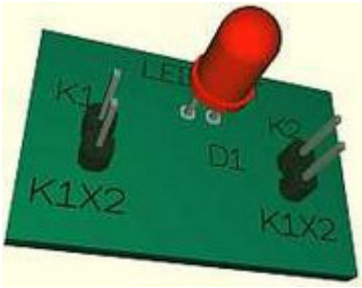
TARGET 3001! CrashCourse2

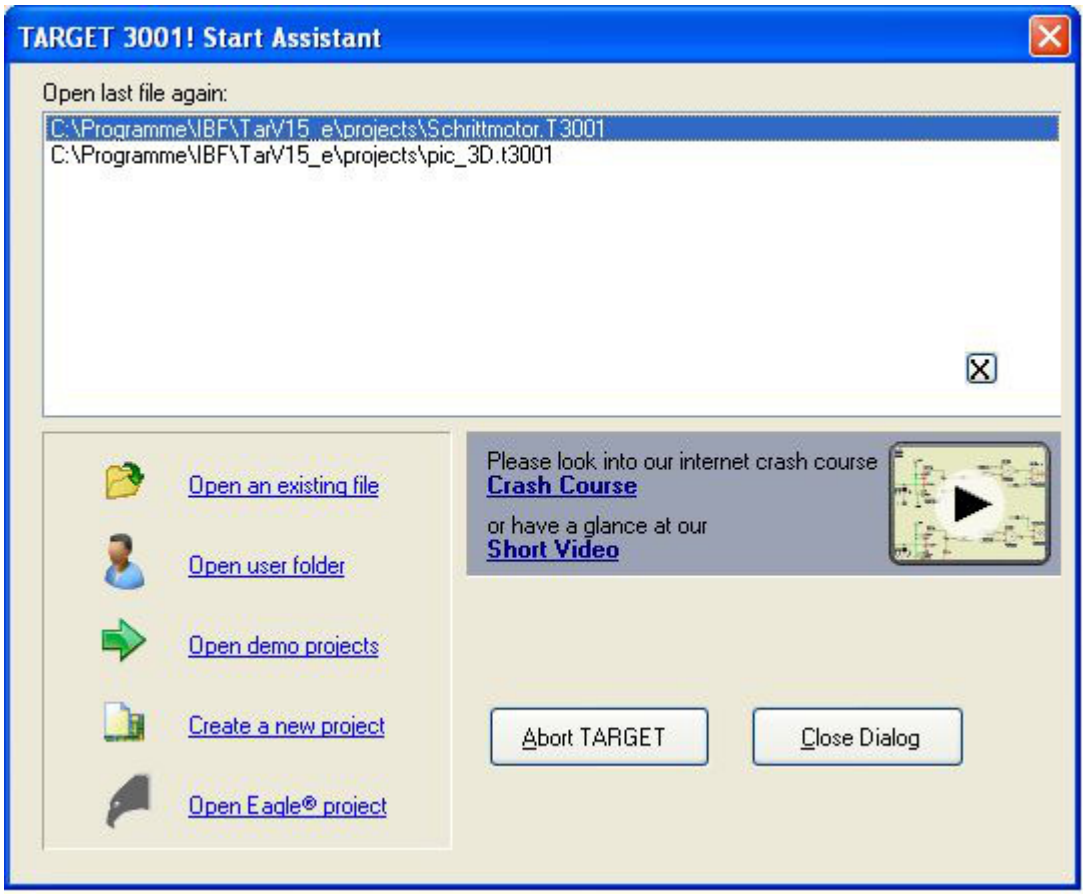


Design a LED between two connectors in minutes!

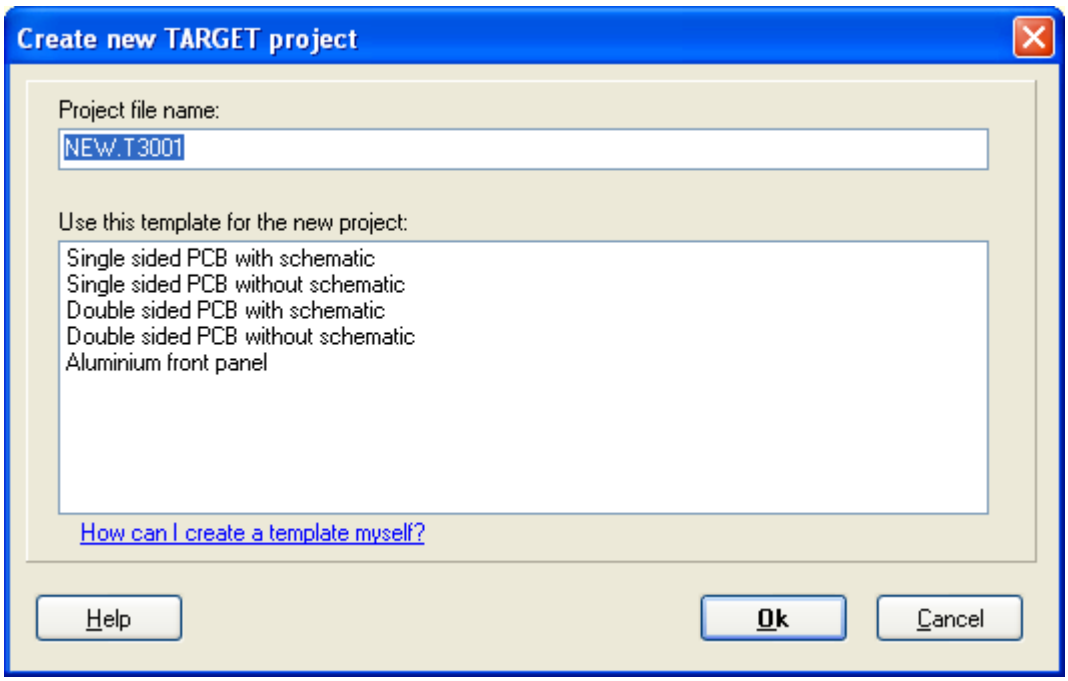
Based on this small project see how easy the use of TARGET 3001! is.

- Begin a new project
- Import a component symbol to the schematic
- Connecting the pins
- Define a PCB outline
- Import matching packages (footprints) to the layout
- Place tracks
- Generate a groundplane
- Simulate the function Part 1
- Simulate the function Part 2
- 3D-view of the layout
- Produce a PCB
- Design and produce a frontpanel

Begin a new project



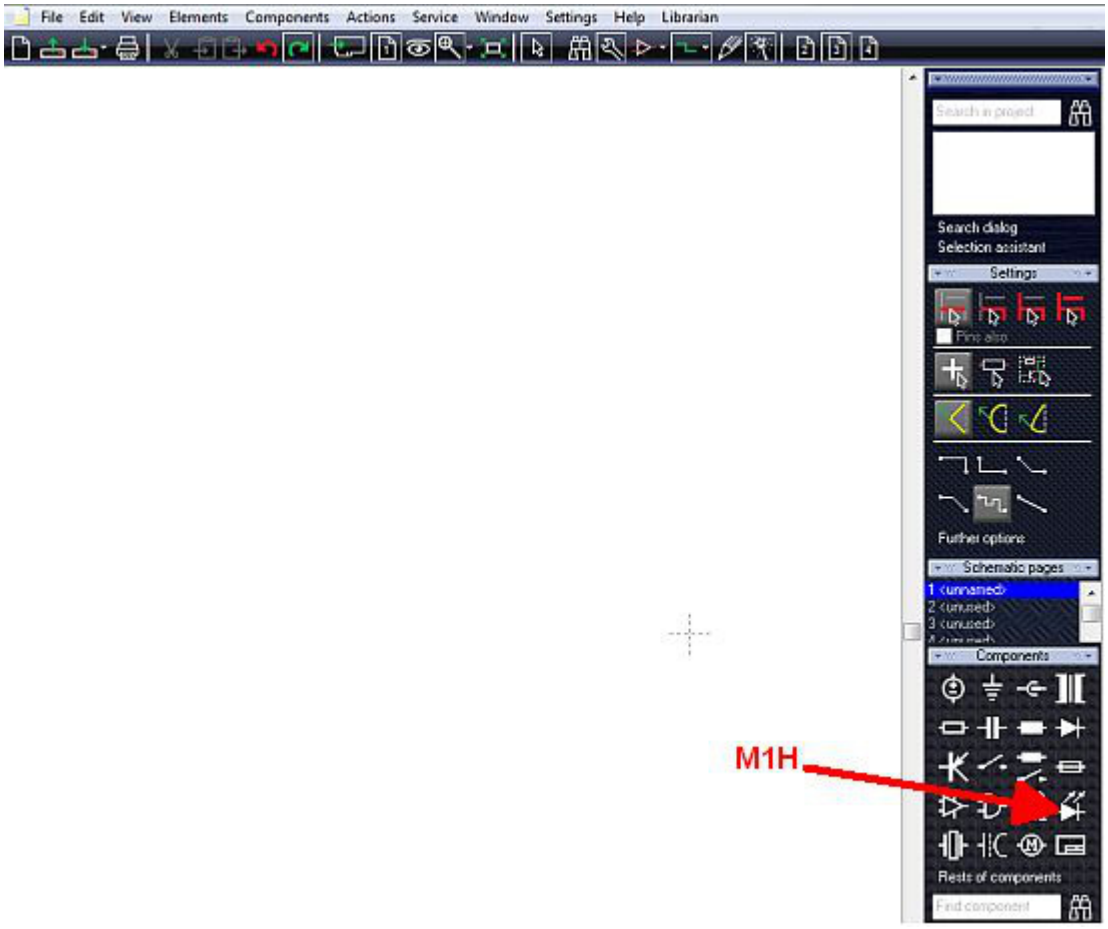
Using the Start assistant you can select one of the 30 recently used projects from the list. You can open a different already existing project, a certain user folder or a demo project. Sure you will be able to create a new project. If you click this fourth option, a dialog opens which allows selection from various project templates:



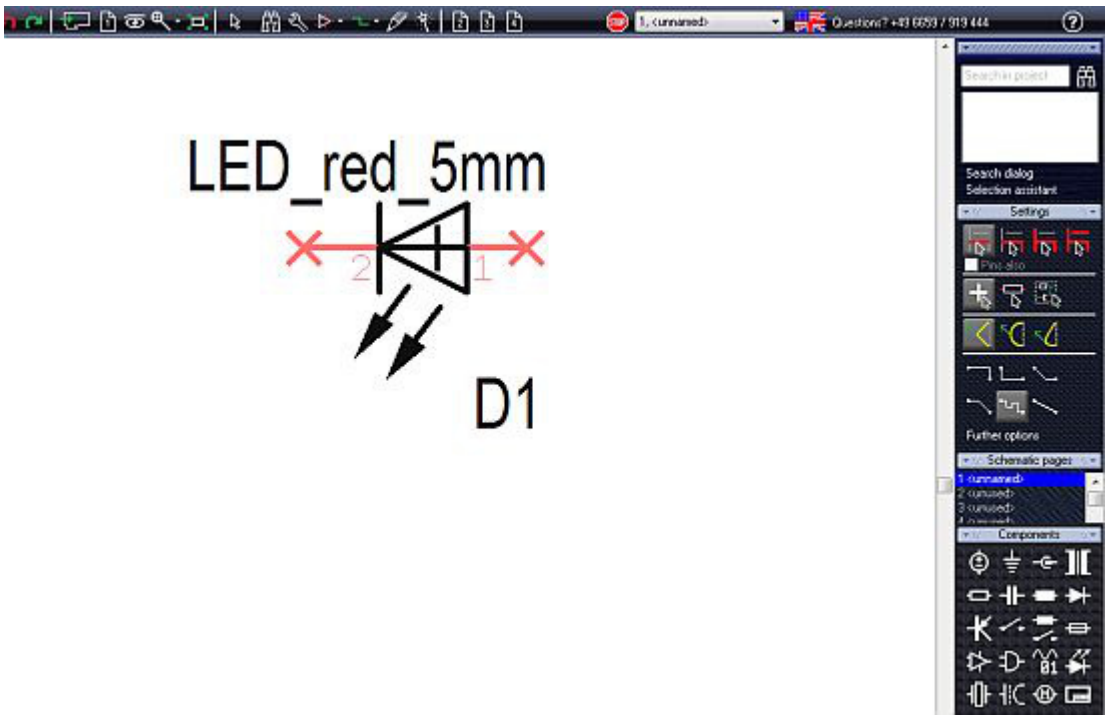
In our case we decide for **Double sided PCB with schematic**. An empty schematic page opens...

Import a component symbol to the schematic

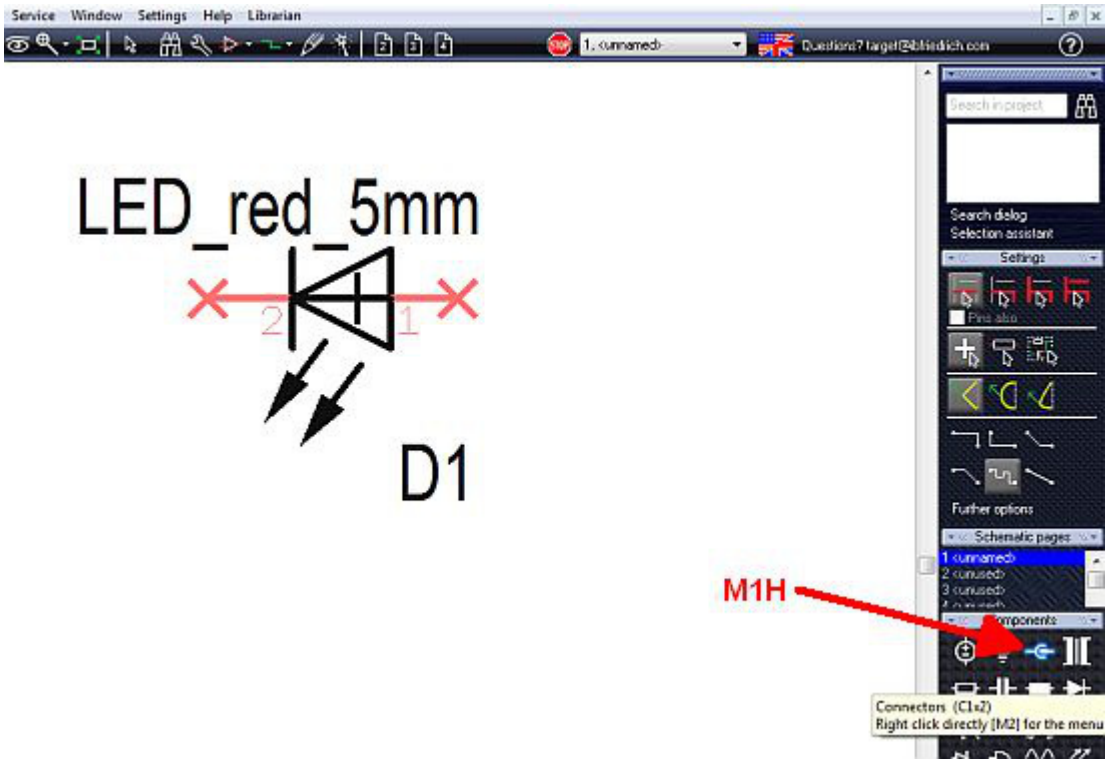
The following image shows an empty schematic page. Now we import a symbol from the sidebar. Click **M1H** upon the symbol pictogram of a LED and bring it into the schematic by drag and drop.



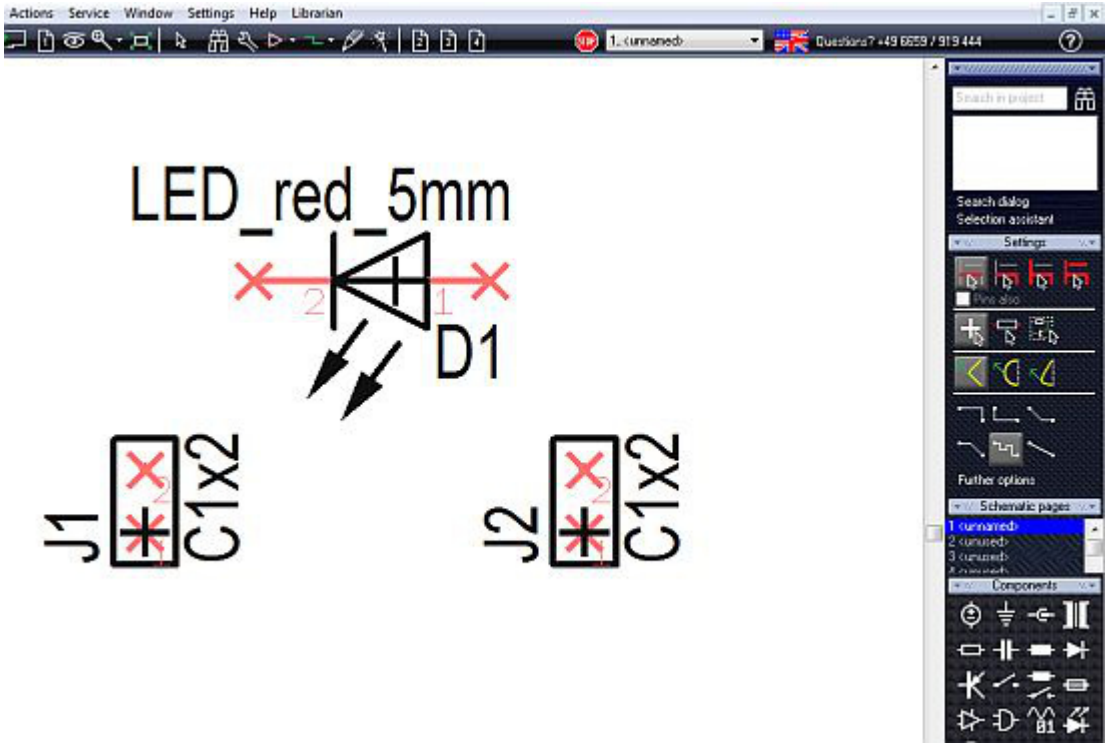
A click **M1** on the pictogram will open the component browser to import further parts of this component family.



In case the component value appears too big, you might change it by M11 upon it's handle cross. The appearing dialog allows to modify the entries. "LED_RED_5mm" stands for Component Value. "D1" stands for "Component Name".



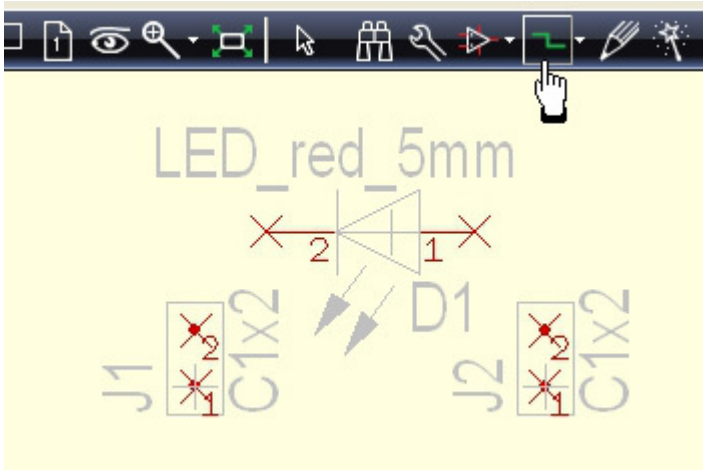
Now get in two dual pole connectors to the schematic by the same procedure...



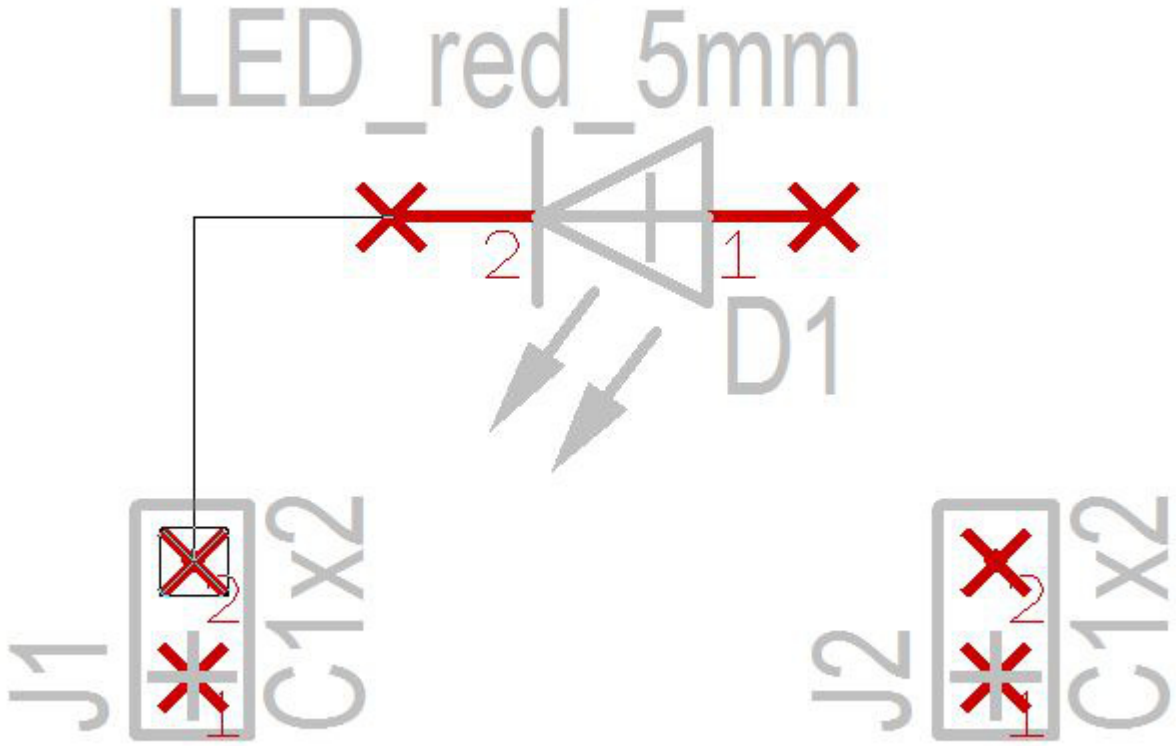
...and place them at a convenient location. Rotate any symbol prior to dropping by **M2**.

Connecting the pins

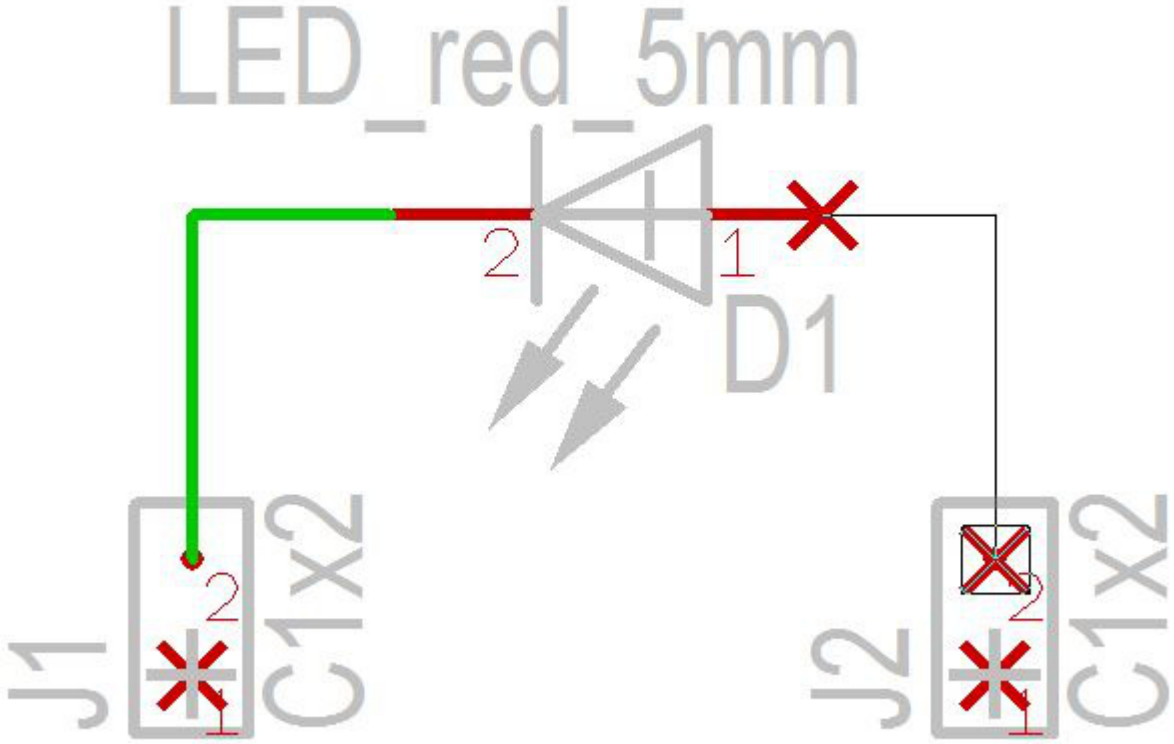
Click **M1** into empty space on your schematic to have all parts unmarked. The pins of component symbols are connected by the function "Place wire" using icon 
You can either use keyboard key **[2]** in order to start this function.



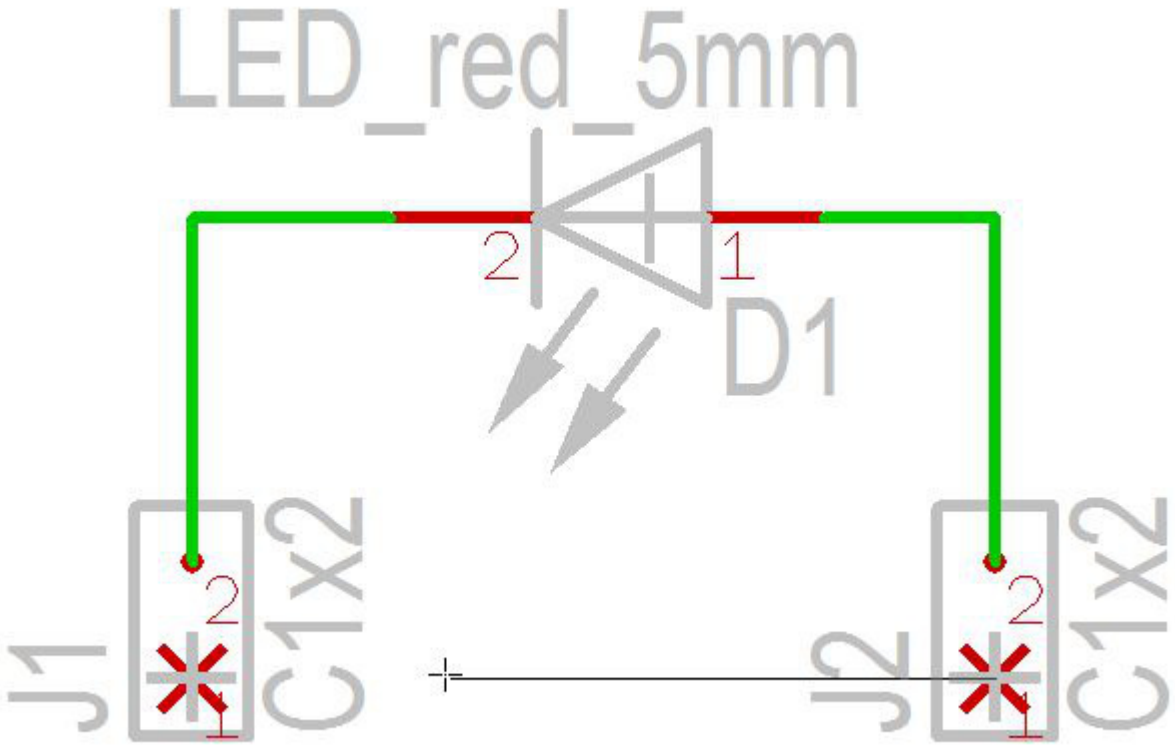
Now lead the signal by **M1H** from the cathode of the diode to a pin of a connector. The signal has adapted the pin function and leads it further as a signal name. After you have created the connection cut the wire by the use of **[Esc]** or by **M12** to proceed with a new connection. Toggle the bending mode by the use of the "space bar".



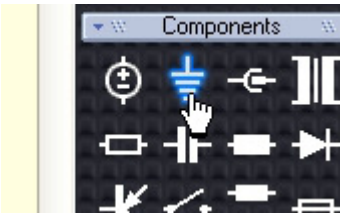
Now connect the anode of the LED...



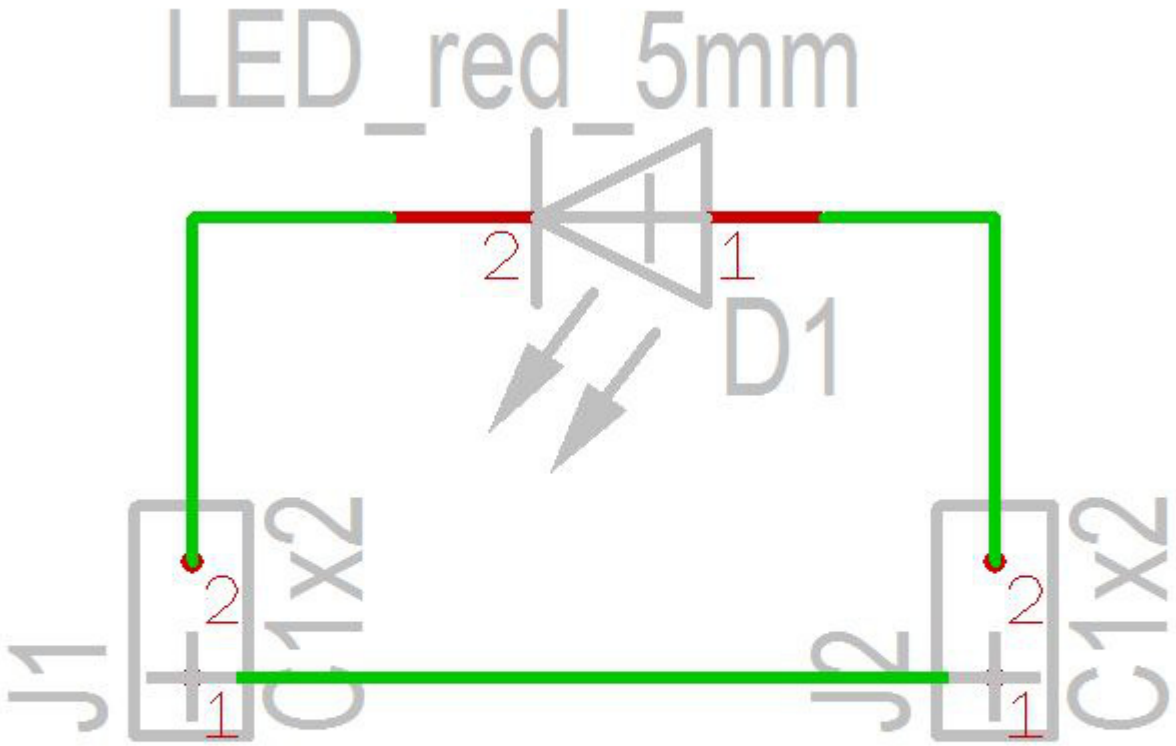
...and connect both remaining pins for ground connection.

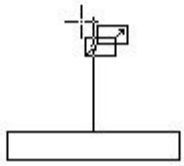


The ground signal is connected by a ground symbol. Please find it within the range of "Reference pins" in the sidebar:

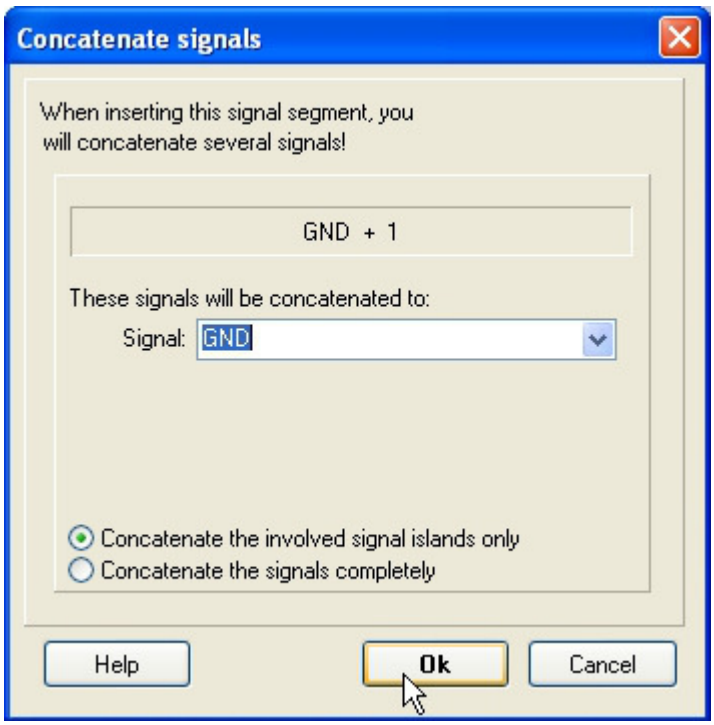


Drag and drop the GND symbol in the schematic by **M1H**...

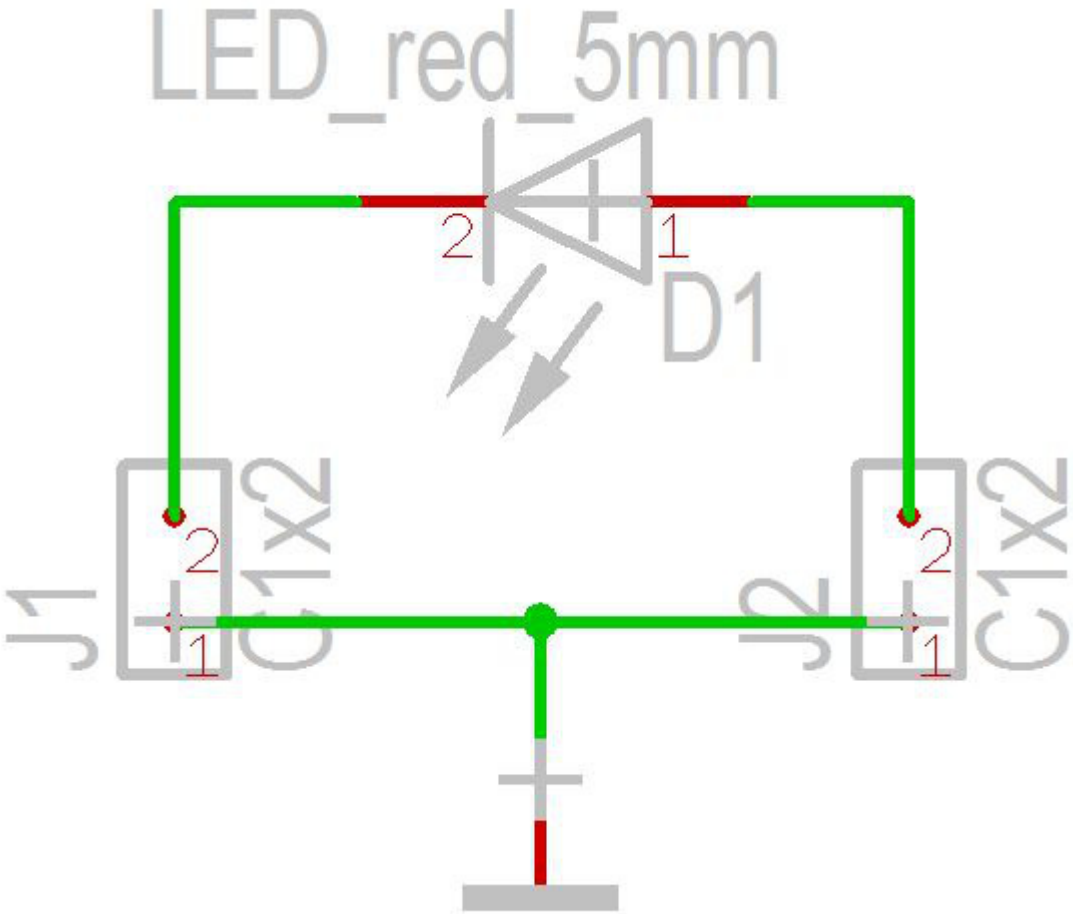




...and connect it that way, that you drag a signal wire from the GND symbol towards the signal track. By this means the signal GND is carried over from the GND-Symbol to the signal track. An intermediate dialog confirms the naming of this signal.

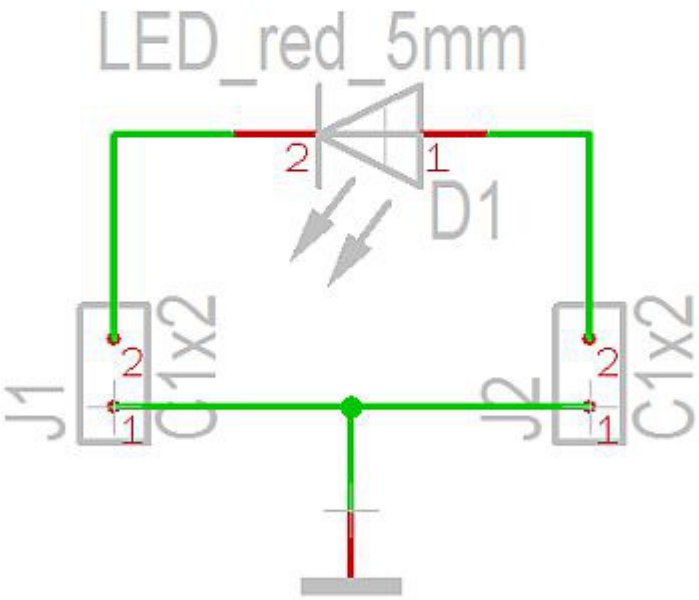


That's what your little schematic now could look like:

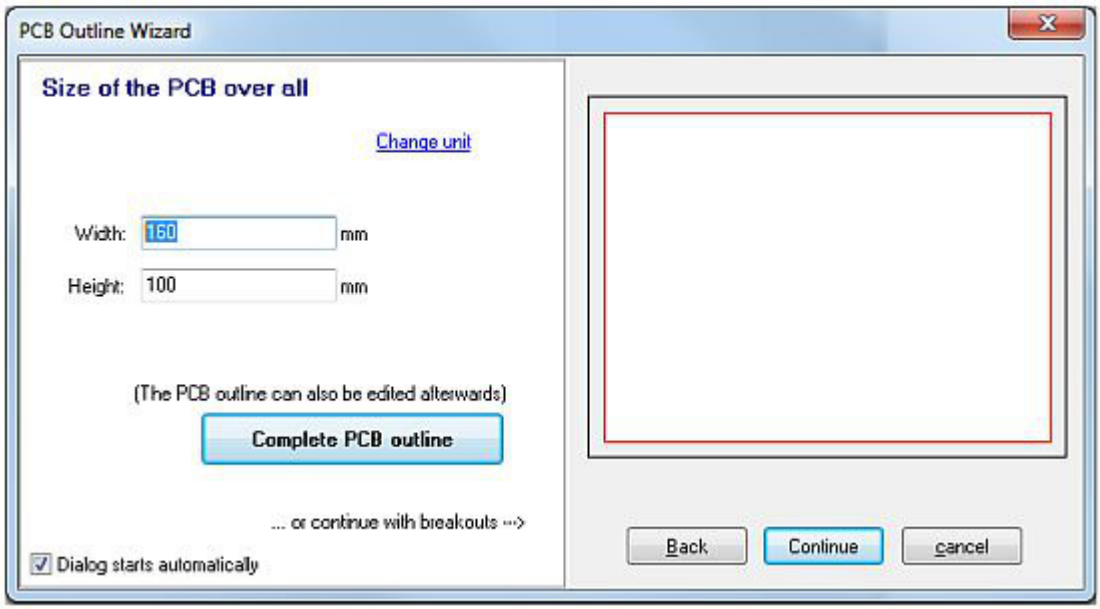


Define a PCB outline

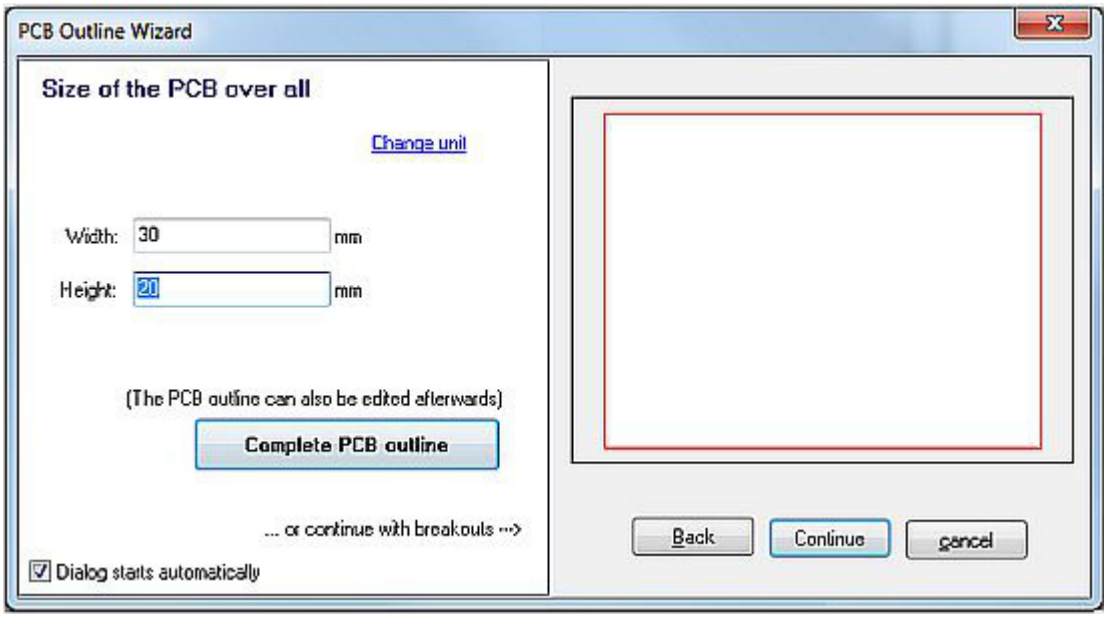
Please switch over to the PCB layout view by the icon "Go to PCB view"  (see cursor in the following image) or use the key **F3**.



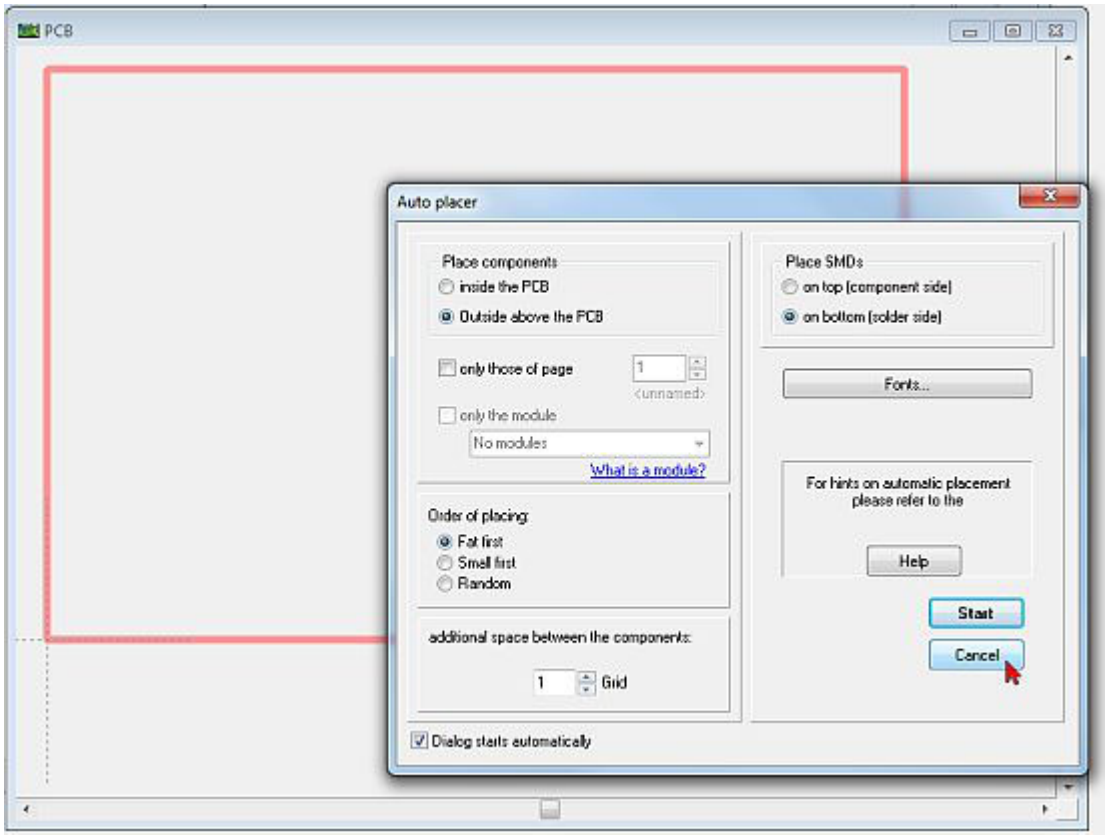
What you see is the PCB outline wizard suggesting a standard PCB outline in light red. It has the measurements of a "Eurocard" (W=160mm, H=100mm). Those measurements are much too big for our little project. Any modification of the outline can automatically be done within the PCB outline wizard in menu "Actions/PCB Outline Wizard":



Immediately the assistant opens and we enter Width=30 and Height=20. The fact that we work in metric measurement is default. It can be adjusted to imperial measurement by the "Change unit" link in the dialog itself or by the icon "Adjust View..." (=the eye).  The grid is set at the same location.



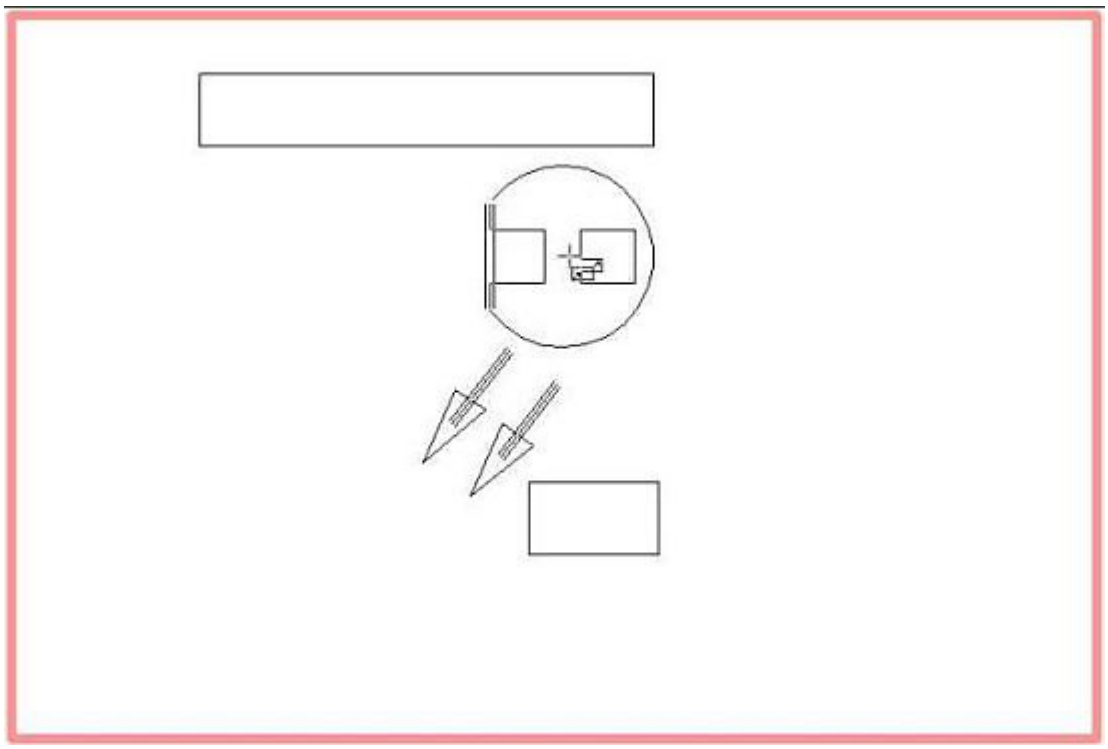
Because we don't have further breakouts at the edges or within the square, we click on button "Complete PCB outline". TARGET 3001! asks whether we want to use the placement wizard which we cancel (we want to place them manually). Immediately the desired outline shape appears on the layout screen. Now we can import the component packages (the footprint pattern, or landmarks) by drag and drop from the sidebar.



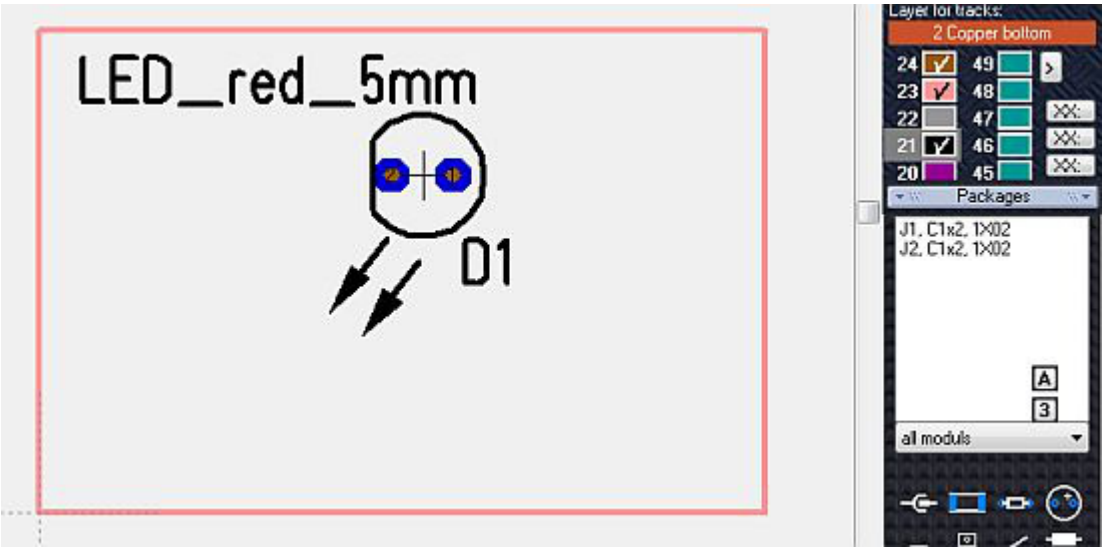
Import packages (footprints) to the layout

Every schematic symbol in TARGET 3001! has a package proposal. The word package in TARGET 3001! stands for "footprints" or more exact: solderable "landmarks". If you want to start with the package import, see the list of all package proposals according to the symbols used in the schematic appearing in the sidebar. Drag and drop from this list each of the packages you'd like to import to the layout. You can lift up the „Layers“ bar by M1H for a more convenient display (arrow).

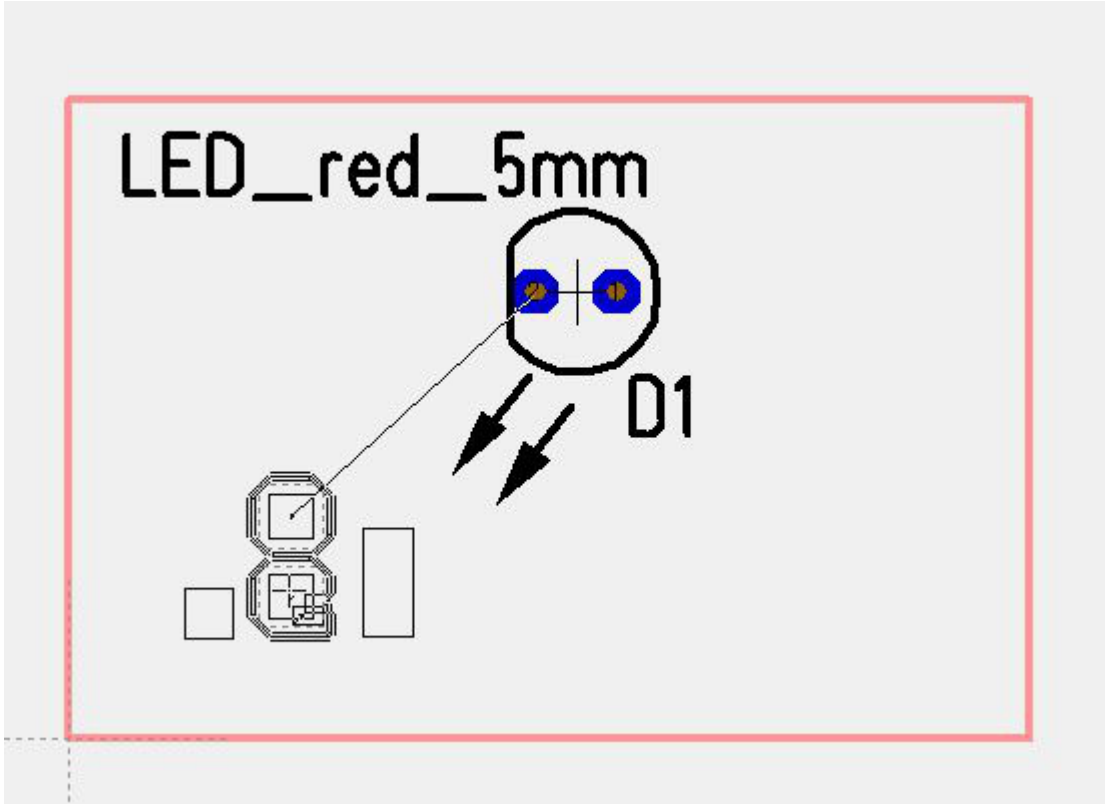
Now it's shape is fixed to the cursor and allows rotation by **M2**,...



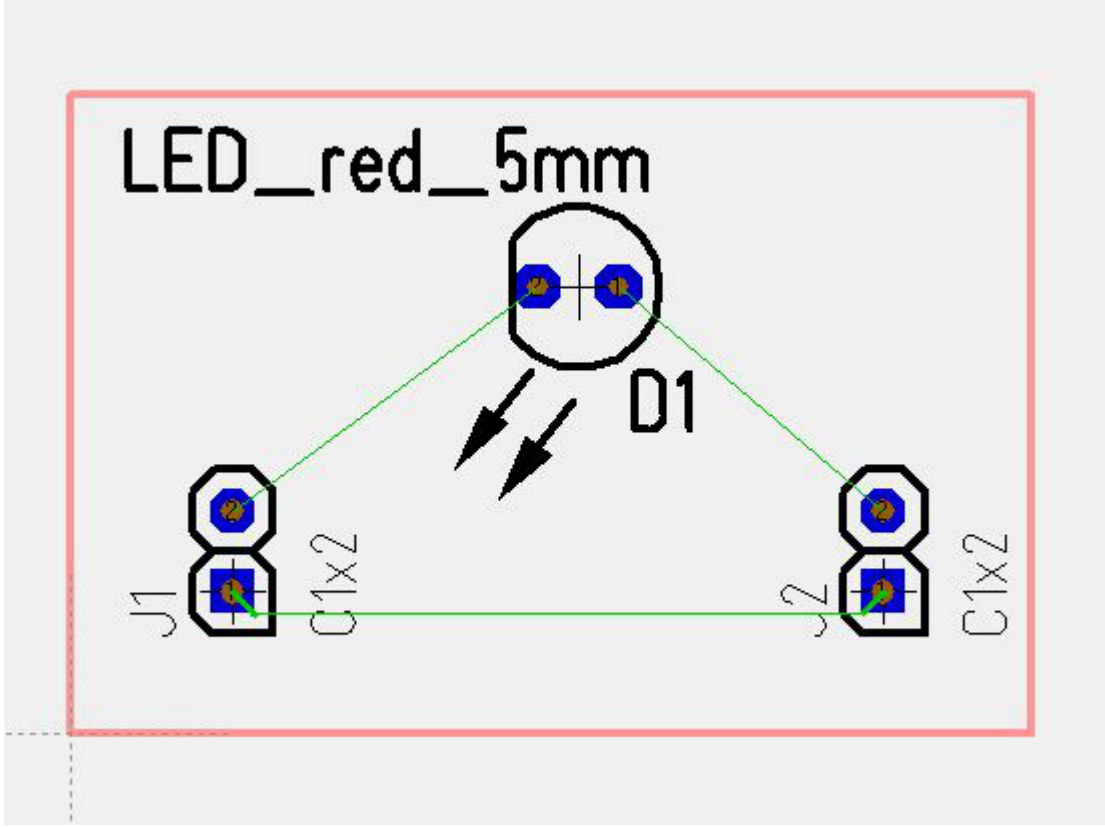
...before it is placed by releasing mousekey **M1**. Now we bring in connector J1 from the sidebar the same way



Again it is fixed to the cursor ready for placement. At the same time an air wire shows the electrical connection. After placement of the connector we see the air wire in green and the pads in blue.



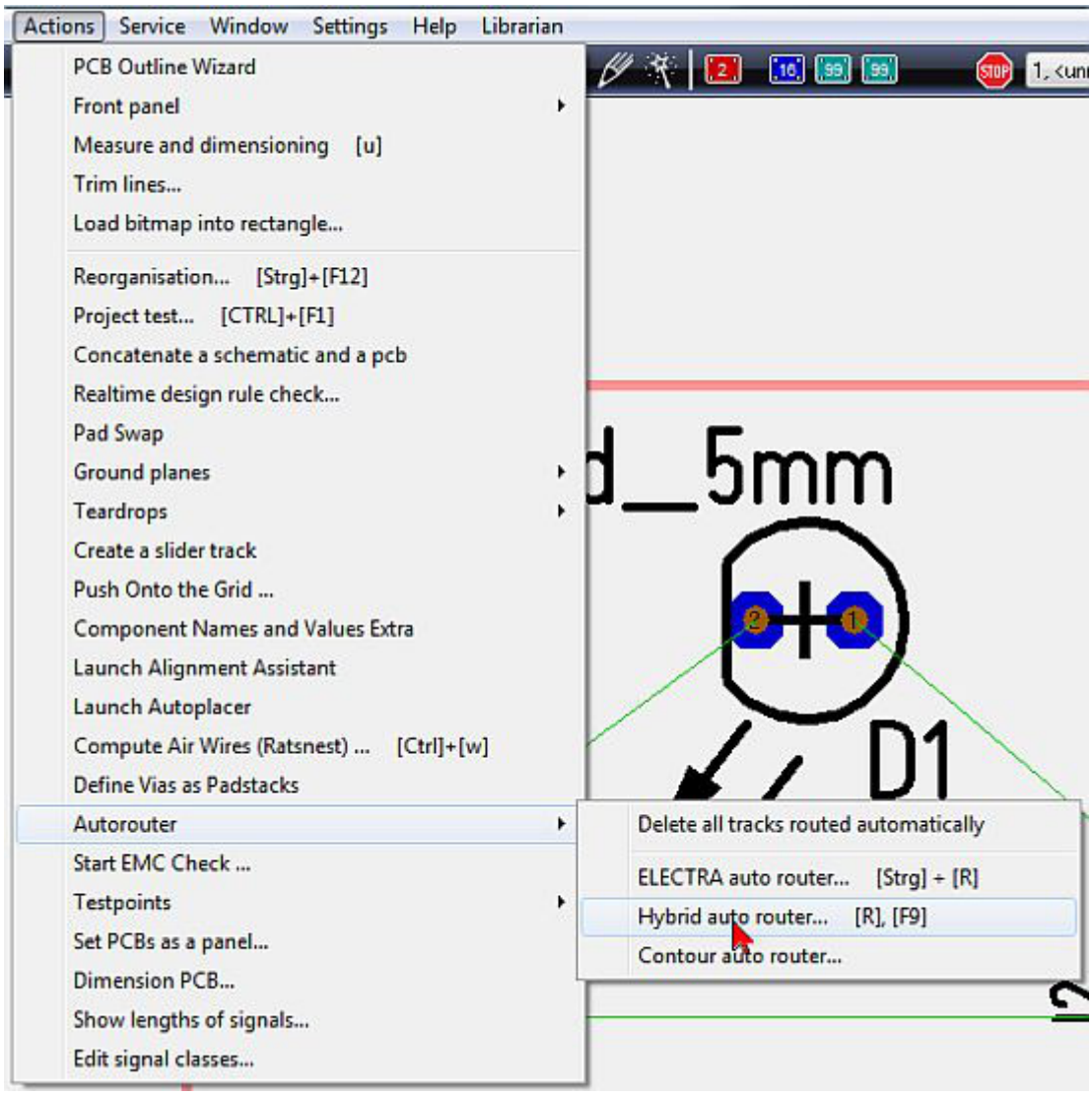
Again the list opens, we select the second connector J2 and place it.



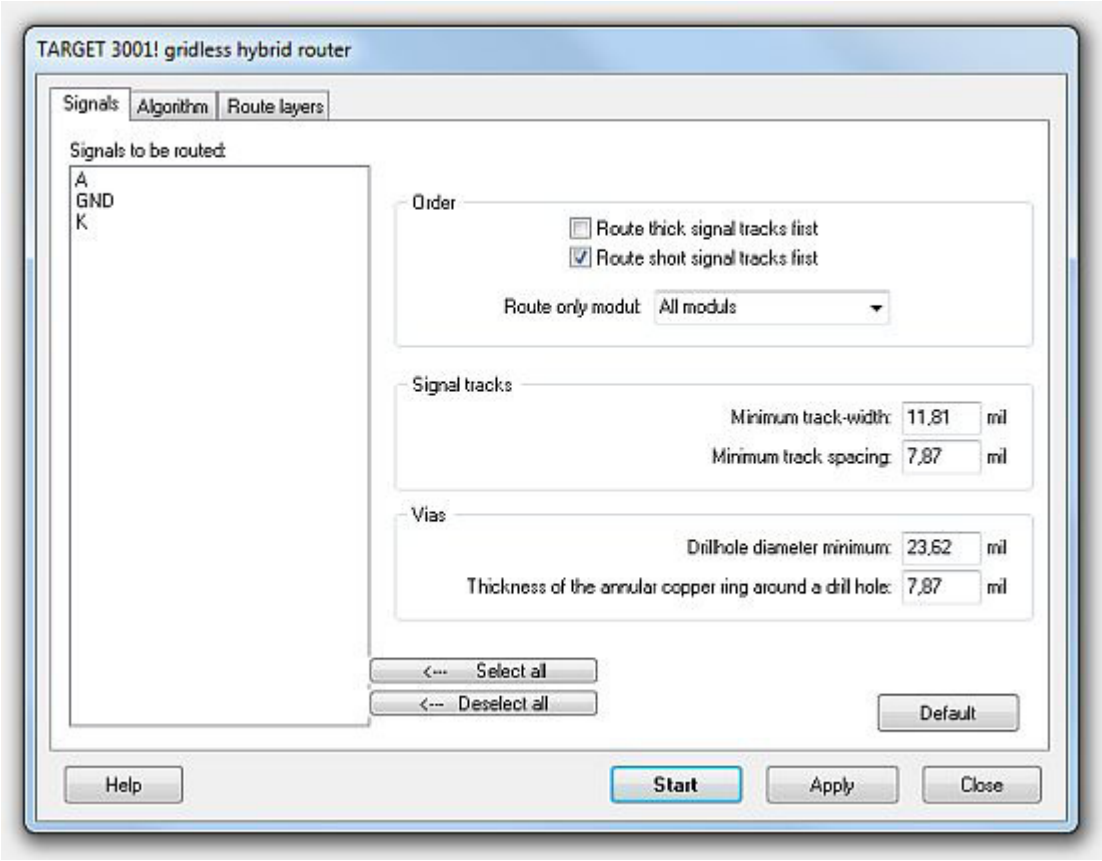
The airwires are no tracks yet. Those must be drawn separately. Do so by hand or by the use of one of the TARGET 3001! autorouters, see the next step.

Place tracks

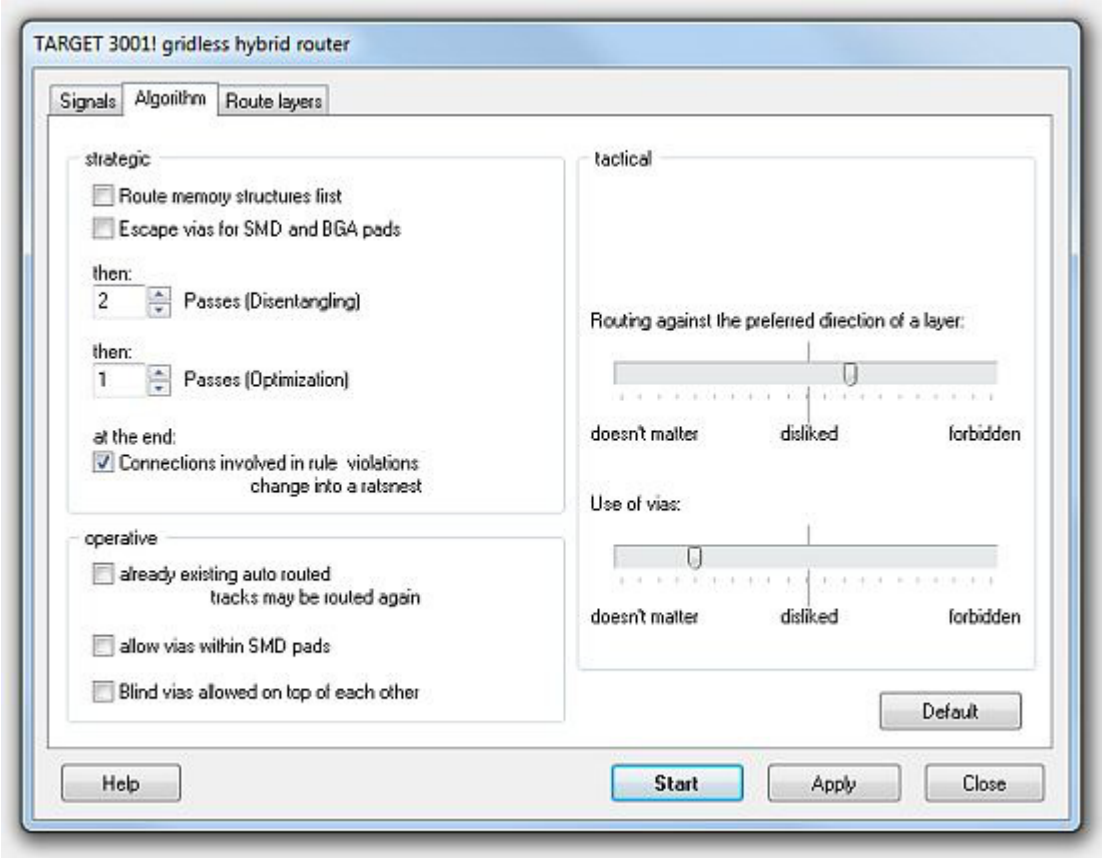
The placement of tracks can be done manually or by the use of one of the TARGET autorouters, in this case we use the "Hybrid autorouter" in layoutmenu "Actions". Also the keyboard key [r] or the function key [F9] will launch the Hybrid Autorouter.



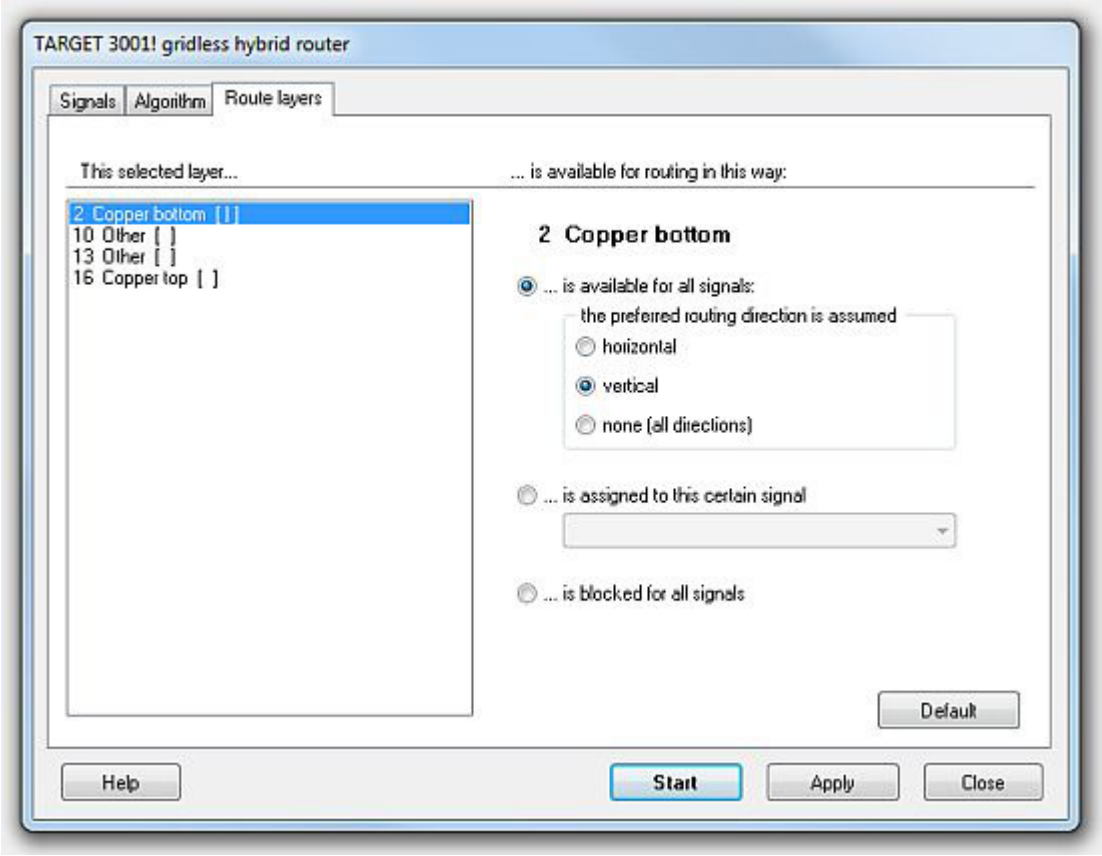
The opening dialog of the hybrid router has three sections divided by tabs. First the section "Signals".



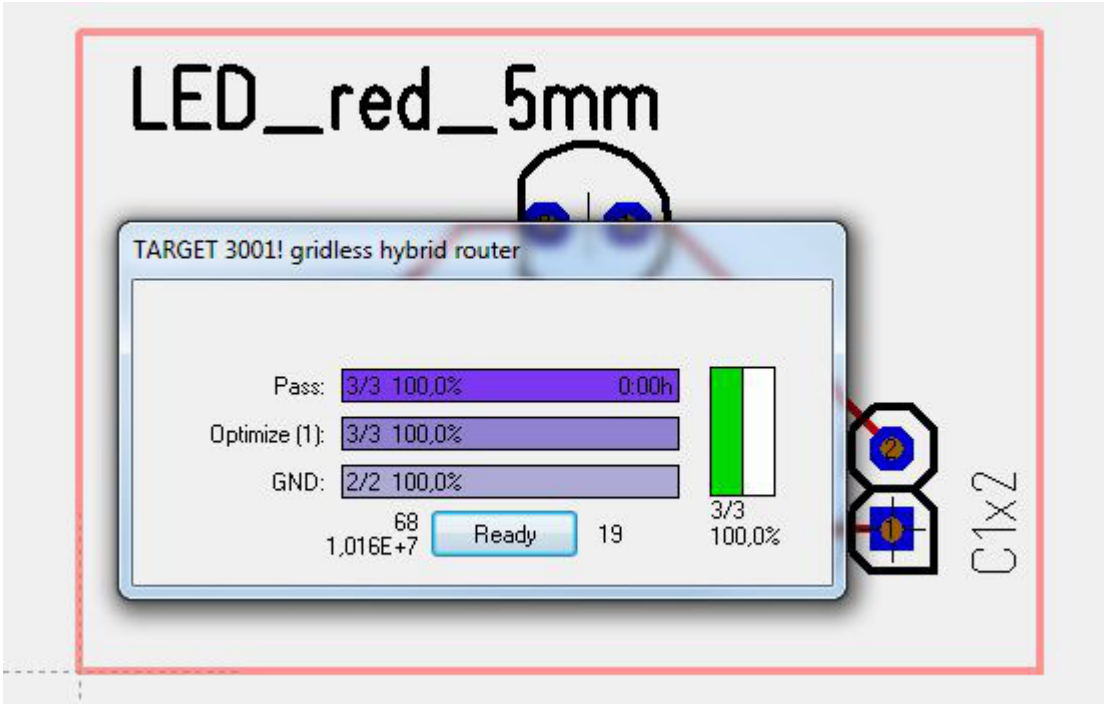
Now choose from the list one or more or all signals. If you select none, all are routed.



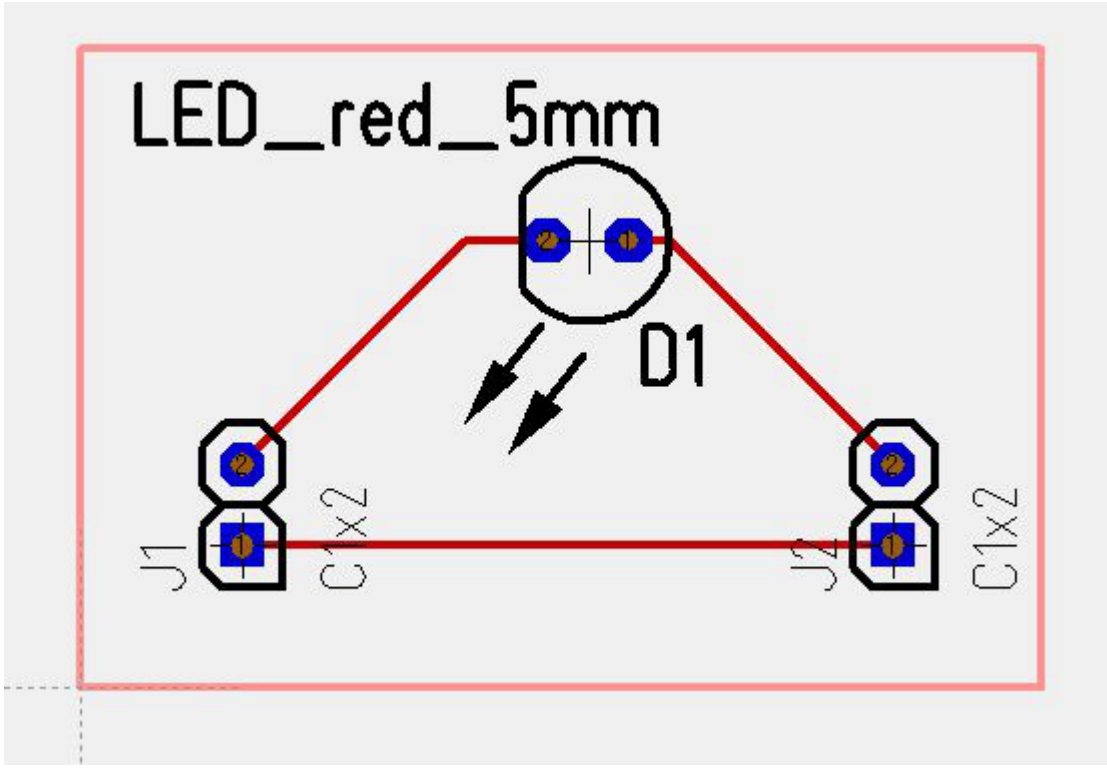
The second tab allows "strategic", "operative" and "tactical" settings. In the strategic area we leave the default settings with "two passes of disentangeling" and "one pass of optimization". In the tactical area we place the slider between "disliked" and "forbidden". The third tab allows the assignment of certain signals to certain routing layers. We enter: "Copper bottom" (Layer 2) is available for all signals, "Copper top" (Layer 16) is blocked for all signals. Click on each layer line at the left to set it's performance.



Now watch the router do it's work...



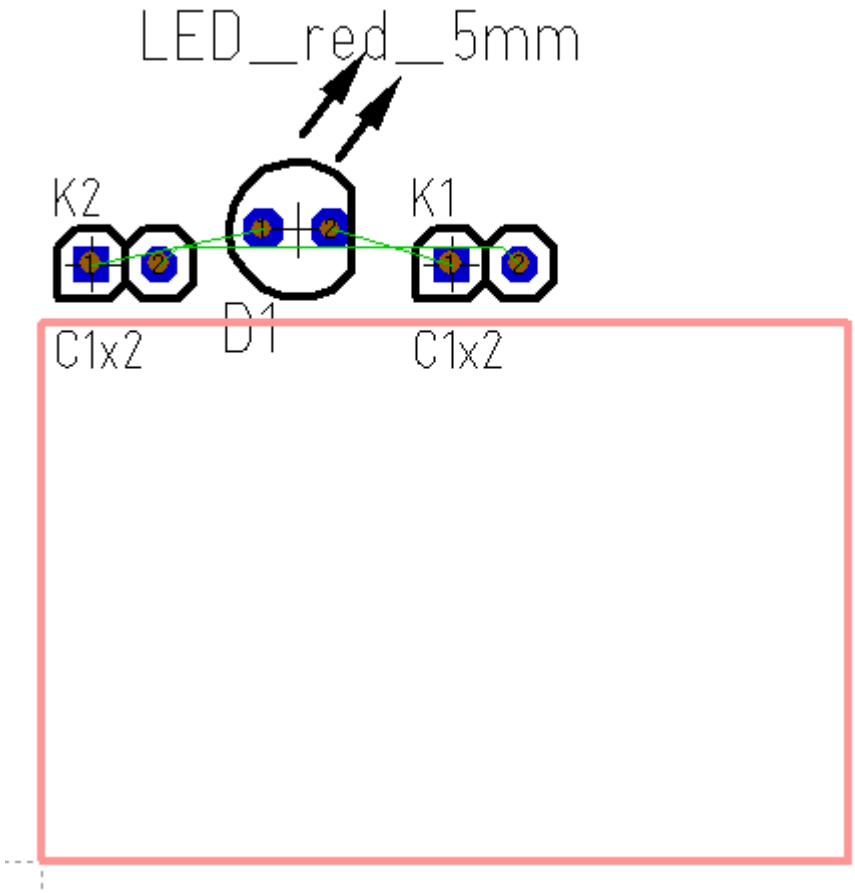
...and expect the following result:



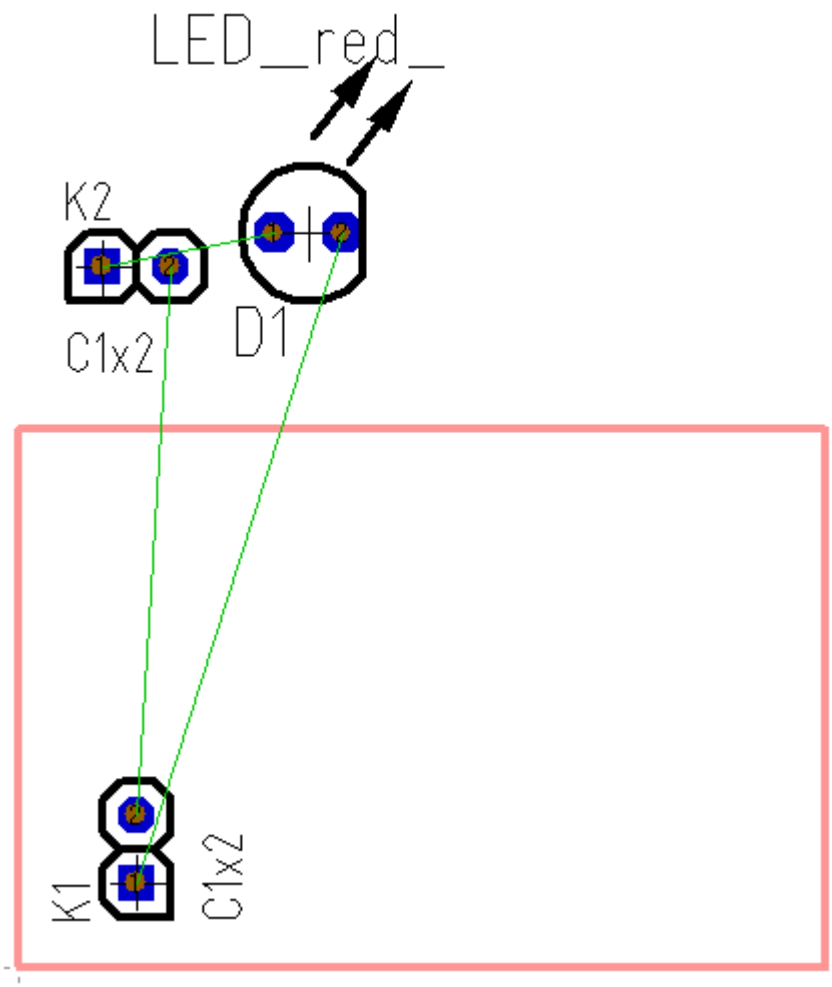
Deletion of the routing result: Drag a highlighting square over all and press the **[Del]** key. Only the autorouted tracks will be deleted.

Place tracks by hand

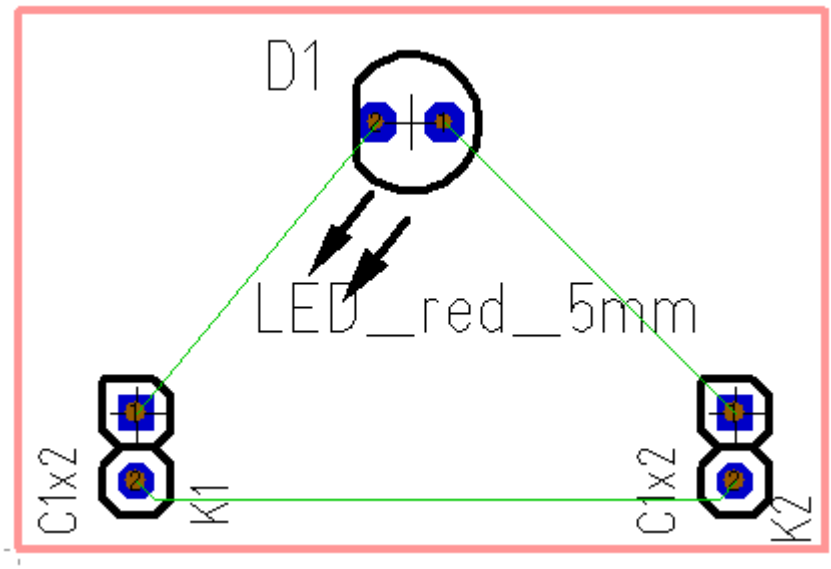
The autoplacer places e.g. randomly your packages at the edge of the board outline:



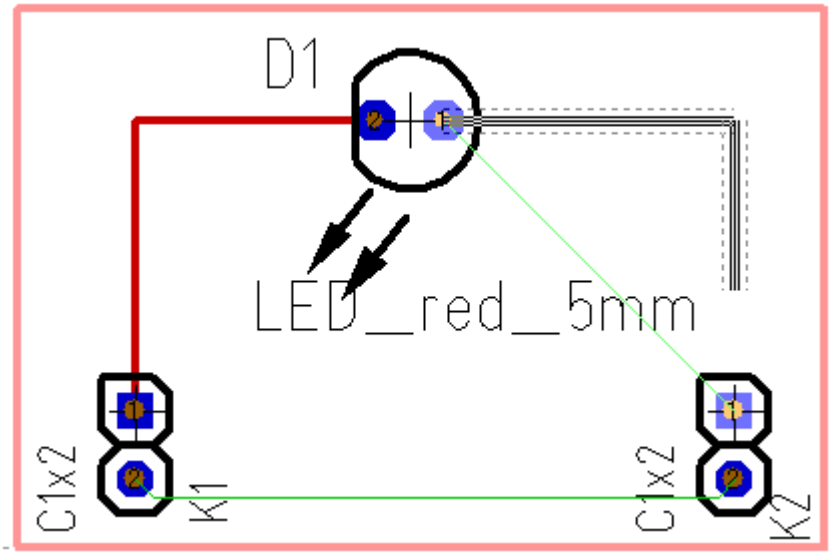
First drag the components into position. For doing so click with mousebutton held (M2) upon the handle cross of a package or at any drawing element of the package for grabbing it. In the second case this button  needs to be pushed, otherwise you won't grab all of the package but only a part of it. You will find this button in the Sidebar in section "Settings".



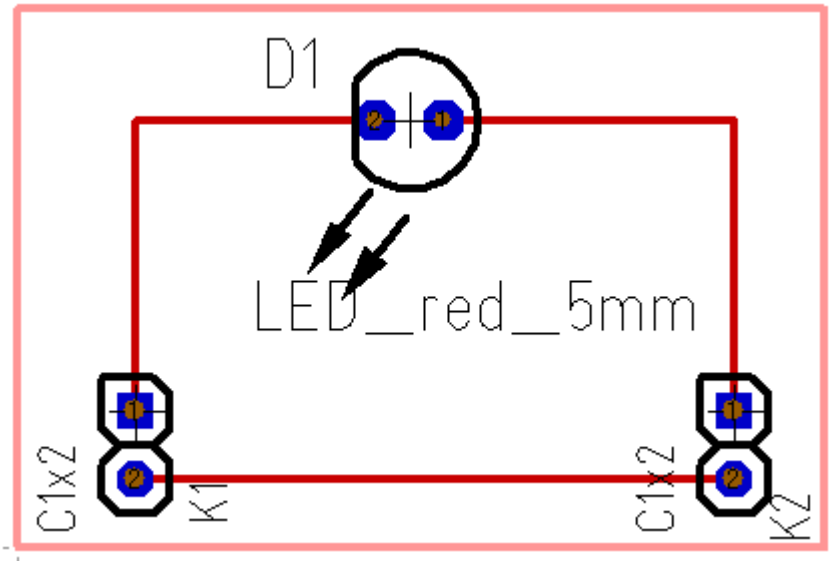
You can rotate the the components during the dragging by pressing the right mousekey **M2**. You also might touch them again for rotation. NOTE: Sometimes little fractions of drawings remain during the displacement. Press key [N] for refreshing the screen! At the end your placement might look like this:



Clicking this icon  will let you start Drawing the tracks. Click on a pad and drag the mouse to another pad for creating the connection. Toggle the bending mode by the use of the space bar or use the buttons in the sidebar in section "Settings":

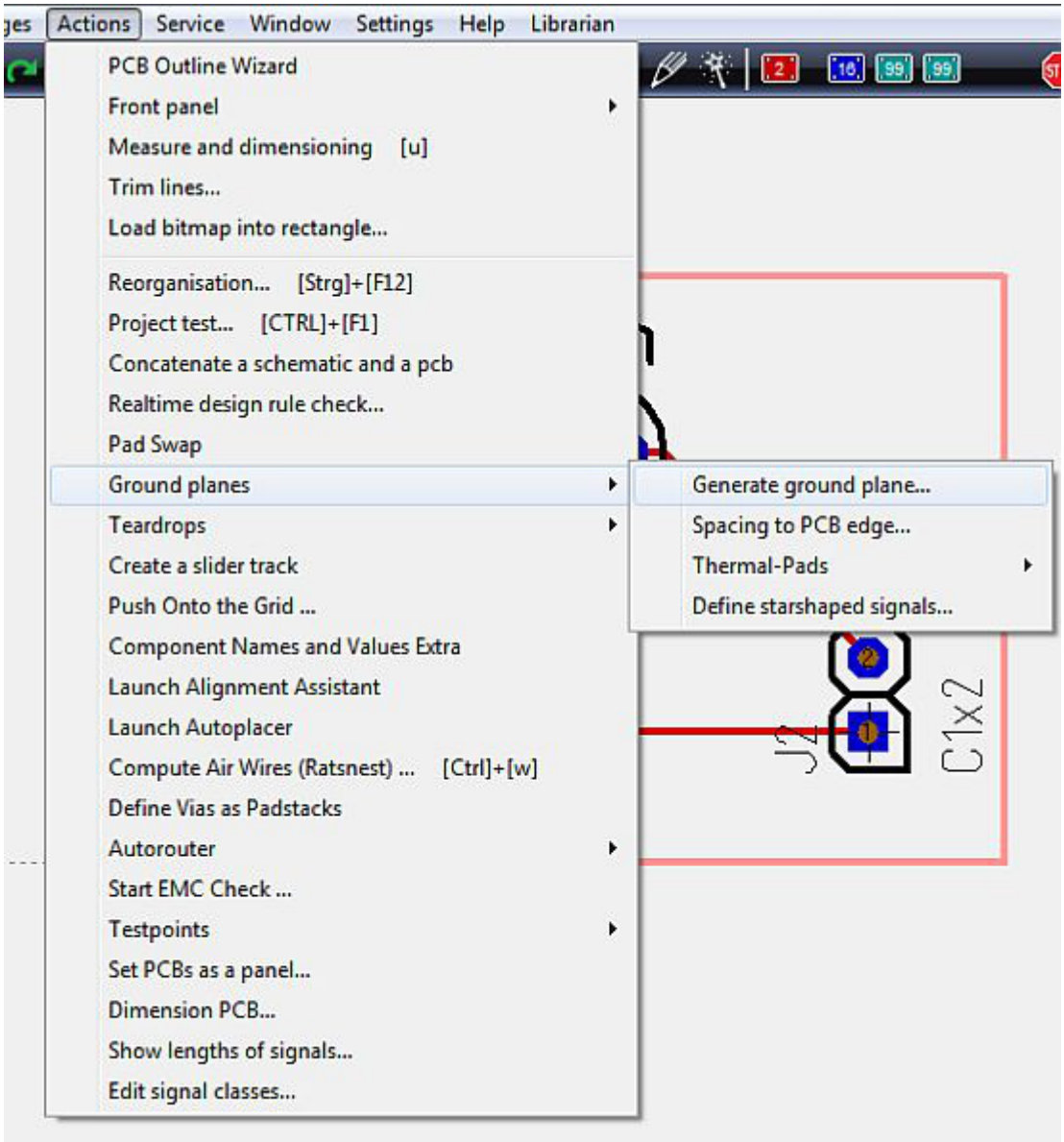


The dashed line which goes along beside the signal line represents a spacing having the the bigger value of both
- the element aura set and
- the minimum spacing of the signal.
But if in the signal's properties the minimum spacing only is set "Normal", the standard spacing is used. This standard spacing is set in "Settings/Settings (Project)" at option "Standard track-spacing (PCB)". After all tracks are placed and all green airwires (ratsnest) have vanished your layout might look like this:

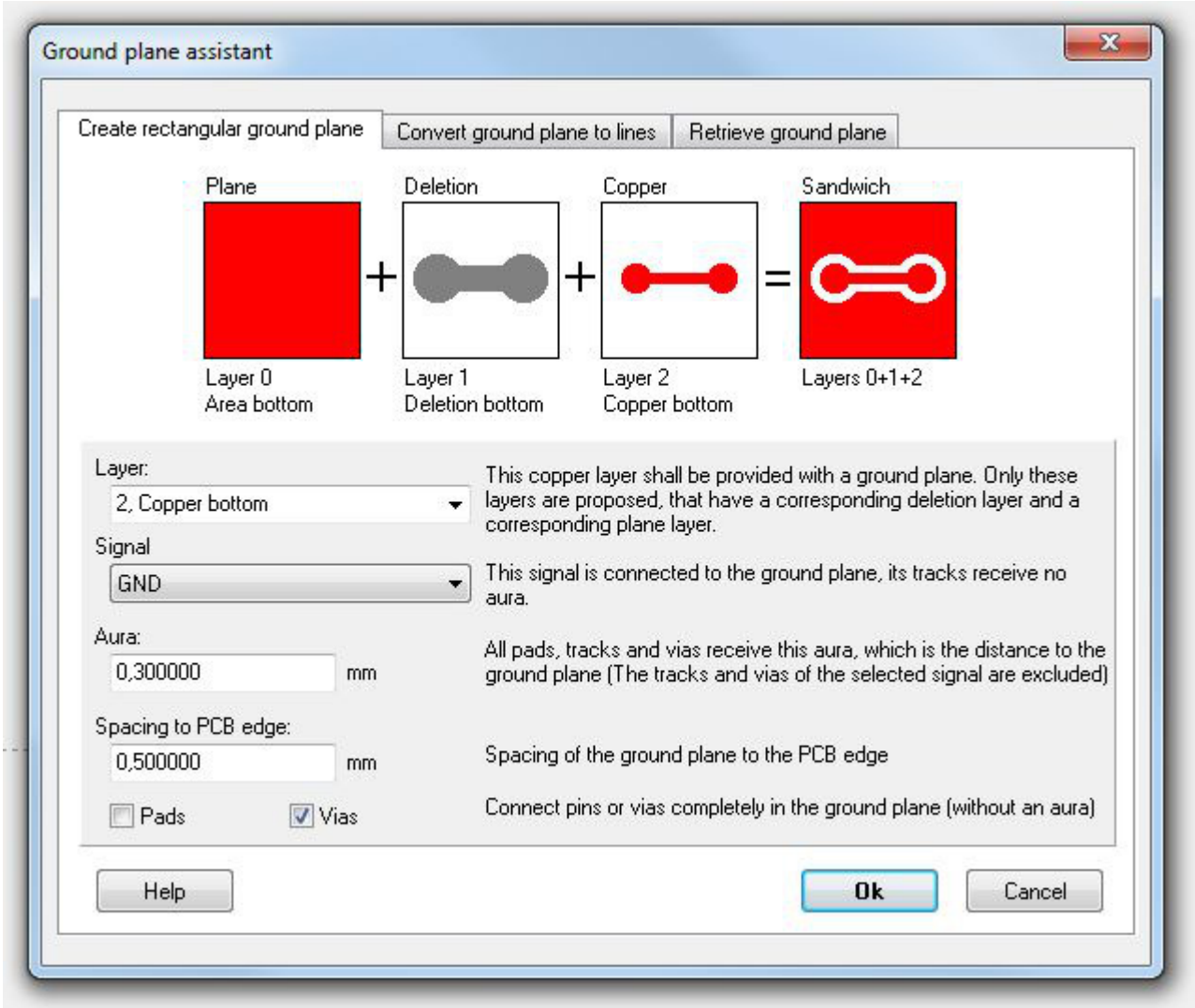


Generate a groundplane

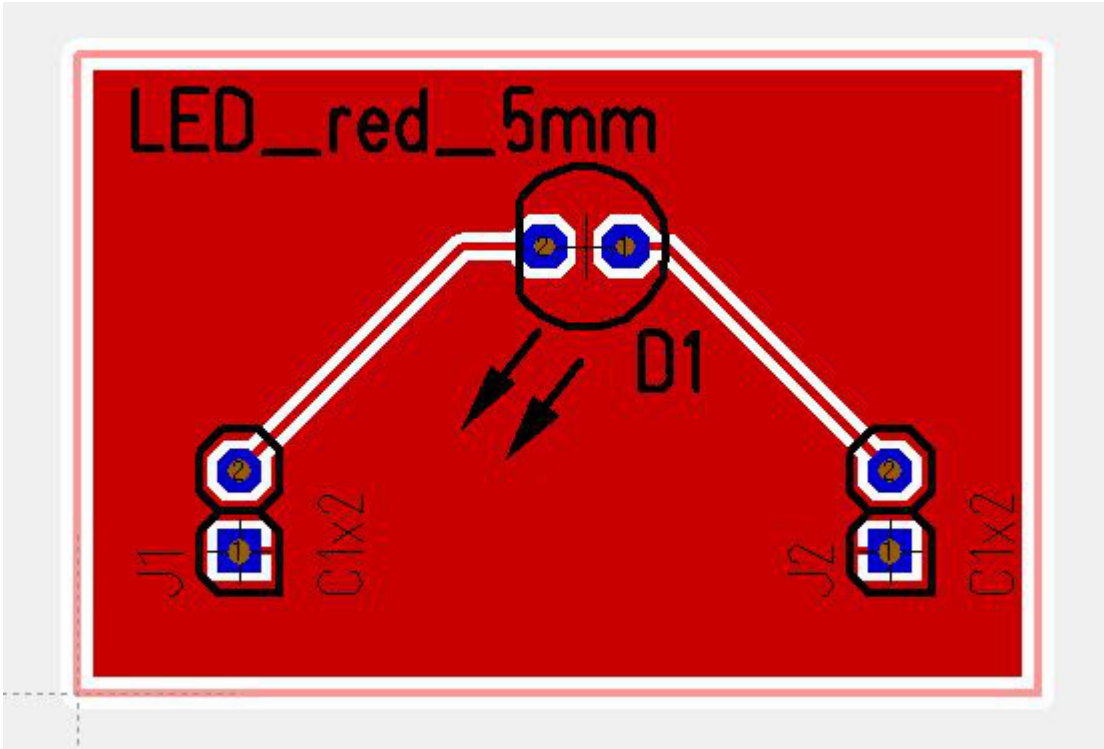
A groundplane having the same dimensions like the layout can easily be created by the use of the groundplane assistant, please see layoutmenu **"Actions"**:



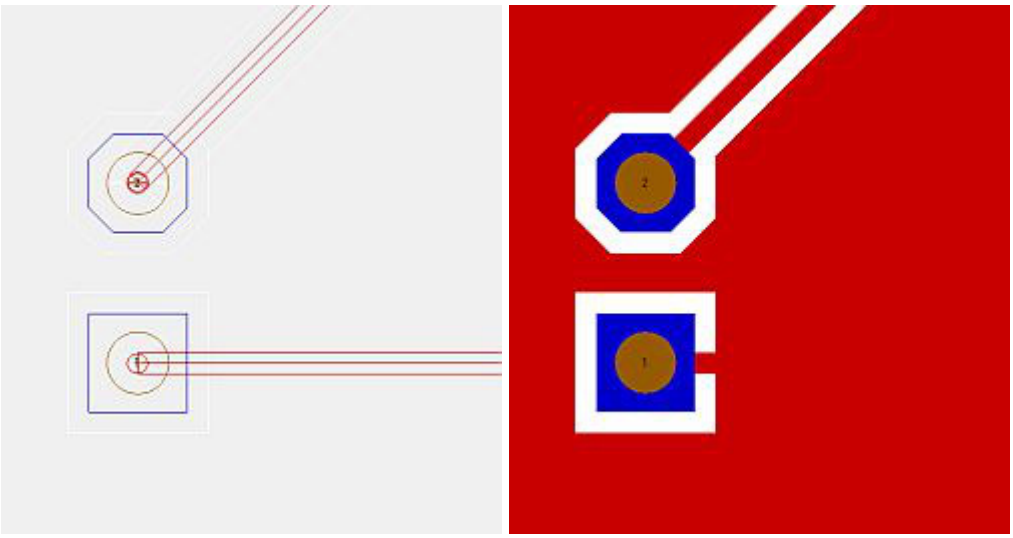
The opening dialog explains that a groundplane in TARGET 3001! is made out of a set of three layers: "copper", "area" and "deletion". The shape of the groundplane is defined on the "area" layer. The "deletion" layer defines the spacings between the groundplane and the non GND-leading tracks. The layer "copper" at least is the one which carries the groundplane and the tracks in copper at the very end. The following example is a groundplane on "copper bottom":



Please confirm the Standard settings and reach the following:

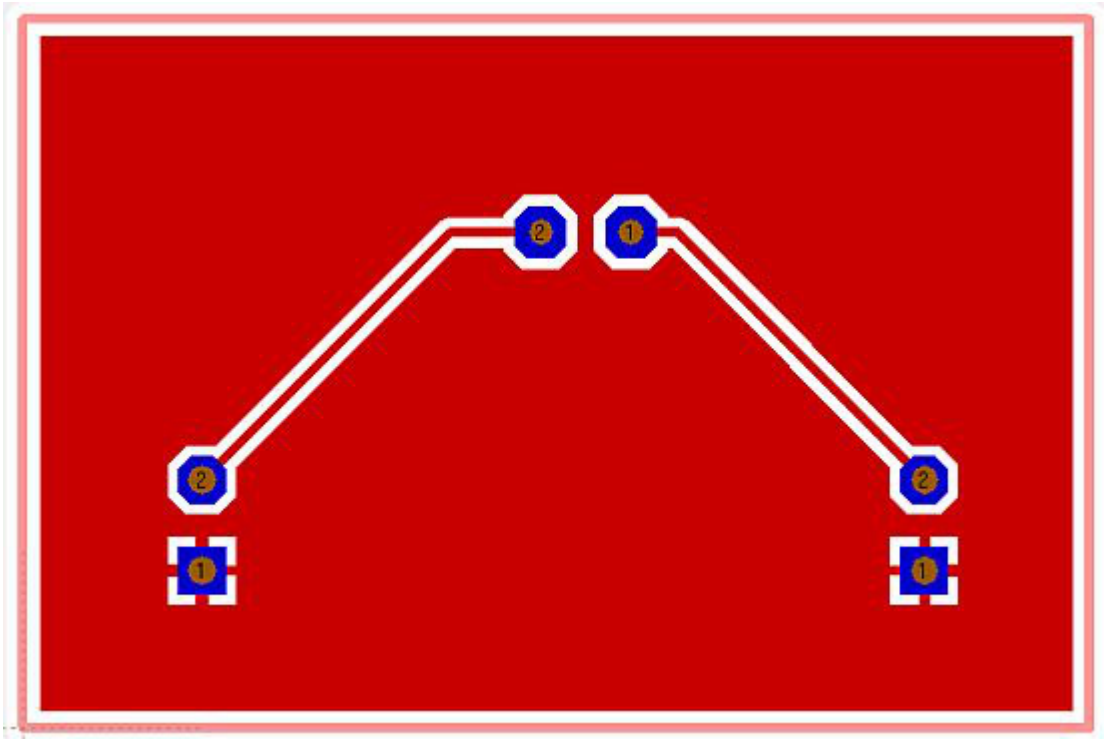


The GND leading track in the south of the layout is embedded completely into the groundplane. Use the hash key (#) for having an x-ray view of the layout. So you can see...

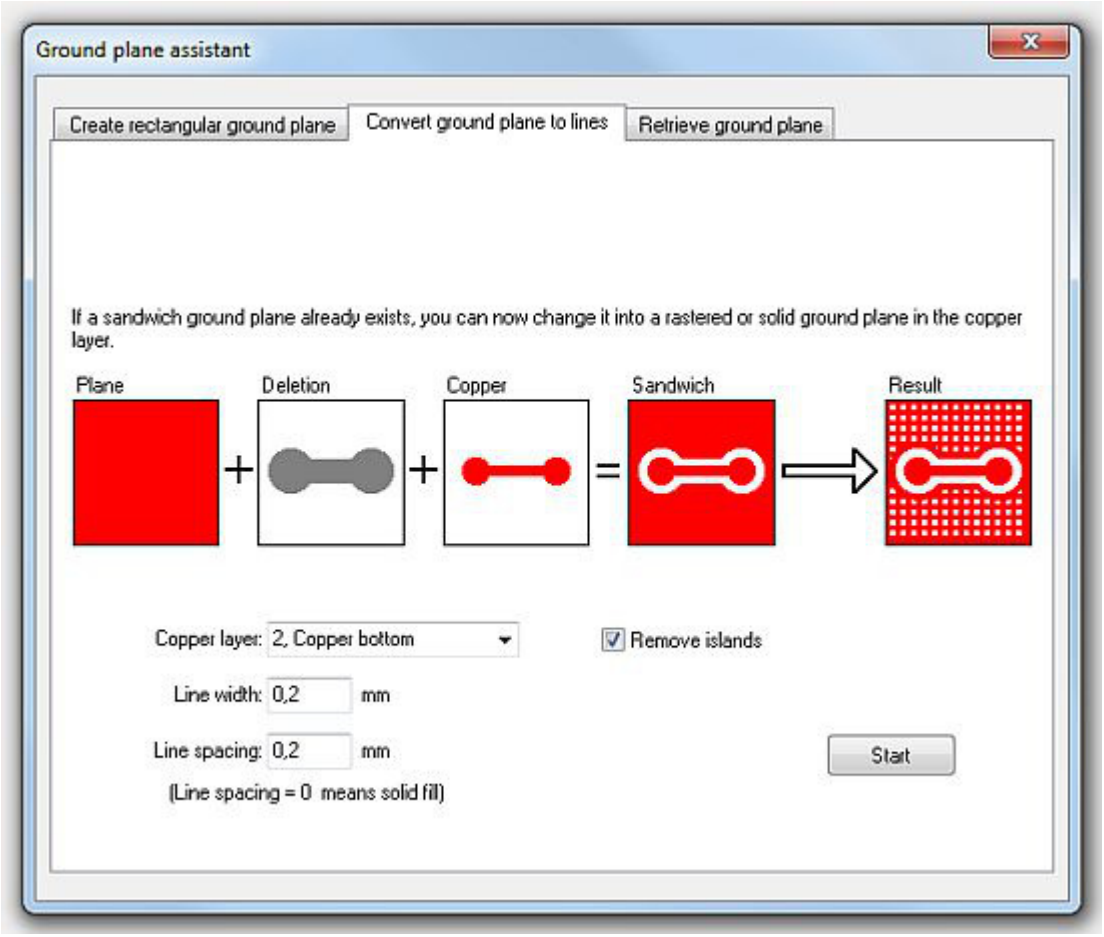


...that the pads seem to be connected correctly. Ant that's the fact (layer „21, Position print“, has been faded out = the colored field at layer 21 is unticked).

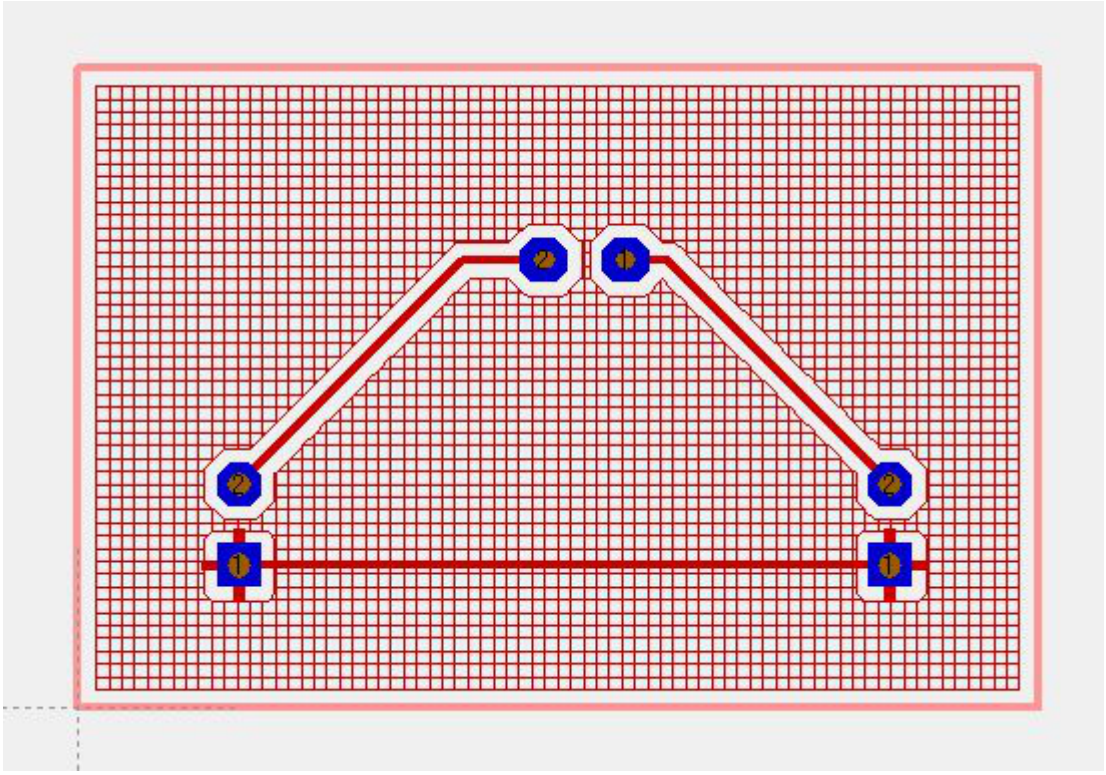
The generation of thermal pads is managed by highlighting the complete layout and choose menu "Actions/Ground planes/Thermal Pads/Create Thermal Pads". Pads whose signal tracks are embedded to the ground plane, receive two or three further ligaments. Result:



The groundplane can also be transferred to a grid. Please again open the ground plane assistant (menu "Actions/Ground planes/Generating a ground plane") and choose the tab in the middle:

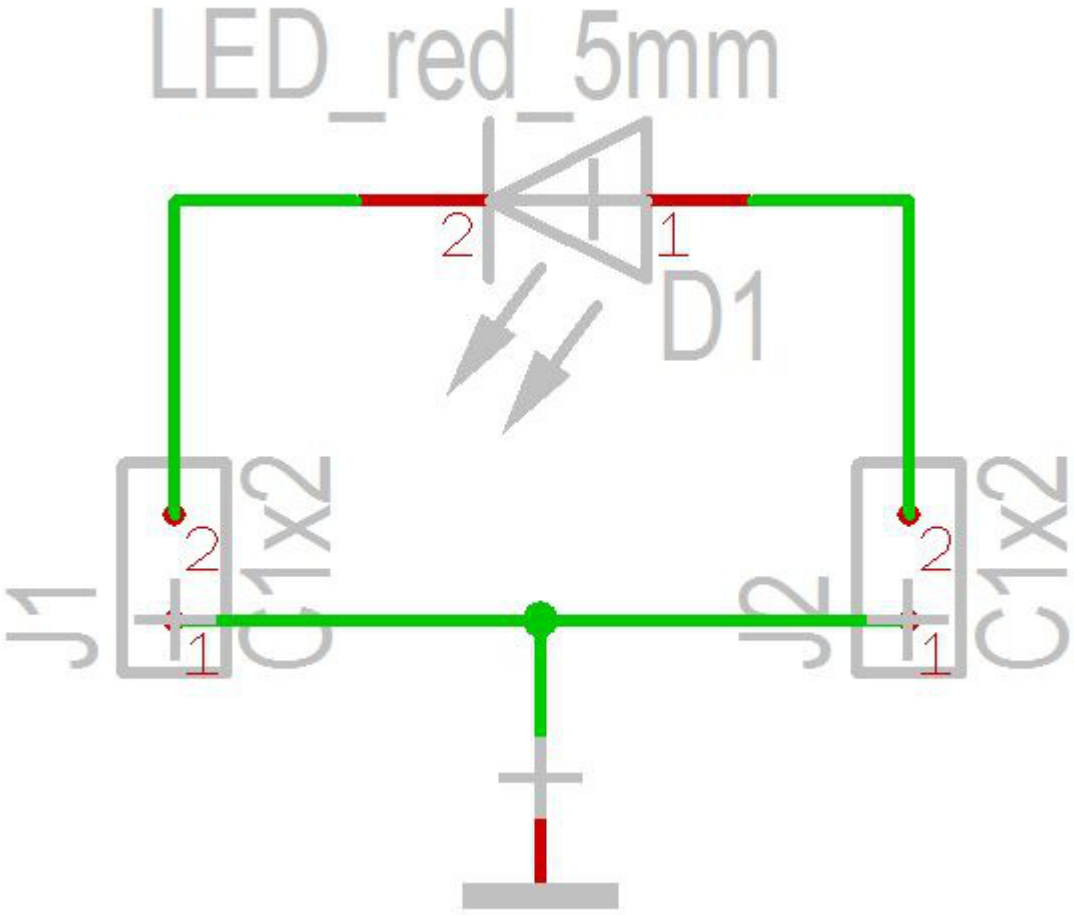


After pressing the "Start"-button, we receive:



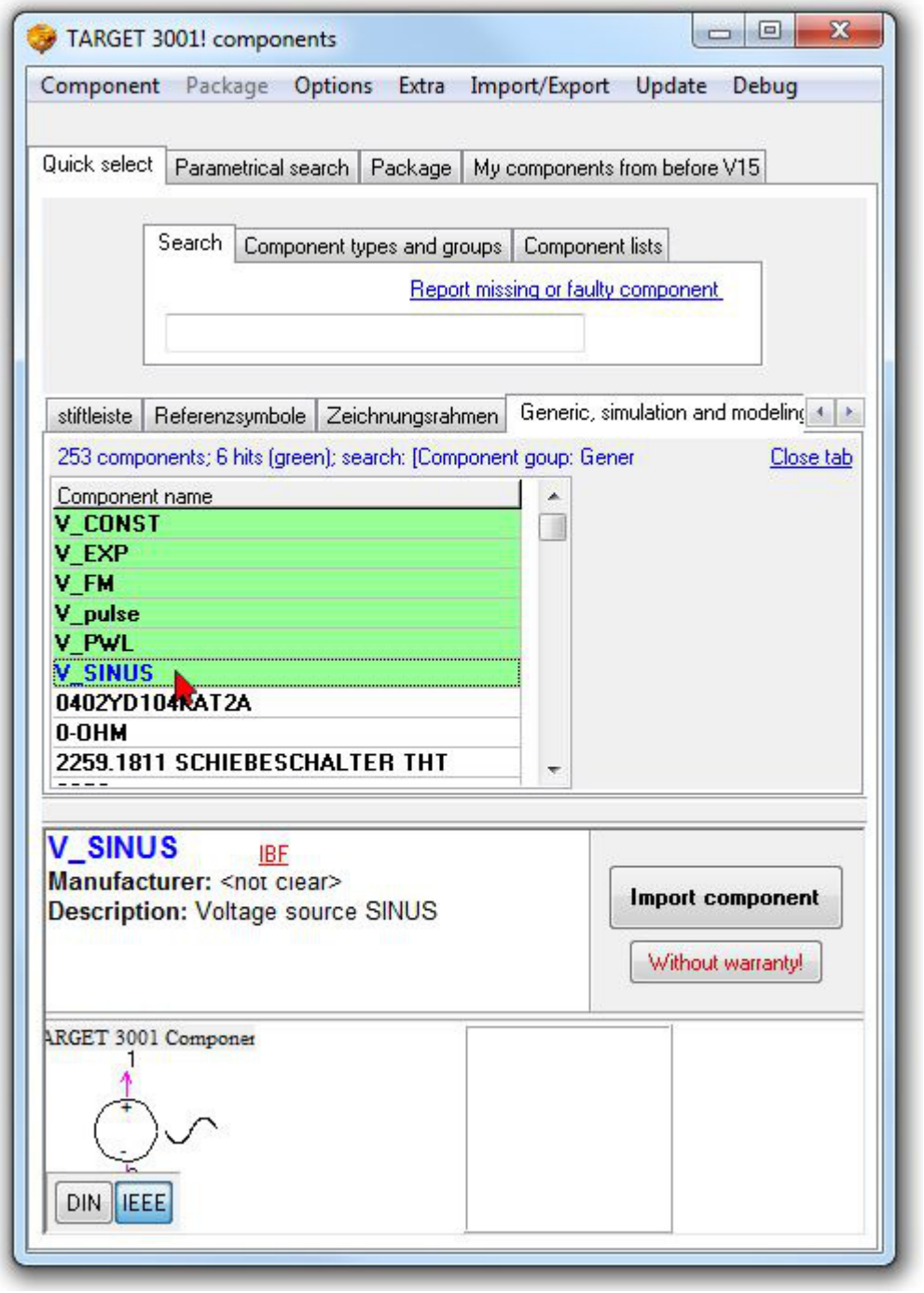
Simulate the function Part 1

The simulation in TARGET 3001! is a matter of the schematic, so we switch over to it:

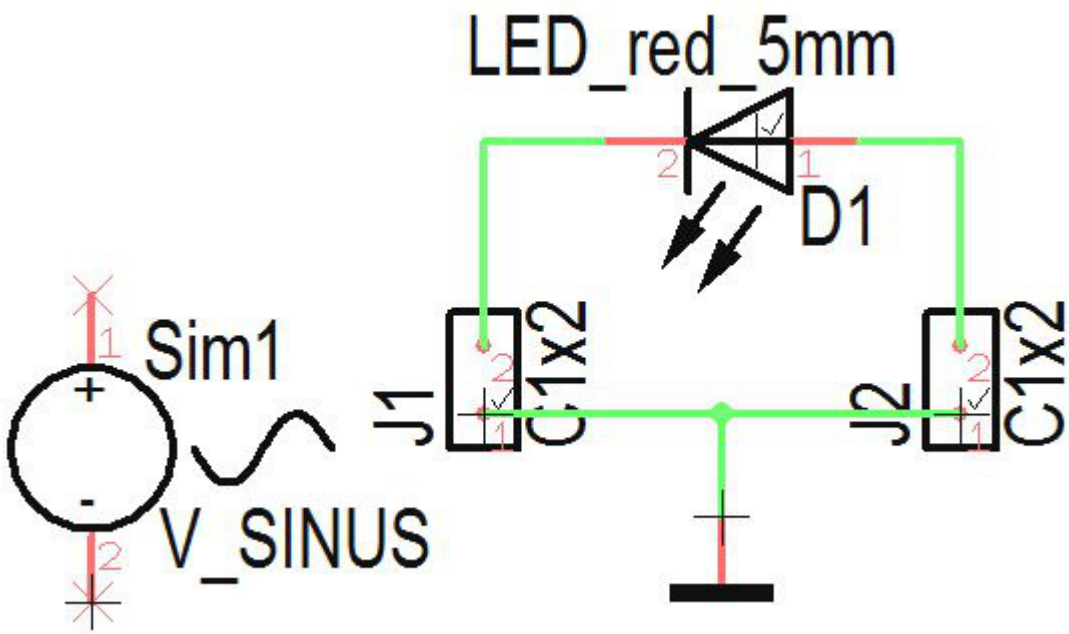


In order to simulate the LED function we need a voltage source, and a load resistance. Those components will not be part of the layout therefore they don't have a package. Find such components in the database. See sidebar icon:

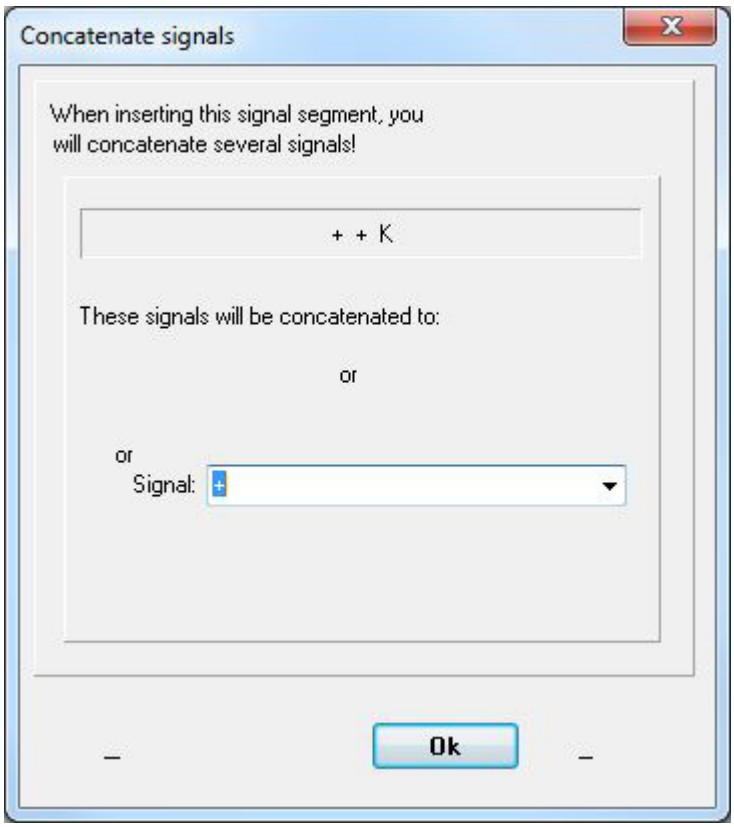
It will open the component browser close to the sources. Select the V_SINUS and import it:

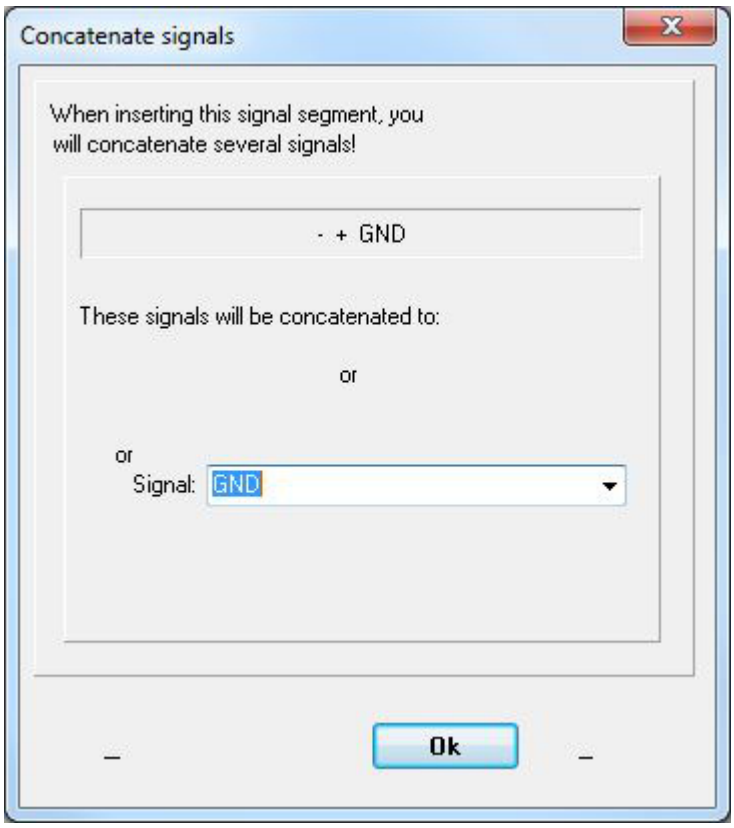


After we have placed it, the schamatic might look that way:

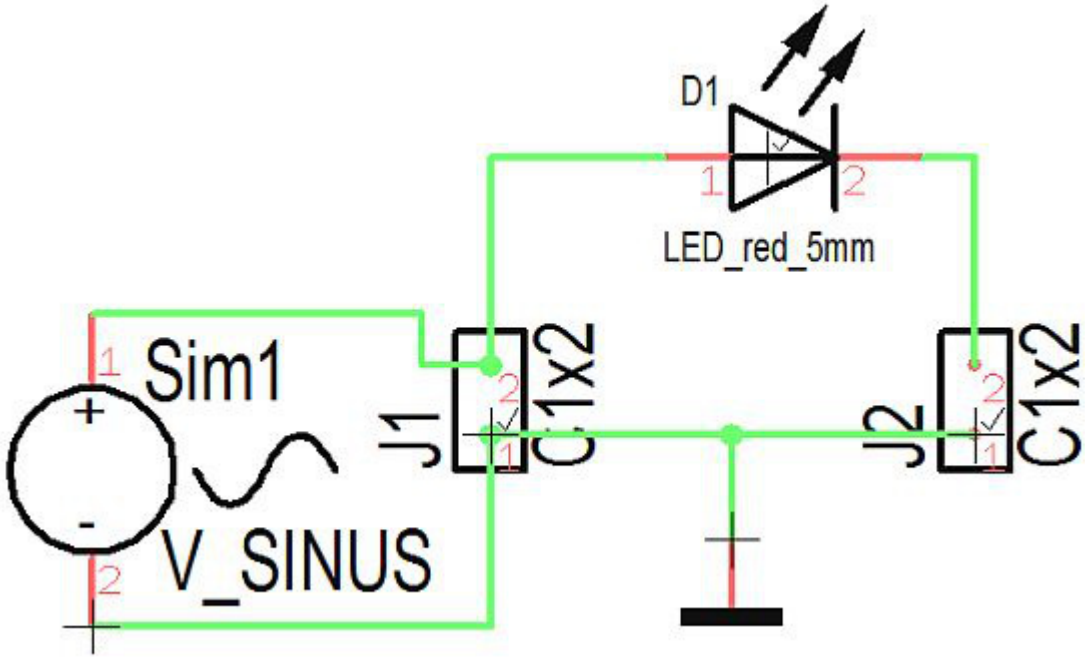


Following the convention that in a schematic the sources should be on the left hand side it is necessary to rotate the diode because we have to recognise it's polarity. Delete the connections then touch and hold the diode at it's handle cross **M1H**. Press right mouse key **M2** two times in order to rotate it. Now connect it new. Reduce the size of the font by a double click each on the text and entering to the dialog smaller font hights and widths. Connect the source pins towards the connector pins and confirm each the suggested signal names (+) and (GND).

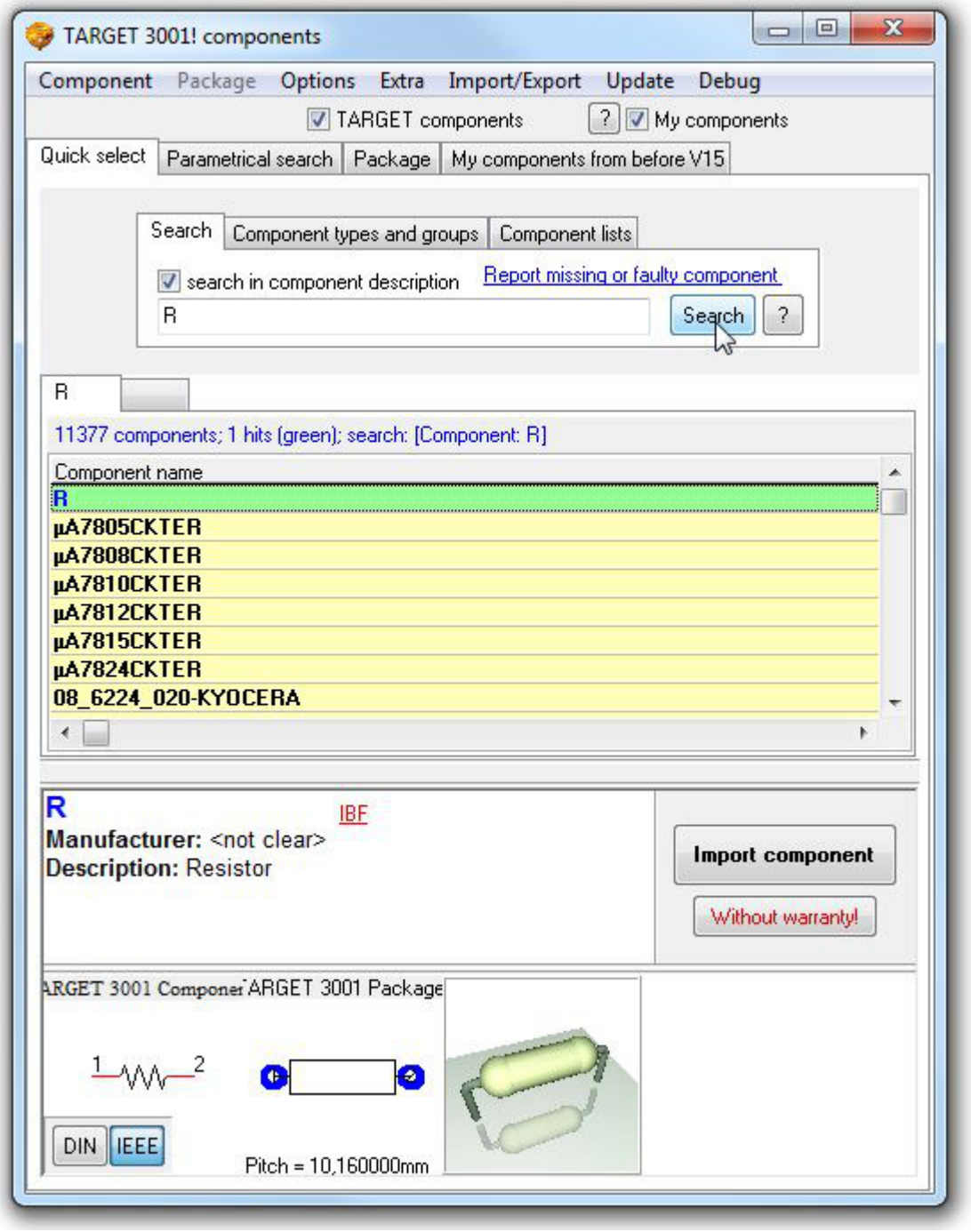




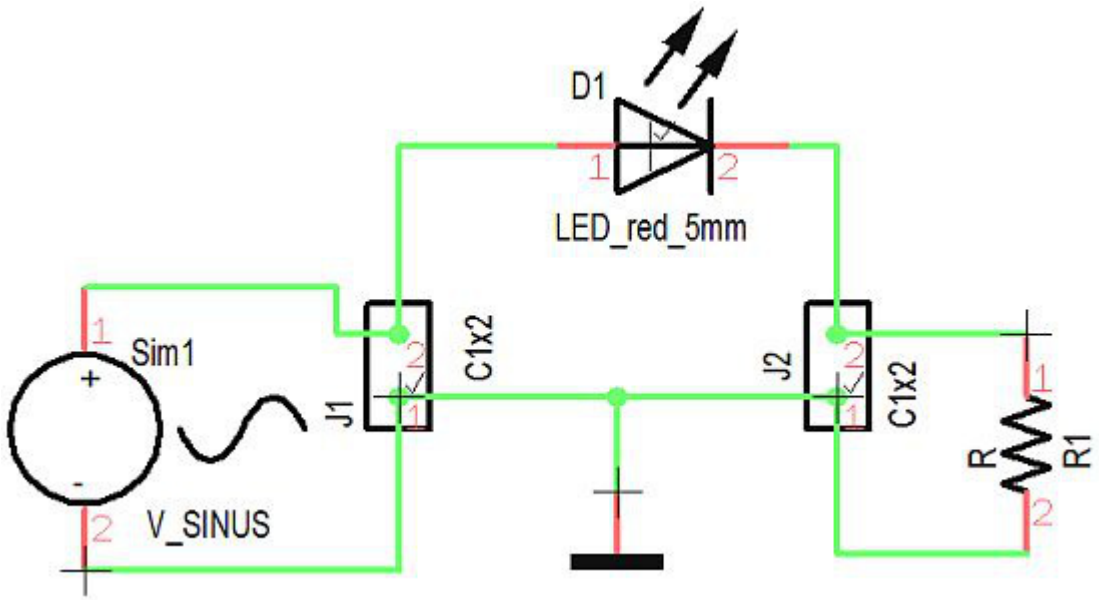
That's the way the schematic looks now:



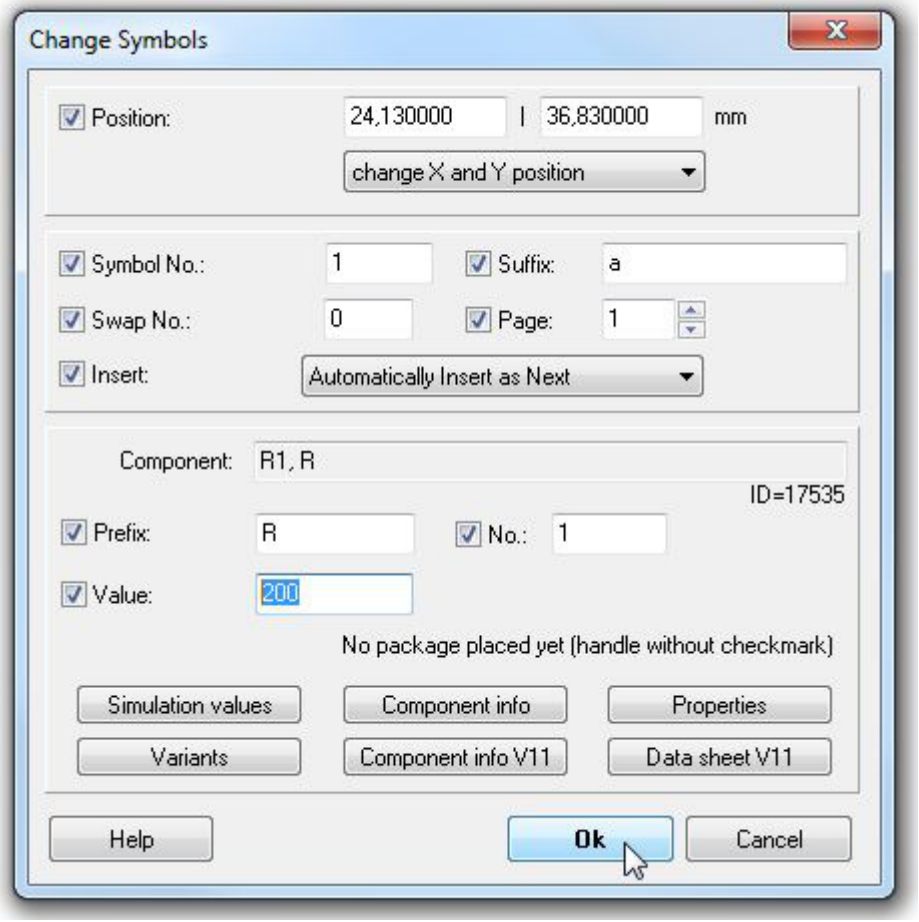
The load resistance we pick from the database. Open it by the key **[Ins]** and enter a bare "R" to the search line of the "Quick select" tab.



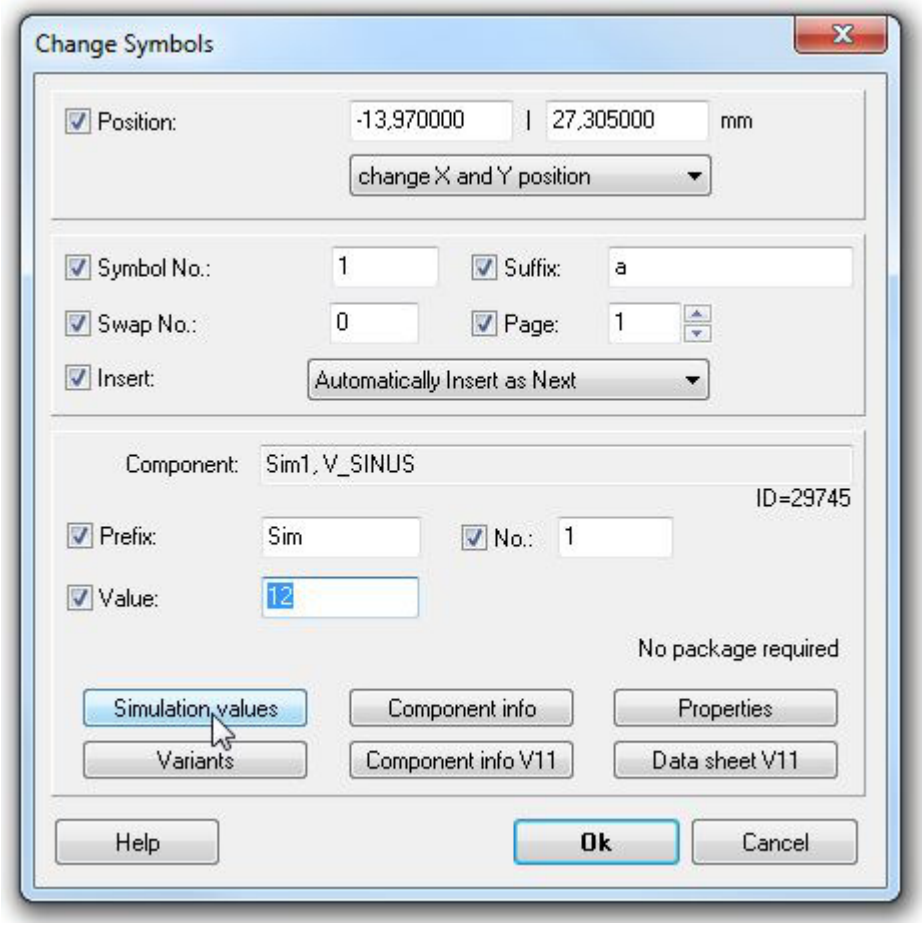
Import it to the schematic and and connect it accordingly. Having done some font adjustments the schematic now might look like this:



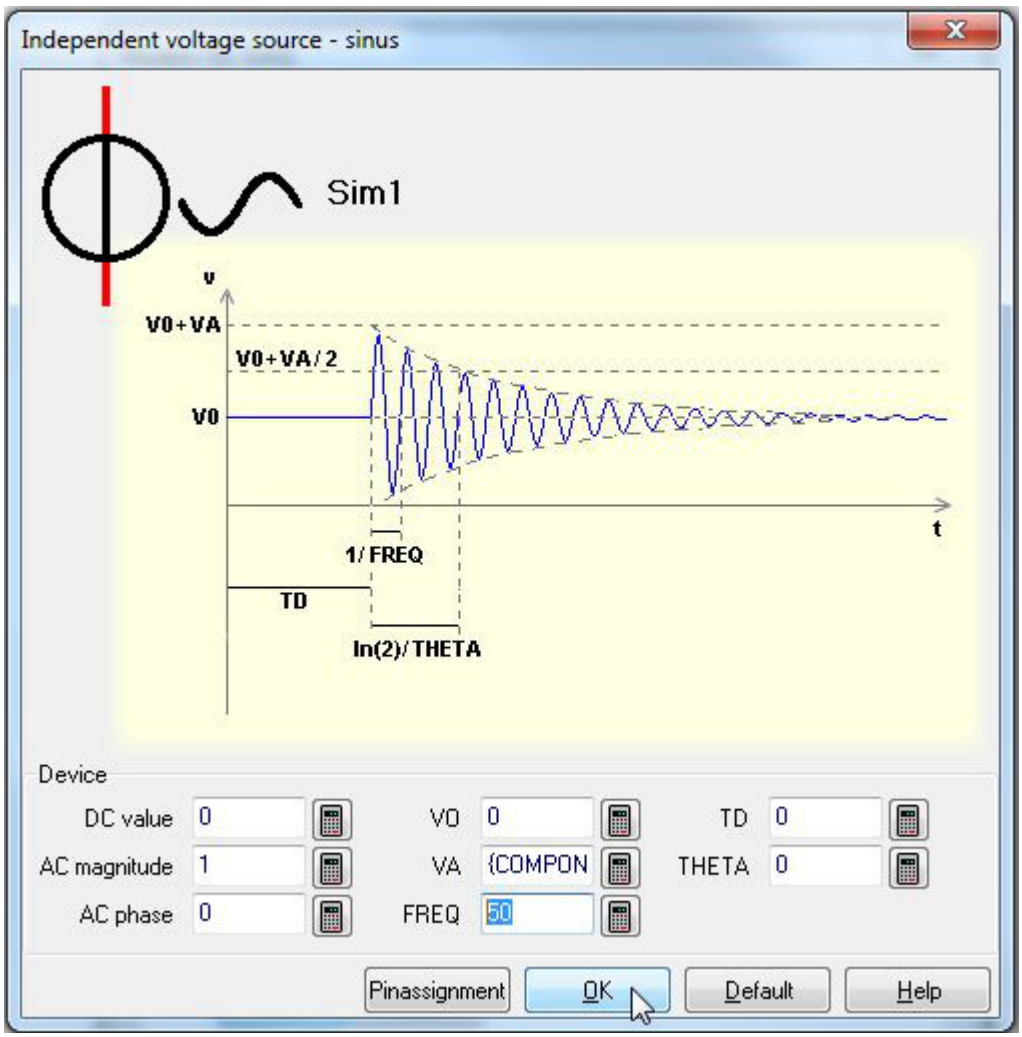
Now we set the values for source and load, first **M11** on the handle cross of the resistor. Now we enter component value "200", this stands for 200 Ohm:



Press OK. Now press **M11** upon the handle cross of the sinus source:



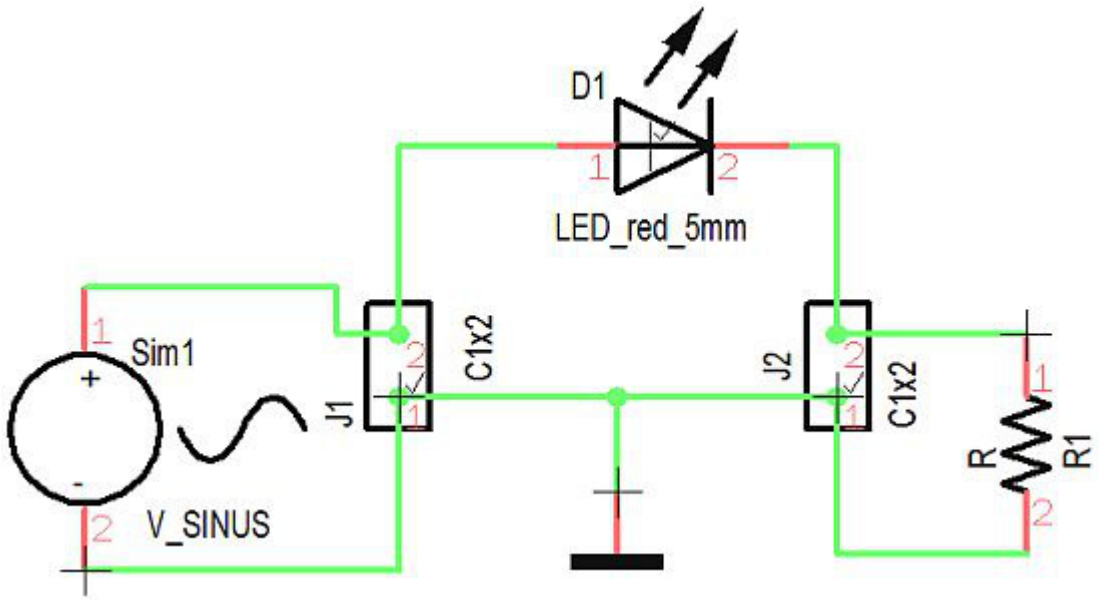
We define it's component value by 12 (meant is 12 V). By the use of the "Simulation values" button in the same dialog we set the frequency. In the following dialog press button "Edit". Afterwards a setting of the parameters can be done:



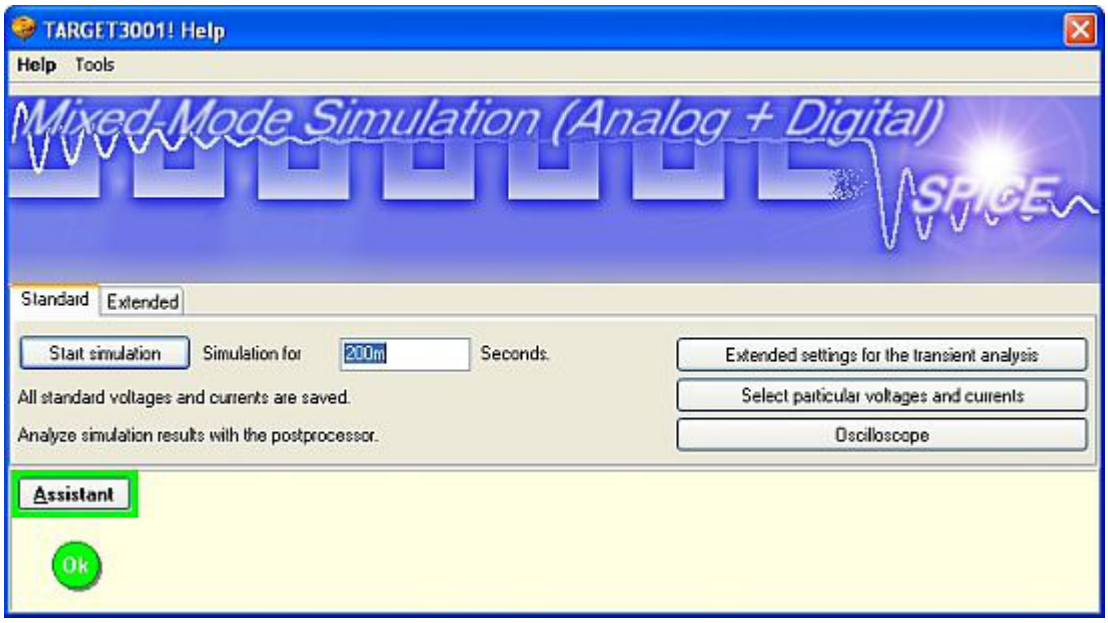
At "FREQ" enter value 50. Now the preliminaries are done. We confirm all dialogs and the schematic is ready for simulation.

Simulate the function Part 2

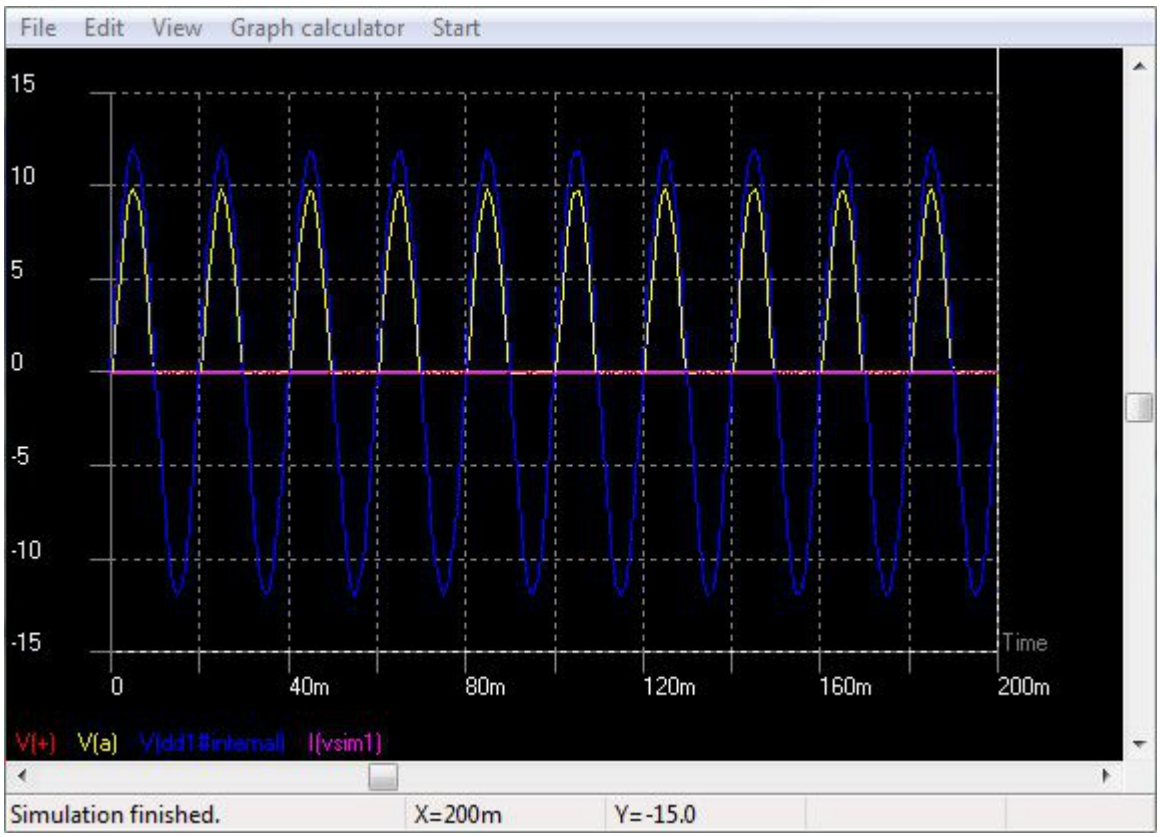
Based on the following schematic...



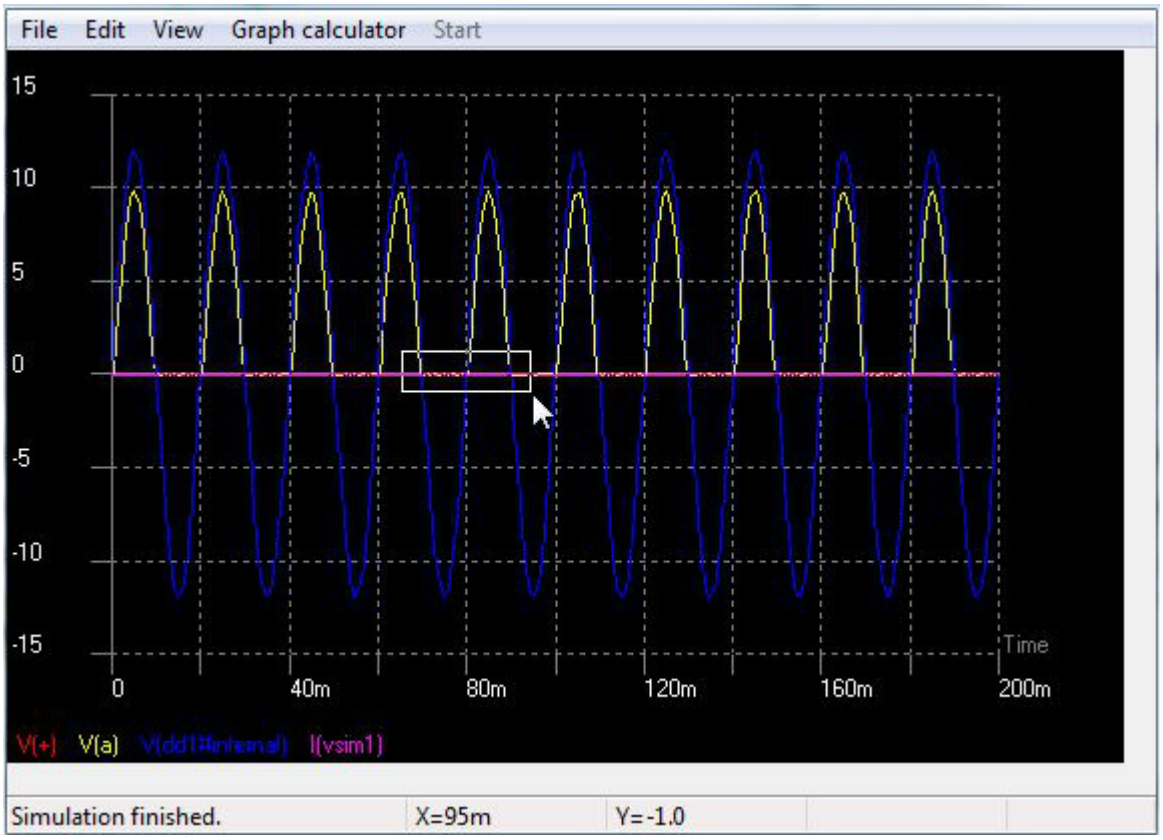
...we start the real simulation. We want to know which current flows in the LED at which tensions. First we start the simulation tool by the use of the function key **[F9]**. The following dialog appears:

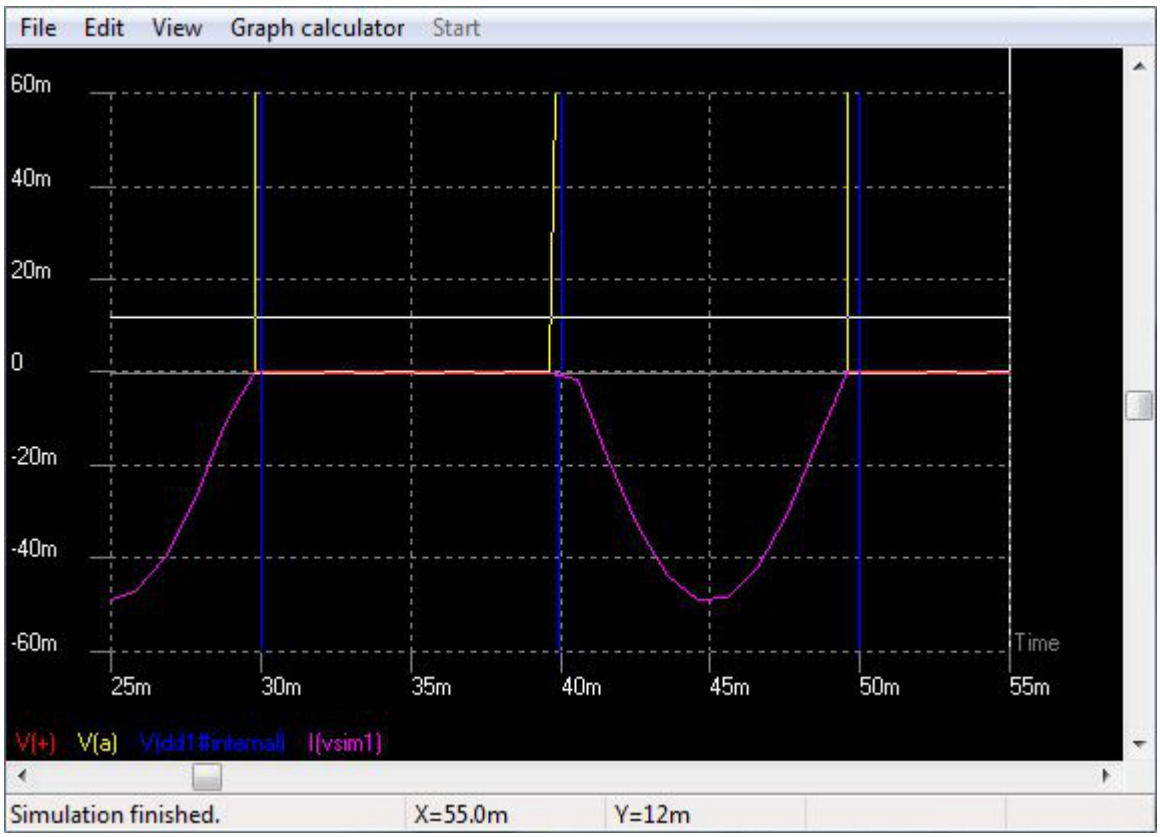


Look at the assistant. It shows green light and so we can "Start the simulation" for a time period of 200 milli seconds. In menu "Graph calculator" you can choose different colors for tension and current and receive for example:



If we now zoom in and view a section of the graph, we can inspect the course of the tension as well as the current:

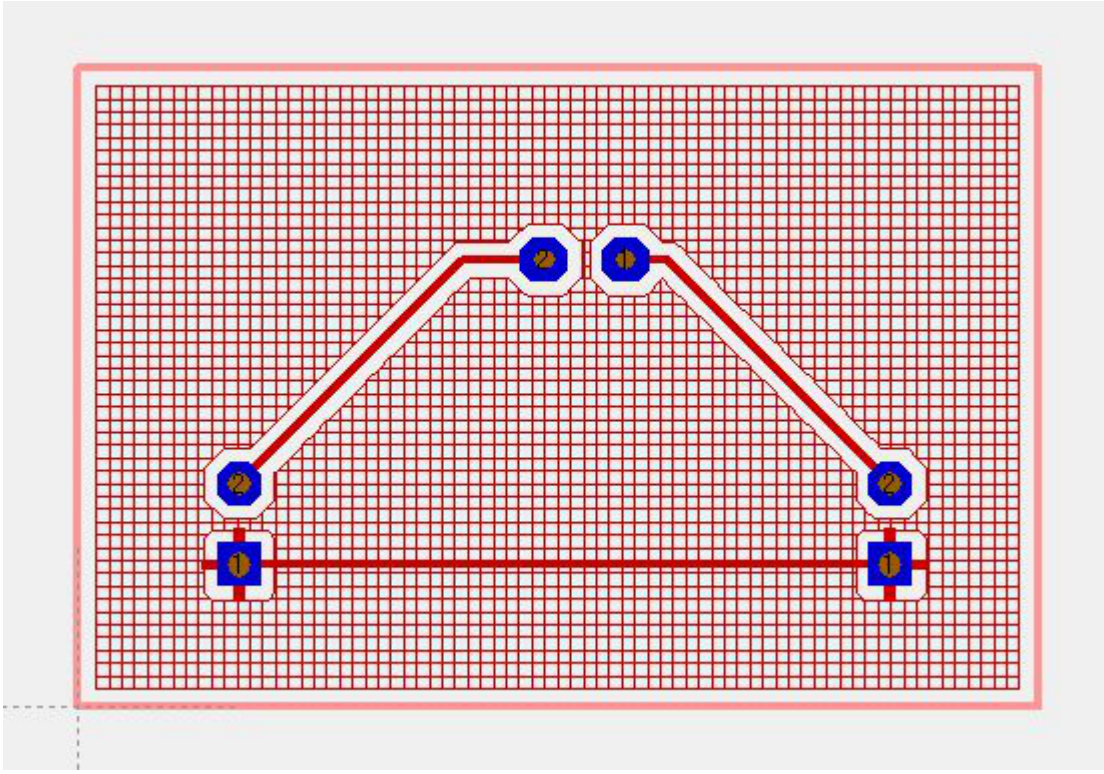




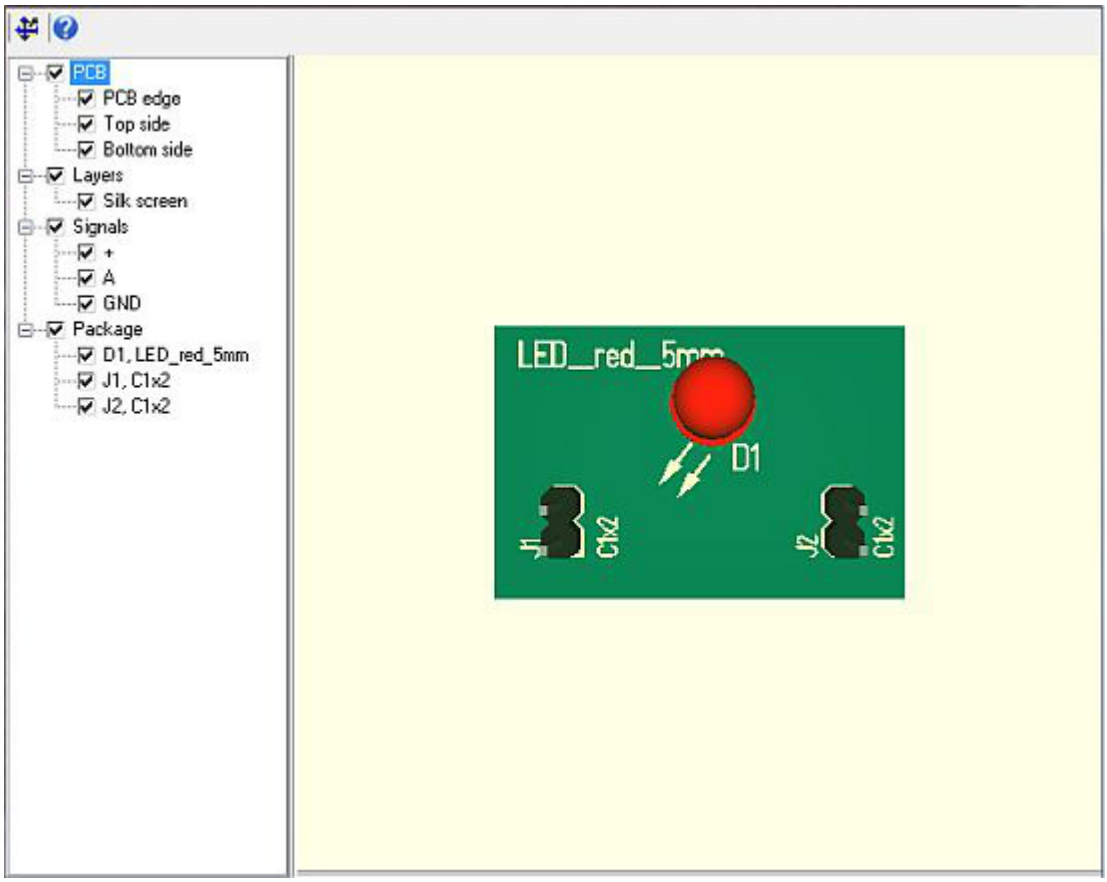
We see that the current (purple) does not exceed 50 milli Ampere during 20 milliseconds...

3D-view of the layout

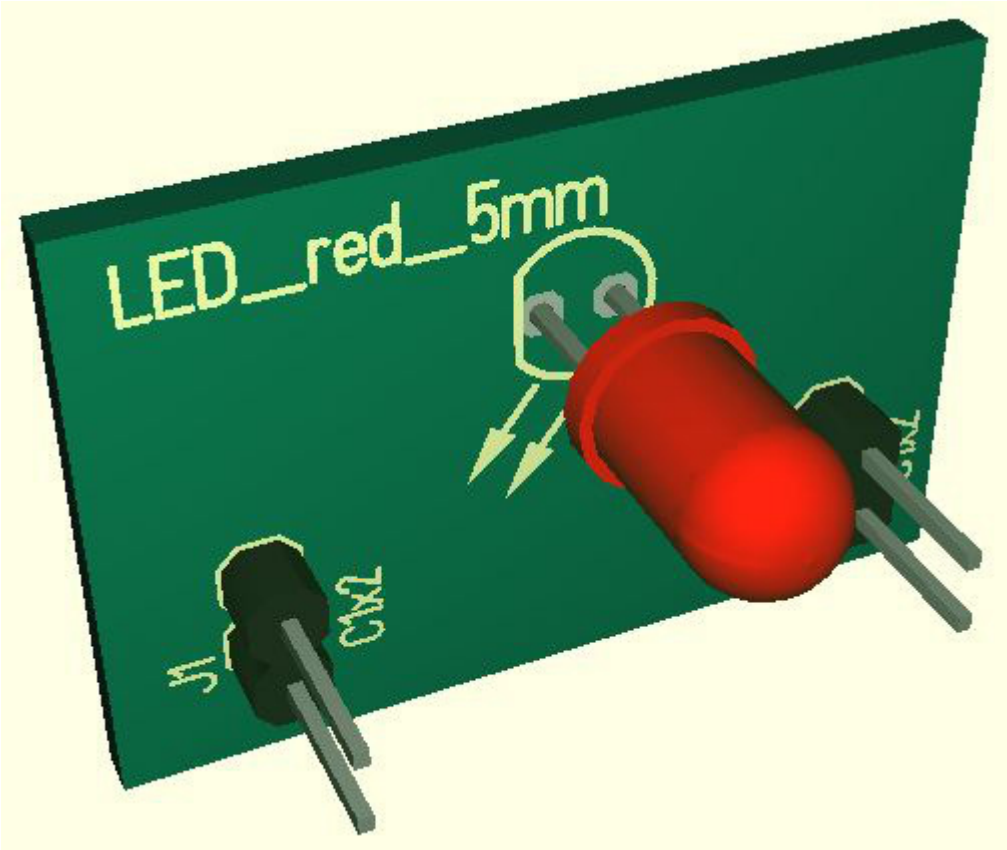
Currently the layout looks like that:



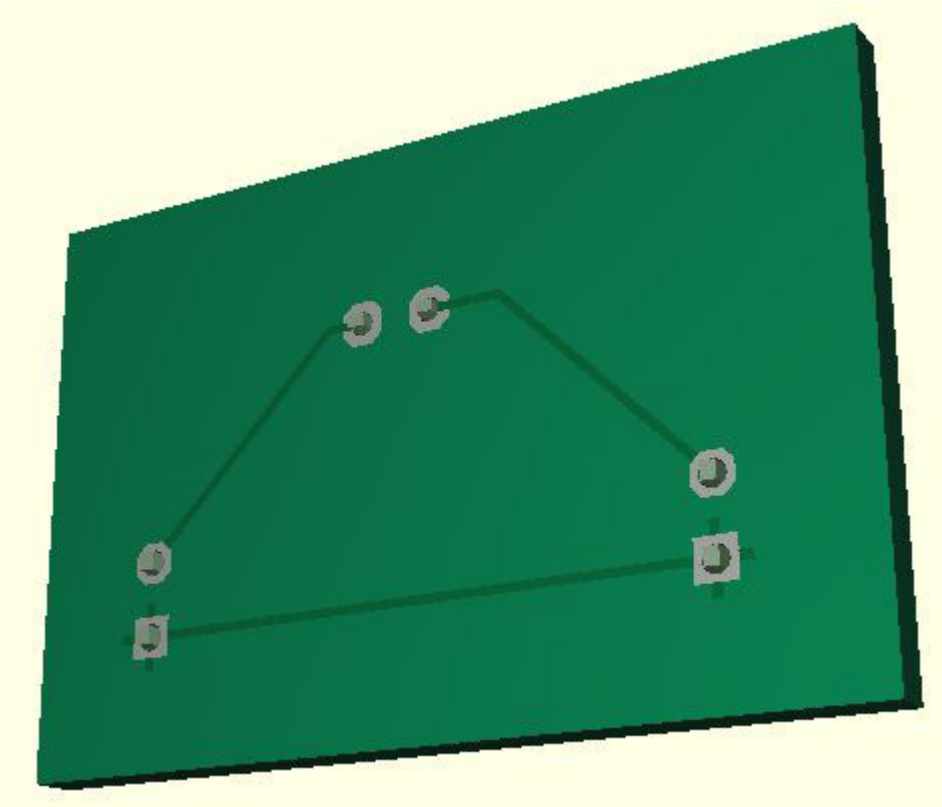
View it in 3D by menu "Actions/3D-View". After a short time of computation the result is:



You can turn the object to all sides by **M1H** or shift it by **M2H**. Zoom in and out using the mouse wheel.



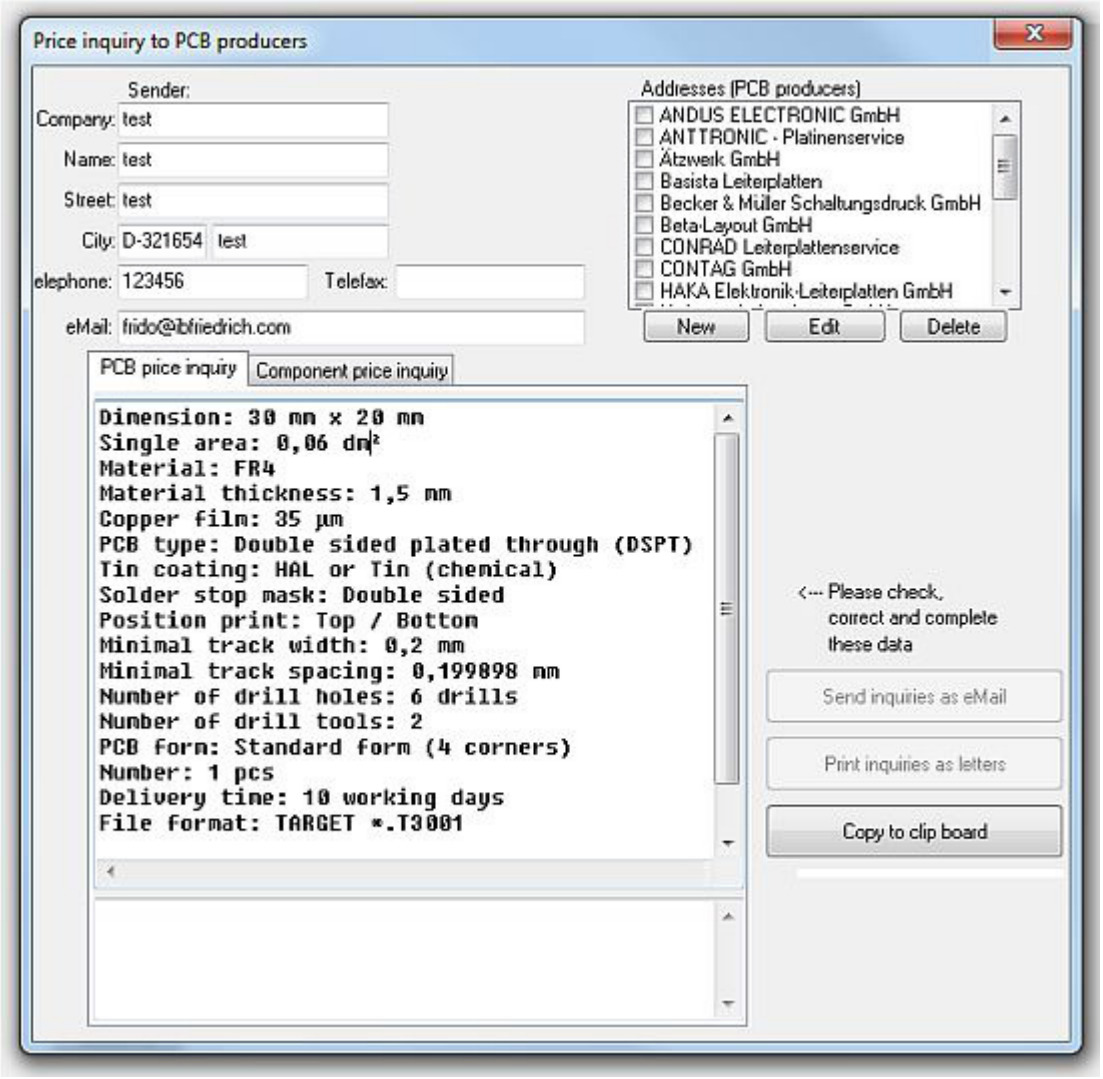
For reasons of computing time the ground plane with it's grid structure on the bottom side is omitted.



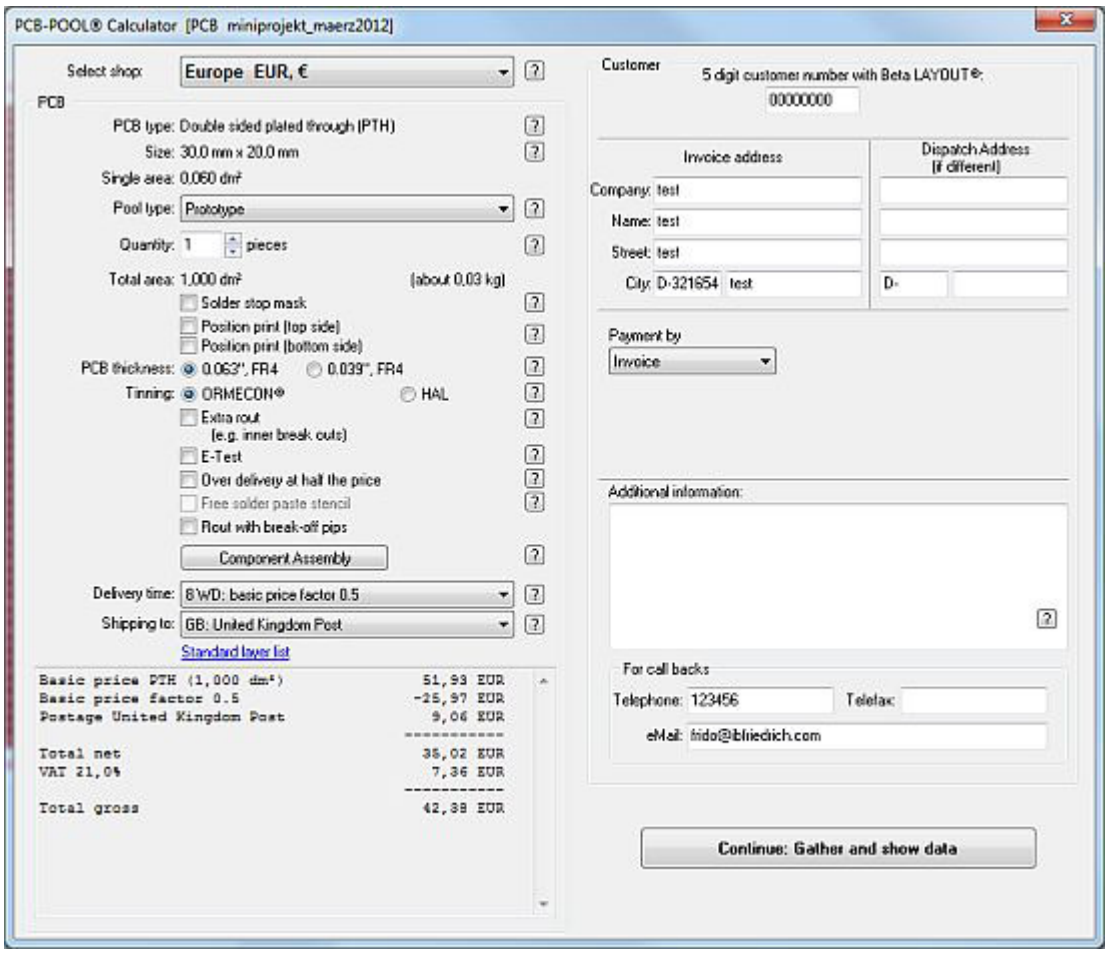
With a double click (**M11**) the PCB can be rotated continuously. Clicking **M11** again switches auto rotation off. During rotating you can interfere with **M1H**. Left side in the browser bar you can toggle parts visible and invisible with ticks. With a **M1** click onto the names you can mark the parts in blue color on the right side in the 3D view.

Produce a PCB

Besides the means of milling the PCB by HPGL data or etching by Gerber data (File/Import/Export Formats/Production/(x)Gerber and drill output PCBout), please see menu "File/Price inquiry PCB production" for generating a couple of inquiries quickly. The layout specifications are automatically overtaken to a form and can be mailed to several PCB houses at a time...

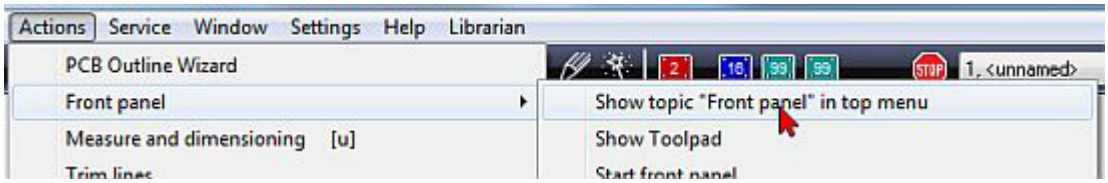


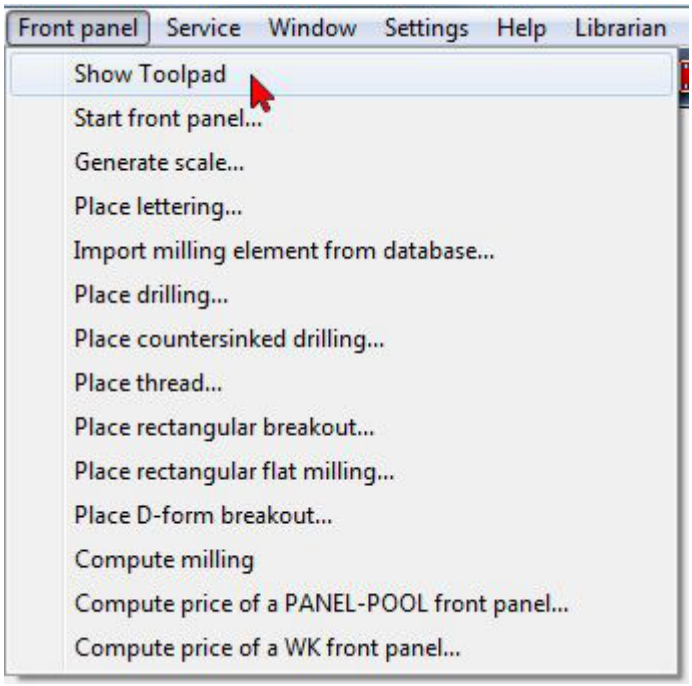
If you want to proceed quicker, calculate your project with the TARGET 3001! PCB-Pool® calculator. See Menu "File/Produce PCB in PCB-Pool®". PCB-Pool® is a German pcb manufacturer who directly accepts TARGET-files:



A prototype (1 piece) of this project costs EUR 42,38 without position print. If you like it, fill in the form, enter your personal data and the credit card data and submit the form. A quicker solution offers the "NextDayDelivery"-Service. That means if you offer the project file before 8:30 am to PCB-Pool®, your PCB is dispatched the next working day. This is a fascinating cooperation between TARGET 3001! and PCB-Pool®. In case you're really in a hurry...

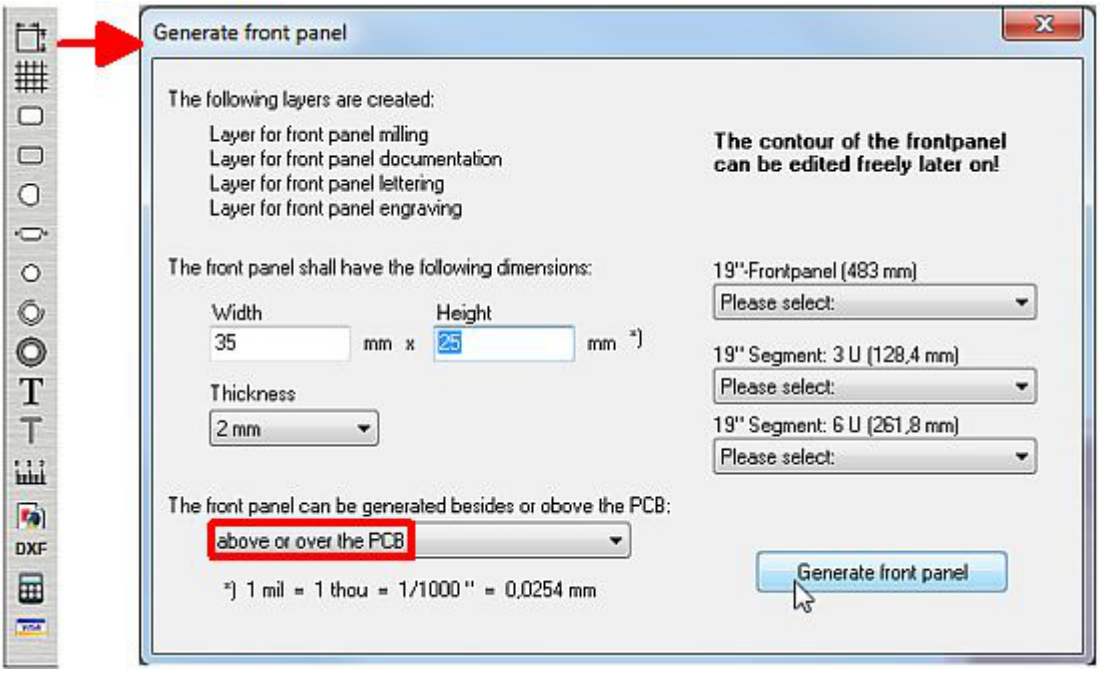
Design and produce a frontpanel

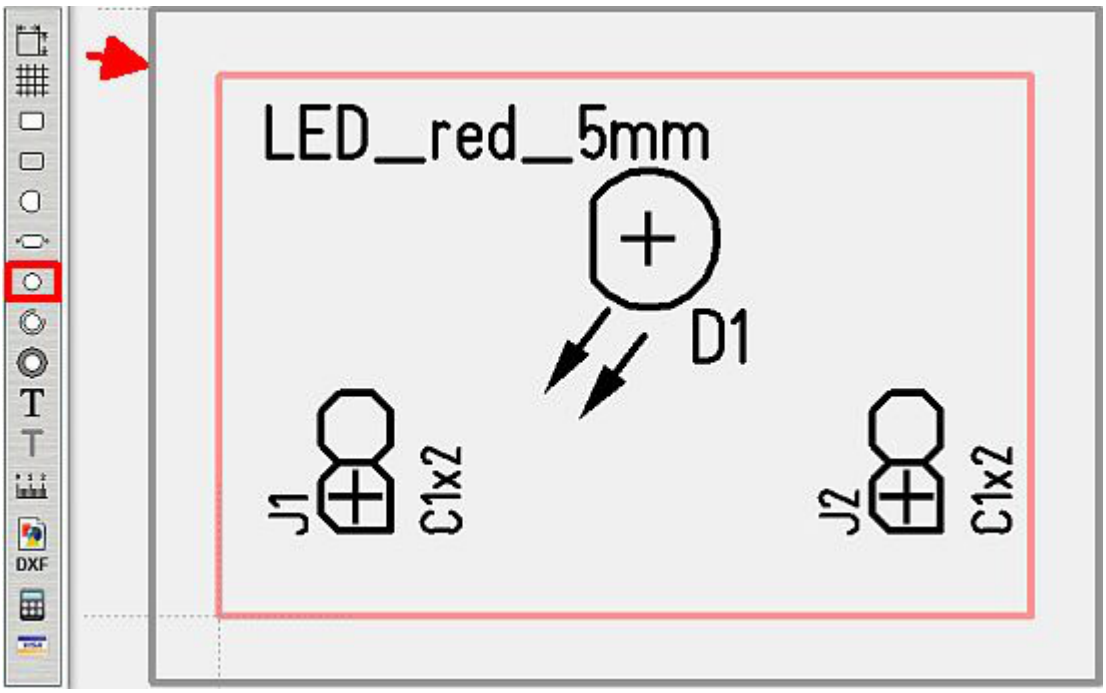



Front panel
Service
Window
Settings
Help
Librarian
Show Toolpad
Start front panel...
Generate scale...
Place lettering...
Import milling element from database...
Place drilling...
Place countersinked drilling...
Place thread...
Place rectangular breakout...
Place rectangular flat milling...
Place D-form breakout...
Compute milling
Compute price of a PANEL-POOL front panel...
Compute price of a WK front panel...

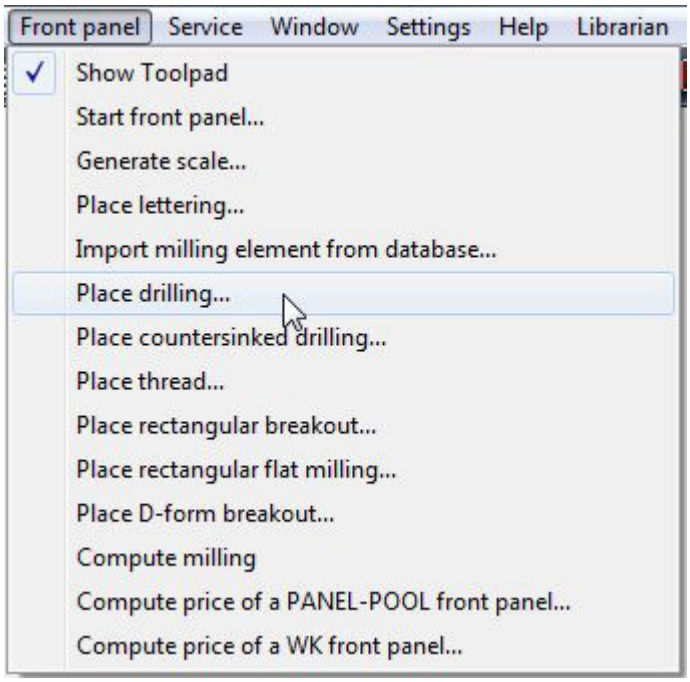

DXF

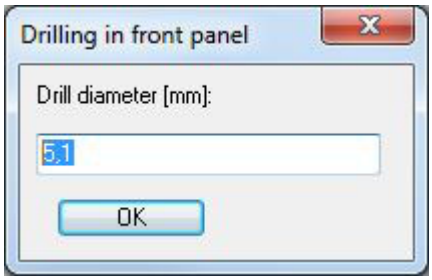
Shows left

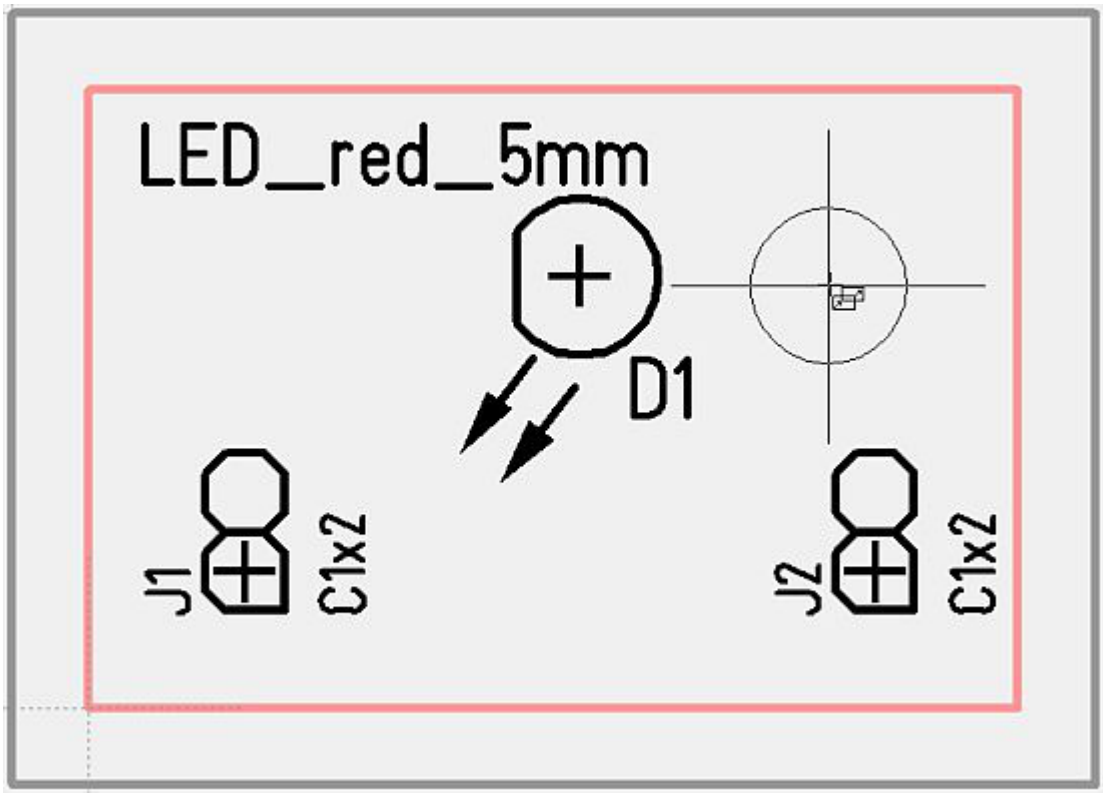

DXF
Generate front panel
The following layers are created:
Layer for front panel milling
Layer for front panel documentation
Layer for front panel lettering
Layer for front panel engraving
The contour of the frontpanel can be edited freely later on!
The front panel shall have the following dimensions:
Width
35
mm x
Height
25
mm *)
Thickness
2 mm
19"-Frontpanel (483 mm)
Please select:
19" Segment: 3 U (128,4 mm)
Please select:
19" Segment: 6 U (261,8 mm)
Please select:
The front panel can be generated besides or above the PCB:
above or over the PCB
*) 1 mil = 1 thou = 1/1000 " = 0,0254 mm
Generate front panel

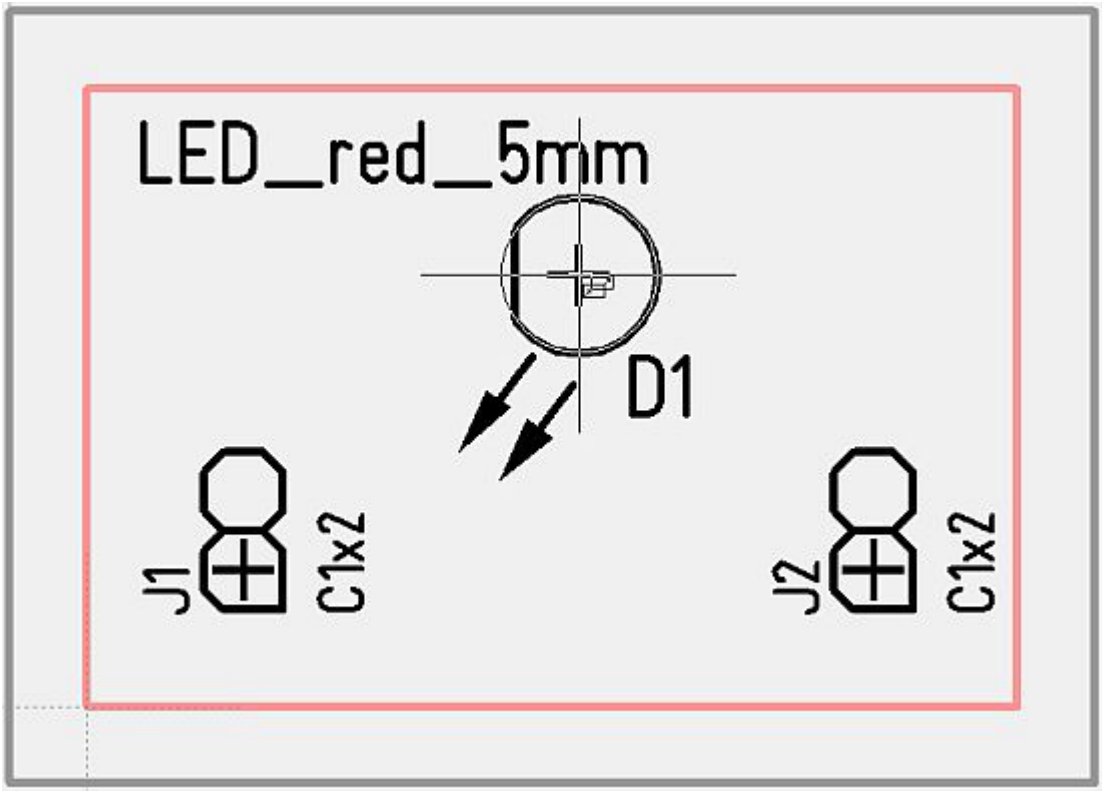


The grey frame is the outline of your front panel. Now tick the icon in order to place a drilling. Alternatively use the the Front panel menu option "Place drilling".

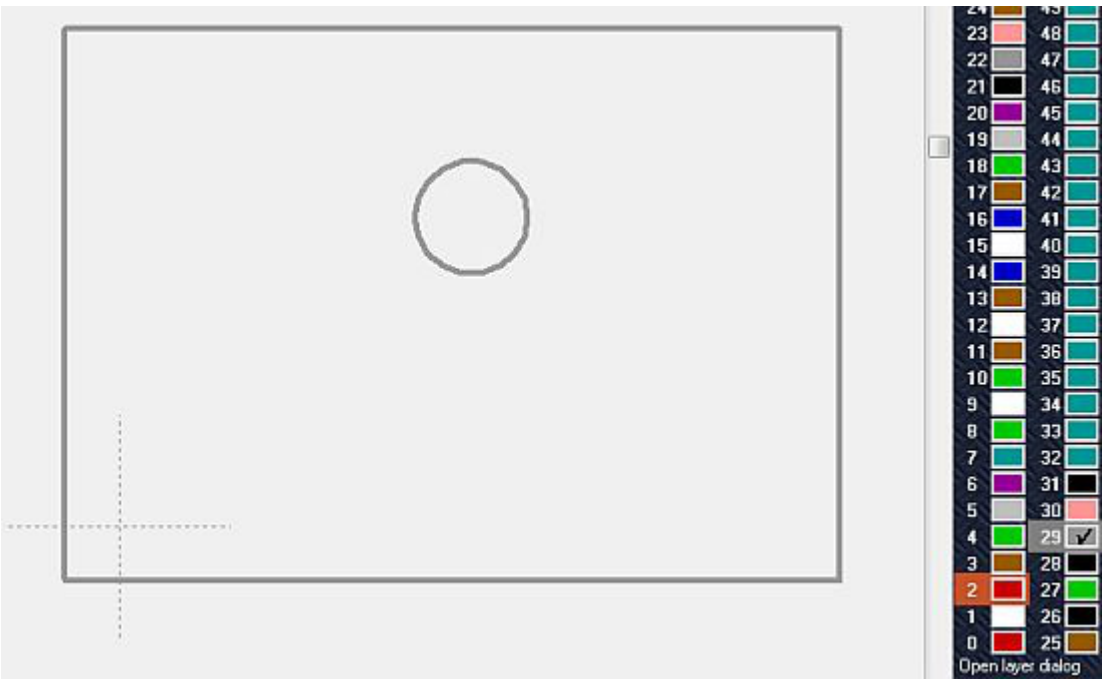






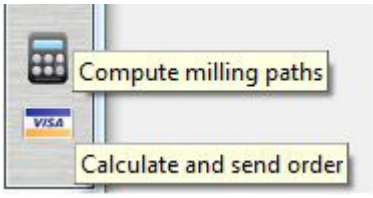


All other layers are faded out (only needed for this view)

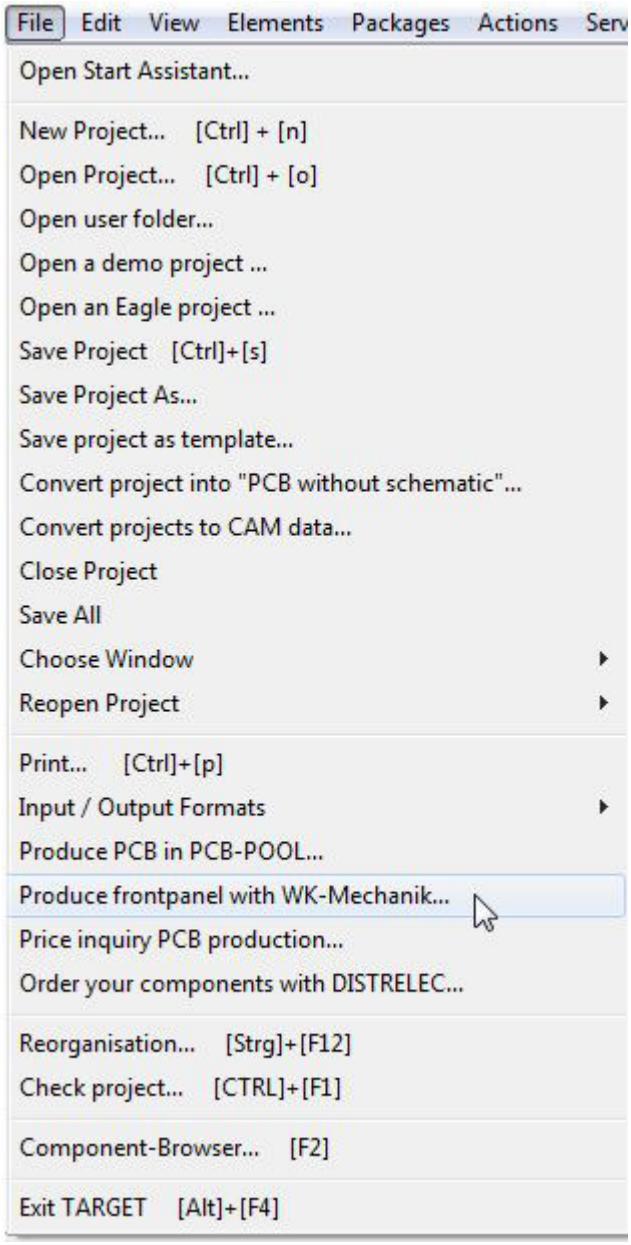


When it comes to production you have three options:

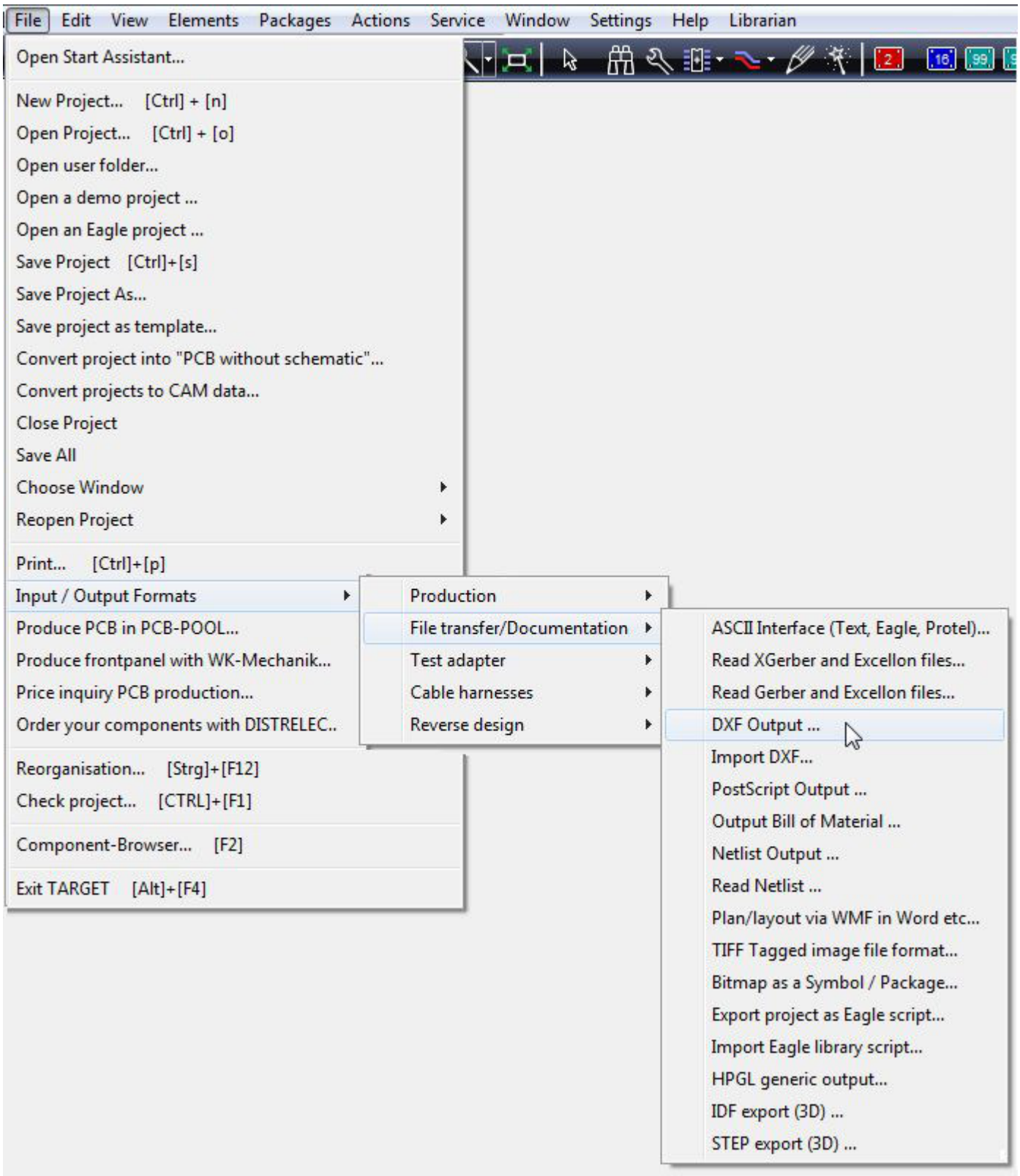
1. Use the sidebar calculator and order your front panel at Panel-POOL®

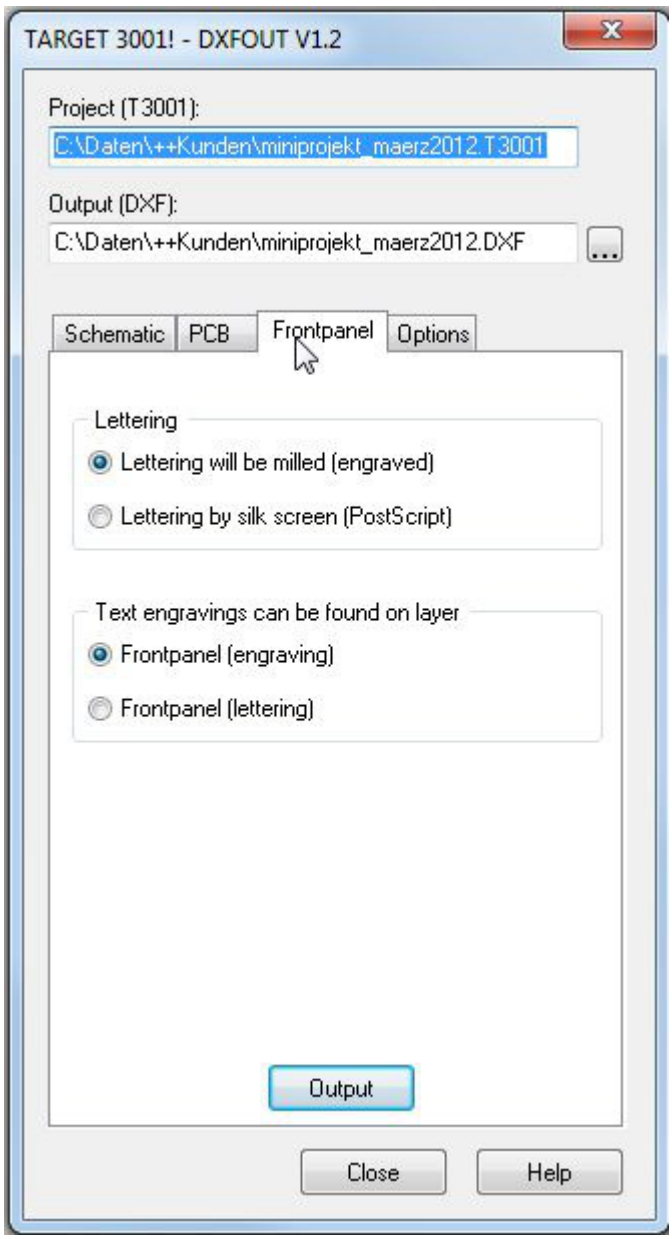


2. Use the WK-Mechanik calculator and order your front panel there. WK-Mechanik does not use digital print technique for printing logos etc. but is able to offer bended panels and casing elements.



3. Produce at any other front panel house. The front panel data are generated in DXF for production anywhere. This is the way how to do it:


File
Edit
View
Elements
Packages
Actions
Service
Window
Settings
Help
Librarian
Open Start Assistant...
New Project... [Ctrl] + [n]
Open Project... [Ctrl] + [o]
Open user folder...
Open a demo project ...
Open an Eagle project ...
Save Project [Ctrl]+[s]
Save Project As...
Save project as template...
Convert project into "PCB without schematic"...
Convert projects to CAM data...
Close Project
Save All
Choose Window
Reopen Project
Print... [Ctrl]+[p]
Input / Output Formats
Produce PCB in PCB-POOL...
Produce frontpanel with WK-Mechanik...
Price inquiry PCB production...
Order your components with DISTRELEC..
Reorganisation... [Strg]+[F12]
Check project... [CTRL]+[F1]
Component-Browser... [F2]
Exit TARGET [Alt]+[F4]
Production
File transfer/Documentation
Test adapter
Cable harnesses
Reverse design
ASCII Interface (Text, Eagle, Protel)...
Read XGerber and Excellon files...
Read Gerber and Excellon files...
DXF Output ...
Import DXF...
PostScript Output ...
Output Bill of Material ...
Netlist Output ...
Read Netlist ...
Plan/layout via WMF in Word etc...
TIFF Tagged image file format...
Bitmap as a Symbol / Package...
Export project as Eagle script...
Import Eagle library script...
HPGL generic output...
IDF export (3D) ...
STEP export (3D) ...


TARGET 3001! - DXFOUT V1.2
Project (T3001):
C:\Daten\++Kunden\miniprojekt_maerz2012.T3001
Output (DXF):
C:\Daten\++Kunden\miniprojekt_maerz2012.DXF
Schematic
PCB
Frontpanel
Options
Lettering
Lettering will be milled (engraved)
Lettering by silk screen (PostScript)
Text engravings can be found on layer
Frontpanel (engraving)
Frontpanel (lettering)
Output
Close
Help