



TARGET 3001!

Crash-course

Ing.-Büro FRIEDRICH
Am Schwarzen Rain 1
D-36124 Eichenzell
Germany
target@ibfriedrich.com
www.ibfriedrich.com
Tel.: ++49 - 66 59 / 919 444
Fax: ++49 - 66 59 / 919 445

Crash course

From TARGET_E

Contents

Introduction

Target 3001! is an object focussed schematic-, simulation-, and PCB CAD-software for PC in English, French and German language. It is a partly automatic tool, supporting you when drawing schematic, running a simulation, and laying out the PCB including an EMC-analysis (electromagnetical compatibility). TARGET manages schematic, simulation, PCB and EMC-check as one so that modifications are implemented simultaneously within the other part or are at least considered. Each TARGET project refers to one single database. No error causing conversions of different data formats!! TARGET is a worldwide unique software system combining the above mentioned properties within one user interface. TARGET supplies enlargeable component-, model- and package libraries. It is able to create a bill of material and netlists. Components can be placed by hand or automatically. They can be edited at every time and easily be displaced after selection. You can route your PCB on screen interactively or use an autorouter. Your PCB can be transformed into CAM-data with the help of the numerous export drivers.

A free version for discover can be downloaded from www.ibfriedrich.com (<http://www.ibfriedrich.com>)

Conventions for this Crash Course

[x], [Alt] expressions in square brackets mean "press key" on the keyboard or press a button in a dialog.



pictograms mean icons on the toolbar.

"Abcd" expressions in quote mean dialogs, menus, menu commands or entries in pull down menus (e.g. name of component in a component library).

Mx Expressions beginning with **M** are **mouseclicks**


Dialog: the word "Dialog" in front of an expression means the header of a window or a dialog field

→ Single arrow means: "press...!", "switch...!", "choose ...!"

⇒ Double arrow means: it follows ..., it arises...

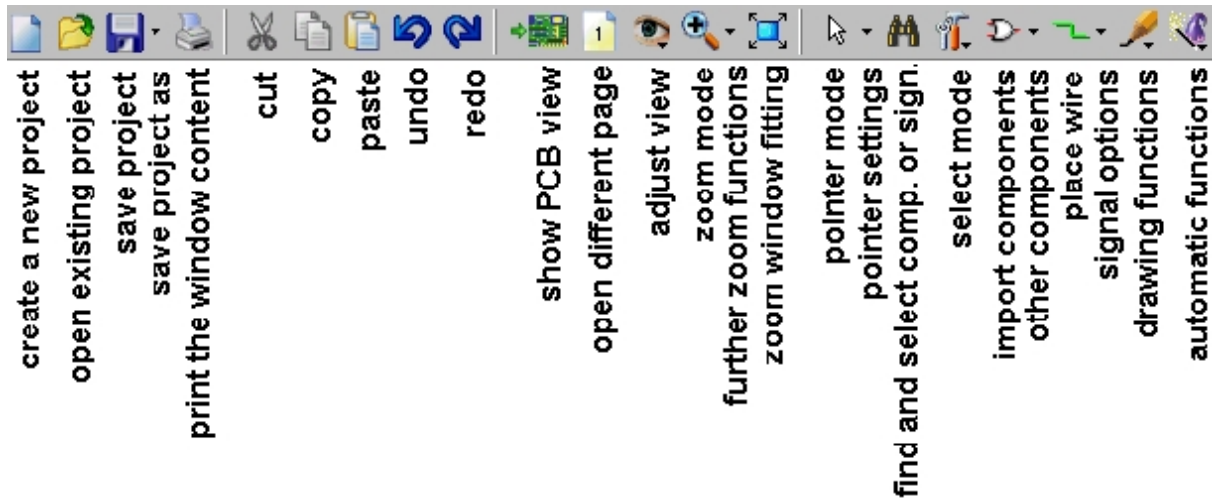
Example: ⇒ [o] ⇒ **Dialog:** Options ⇒ Track width 0.3 mm

Means:

Press key , the dialog "Options" will open, set track width to 0.3 mm.

After starting TARGET 3001! the start assistant opens. With its help you can open an already existing project or start a new project (with or without PCB). After selection the "schematic mode" is shown. You can see the header, the menu line, the toolbar, the working area (canvas) and the status line. If you just move the cursor over a button without clicking, an info flag is shown describing the function of the tool.

The buttons of the toolbar in the schematic mode



Tool extension: ▾

Tools with extensions offer a selection of further functions, sub-functions, options or entries of parameters.

Examples:



→ **M1** on transistor-symbol ⇒ the main function *import component* is activated.

→ **M1** on ▾ ⇒ further component import functions e.g. *import rests of components* or *insert reference symbols* can be selected.



→ **M1** on signal button ⇒ the main function *place wire* is activated

→ **M1** on ▾ ⇒ a menu of *signal options* opens e.g. *bending mode* or *track width* can be selected.



→ **M1** on wizard symbol ⇒ a pull down menu containing *automatic functions* is opened e.g. *start simulation* or *check project*.

Mousekeys in normal mode (pointer mode)

Left: **M1**, **M11**, **M1H**... Aselection and edition of all drawing elements

Right: **M2** opens a context menu concerning the view of the schematic

Zoom functions:

M22 zoom out (diminish)

M2H and drag right: catch a window and enlarge

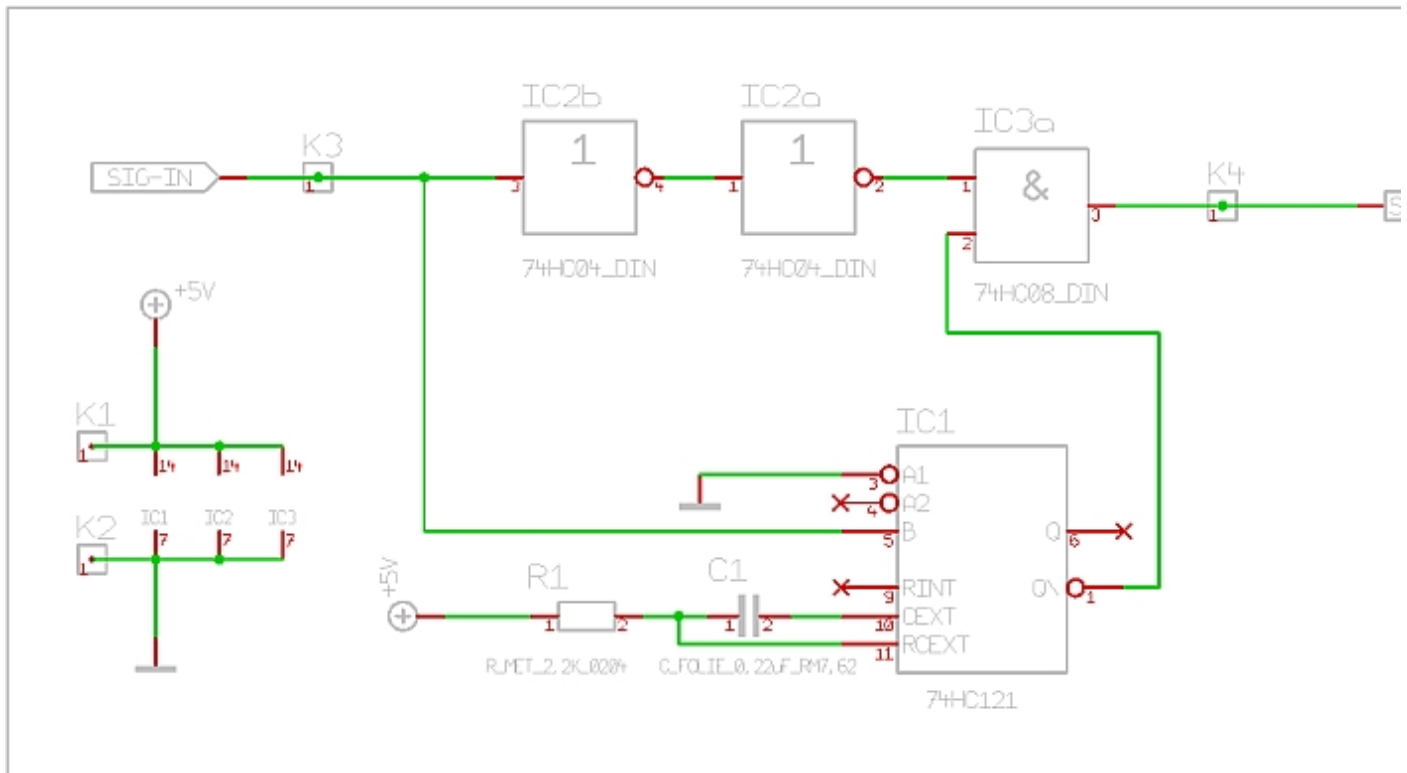
M2H and drag left: the entire schematic is shown fit to screen

Middle: **M3** zoom in (enlarge the area around the pointer)


Left and right simultaneously: **M12** current action is cancelled ([Esc]).

Schematic mode


This chapter will show you by an example, how to draw a schematic. The following schematic of a switch-delay by monoflop is to be drawn:



Start a new project titled: s_delay

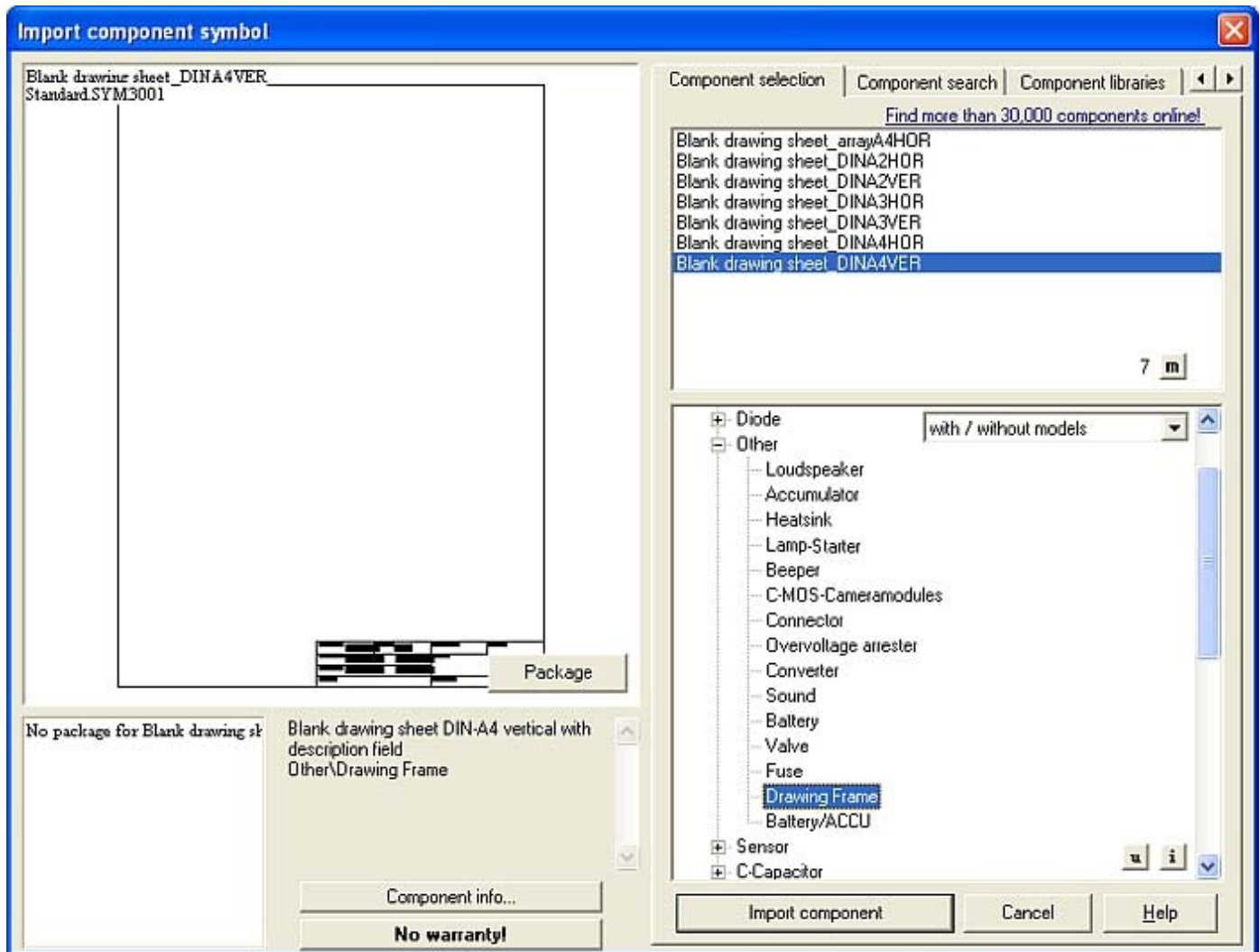
→  ⇒ Dialog: Create new TARGET project. → file name: s_delay . TARGET creates the extension .T3001 automatically. In spite of drawing a schematic, you mustn't activate the switchbox "Create PCB without schematic".
 →[OK] ⇒ **1 Schematic s_delay.T3001 Page1 "<unnamed>"**
 Maximize the empty schematic fit to screen. The area of the opened schematic page (canvas) has the size of 1,44 m² or 5511.58 in². For an easy handling of your drawing...

Insert a frame DinA 4

→  ⇒ Dialog: Import component symbol. The opened window shows:

- To the right the tabs: "Component Selection", "Component Search", "Component Libraries" and "Update". Below a tree view of the component categories.
- To the left is space for the image of the drawing of the symbol.



→ STANDARD.SYM3001 (from library list) or → section "Other" from tree:⇒



→ Drawing Frame → Blank drawing sheet_DINA4VER (from symbol list), → **[Import Component]**

The frame now appears on your screen. It is fixed to the cursor and can be dropped by **M1**. **M12** quits this action. Zooming out by mousewheel makes the entire frame visible on screen. Before inserting components you should...

Switch on grid

→  → , ⇒ Dialog: **Window grid**.

Select "Grid as points" instead of "Grid invisible". The other settings on this window remain as preset. Especially the grid spacing should remain 0,635mm (=1/40" =25/1000" =0,025" =25 mil) because the components of the libraries, are based on this grid. After → **[OK]** a grid of points is visible. If not the scale may be too small. Zoom in by **M2**. The grid will ease the drawing of signal wires and will help you when...

Inserting components

act like described in Section "Insert a frame DinA4"!



NOTE: Shall the matching of your circuit later be simulated please select only components fitted to a simulation-model. Choose in Dialog: Import symbol "only components with models"; then only those components are offered. In this crash course we abandon the simulation so we choose "with / without models".

The components **74HC04_DIN**, **74HC08_DIN** and **74HC121**, the resistor **R_MET_2,2K_0204**, the capacitor **C_FOLIE_0,022µF_RM7,62** as well as the connectors **K1...K4** you will find in the library **Standard.SYM3001**.

The connectors K1 ... K4 you will also find within library Standard.SYM3001 named K1X1 (without simulation model). They will only be used for a later creation of a PCB with connectors (e.g. soldering nails, edge socket connectors, receptacles).


Consider clever placement of the components so that subsequent placing of the signal tracks can be done with a small number of bendings. During the import of a component it will be fixed to the cursor like a phantom. **M2** causes a 90° rotation of components (counterclockwise). After insertion of the symbols/components please insert the symbols for ground (GND), power supply (VCC) and input and output signal. You will get this by...

Import of reference symbols

→  of the button  → [import reference symbol] ⇒ Dialog: Import reference symbol.

The symbols required (GND, V+, SIGN-IN and SIGN-OUT) can be imported the same way into your schematic.

Import rests of components




Some real components are usually not drawn completely in schematics. Especially when drawing digital elements the power supply is for reasons of an easier overview not drawn. But if you want to manufacture a PCB you have to make sure that the corresponding connectors do exist. TARGET offers power supply symbols as "Rests of components" to be inserted into the schematic and to be connected to the reference-symbols (e.g. +5V and GND): →  → [Import Rests of components] ⇒ Dialog: Rests of components. → Select component (ICx,xxxx), → ICxp import =as rest (p stands for power) → [OK] → Place power symbol into schematic.



Displace symbols

Click with **M1H** on the handle (snapping cross) of the component and displace it with mousekey held (drag & drop). If this action does not work, please adjust the pointer mode:

→ Window → Settings/Options ⇒ Dialog: TARGET preferences and options

Activate the option on the right side of the dialog "Component handle active" and deactivate "Component always complete".

You might also reach these options by →  →  and →  Now you can displace single elements of a component (e.g. writings) by **M1H**. (Danger: components now can be torn apart. A warning appears).

For a quick change of pointer mode settings you might fix the toolbar on screen by →  → 

For displacement of several symbols as one, you first have to highlight each of them by [Shift]+[**M1**] or by opening a marking area (catching window) by **M1H**.

Entries of component values

Click M11 on the handle of a component ⇒ Dialog: Change symbols → value: enter any other value (e.g. for C1: 20nF instead of BIPOL)

Connection of components


TARGET 3001! calls connections "signal tracks" (short: signals or wires). →**[place wire]** leads you to this mode and the cursor is shown as cross. The hit of a pin by **M1** is confirmed by a beep and the connection cross fades. A thread is fixed to the cursor now. By **M2** or the spacebar the bending mode during placing wires can be changed. TARGET 3001! itself chooses a name for the signal (see status line). If you don't like this name, change it while placing wires: → **[[M11]]** or → **[o]** ⇒ Dialog: Signal wire options → signal: enter new name. Place all segments of a signal by **M1** and quit every signal with **M12** or **[Esc]**. The latest placed segment can be deleted with **[Del]**. Have you placed all signals, leave the place-wire mode by **M12** or **[Esc]**.

TARGET offers lots of further options and tools for editing signals.

PCB - mode

Now we want to create a PCB from our above drawn schematic. You must now ...

Switch to PCB-view

→  ⇒ 2PCB...s_delay.T3001 Now you see an empty PCB on screen. New buttons are:



to the schematic view (toggle) highlighted component is shown in the schematic



configure layers



import package / else



draw track / track options



select other layer for drawing / use layer x (Left shows the currently used layer)

Show grid

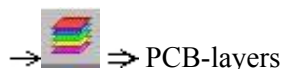
See above. Advice: When layouting PCB the window grid scale is of importance. In most cases a grid of 0,635 mm = 25 mil (=25/1000 inch) is best, because normal spacings of component pins are based on inch scale. E.g. the spacing of two besided IC pins =2,54 mm = 4*0,635 mm =100 mil. Deviate from grid value 0,635mm only for important reasons (e.g. 0,3175 mm, if the track width has to be smaller).

In the grid dialogue you should tick the box: "Cursor snaps on grid".

Definition of the layers

The elements of a PCB (package, signal tracks, drill holes ...) are placed (drawn) upon different layers. All layers are

placed over each other. So what you see in the PCB view is a look through all layers from top to bottom. TARGET offers layers 0 to 99. For reasons of saving screen time you should only fade in the layers in need.




For a usual single sided PCB fade in (tick) layers:

- 2 for copper bottom
- 21 for packing on top
- 23 for drawing the outline of the PCB
- 24 for drill holes
- 26 for soldering pad numbers and
- 27 for air wires

Set the function of layer 16 to "other" (instead of "copper top"). To do so, please use the layer tool and doubleclick the line which shows the layer function. By this means you will avoid placing signal tracks on this layer.

Define the size of the PCB

a) You can define the size by →Menu: "Actions/PCB Outline Wizard" ⇒Dialog: PCB Outline Wizard. Follow the dialog to define the outline of your PCB on layer 23.

If the size later isn't exactly to your wishes, catch the right or the upper half of the PCB with a catching window and move it by **M1H**. Make sure to inactivate the resistor-symbol within the pointer mode toolbar .

You can read the measurements directly within the status line (bottom of screen).

b) As an alternative you can draw an outline → Drawing functions → Draw open rectangles on layer 23 of desired size.

The PCB outline appears as a rectangle, coloured the same as the colour of the outline layer, pink by default. If you do not see anything, maybe the layer 23 is not faded in (by a tick in the colored box of layer 23 in the layer tool). Help: select the complete outline → **[e]** →Change lines set layer to 23.

Define mounting holes

For a later mounting of our PCB with screws we first insert drill holes at the corners of our PCB. We place them by → Menu: Elements→Place mounting drill hole. Drop them by **M1** to an appropriate place on the board. Edit them by following the dialog after **M11** on each of the drill hole.

Import and placement of component packages

The remaining area offers space for import of the component packages corresponding to the symbols used in schematic.

General remarks: First place those components with a fixed (non variable) position to the PCB (e.g. interfaces at the edge of a PCB). Then place the biggest ones (those components of central meaning or those with lots of pins). Last but not least place the smaller ones into remaining spaces. As a help for placing, TARGET shows air wires to the pins of the component most close to when importing or displacing components. This display is computed after every movement and eases a favourable placement of the packages. A crossing of two air wires can often be avoided by rotating one of the packages by **M2** at 180°. Place packages carefully and leave enough space for later placement of the signal wires (on the edge of the PCB as well). In other words : The worse you prepare your PCB the worse signal tracks are to be placed and routed without touch.

Now we want to get the corresponding packages out of the library:

→[Import package] or

→[Ins] or



⇒ Dialog: Import package for

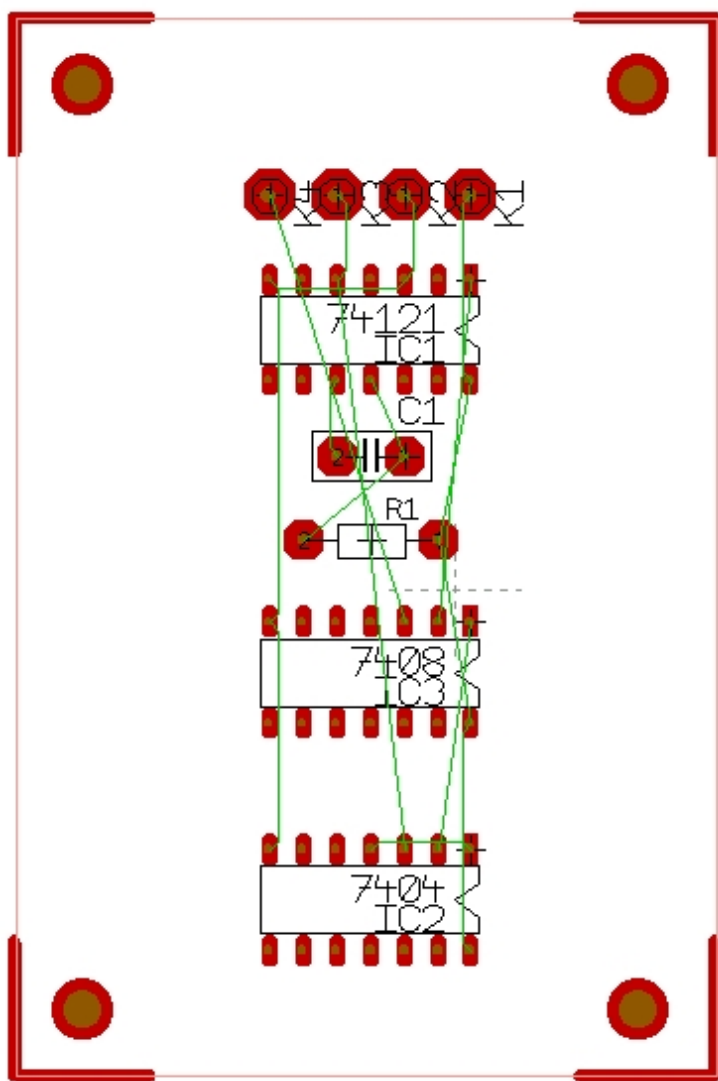
The list at first shows all package proposals. The kind of ranking within this list can be adjusted. Corresponding to the above mentioned rules first select one of the contacts (K1, ..., K4), e.g.:

→K1,K1X1 →[OK] ⇒ select package (vgl. Import component, described as in 1.2)

Within the opened library TARGET 3001! has yet highlighted the recommended package 1X01 . Accept the suggestion by →[OK] and place it by **M1** (e.g. in the corner top left beneath the drill hole) on the PCB.

Place now all packages as suggested each after another K2, K3 and K4 and the ICs 74HC121, 74HC04DIN and 74HC08DIN with DIL14-packages (you can also choose different packages e.g. SMD instead of DIL). You can rotate the selected package by **M2** before placing. Now only the resistor and the capacitor are left. Please pay attention to the real size of the components to be placed and soldered later. The figure x of the package namings Rx , Cx... mean the spacing between both of the connection pins, e.g. R4: 4-times IC-spacing of the pins of the IC (=10,16mm =400mil). According to tis the packages R1 and C1 are only suitable for an upright mounting. Place R und C close to the connections 10 and 11 of IC 74HC121.

Your PCB might look like this now:



Before starting the placement of the signal tracks, please note these ...

Remarks: According to the method of your PCB-manufacturing (etching or milling) special minimum widths and minimum spacings of tracks and soldering pads have to be realized. The following values seem to match well in both of the methods:

Power supply tracks (GND, +5V, Vcc, Us...) should show a minimum width of 1 mm (better more!). Place them first.

All other tracks should show a minimum width of 0,5 mm (better: 1,0 mm). The minimum spacing between tracks and respective to pads (aura) should amount 0,3 mm. The soldering pads of IC's have in TARGET 3001! a standard size of 2,5x1,27 mm. So a spacing of 1,27mm between two pads results. Signal tracks, running between two IC-soldering pads may reach a maximum width of 0,67mm. Other pads, should have a minimum diameter of 2 mm (better 3 mm if space enough), connectors (e.g. soldering nails): 4mm.

Setting of the soldering-pad size

Highlight by **[Shift]+ M1** the components R1 and C1, then → **[e]** ⇒ Dialog: Edit the selected elements → pads → Individual properties for the selected →[OK] ⇒ Dialog: Change pads → Pad height: 3 mm → Pad width: 3 mm → Drill hole: 0.9mm →[OK]. Don't forget to tick the options!

Adjust the pads of K1 ... K4 to a size of 4mm with drill hole 1.3mm.

Note: According to this method you can edit any element of your project (lines, texts, signal tracks, pins...)

Placement of signal tracks

The placement of signal tracks on PCB by hand is corresponding to the placement of wires in the schematic mode. The air wires show you the pads to be connected.

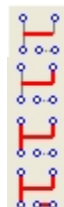
Start with signal +5V or GND: → **M1** Draw track → **[o]** ⇒ Dialog: Track options → Track width: 1.2 mm → track aura: 0.3 mm, 1 layer: 2 (copper bottom) → **[OK]**. Now highlight contact K1 by **[[M1]]** and (see chapter 1.9)

After placement of a signal tracks, the corresponding air wire fades. If the pad isn't hit in the right matter the airwire won't fade. In this case delete the signal segments starting from a pad and draw them new. As far as the space on your PCB is enough draw all tracks using the same width. If there are narrow spaces choose a smaller width.

Adjustment of signal tracks

Already placed tracks can be edited afterwards.


Those three buttons in this respect have a special meaning. You find them → [Pointer mode settings] (see chapter 4.6):



Select the desired highlighting mode and highlight the signal tracks you wish to change by **M1** or **[Shift]+M1**. Act now like described in chapter 4.6 . Single segments can directly be edited with **M1** or displaced by **M1H** (e.g. for getting more space for placing signal tracks). Do you suffer problems, place the cursor close to the worrying element and press **[s]** as long until the desired element flashes. Press enter to highlight it finally. With **M1H** you can open a catch-window above all desired elements. They get hightlighted that way.

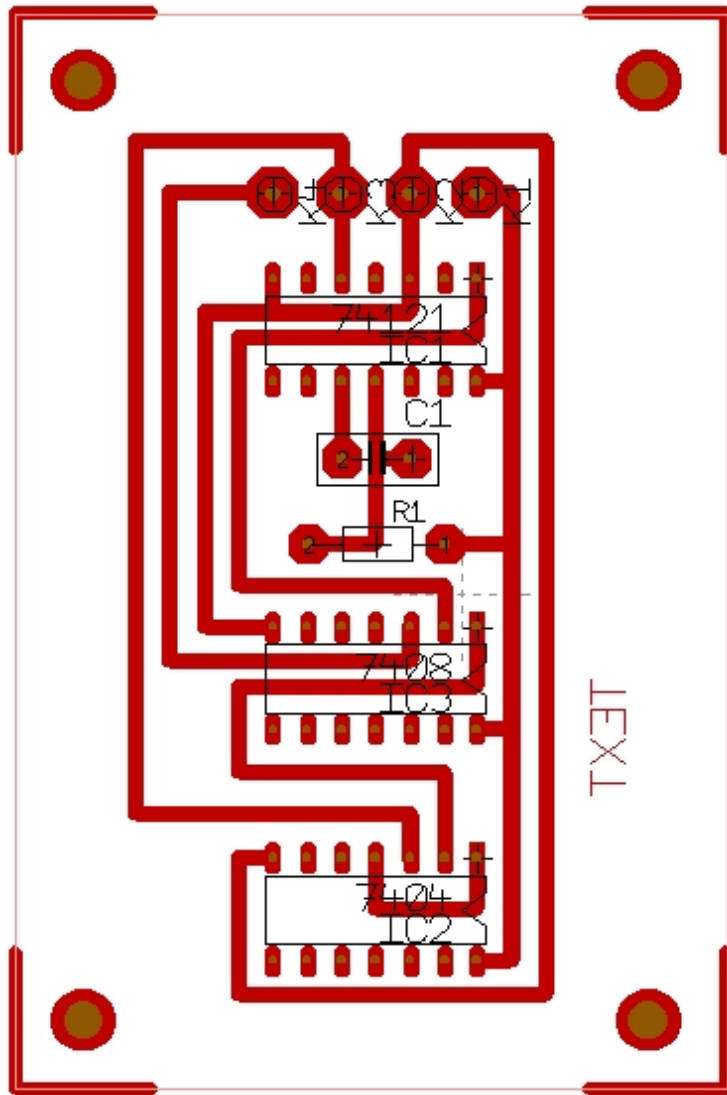
Insert writings upon your PCB

You can easily insert writings upon any layer e.g. to point at certain connections or to drop other messages (data) on the PCB. On a single layer PCB as we have it here, only the copper bottom layer is used. So we drop our writings on this layer:

→ [Drawing functions] → [Place texts] 2cursor transforms into  **T**
 → **M1** on this point where your text shall begin ⇒ Dialog: Text options → insert desired text → font height: min 2,5 mm → font width: about 2/3 of the height → layer: 2 → activate "mirrored"!!!

Check by the colour (normally dark red) whether the text really is placed upon the copper layer.

Did you work out all the mentioned actions, your PCB could look like this:



Check project

Now you can make TARGET check your project (schematic and PCB) concerning the design rules (Design Rule Ckeck). It is useful checking especially the track width and the spacings between the tracks and pads, as well as the check concerning not connected signals and short circuits (crossing tracks).

→ File → Check project... → Dialog: check project change only the parameter Track width to 0.5mm and let all other parameters remain at predefined value.

The spacing between each of the signals and the thickness of the rest ring at pads should be bigger than 0.3mm. But especially when using IC's lower values can often not be avoided. Start your check with →[OK] ⇒ project checked. The appearing list shows all errors and additional results of the check. For interpretation of the errors click **M11** upon the purple coloured error flags (visible as far as the error-layer 29 is activated). Inactivate layer 29 so the error display is switched off.

Output of a file for engraving isolation channels in HPGL-format

a) →File →Input-/Output-Formats → Production → Engraving isolation-channels ⇒ TARGET

3001! isolation milling

b) Follow the dialog to be guided through the process (double sided layouts work very convenient also).

Output of GERBER Data

a) →File →Input-/Output-Formats → Production → (X)-Gerber and drill output PCBout...

b) Follow the dialog, the Gerber data are generated to a directory of your choice.

That's it.

If you have further questions, feel free to contact us.

TARGET 3001!

Ing.-Bueru FRIEDRICH

Am Schwarzen Rain 1, D-36124 Eichenzell, Germany

target@ibfriedrich.com, www.ibfriedrich.com

Tel.: ++49 (0)66 59/919 444, Fax: ++49 (0)66 59/919 445

Retrieved from "http://www.friedrich-eichenzell.de/ibfwiki_e/index.php?title=Crash_course"

■ This page was last modified 10:35, 30 January 2007.