from digital structures regarding the placement of components. That means not to lead digital signal tracks through analog signal systems and to place analog circuits to suitable areas on the PCB.



Left: Suboptimal placement of tracks
Right: Favourable placement of tracks

It is important furnishing digital and analog systems on a board with seperated referring potentials (GND).

9.4 Improvement of EMC properties

There are lots of measurements for improving EMC properties. First the disturbances (paths of coupling)need to be localized. Is the kind of disturbance known, individual measurements can be started.

Contents:

9.4.1 Advice regarding galvanic coupling
9.4.2 Advice regarding inductive coupling
9.4.3 Advice regarding capacitive coupling
9.4.4 Advice regarding radiation coupling
9.4.5 Advice regarding coupling factor
9.4.6 Advice regarding the length of signal tracks
9.4.7 Advice regarding blocking and supporting capacitors.

Please note that about 95% of all disturbances are coused by the following kinds of coupling:

- common use of tracks (galvanic coupling)
- tracks too close to each other (capacitive and inductive coupling)
- field influence (radiation coupling)

Against generic EMC disturbances the following measurements can be helpful:
- reduce any disturbance on its source (e.g. shortening, drilling or shielding the tracks)
- interrupt the coupling paths of the disturbance (e. g. insert shielding metals, increase distances)
- save disturbing sinks (e. g. increase immunity, insert overvoltage arrester)

9.4.1 Advice regarding galvanic coupling

The reduction of galvanic coupling depends on the aspects:
- track length (as short as possible)
- track diamerter (track width as wide as possible)
- the area of current loops (as small as possible)

Reduction of galvanic coupling on reference signals (GND):
Placing tracks of reference signals please separate signal-reference signals (loww need for current) from current reference signals (high need for current). Make sure that digital reference signals are seperated from analog reference signals. Realize it by using multilayer PCB on which copper coated reference conductor areas resp. shielding areas are planned or realize it by using meshed tracking on double sided PCB. Meshed reference signal tracks hinder the track routing of the other signal tracks relatively few. It can be reached by dividing the meshing on to the tlower and the upper copper layer. With normal track dimensions the measurements of the meshes will be realized at some inch, so that radiation to be mentioned will only occur at more than 300 MHz .

Reduction of galvanic coupling on power supply tracks:
A high track width and short lengthes are very important. Multilayer PCB could show a single layer solely as power supply. Supporting capacitors can be inserted if indifferencies in the supply tension occur. Return lines of power supply (0V) should be fixed to the reference potential (GND) on one allocation point. If we have several supply voltages with individual potential free 0V-connections for any track different connection points to the reference potential can be choosen.

9.4.2 Advice regarding inductive coupling

Low coupling in this respect is reached if signal tracks as well as reference potentials show small areas. Realize it by:
- placing short tracks and
- fixing them close to the return line.

Shield the signal tracks by placing GND tracks besides.

9.4.3 Advice regarding capacitive coupling

For minimizatin of capacitive disturbing coupling make sure that
- the tracks of disturber and disturbed are short
- their parallel tracking is short
- their spacing is wide

Parallel signal tracks lying on one layer, should use beside-tracks as return lines. So their area can be minimized. Here the return lines should run potential free. The signal tracks should run shielded by GND tracks besides.

9.4.4 Advice regarding radiation coupling

For to reduce the radiation of a signal the loop area (the area between disturbing signal and the signals reference signal) should be minimized. That means that besides the disturbing track the return line directly should be placed. If a reduction of the loop area can only be realized partly we have the opportunity shielding the radiation. Disturbing tracks can be shielded by metal sticks, coverings or little walls between (Al or Cu of 0,5 - 1mm).

Multilayers can have whole reference signal layers as shields. If there are not enough layers available the shielding can be realized by floating tracks. Especially those components need to be shielded, wich are sensible on the one hand and on the other hand emit radiation relatively high. Often this is the case between digital and analog groups. Single components can be shielded by Al- caps (0.5 mm) touching the PCB. Transistors can use their own metal package being connected to GND. Simple functional blocks can be put into small metal boxes.

9.4.5 Advice regarding coupling factor

The coupling factor contains inductive and capacitive coupling as well as wave resistance. A reduction of the coupling factor can be read in chapters:

- Advice regarding inductive coupling (chapter 9.4.2)
- Advice regarding capacitive coupling (chapter 9.4.3)Advice regarding the length of signal tracks (chapter 9.4.6)

In general: the longer the tracks of disturbed and disturbing signal are and the bigger their parallelity is, the stronger their influences will grow.

9.4.6 Advice regarding the length of signal tracks

A track should be designed the same over the entire length and show no bendings and forks. In most cases this cannot be realized. Please recognize the following aspects:

- short tracks
- no bendings over 45°
- avoid change of layers (vias)
- tracks which are longer than the value the EMC analysis suggests and which can not be shortened, should be closed up using the same wave resistance like the track itself for to prevent reflexions as far as possible.
- replace T-connections by star point shaped tracking
- enough spacing between digital and analog tracks. Place shielding tracks (GND)

9.4.7 Advice regarding blocking and supporting capacitors.

Upon GND and supply track sudden changes in current flow (peaks) could arise, which cause disturbing voltage by inductivities within this tracks. Those disturbing voltages are the bigger, the longer the track is and the quicker the change in current flow is. To avoid it use supporting capacitors (ceramic) which are poor of inductivity. In the moment of peak they effect like a store and take over the peak. So it is kept out from further supply lines.

Supporting capacitors should be placed close to the IC's (at 1 to 10 nF). Every capacitor should be effective for 1 to 4 integrated circuits. To the exits of voltage controls supporting capacitors (about 10 F) are fixed in practice. Every PCB should have supporting capacitors of 1F on any power input. On tracks evry 5 cm distance supporting capacitors of 0.1 F should be placed.

A clever arrangement of power supply tracks can help minimizing disturbances.(see advice regarding galvanic coupling)

Use the following typicalcapacities of supporting capacitors:

- for fast signals (f>80 MHz; risetime <5 ns): 1000 pF
- for medium fast signals (f = 10 to 80 MHz; risetime = 5 to 40 ns): 10 nF

- for slow signals (f < 10 MHz; risetime<40 ns): 33 nF to 0,1 F

Chapter 10: Data Interfaces and I/O Device Drivers

This chapter explanes all interfaces and export drivers TARGET 3001! offers. TARGET 3001! is an "Open System" that means you may create individual interfaces e.G. to pic and place machines or in terms of data transfer to other CAD-systems. Don't fear the generation of Gerber files. Using the gerber viewer you obtain a control-view of the generated Gerber files before producing a PCB.

Contents:

10.1 Gerber Output (Photoplot)
10.2 Read in Gerber data
10.3 Drilling
10.4 Gerber and drill output
10.5 Post-Script driver
10.6 DXF-driver
10.7 Engraving insulation channels
10.8 ASCII- Interface
10.9 Converter TARGET 2.1 to TARGET V3
10.10 Converter TARGET V3 to TARGET 3001!
10.11 Read in EAGLE files and EAGLE libraries
10.12 XGerber-driver

10.1 Gerber Output (Photoplot)

First make sure, that your producer would not prefer your data in XGerber Format! See also chapter 10.12

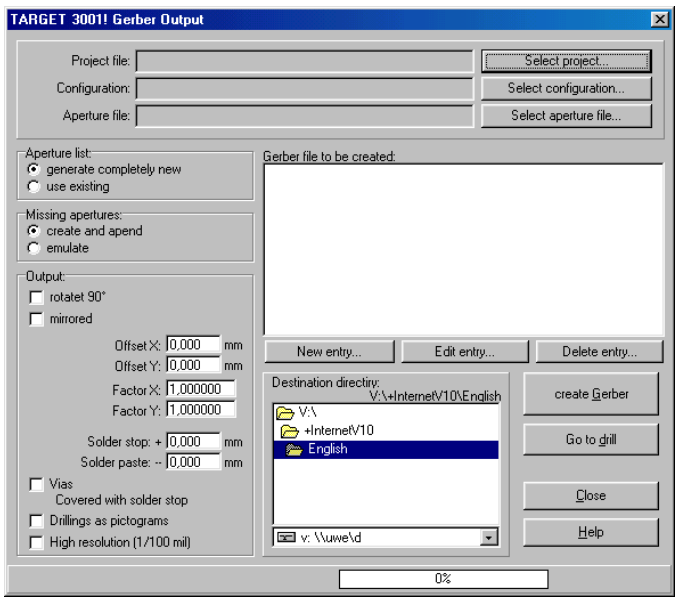
Gerber Format RS 274 D is a far spread driving format for photoplotters. A photoplotter is a x-y-plotter with a light-pencil drawing on a film. The source of light can adopt several forms depending on apertures preponed. E.g. for drawing of lines often a circular aperture is used , pencil light is switched on and the route of your copper trace is driven. For creation of solder pads no movement is needed but they are flashed on the film with a special kind of aperture. Every aperture corresponds to a specific D-code (D10...Dxxx). The aperture and its corresponding D-code are agreed in an aperture file which are not normative and often not readable to machines. TARGET is able to work with predefined aperture files or generate those on it`s own. Thus for a film a Gerber file and an additional aperture file is needed. The aperture file contains a list of all apertures, the Gerber file shows how they are to be moved.

Generation of Gerber data on an example

For generating Gerber data from your file act as follows:

a) Draw a PCB in TARGET and save it. Write in the layer **Copper Top** the word "**TOP**" and in layer Copper Bottom the word " **BOTTOM**" in mirrored manner, thus beeing readable on bottom view. Internal copper layers should be numbered by K1 to Kx, if they are used.

b) Choose Options "Input / Output Formats" in menu FILE. Click on "Gerber output (RS 274 D)". The Gerber driver is automatically started.



c) Click on "select project" and select it.

d) The project-file and the aperture file is shown in the window. If you want to rename the aperture file , click on "Select Aperture List".

e) If available and useful for your purposes load a configuration file. It contains information on zhe fact which TARGET layers are quoted in which Gerber files. Otherwise define by "new entry..." individually a Gerber file to be put out. Choose a name for the gerber file and define, Which TARGET layer(s) should be mentioned in it. Possibly it makes sense to activate layer 23 "outline" in combination with the gerber file to be put out. So you generate a relation frame for easier viewing. The following configurations are offered by TARGET:

- DSDK-LSL.GC
- DSDK-without -LSL.GC
- DSDK-MASSE-LSL.GC
- DSDK-MASSE.GC
- 4MULTI-MASSE-LSL (ground on inner layers)

Meaning: DSDK = double sided, plated through, LSL = solder mask (dielectric), MASSE = PCB with ground plane 4MULTI = four layer multilayer

f) Naming the gerber files please use sense making names, best using not more than 8 digits for to make sure that your PCB manufacturer is able to compute your files.If you want to edit the configuration settings please use "edit entry" or "delete entry".

g) If the remaining settigns are OK for you click on "generate Gerber". The Gerber files are now generated into the dirctory shown below. Advice: Save Gerber files on your harddisc before copying them on floppy disc.

By "**90° rotated"** the Gerber output is rotated, by "**Mirrored"**, such is done . By "**Offset**", an +/-offset can be set, by **"Factor"** you may adjust the output. **"Solder mask+"** **"Solder paste--"** diminishes the solder paste a little for SMD-components. **"Vias covered with solder stop"** makes sure that no solderable areas are left when plating through. Nevertheless for every via a little cap for the drill hole is generated so that the solder mask fluid does not inflict the via when printed by silkscreen. **"Drillings as pictograms"** corresponds to the same utput driver in TARGET. For every drill hole with different diameters a little symbol is generated (pictogram). It helps when manually drilling. For having results in higher resolution choose **"High Resolution (1/100 mil)"**.

Aperture List

Possibly the generated aperture list is grown a littlw at every output step: at "Copper bottom" it is generated new: All apertures required of it are listed. If useable apertures are found when generating a new layer, those are used. is e.g. at "copper top" an aperture used not having been used already, it is added to the aperture list.

TARGET uses the following kinds of apertures::

- FK:Fahr-Kreis: circular aperture for traces, triangles, circles (draw)
- FQ:Fahr-Quadra: square aperture (square)
- BE: Blitz-Ellip: elliptic / round flash aperture for solder pads (possibly. round or oblong)
- BA: Blitz-Achte: octagon oder poss. oblong flash aperture for solder pads
- BR: Blitz-Recht: rectangular flash aperture for solder pads
- BO: Blitz-Oval: oval / oblong flash apertures for solder pads
- BP: Blitz-Pass: Fiducial
- BB: Blitz-Bohr: symbol representing a drill hole

Aperture List: generate new: and **Missing Apertures: generate and add.** If your photoplotservice is able to compute your aperture file (speaking your language) you deliver to him, don't work out one but let it generate by TARGET (Emulation = no) and deliver it to him.

Aperture List: use existing and **Missing Apertures: emulate.**
speak his language), you have to generate one in ASCII inthe following format (write down e.g. in Windows-Editor NOTEPAD.EXE) :

e.g. EXAMPLE.BLN:
D-Code, kind of aperture, width, height

D10 FK 4 4
D11 FK 6 6
D12 FK 8 8
...
D46 FK 315 315
D47 FK 394 394
D48 FQ 31 31
...
D67 FQ 157 157
D82 BP 200 200
D83 BP 394 394
...

D-Code is for "D10" to "D999", kind of aperture is as above (2 letters), width and height is for the measurements of the aperture in MIL (= 1/1000 inch = 0,0254mm). The notations have to be seperated by blanks. Additionally you need to deny TARGET to add missing apertures to the list (emulation). TARGET then has to try to emulate the missing apertures with smaller apertures. That means e.g. a big octagonal solder pad is drawn by using a smaller circular aperture. This emulation wastes plotting time and generates often huge files.

If you need several plots, (e.g. copper bottom, copper top, ...), it is disadvantage if you deliver a newly generated aperture list to each of the gerber files. Your photoplotservice then needs to enter the aperture liss manually. This is costly! Let the Gerber files be generated by configuration file at once. So only one aperture list is generated being used by all Gerber files.

Send your Gerber file(s) and your aperture list (files) to your photoplotservice. In many cases they need a description of the kinds of apertures used additionally (FK,FQ,BE,...). Also you should tell them which Gerber- and aperture files match together and what is made visible on the files:

OK but suboptimum:

Gerber file□Aperture file

XY.TOP□□XYAPR.BLN copper top
XY.BOT□□XYAPR.BLN copper bottom

XY.PT□□XYAPR.BLN silkscreen (positioning of components)

better:

File□□contains
XY.TOP□□copper topKupferseite oben
XY.BOT□□copper bottomKupferseite unten (Lötseite)
XY.BTB□□Silkscreen (positioning of components)Bauteilaufdruck oben
XXY.BLN□□Aperture list for all Gerber files

Place writing on the copper layer to be read unmirrored in reality. So you show your photoervice or your PCB manufacturer undoubtedly how the copper layer is meant. If there is few space, place "EL" for electric mirrored on copper bottom or "POS" for positioning on copper top. So you may avoid misunderstandings.

Generation of ground planes

Ask, whether your photoplotservice is able to subtract indiviual Gerber files optically from each other and create a film of it. Now you are able to work with deletion layers or open drill holes.

For the automatical generation of ground planes using deletion layers and auras, more than one gerber file for one copper layer are needed: e.g. "copper bottom":

Layer□□□Gerber file

area : layer 0□□TEST.B1: Bottom, , positive
deletion : layer 1□□TEST.B2n: Bottom, compute second, negative, above the first
copper: layer 2□□TEST.B3: Bottom, compute last, positive, above all

This only will work if your photoplot service is able to create several layers with different operators on one film. Nowadays plotters and their software always are able to do so. Some plotters can only work out one layer negative (as for solder stop) for to place another layer positive above it:

Layer□□□Gerber file
copper: layer 2□□TEST.B2: Bottom, compute second, positive, above the first
deletion : layer 1□□TEST.B1n: Bottom, compute first, negative

For acting that way it is necessary to fill layer 0 (area bottom). with a filled rectangle over the complete PCB size! If then e.g. the PCB needs to be provided with a ground plane half a side, the other half must be made rid of the ground plane copper on deletion layer 1. On the deletion layer you also can place other geometric figures (rectangles, triangles, circles, tori,...)

- **Attention:** Generating Gerber data an subsequential computing on films and further manufacturing media is not trivial (CAM-Prozess). Please check in any case your data before ordering films, silks, stencils or PCB. It is useful loading "GC-Prevue" (a Gerber viewer) from our homepage.

Order a copy of the plots before for correction reasons.

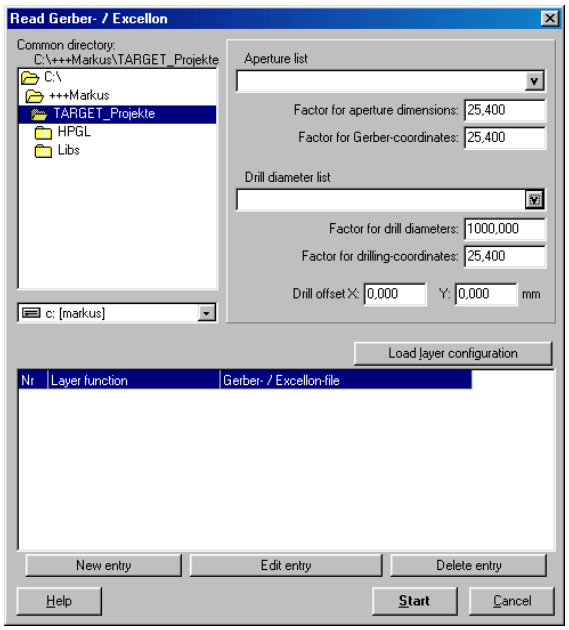
If this all seems too complicated to you, let your PCB be manufactured by an expert manufacturing by directly using your TARGET *.T3001 files. Manufacturers doing so are listed in the program TARGET under PCB service or visit our homepage http://www.ibfriedrich.com/service/services.htm.

10.2 Read in Gerber data

Basics

Using this tool you can view the Gerber data of your project before producing a PCB. The Gerber data are directly read into a TARGET project. By selecting the PCB layers set visible, the corresponding views can be seen and checked with the TARGET project in seperated windows besides each other.

Additionally you can import Gerber data from other layout CAD packages and edit those within TARGET. For to read in other formats please pass a detailed description of aperture and drilling list and sample aperture and drilling files out of your system to us. We can create a new tar3001.exe for you quickly free of charge.



Read Gerber-Files

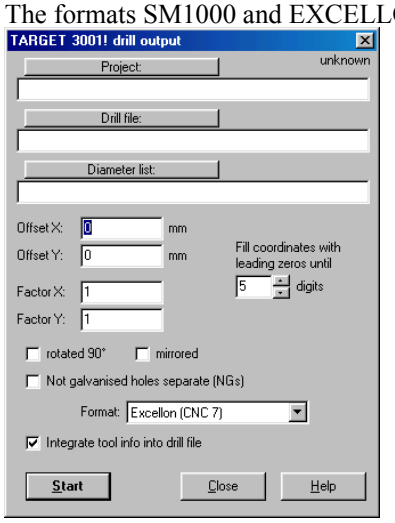
- Open a new project "pcb without schematic"
- Click on "Read Gerber & Excellon files" in menu File/Input / Output Formats
- Choose your **standard directory** which contains the Gerber data desired
- Choose the corresponding **aperture** and **drilling list**.
- Click on "load layer configuration", if you want to use the existing layer configurations of the TARGET directory, or if you already edited an individual configuration. Here you have the opportunity to open "Gerber configurations" (edited by the Gerber Out function) or "GerberIn Configurations" (edited by "GerberIn")
- Now the dialog opens "Load Configuration". You may enter a new name for the configuration files. Those are saved in ASCII format. All TARGET supplied configuration files have stars in their file names (e.g. *.BOT for copper bottom). Thus enter an exxpressive file name regarding to your Gerber file
- If you cannot use one of the proposed configurations, edit a new one on your own. Drop information which layer (e.g. copper bottom) is to be found in which file (e.g. TESTFILE.BOT)
- Click on **New Entry**, for to define further layers, e.g. drill_file *.DRL.
- If all settings are made, enter "START".

REMARK: Gerber data concerning the layer "solder stop" should be read in from another layer than "Solder Stop Bottom", because TARGET recognizes the pads of the Solder Stop Layer automatically by activating them in the normal layer dialog. Gerber data of the solder stop layer with topping of the vias are ignored then or dominated by those "recognized".

If the external CAD format you need concerning the aperture and drilling list not is offered by TARGET, please describe it to us (evtl. using an example - file). We add all formats you drop to us into the list as we estimate it reasonable.

10.3 Drilling

The formats SM1000 and EXCELLON are far spread driving formats for drilling machines.



Setting of the process parameters in the enter-box

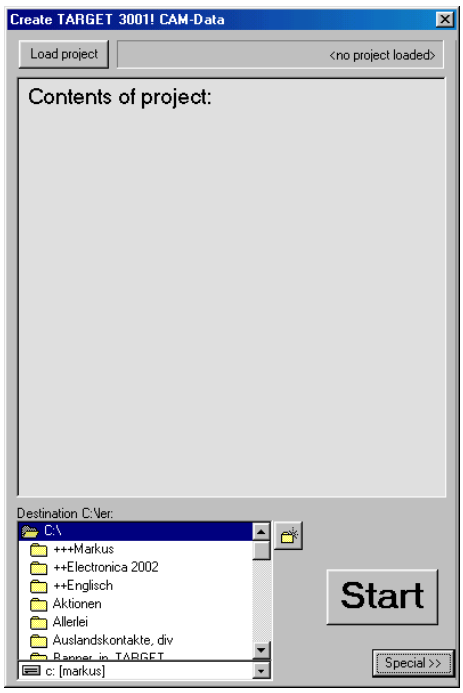
a) Open a TARGET-project with button "**Project**"
b) Enter the drill file with the extension *.DRL . The corresponding one is automatically generated by TARGET.
c) Normally no other settings are requested. Choose START.

EXCELLON-format is: **Unit**: 0,0254 mm, **File Header**: %.
Sieb&Meyer-format is: **Unit**: 0,01 mm, **File Header**: %%1000.
With offset X and Y you may enter additional Offset-data for the drilling machine. Default: **Offset X**: 0 mm **Offset Y**: 0 mm
With factor X and Y individual adjustments of the scale of the drill image for both of the directions can be made.
Default : **Factor X**: 1 , **Factor Y**: 1
The drilling image can be put out rotated by 90 degrees and / or mirrored. Tick the corresponding box. No default is set.

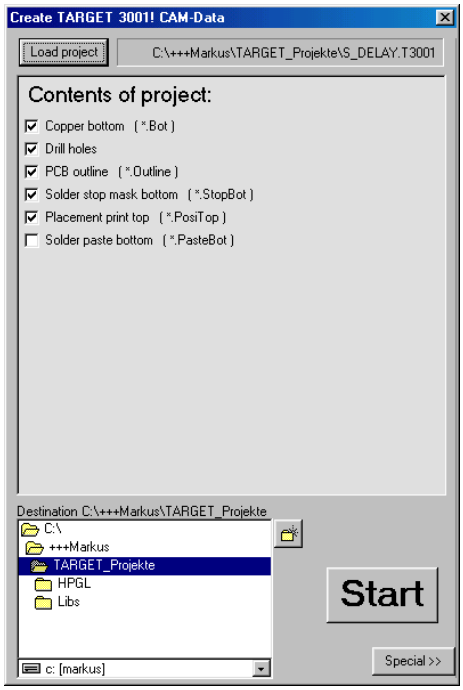
Some PCB manufacturers need the drilling coordinates shown with leading zeros. Set the number of digits here. Default: 5 digits.
NGs separate: a tick here generates an additional drill file for not soldered through holes with the extension *.NG. NGs (not galvanized) are through holes or soldering pads with pin number=0, height=0, width=0 but drilling >0.

10.4 Gerber and drill output

Clicking this function, the following dialog opens:



Initially you are requested to load a project. After clicking "Open" , the following picture could appear:



Select a target directory (dialog bottom left), in which the XGerber-data shown are saved and press START. The XGerber-data are now generated to a target-directory. With button "Special" kyou may first define further specifikations. Transmit the data now to your PCB manufacturer.

Attention: If you are PCB manufacturer working with TARGET, show the plots to your customer for having them confirmed.

The specialty of this XGerber driver is, that XGerber data are generated but they can be read from "older" Gerber machines because only "old" Gerber commands are used. For example Gerber aperture lists are furnished, redundant for CAM-companies working with XGerber, but necessary for CAM plants using Standard Gerber.

10.5 Post-Script driver

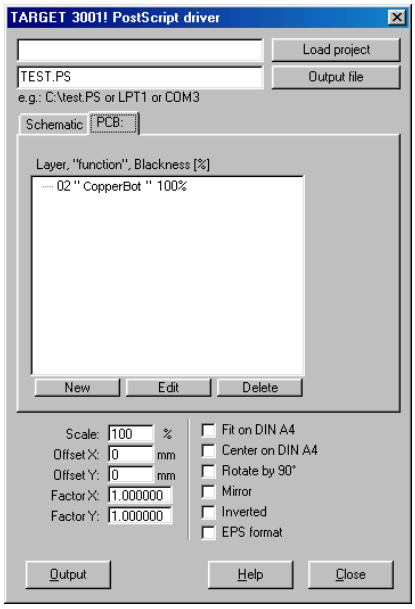
Allgemeines

PostScript is a language for description of pages initially developed by Adobe Systems Incorporated, USA for printer's industries and for desktop publishing (DTP). Fortunately it does not matter whether it prints letters and pictures or soldering pads and signal tracks on paper (or film).

Take your PostScript-file to a local printer or an exposure studio and have a high quality film (in most cases over 1000 dots per inch) made out of your PCB layout.

PostScript offers convincing advantages: high resolution, sharpest outlines, high contrast, quick done and moderate price. Additionally it is possible writing with"white ink". So constructions using deletion layers (HF-Technique) are possible. Also the generation of open drill holes are possible (contrary to the results of several plotters). On documentation reasons the layers can be put out black and white in steps of grey. Thus in the layers dialog additionally to "visible" a value of grey for every layer has to be set (0.00 is black, 1.00 is white). Normal layers must be set 0.00 for black, deletion layers and open drill holes must be set 1.00 for white! All intermediate values defining grey are allowed.

Export PostScript-files



- Save your project in "TARGET"
- Open the PostScript driver in menu "File/Input-Output Formats" .
- Load your project by click on "load project".
- Select "output file" and define a file name.
- Here you can redirect an output directly to the interfaces existing.
- The options "Schematic" and "PCB" are for setting the desired output views. Button "new" sets the corresponding PCB-layers.
- Choose the desired parameters. If you are insecure take the default-settings by clicking "OUTPUT".
- For to insert a PostScript-File into other PostScript-projects activate "EPS-Format"
- Click on "OUTPUT"

10.6 DXF-driver

General

DXF (Drawing Interchange Format) is a format for data transfer between different CAD programs from company Autodesk, Inc.

TARGET saves all details of the schematic and the PCB-layout whithin an internal data structure. DXF format gives opportunity making those internal data available and visible that way that they can be read and computed by other systems.

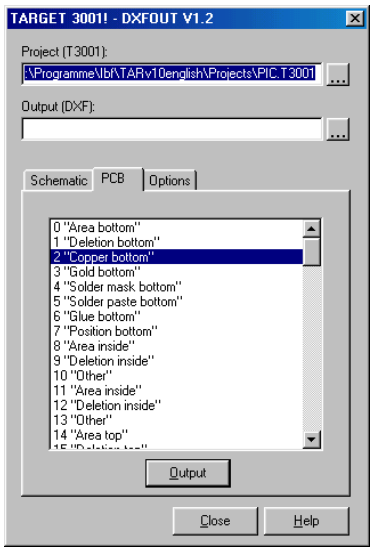
Two conditions must be fulfilled. The other system must be able to

- read an ASCII file
- load and analyze lines in pairs out of this file.

Unfurtunately the data transfer using DXF is problematic because of the different qualities of the DXF converters.

- DXF ist not normative, it has grown historically and is growing yet
- DXF, as far as we know, is not valuable documented
- All coordinates and values of length in a DXF file are independent from units. The system of measurements used in the drawing (millimeter, inch, ...) is not part of the information given.
- DXF is useful for drawing lines but not for drawing filled areas.

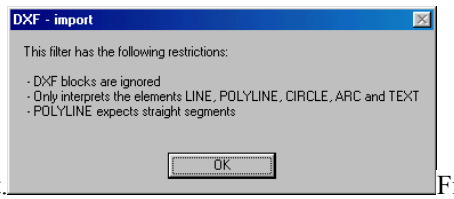
Export DXF-file



- Save your project in "TARGET"
- Open the DXF driver in menu FILE/INPUT-OUTPUT-Formats
- Select a TARGET project
- Click on register "OPTIONS"
- Select the desired parameters. If you are insecure leave them as they are set default (no box activated).
- Click on register Schematic bzw. PCB, and select the desired schemstic page or the desired PCB layers.
- Enter a file name to "Output (DXF)
- Clicking "OUTPUT" your file is generated.

Import a DXF-files

- Open TARGET and choose the schematic view if you want your DXF-project as part of the schematic.
- Open TARGET and choose the PCB view if you want your DXF-project as part of the PCB layout



- Choose in menu FILE/ Import-Export-formats and then DXF import. First a dialogue declares few restrictions of the DXF import. Accept those and press OK.
- Choose now by the windows-dialog the desired dxf-project and click on "Open"
- A further dialog-window enables you to fix import-options before . Set the scaling, define the layer to which the DXF-Project shall be inserted or define whether polygons shall be shown filled or not.
- Press "START" for activating the import..

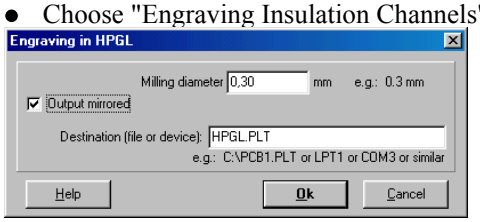
10.7 Engraving insulation channels

General

TARGET calculates the outlines of the signal tracks and generates them in HPGL-format. Those milling data can be read by CNC milling machines for the production of samples, prototypes and small series.

Generation of your Project in HPGL

- The PCB view in the current window is going to be milled. So please set only the referring layer visible (for a single sided BCB only copper bottom and possibly the outline (Layer 23)). The other layers (soldermask, padnumbers...) need to be set invisible (hook off)
- Choose "Engraving Insulation Channels" and "HPGL" in menu "File/Import/Export Formats. The following dialog appears:



- Enter the name of the HPGL target file with extension .PLT and the diameter of the drilling tool in mm.
- "Mirrored" is set as a default e.g. for to have copper bottom milled in mirrored manner.
- Press OK -button

TARGET generates the desired File in HPGL- format. The following pencils are used:

- Pencil 1: PCB outline
- Pencil 2: Outlines of signal tracks
- Pencil 3: Drill holes

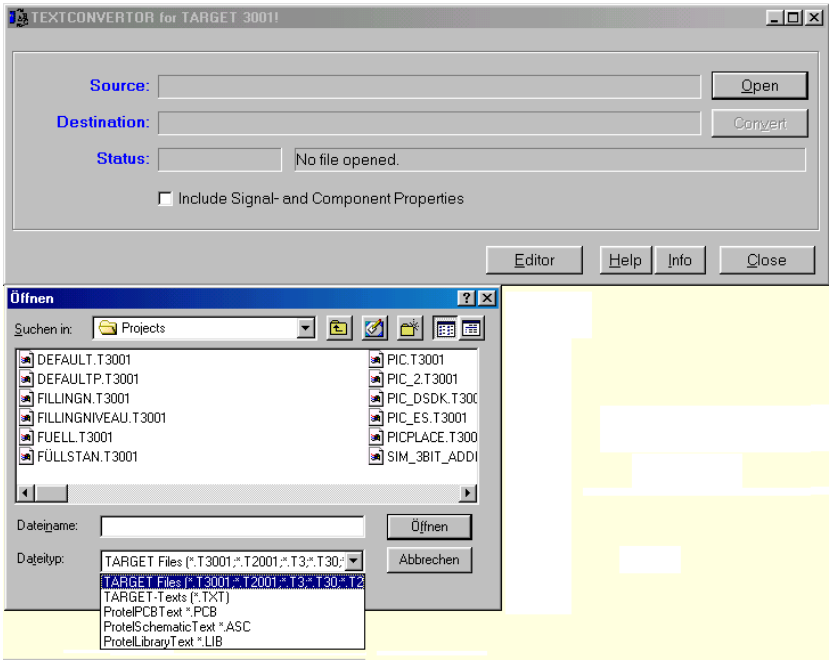
Please note: Ground planes are not supported. Areas currently cannot be milled off (set free).

10.8 ASCII- Interface

General

Using this tool you can convert your TARGET file into standard ASCII - format. You may develop individual automatisms and tools

Start this function in menu File/ Input/Output Formats/ASCII Interface. Clicking the button OPEN you have the opportunity to read in TARGET-files as well as text files e.g. Protel Text files.



You can generate TARGET- files as well as text-files using the button "Convert". After using the Button "Editor" a text editor opens with the contents of the whole ASCII file.

Description of the ASCII file structure

Comments are written in braces e.g. **{this is a comment}**

General Parameters:

- **Layer/Page:** 0..99: PCB layer, 100: on all copper layers, 101..200: schematic pages, 101 = page 1 , 102 = page 2 etc..
- **ComponentNo.:** Number of the component (WORD = 2 Byte)
- **SignalNo.:** Number of the signal (WORD)
- **Invisible:** Element visible/invisible (0/1)
- **X1:** X-Coordinate in nanometer (LONGINT = 4 Byte)
- **Y1:** Y-Coordinate in nanometer (LONGINT = 4 Byte)

A command ends with end of line (ASCII 10 = return).

Parameters are divided by tabs (ASCII 9 = tab).

Coordinates, width,... are to be defined in nanometers.

Commands:

- **BEGIN:** New TARGET project
- **LINE:** insert signal track or line into schematic
- **PAD:** insert pad or dot in schematic
- **CONNECTION**: insert connection
- **TRIANGLE:** insert triangle
- **RECTANGLE:** insert rectangle
- **TORUS:** insert torus
- **CIRCLES:** insert circles
- **TEXT:** insert text
- **FIDUCIAL:** insert fiducial
- **COMPONENT:** insert component
- **SYMBOL:** insert symbol
- **PACKAGE**: insert package

- **SIGNAL:** insert signal
- **SPLINE:** insert Bezier-curve
- **LAYER:** edit layer (function, color, invisible)
- **END:** save project in T3001-file and close (no parameters)

Commands in detail

COMMANDNAME:
parameter(1).name , meaning , (Pascal)type
parameter(1).name , meaning , (Pascal)type
.
.
.
parameter(1).name , meaning , (Pascal)type

BEGIN:
OnlyPCB , without/with schematic (1/0) , Boolean

SIGNAL:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
name , Name of the signal e.g GND , String[80]
property1 // not set
property2 //not set
property3 //not set

COMPONENT:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
searchname, searchname in library, string[80]
value, componentvalue, string[80]
praefix , presyllable , string[80]
library , DOS-filename , string[80]
packagename, default package, string[80]
packagelibrary, DOS filename of library, string[80]
number , componentnumber, Word
numberofpins, amount of really existing pins, Word

PACKAGE:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter

packagename, really used package, string[80]
packagelibrary, DOS filename of library, string[80]
numberofpads, number of really existing pads, Word
mirrored, top/bottom (0 / 1) , Boolean
direction , angle, single

SYMBOL:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
name // set by reorganisation
Suffix, string[80]
number, Symbolnumber, Word
insert, next/must/request (0/1/2), 0..2
swap, 0 = no swap posible, Word

CONNECTION (PIN):
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
length, connection length, integer
function , e.g. input , 0..8
swap, 0 = no swap posible, Word
inverterpoint, , Boolean
clockarrow, , Boolean
shownumber, , Boolean
referencepin, , Boolean
AnschlussNr, AnschlussNummer, Word
Richtung , Winkel , single
SymbolNr , SymbolNummer, Word

LINE:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
X2 , EndPoint (X) , LongInt
Y2 , EndPoint (Y) , LongInt
width , linewidth, integer
aura , width of aura , integer
hatched , , Boolean
roundet , , Boolean
symbolno , symbolnumber, Word

SPLINE:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter

signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
X2 , endpoint (X) , LongInt
Y2 , endpoint (Y) , LongInt
X3 , attractionspoint (X) , LongInt
Y3 , attractionspunkt (Y) , LongInt
width , width of spline , integer
aura , width of aura , integer
symbolno , symbol number, Word

PUNKT:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
dotform , round /octagonal/rectangular/oval , 0..3
width , width of dot, integer
hight, height of dot, integer
aura , aura of dot, integer
drillhole , integer
widthoffset, , integer
pinno, number of pin, Word
direction , angle, single
symbolNr , symbolnumber , Word

TRIANGLE:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
x2 , , LongInt
y2 , , LongInt
x3 , , LongInt
y3 , , LongInt
symbolno , symbolnumber , Word

RECHTECK:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
width , width of rectangle, LongInt
height, height of rectangle, LongInt
direction, angle, single
symbolno , symbolnumber , Word

TORUS:

layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
y3 , center (y) , LongInt
radius , radius , LongInt
width, width of torus , integer
aura , , integer
start, startangle, single
end , endangle , single
symbolno , symbolnumber , Word

CIRCLE:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
radius , radius of circle , LongInt
symbolno , symbolnumber , Word

TEXT:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameterInhalt , Text , string[80]
hight, hight of fonts , integer
width, width of fonts, integer
bold, % of width. bold is more than 11% , 0..100
fixed, fixed/dynamic, Boolean
mirrored, , Boolean
function , normal/component/value/symbol/signal/connection/type/itemNo/manufacturer/
distributor/orderNo/purchasing price , 0..10
connectionno , Connection number , Word
fontcharacter , "" = vectorfonts, string[80]
direction, angle, single
symbolno , symbolNumber , Word

FIDUCIAL:
layerside // see 1.) - gen. parameter
componentNo // see 1.) - gen. parameter
signalNo // see 1.) - gen. parameter
invisible // see 1.) - gen. parameter
X1 // see 1.) - gen. parameter
Y1 // see 1.) - gen. parameter
form , targetiel/arrow/two fourth, 0..2
radius , fiducial radius , integer
bold , , integer
direction , angle , single
symbolno , symbolNumber , Word

LAYER:

layerside // see 1.) - gen. parameter
layerfunction , e.g. copper bottom, Byte
layerrelation, which other PCB layer is the layer related to, Byte
layercolor, color of the elements on this layer , 0..15
layervisible , visible/invisible (1/0), Boolean

10.9 Converter TARGET 2.1 to TARGET V3

Not for English versions!

10.10 Converter TARGET V3 to TARGET 3001!

Not necessary for English version

10.11 Read in EAGLE files and EAGLE libraries

General

You can import by program EAGLE3.ULP (ULP=User-Language-Programm) Eagle 3.5 files 1:1 into TARGET format or by programm Eagle4.ulp Eagle files younger than Eagle version 4 (included) into TARGET format. Both programs you can find on your CD in the directory "Drivers" or you may download them free of charge from the internet http://www.ibfriedrich.com/english/download.htm within the download file "drivers.zip" .

Read in Eagle-files (Schematic and Board-files)

- copy the file **EAGLE3.ULP** or **Eagle4.ulp** into the directory your Eagle files are in.
- StartEagle 3.5 oder 4.*.
- open the Eagle project to be converted
- enter to the command line **RUN EAGLE3.ULP** or **RUN Eagle4.ulp** . The converter automatically generates a text file in ASCII fomat (*.TXT)
- start programm " ASCII I/O interface" within program group TARGET
- clich in the opened dialogue on button "Open"
- set file type to "TARGET TextFile" and choose in "file name" the desired ASCII file.
- click on button "Convert"
- set in "file type" "TargetFile" and choose the corresponding TARGET-T3001 file name.

Conversion of EAGLE libraries

- In the Eagle control panel first open the library you want to convert under File/Open/Library e.g. 41xx.
- now save the selected library in *.scr format to a directory of your choice.To do so choose in opened library File/Export/Script.
- name it e.g. 41yy.scr and press button "Save"
- start TARGET and choose File/Input output formats/Import Eagle library as script...
- select the generated .scr file and press "Open"
- the conversion now is made and you receive an dialogue telling this.

Two files have been generated to the same directory, one with extension .SYM3001 for the symbols of the components the other .PCK3001 for the packages.

- now copy both of the files into the directory of your symbol and package directory. Normally the directory "TARGET". If you'd prefer another directory you must tell TARGET the path to it under "Window/Settings/Options" within the dialogue window bottom left.

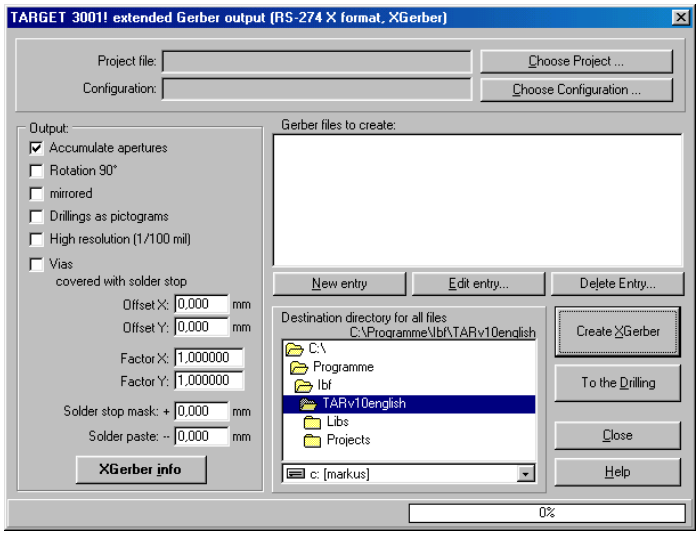
- Eagle is a registered trade mark of Fa. CadSoft, Pleiskirchen, Germany.

10.12 XGerber-driver

General

XGERBER format RS-274-X is the new variant of the Gerber format and is used increasingly. The greatest advantage of XGerber is the fact that in the header aperture information is provided. Additionally in TARGET XGerber format a copper layer contending ground areas can be put out within one file. This sandwich made out of three TARGET layers **(Area-, Deletion- and Copper) fits into one file**. Ask your PCB manufacturer if he is able to read in XGerber.

XGerberdatenerstellung



Generate XGerber as follows:

- Draw a PCB in TARGET and save it. Write in the layer **Copper Top** the word "**TOP**" and in layer Copper Bottom the word " **BOTTOM**" in mirrored manner, thus beeing readable on bottom view. Internal copper layers should be numbered by K1 to Kx, if they are used.
- Choose Options "Input / Output Formats" in menu FILE. Click on "XGerber output (RS 274 X)". The Gerber driver is automatically started.
- Click on "select project" and select it.
- If available and useful for your purposes load a configuration file. It contains information on the fact which TARGET layers are quoted in which Gerber files. Otherwise define by "new entry..." individually a Gerber file to be put out. Choose a name for the gerber file and define, Which TARGET layer(s) should be mentioned in it. Possibly it makes sense to activate layer 23 "outline" in combination with the gerber file to be put out. So you generate a relation frame for easier viewing. The following configurations are provided by TARGET: DSDK-LSL.GC, DSDK-without -LSL.GC, DSDK-MASSE-LSL.GC, DSDK-MASSE.GC, 4MULTI-MASSE-LSL .GC (ground on inner layers). Maning: DSDK = double sided, plated through, LSL = solder mask (dielectric), MASSE = PCB with ground plane 4MULTI = four layer multilayer
- Naming the gerber files please use sense making names, best using not more than 8 digits for to make sure that your PCB manufacturer is able to compute your files.If you want to edit the configuration settings please use "edit entry" or "delete entry".
- If the remaining settings are ok to you, select "Generate XGerber". The Xgerber files are now generated within the desired directory.
- Die Gerber-Dateien werden in das unten angegebene Zielverzeichnis geschrieben.
- Tip: Save your Xgerber data on harddisc before copying to floppy disc.

- **Accumulate apertures** means that every file begins with a compatible set of apertures. This option ist advantageous, if a manufacturer gathers problems that a project D11 in its first XGerber file has a different meaning than D11 in the second file of the same project. Else the files only get bigger.
- By **Rotation 90°** the output is rotated by 90°
- By **mirrored** kan output turned over can be chosen.
- **Drillings as pictograms** corresponds to the same export-driver in TARGET. For every drillhole with different sized diameter a

little symbol is generated (Piktogramm). Its for better view when drilling by hand. See also chapter .

- **High resolution (1/100 mil)"** for higher resolution
- by"**Offset**", an +/-offset can be set,
- by **"Factor"** you may adjust the output.
- **"Solder mask+"**
- **"Solder paste--"** diminishes the solder paste a little for SMD-components.
- **"Vias covered with solder stop"** makes sure that no solderable areas are left when plating through. Nevertheless for every via a little cap for the drill hole is generated so that the solder mask fluid does not inflict the via when printed by silkscreen.

Attention: Generating Gerber and XGerber data and subsequential computing on films or further manufacturing media is not trivial (CAM-Prozess). Please check in any case your data before ordering films, silks, stencils or PCB. It is useful loading "GC-Prevue" (a Gerber viewer) from our homepage www.ibfriedrich.com.

Order a copy of the plots for correction reasons.

If this all seems too complicated to you, let your PCB be manufactured by an expert manufacturing by directly using your TARGET *.T3001 files. Manufacturers doing so are listed in the program TARGET under PCB service or visit our homepage http://www.ibfriedrich.com/service/services.htm.