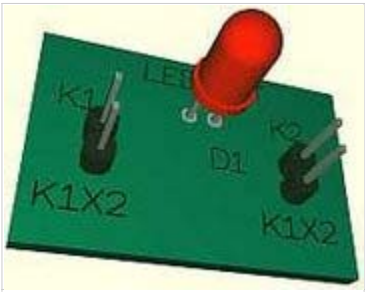
CrashCourse2

From IBF-Wiki

Based on a small project (one LED between two connectors) see how easy the use of TARGET 3001! is:



Design a LED between two connectors in minutes!

 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

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=CrashCourse2"

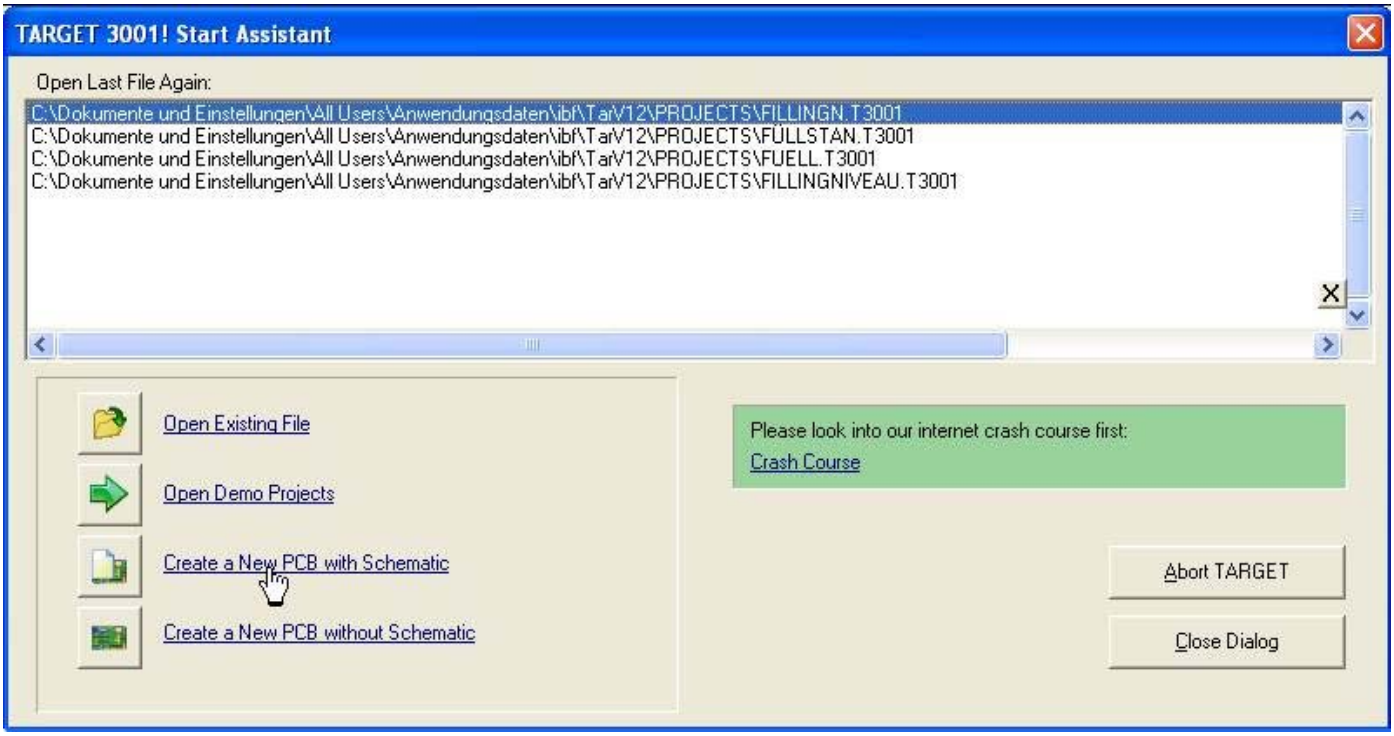
Categories: First steps

- This page was last modified 12:13, 10 May 2007.

Begin a new project

From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File



With the Start assistant you decide whether to open an existing project or to create a new project either with schematic or without. In our case we decide for a new project **with** schematic. An empty schematic page opens...

Next step
One step back

Back to overview

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Begin_a_new_project"

Categories: About TARGET 3001!

- This page was last modified 09:37, 30 August 2007.

Import a component symbol to the schematic

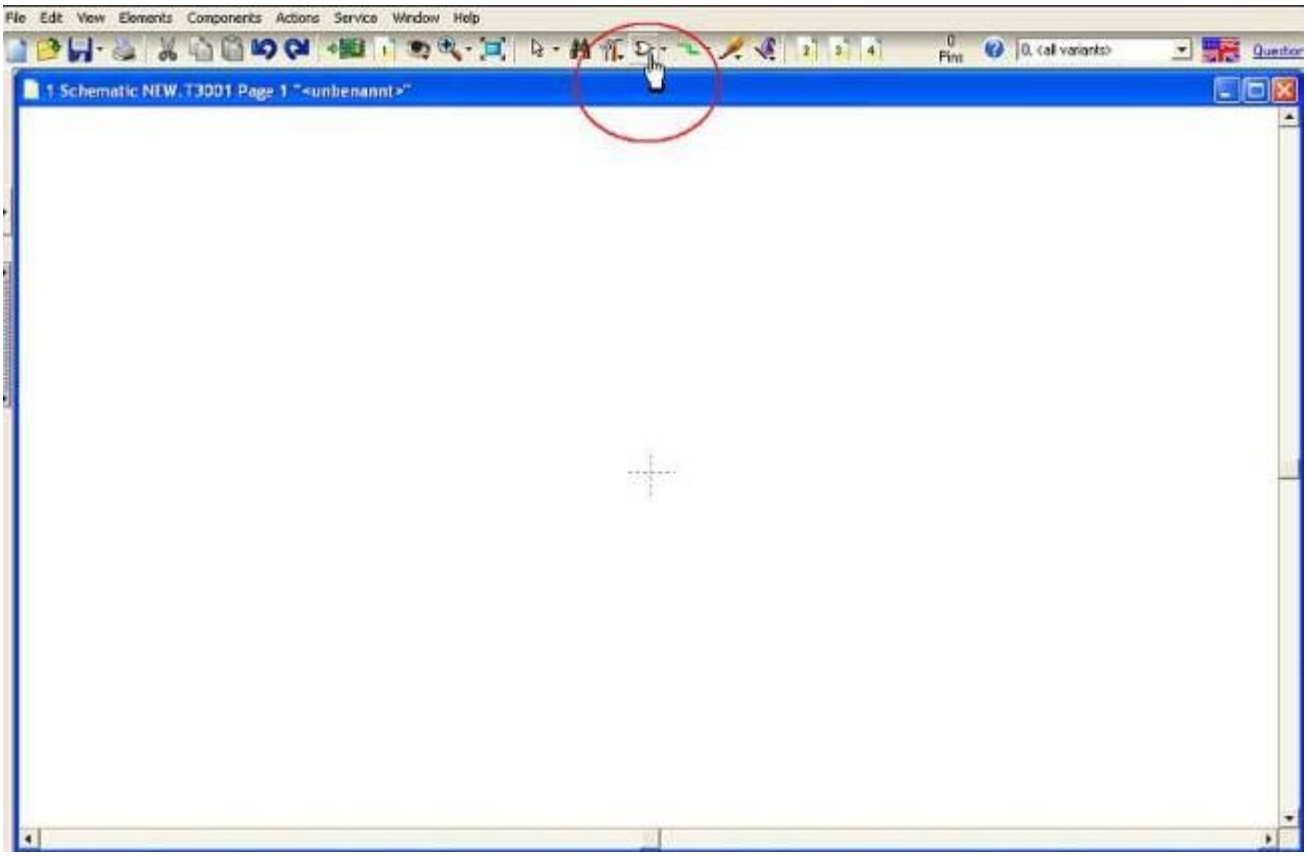
From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

The following image shows an empty schematic page. Now we import a symbol from the component library. Click **M1** upon the symbol button on the icon bar. You might also press button **[Ins]** from the keyboard.

More details?

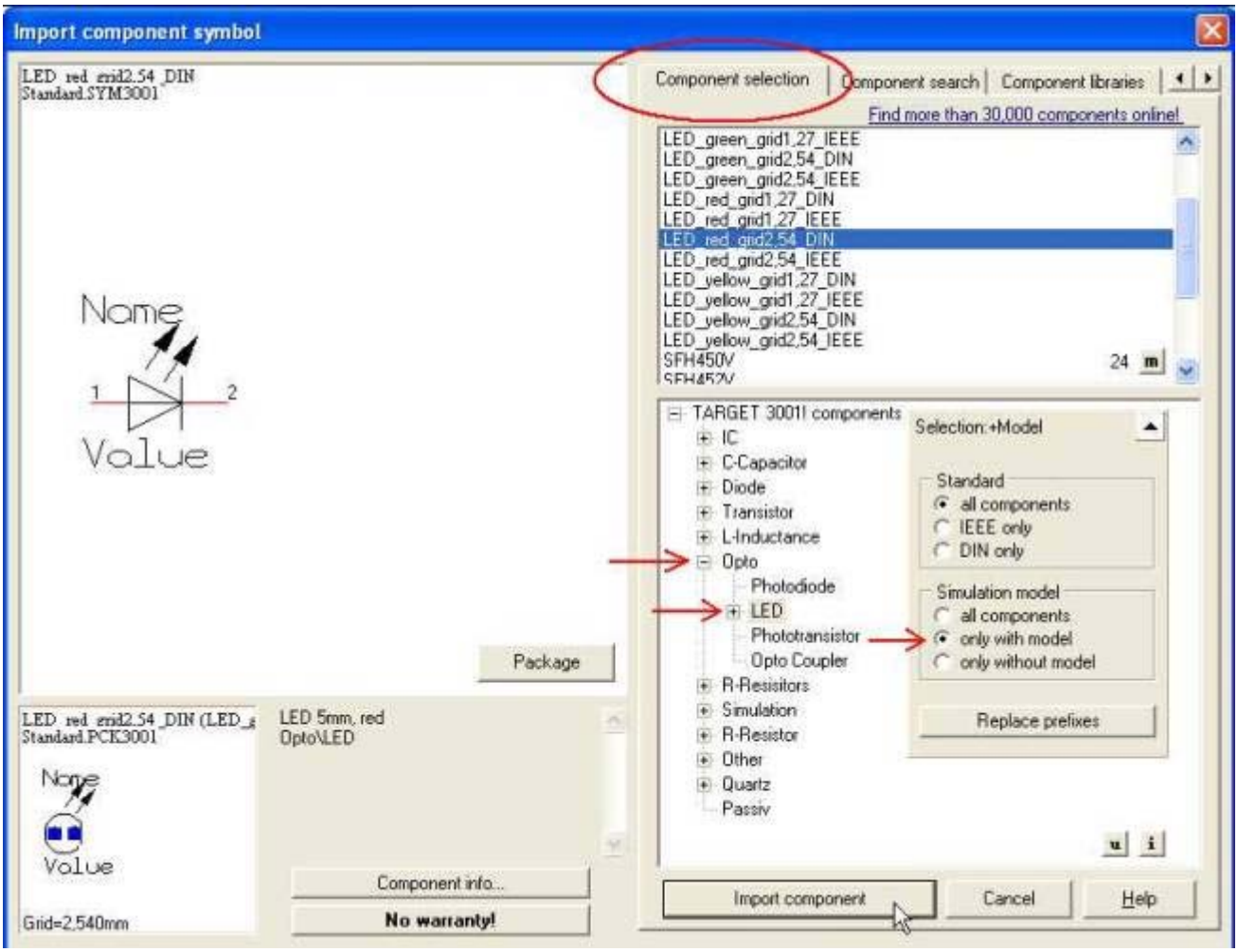
Origin
Grid
Mousekeys
Key combinations
Zoom



The library browser opens and allows component search and import. In our case we pick a **LED red, 5mm**.

More details?

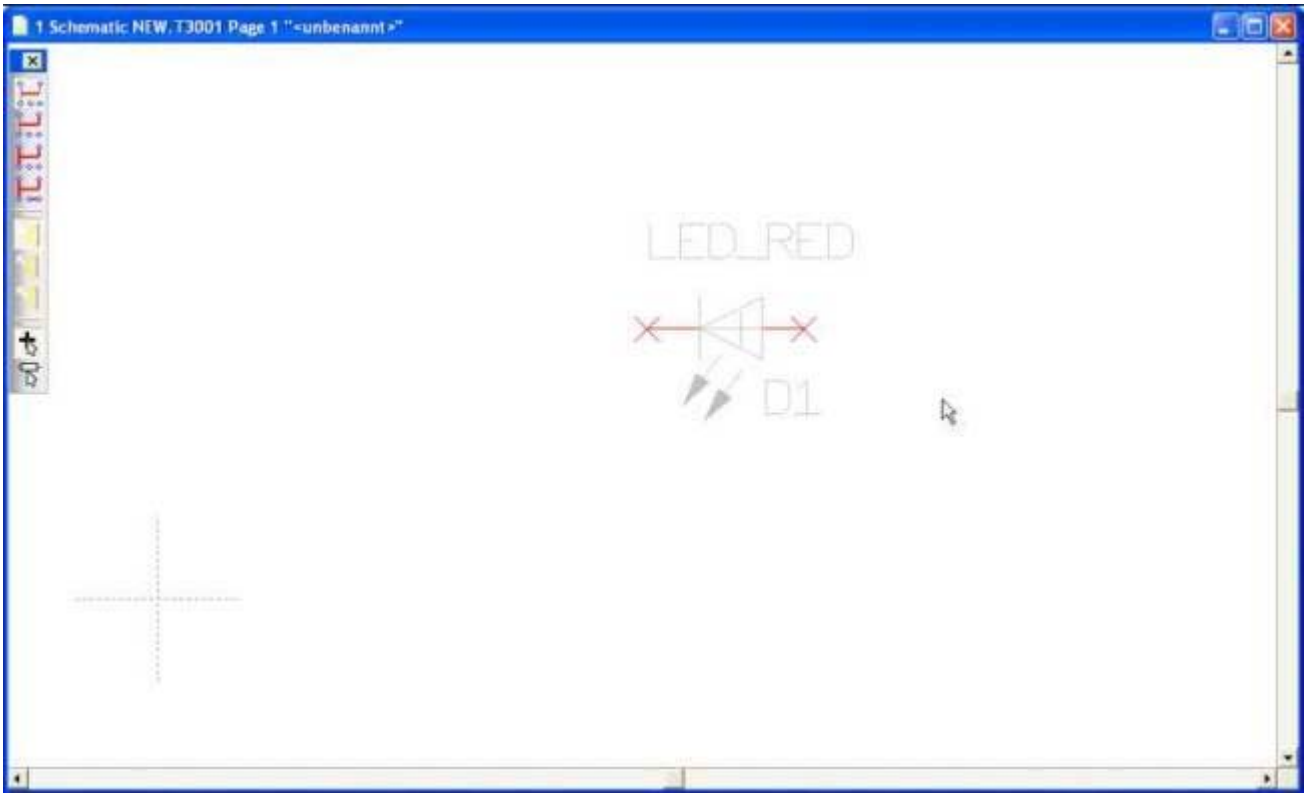
Libraries



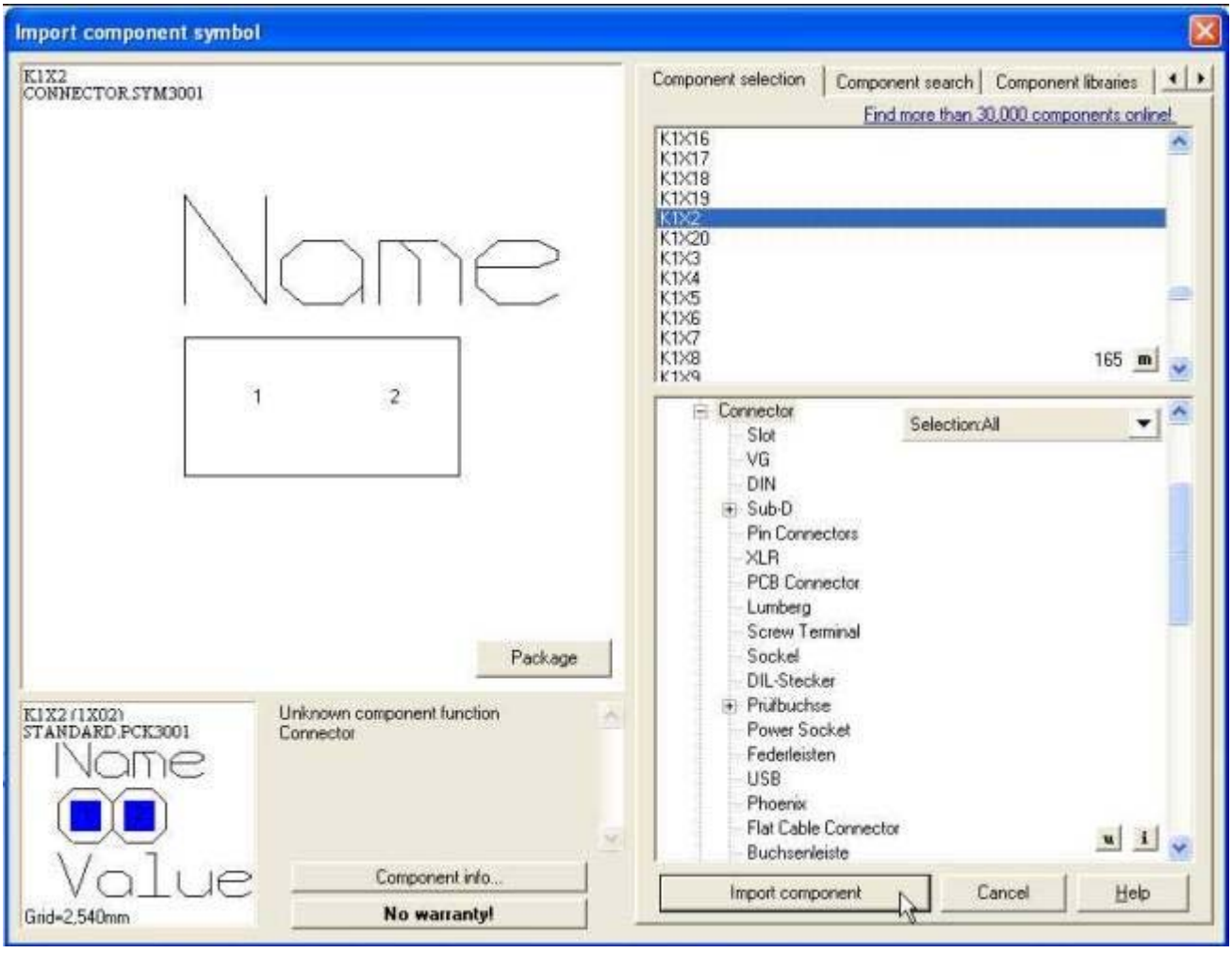
After clicking **"Import component"**, place the symbol on the schematic by M1 preferably within the first quadrant of the underlying coordinate system. In case the component value appears too big, please change it by M11 upon the handle cross. In the appearing dialog modify the entry at "Component Value" to zu LED_RED. "D1" is the "Component Name".

More details?

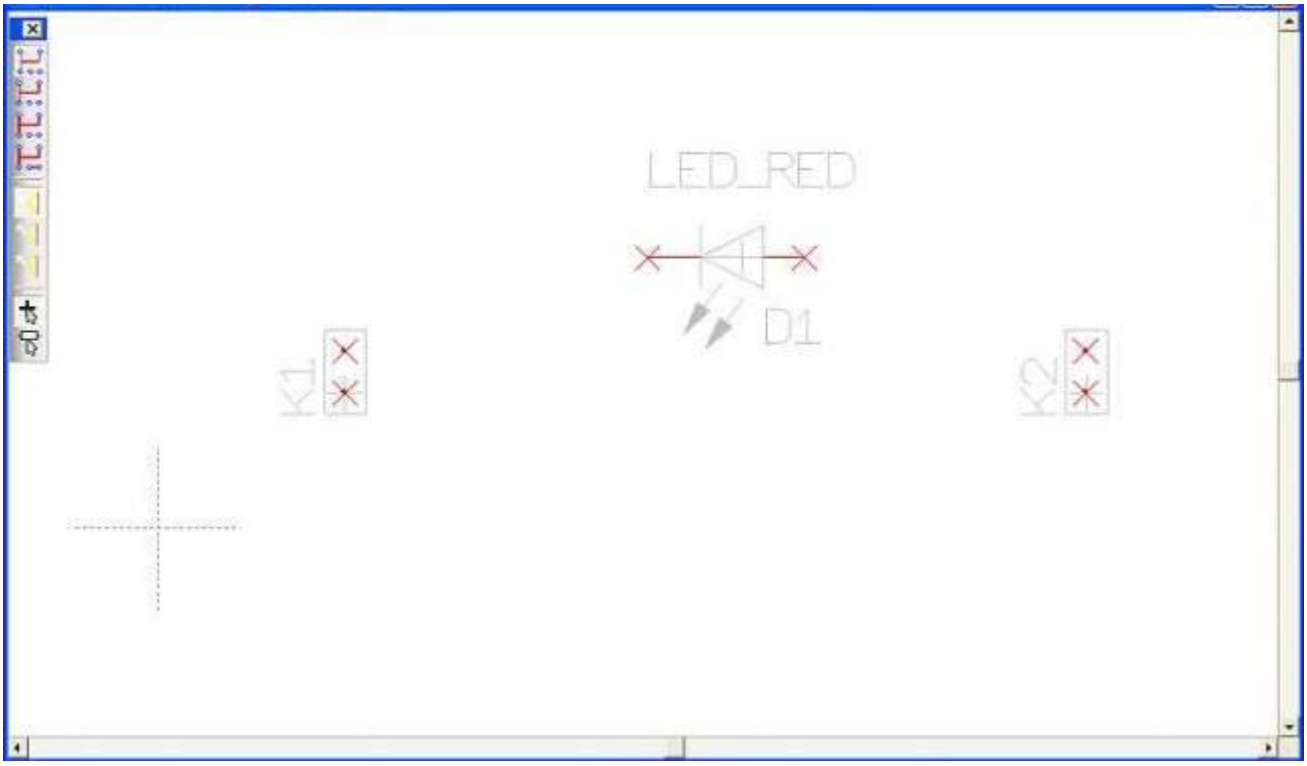
Select Displace / Move / Relocate from



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 placement by **M2**. That's the way our schematic might look like by now:



Next step
One step back

Back to overview

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Import_a_component_symbol_to_the_schematic"

Categories: Components/Libraries | Schematic

- This page was last modified 09:40, 30 August 2007.

Connecting the pins

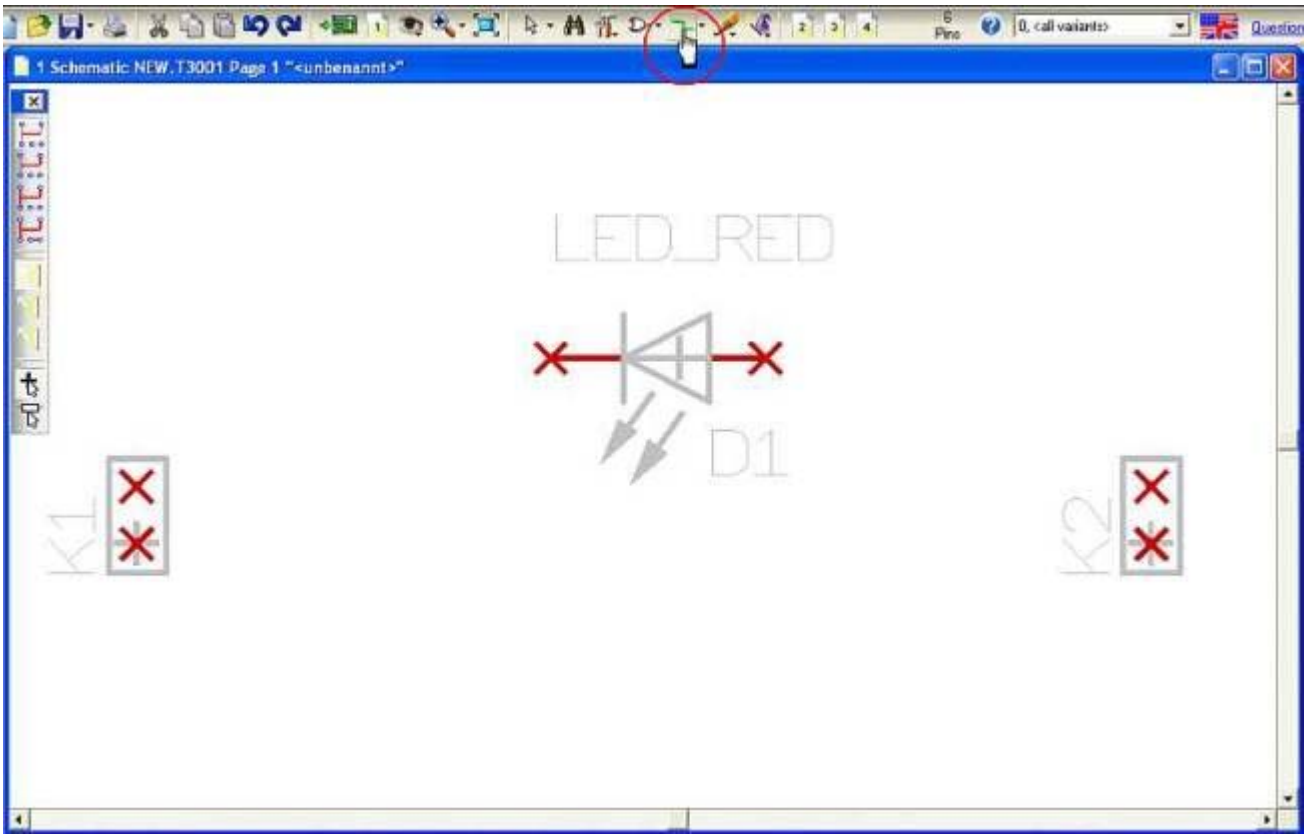
From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

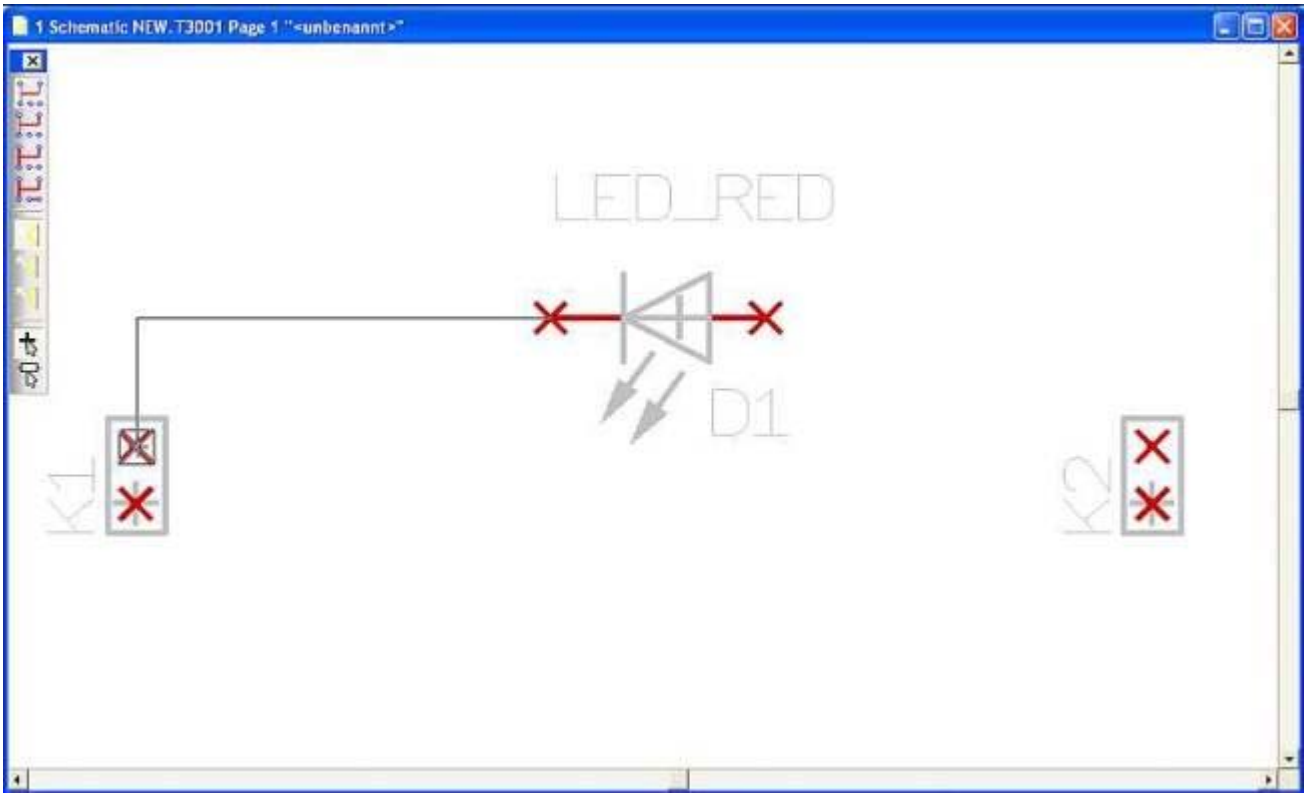
The pins of component symbols are connected by the function "Place wire" under the icon  You can either use key **[2]** from the keyboard to open this function.

More details?

Signal Page
Lead a signal to a next page
Grid
Component Server
Component Management System
Block Diagram
Bus



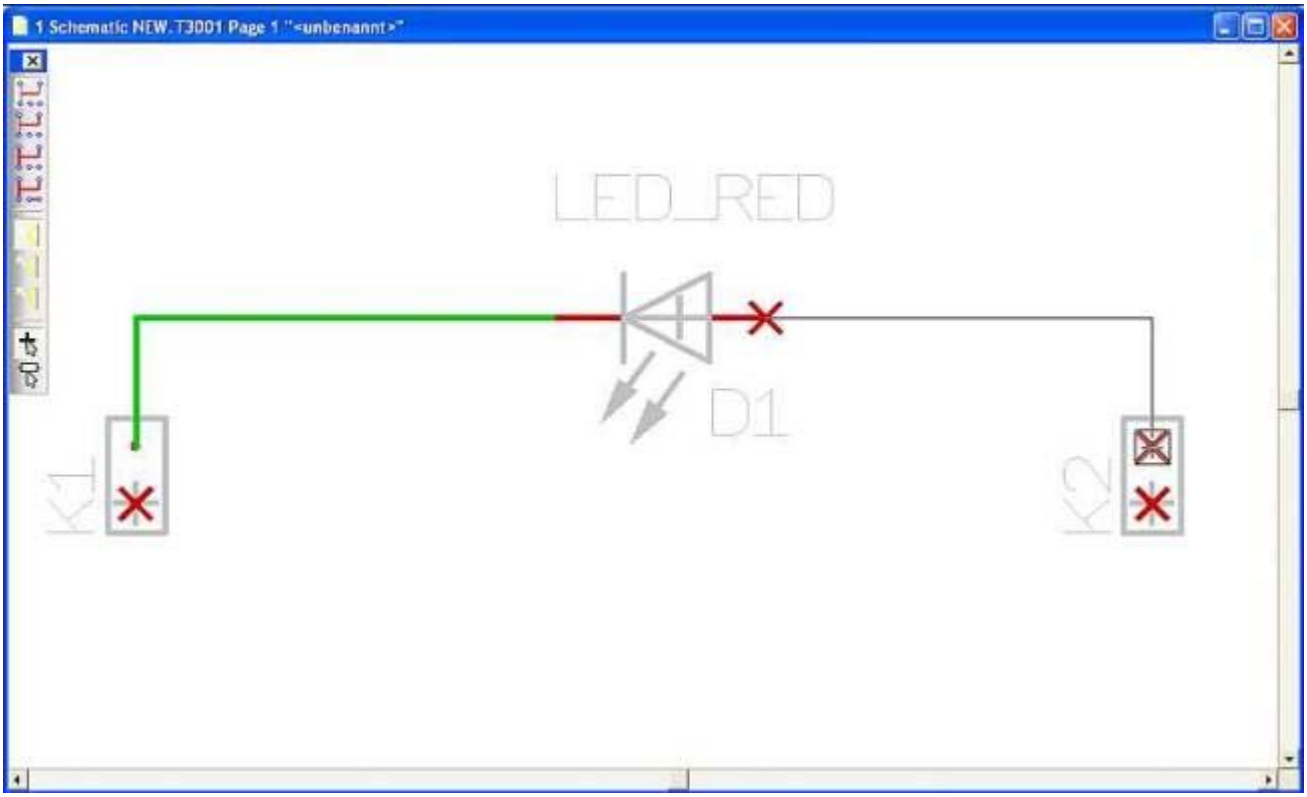
Now lead the signal by **M1H** from the cathode of the diode to a pin of a connector. The signal has adapted the pin function and leads it further as a signal name. When you have created the connection cut the wire by the use of **[Esc]** or by **M12** to proceed with a new connection. Toggle the bending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



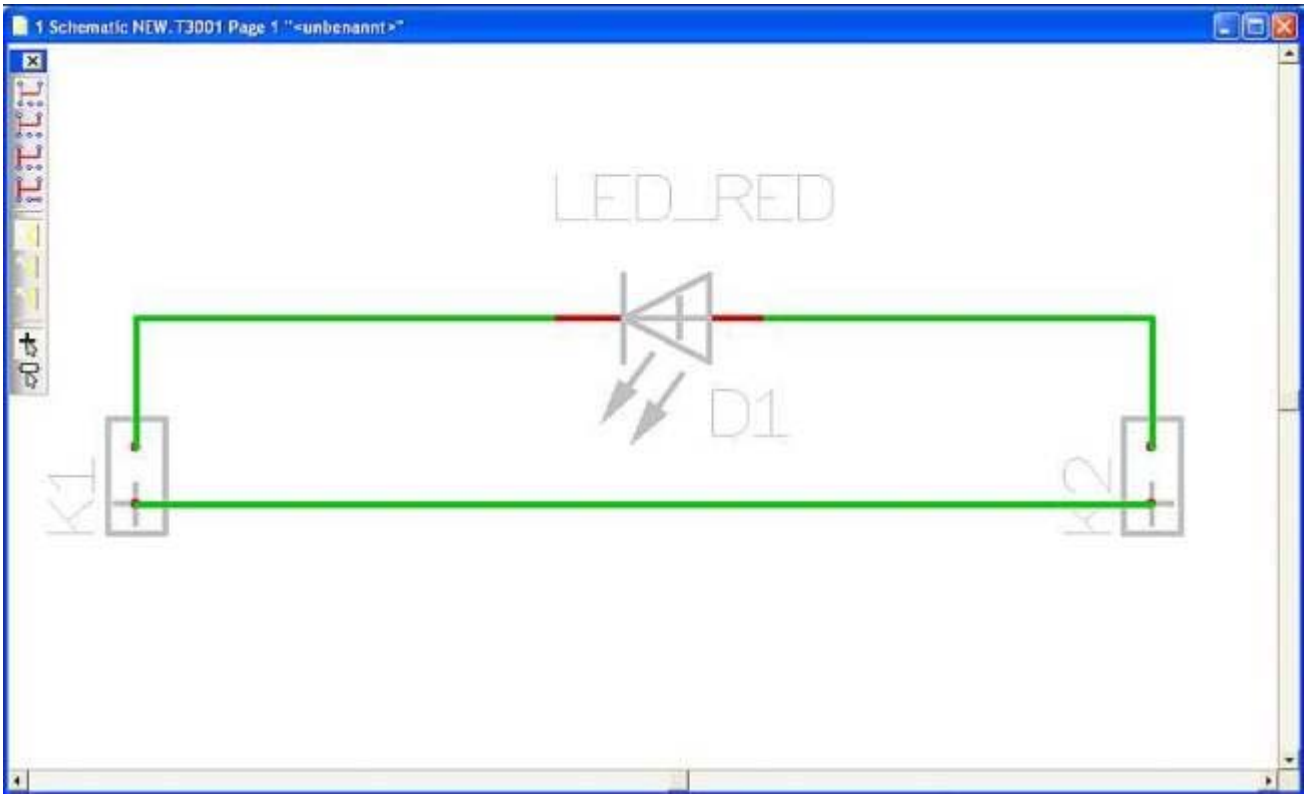
Now connect the anode of the LED...

More details?

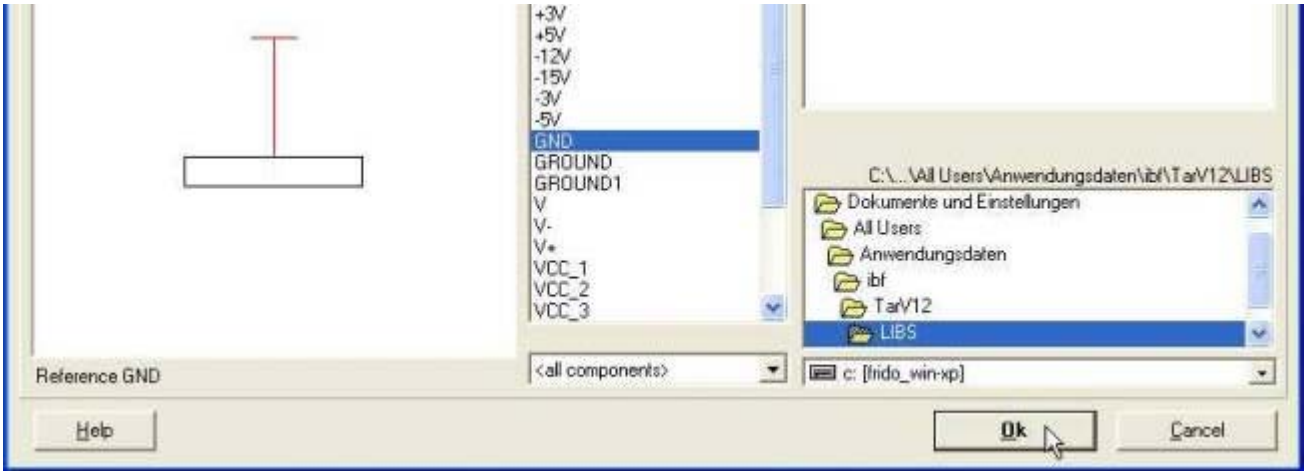
Bending modes
Undo
Redo



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



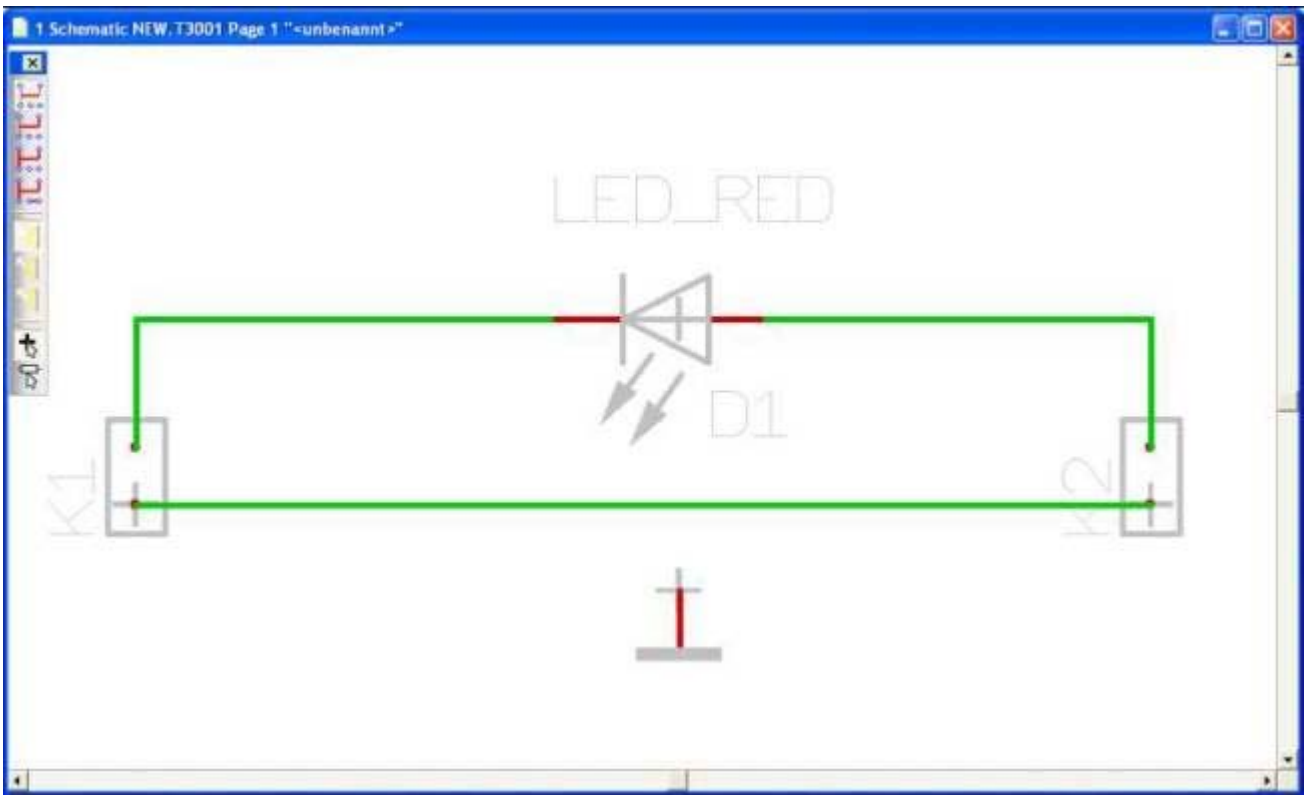
The round signal is connected by a ground symbol. please find it within the range of "Reference pins", see schematic **Menu Components / "Place reference symbols..."**. Do it quicker by key **[r]**.



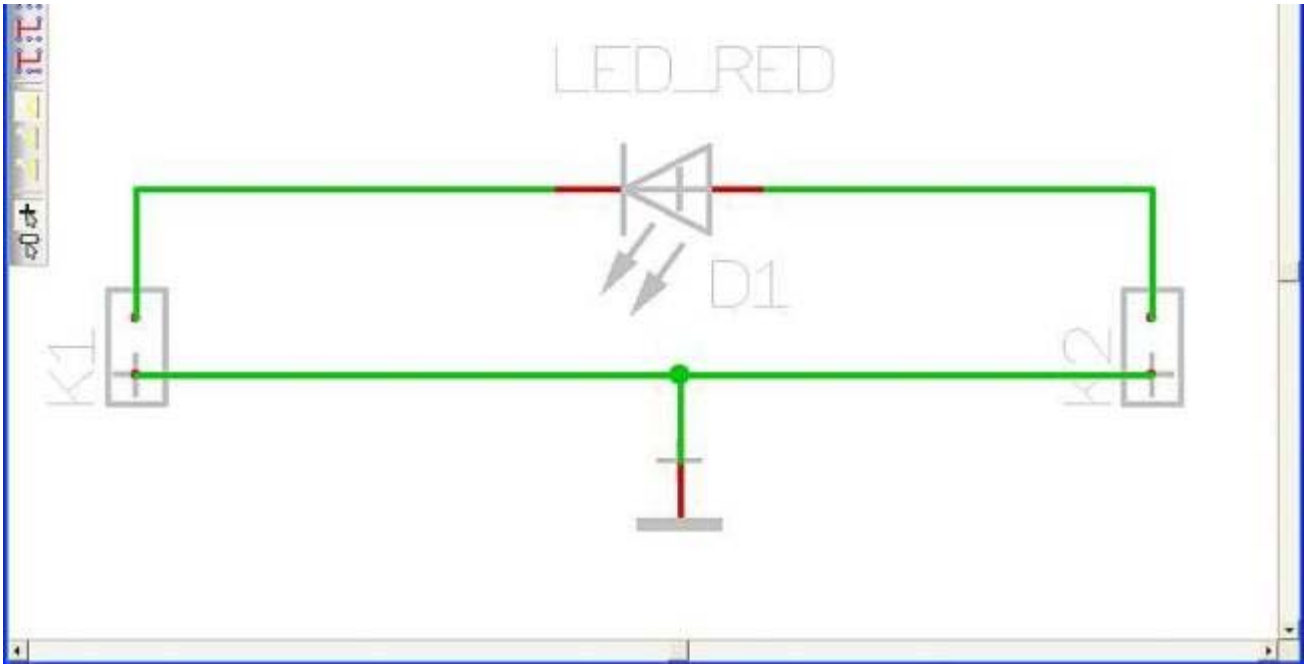
Place the GND symbol in the schematic by **M1**...

More details?

Reference
Symbol
Reorganisation
Settings / Options



...and connect it that way, that you, drag it 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.



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

Next step
One step back

Back to overview

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Connecting_the_pins"

Categories: Actions | Schematic

- This page was last modified 09:46, 30 August 2007.

Define a PCB outline

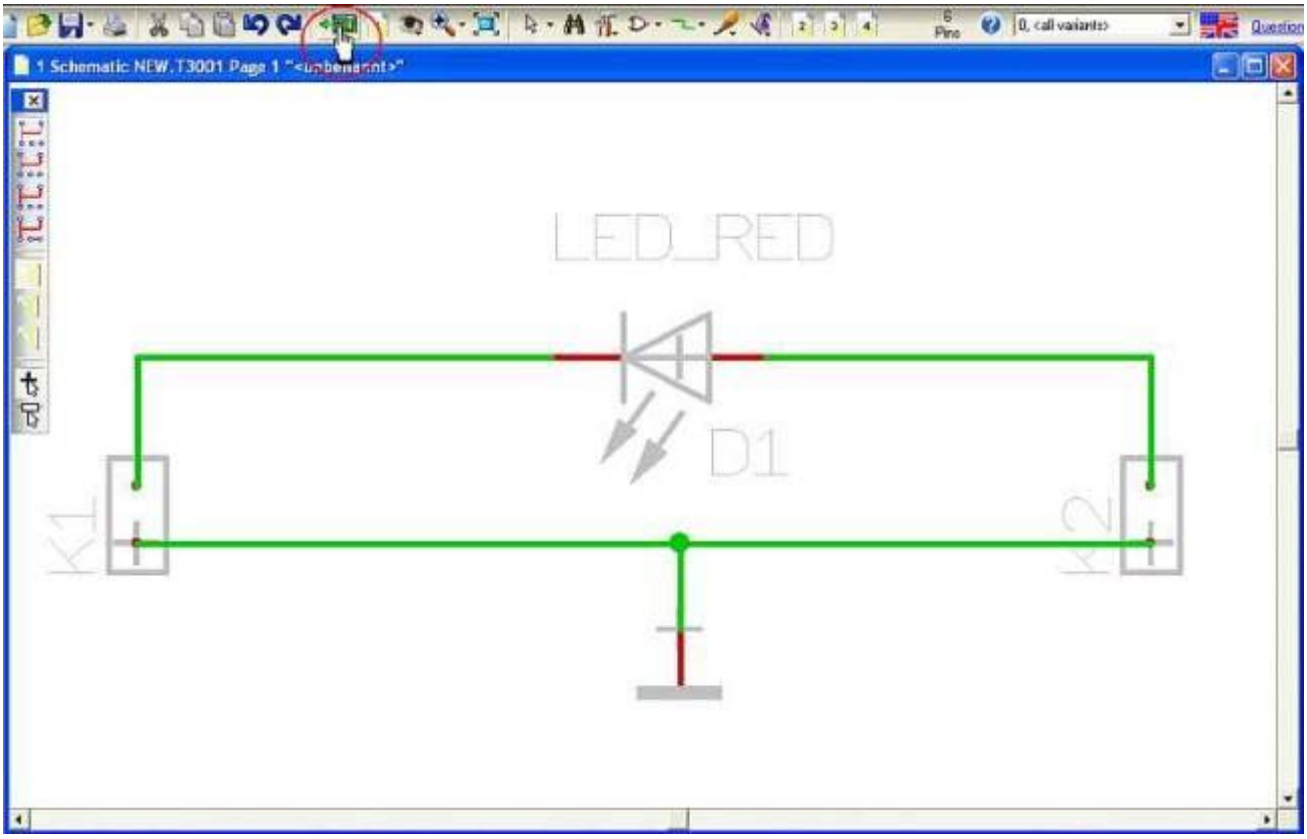
From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

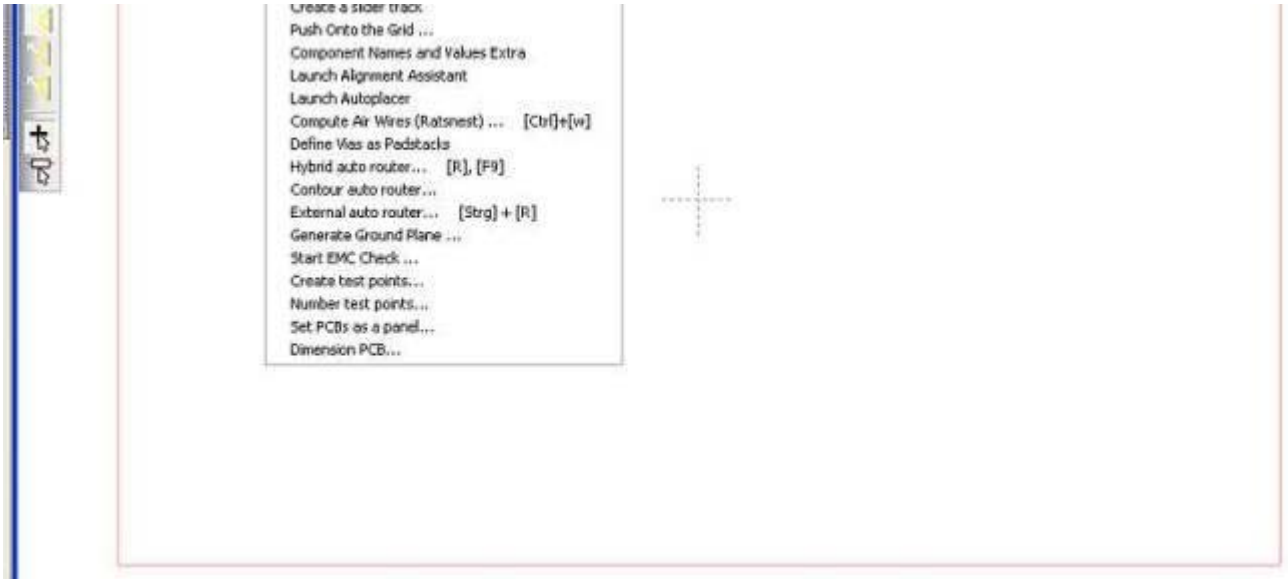
Please switch over to the PCB layout view by the icon "Go to PCB view" (see cursor) or by the key F3.

More details?

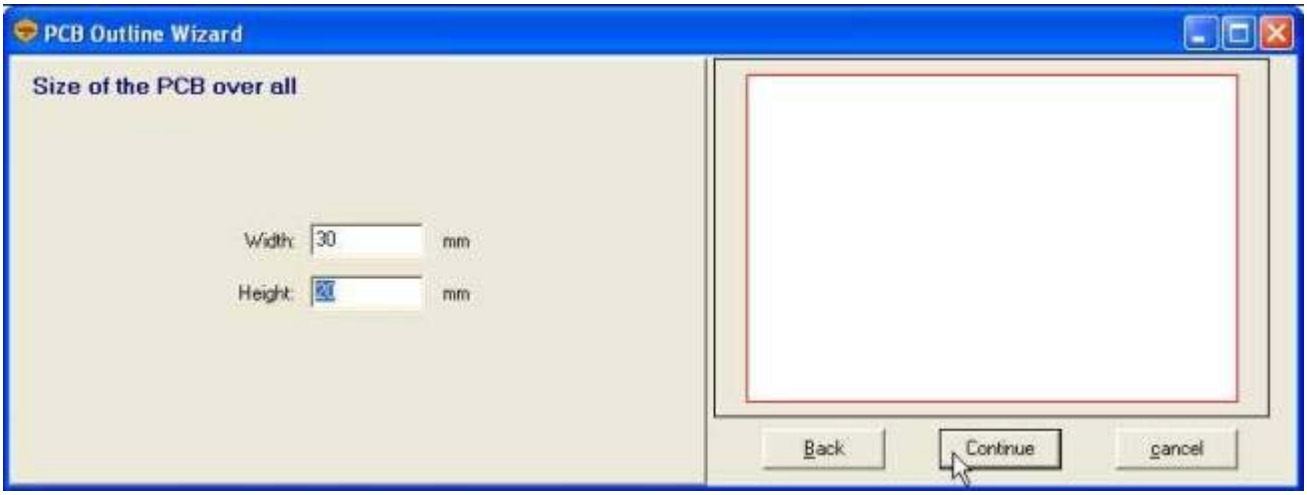
Outline Layer



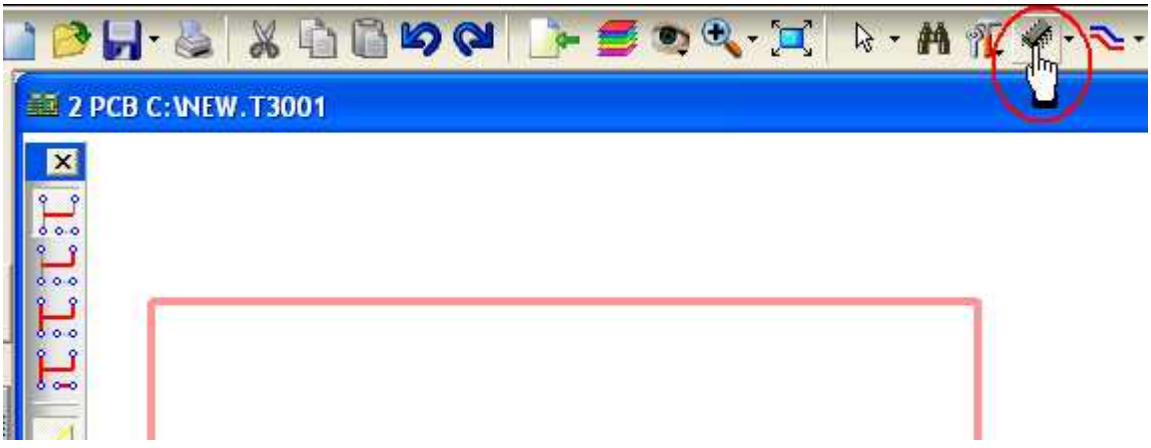
What you get is the Layout view showing a standard PCB outline in light red. It has the measurments of a "eurocard" (W=160mm, H=100mm). Those measurments are much too big for our little project. Any modification of the outline can automatically be done by the PCB outline wizard in menu "Actions/PCB Outline Wizard":



Immediately the assistant opens and we enter Width=30 and Height=20. The fact that we work in metric measurment is predefined by standard. It can be adjusted to imperial measurment by the icon "Adjust View..." (=the eye). The grid is set at he same location.



Because we don't have further breakouts at the edges or within the square, we click on button "Continue" as long as the desired shape appears on the layout screen. Now we can import the component packages (the footprint pattern, or landmarks), see cursor...



More details?

Drawing
Drag
Line
Text

Next step
One step back

Back to overview

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Define_a_PCB_outline"

Categories: Actions

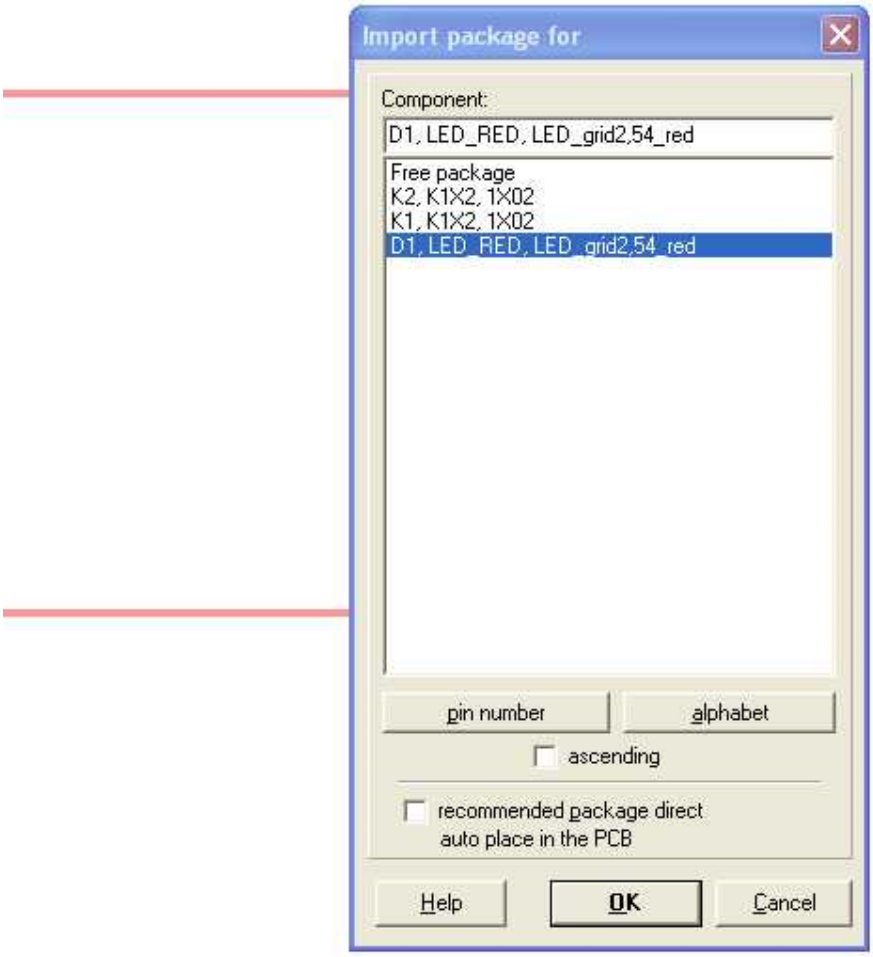
- This page was last modified 09:52, 30 August 2007.

Import matching packages (footprints) to the layout

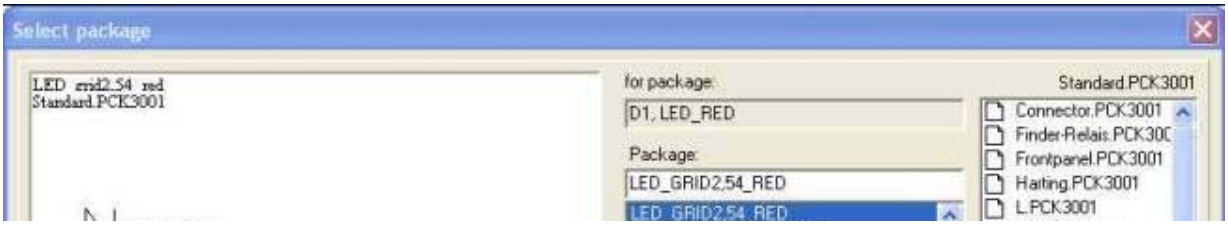
From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

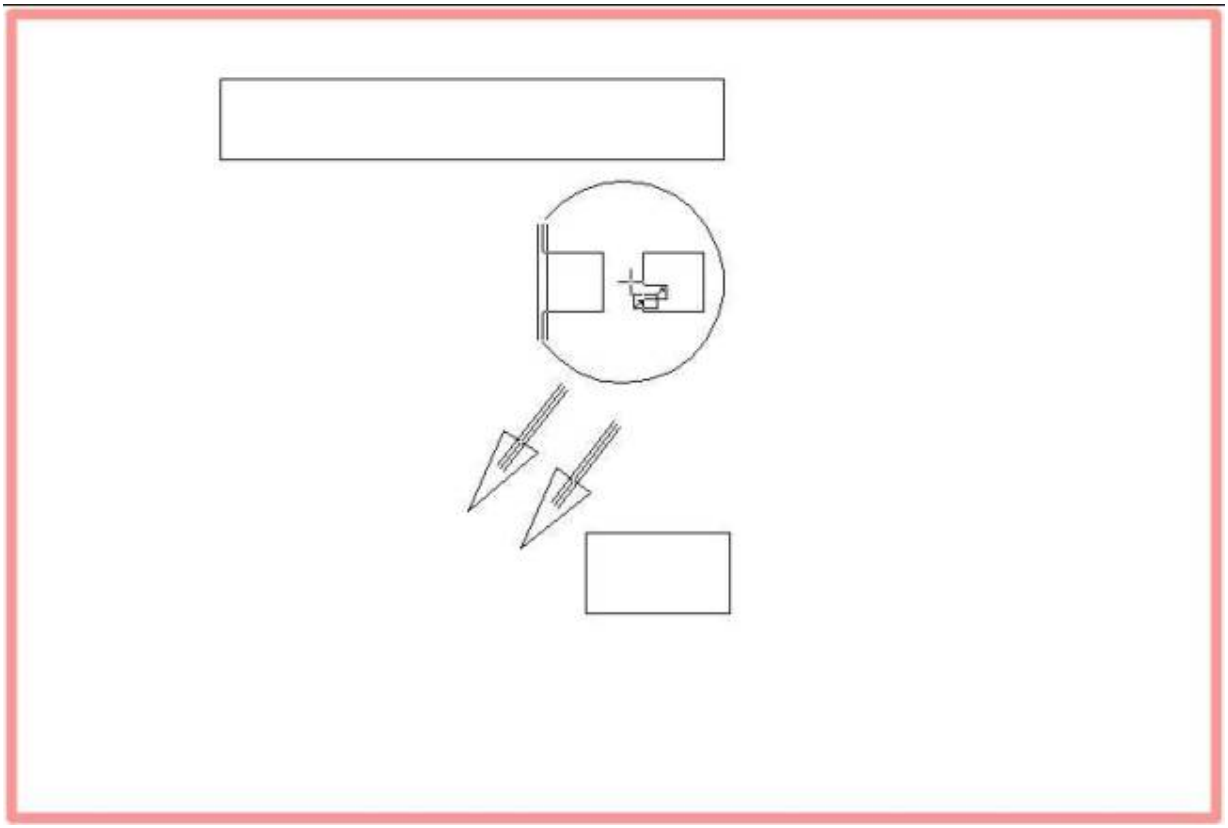
Every schematic symbol in TARGET 3001! has a package proposal. The word package in TARGET 3001! stands for "footprints" or more exact: solderable "landmarks". If you want to start with the package import, see cursor, a list of all package proposals according to the symbols used in the schematic appears. Select from this list each of the packages you'd like to import to the Layout:



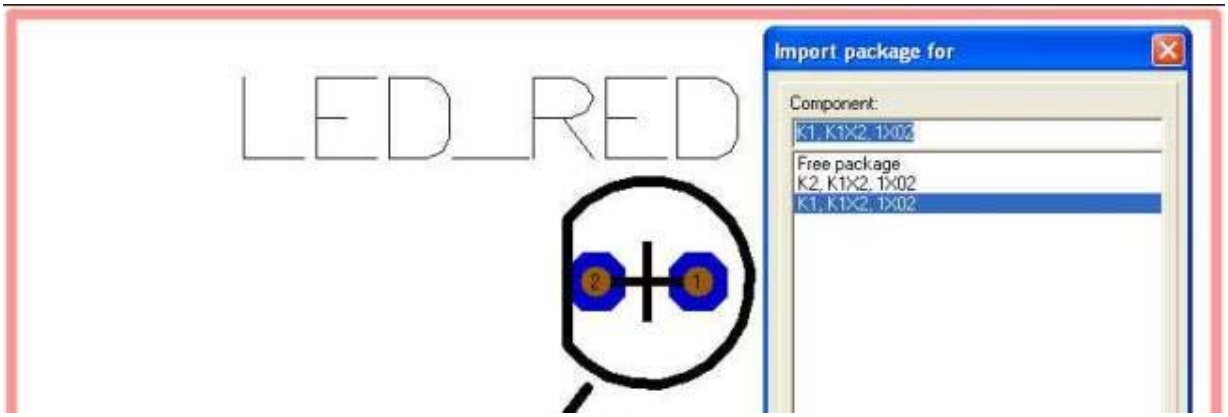
If you decide for the import of the LED, visualize the package in the browser prior to insertion. Because we want to use it, we confirm the dialog by **"OK"**.

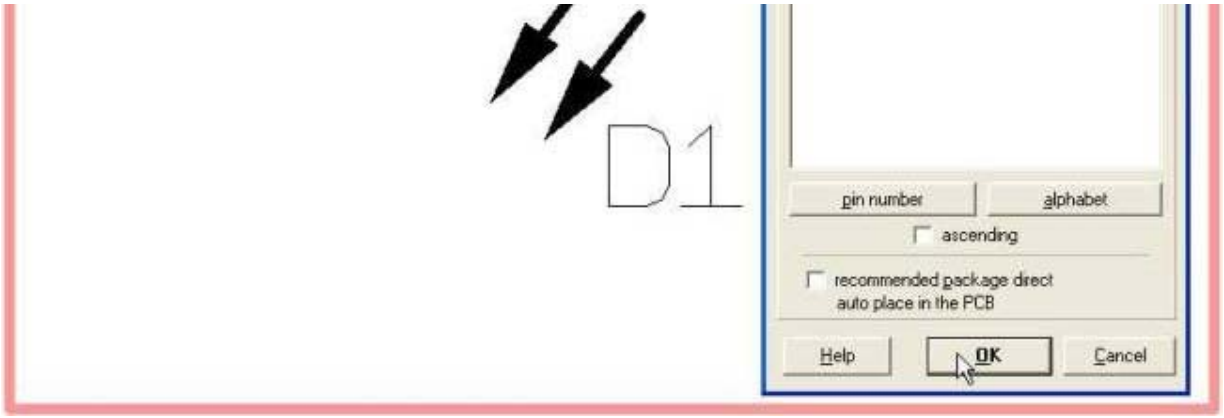


Now it's shape is fixed to the cursor and allows rotation by **M2**,...

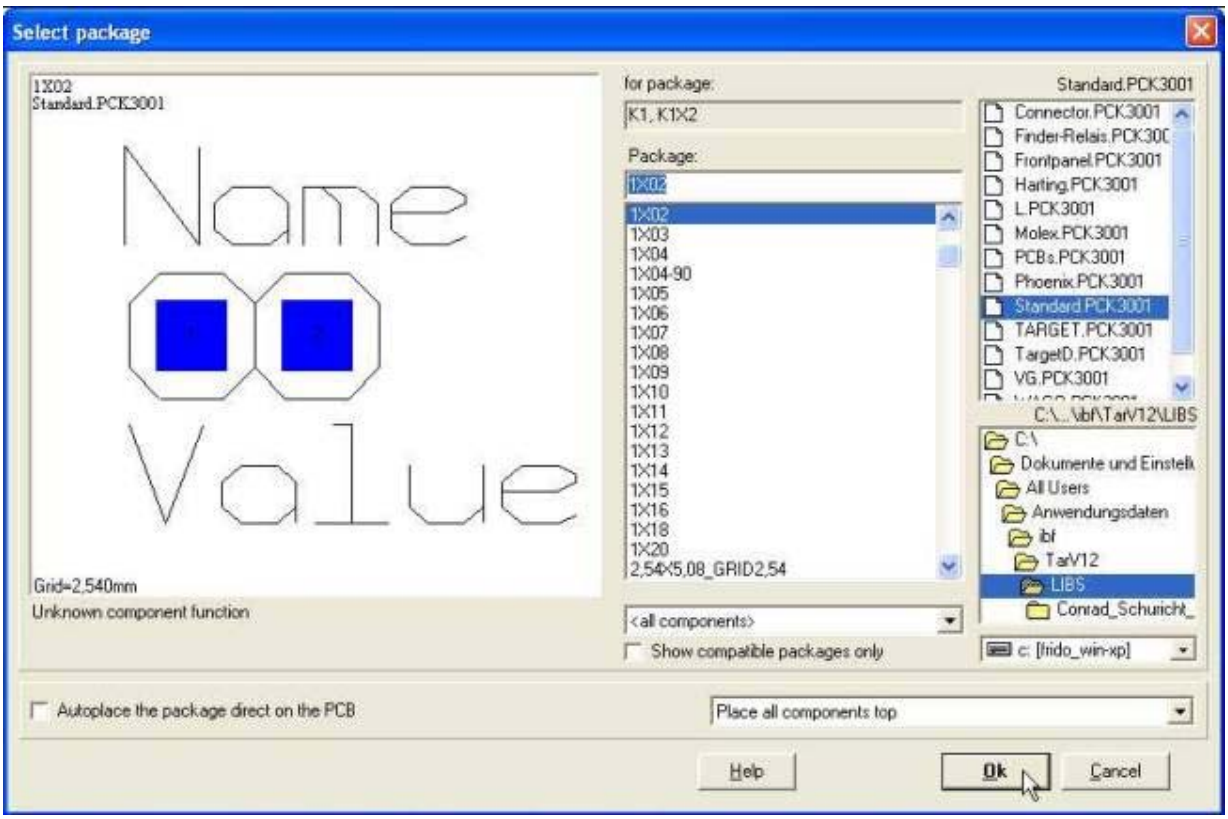


...before it is placed by **M1**. Immediately afterwards the selection list opens for insertion of another package. We choose the connector K1,...

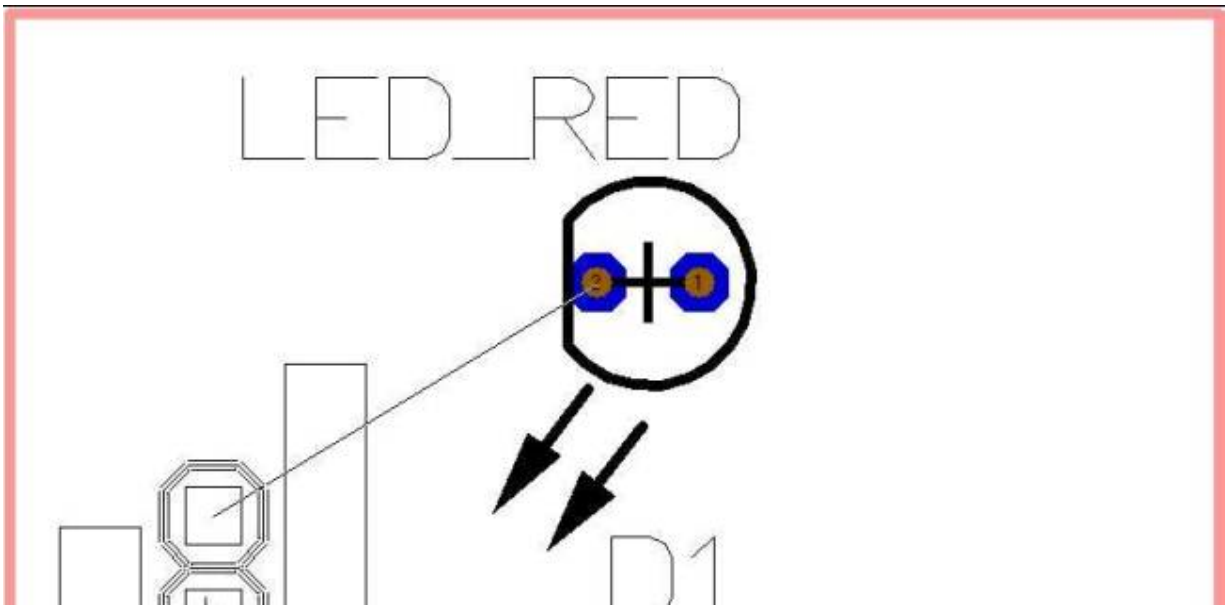




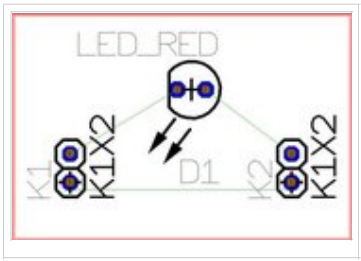
...confirm it by [[**OK**]]...



...and again it is fixed to the cursor ready for placement. At the same time an [[Airwires / ratsnest|airwire] shows the electrical connection. After placement of the connector we see the airwire in green and the pads in blue.



Again the list opens, we select the second connector 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-internal autorouters, see the next step...

Next step
One step back

Back to the overview

Categories: Layout

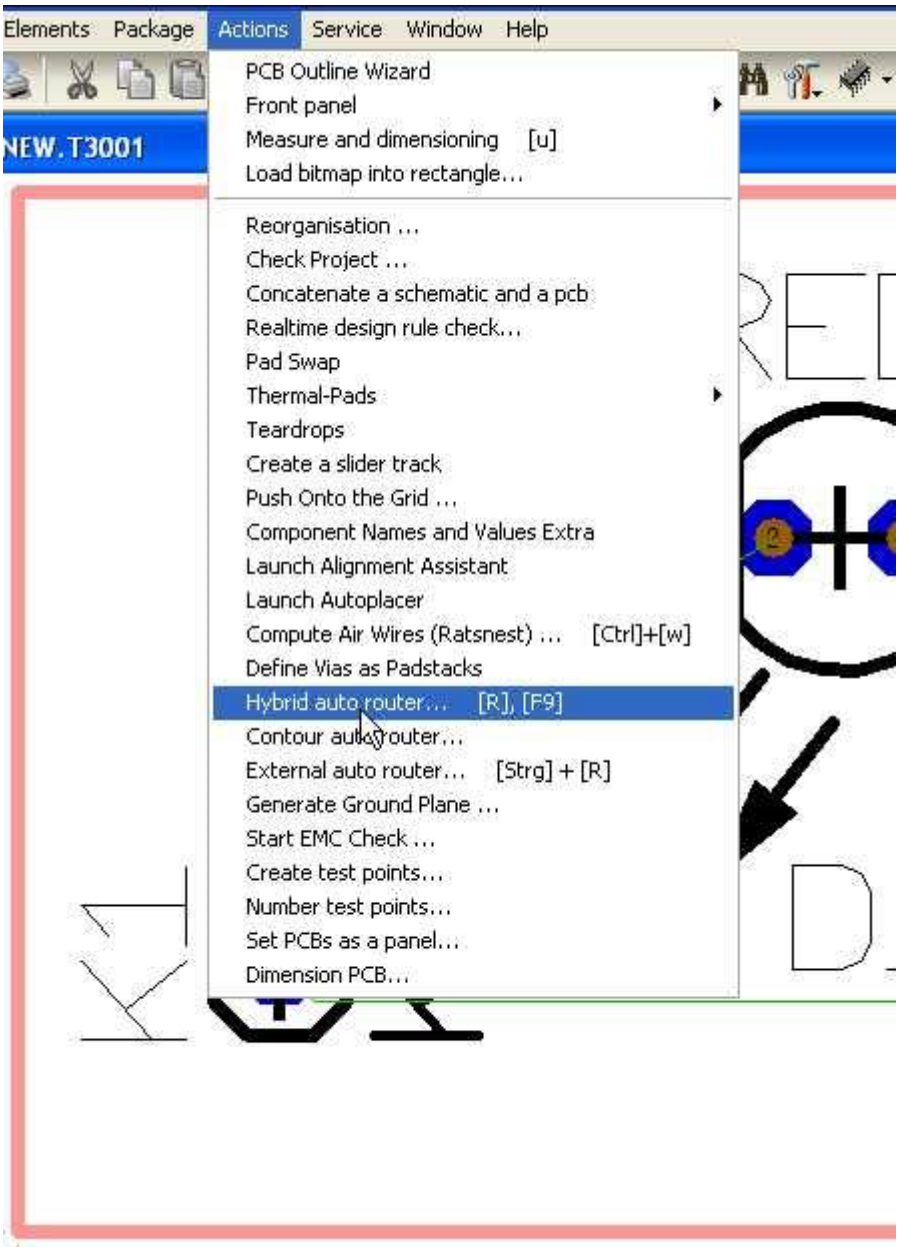
- This page was last modified 09:59, 30 August 2007.

Place tracks

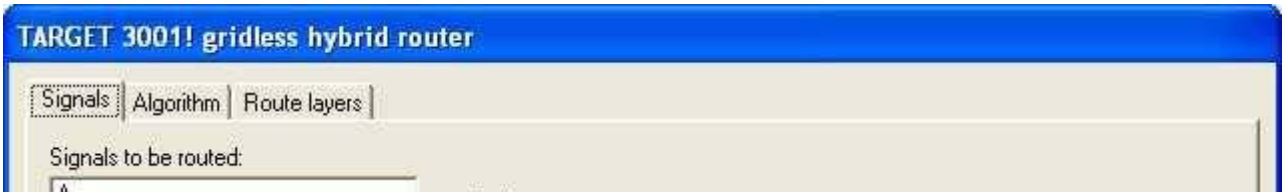
From IBF-Wiki

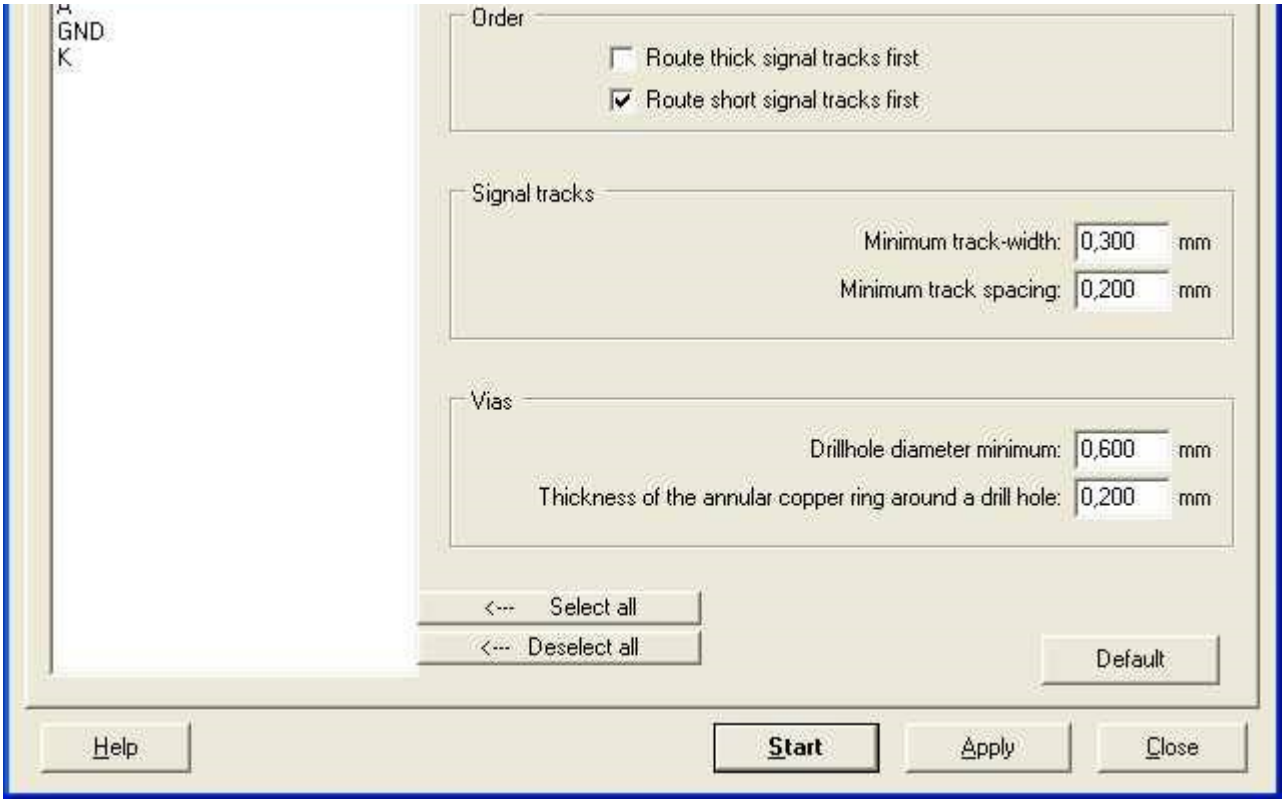
This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

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.

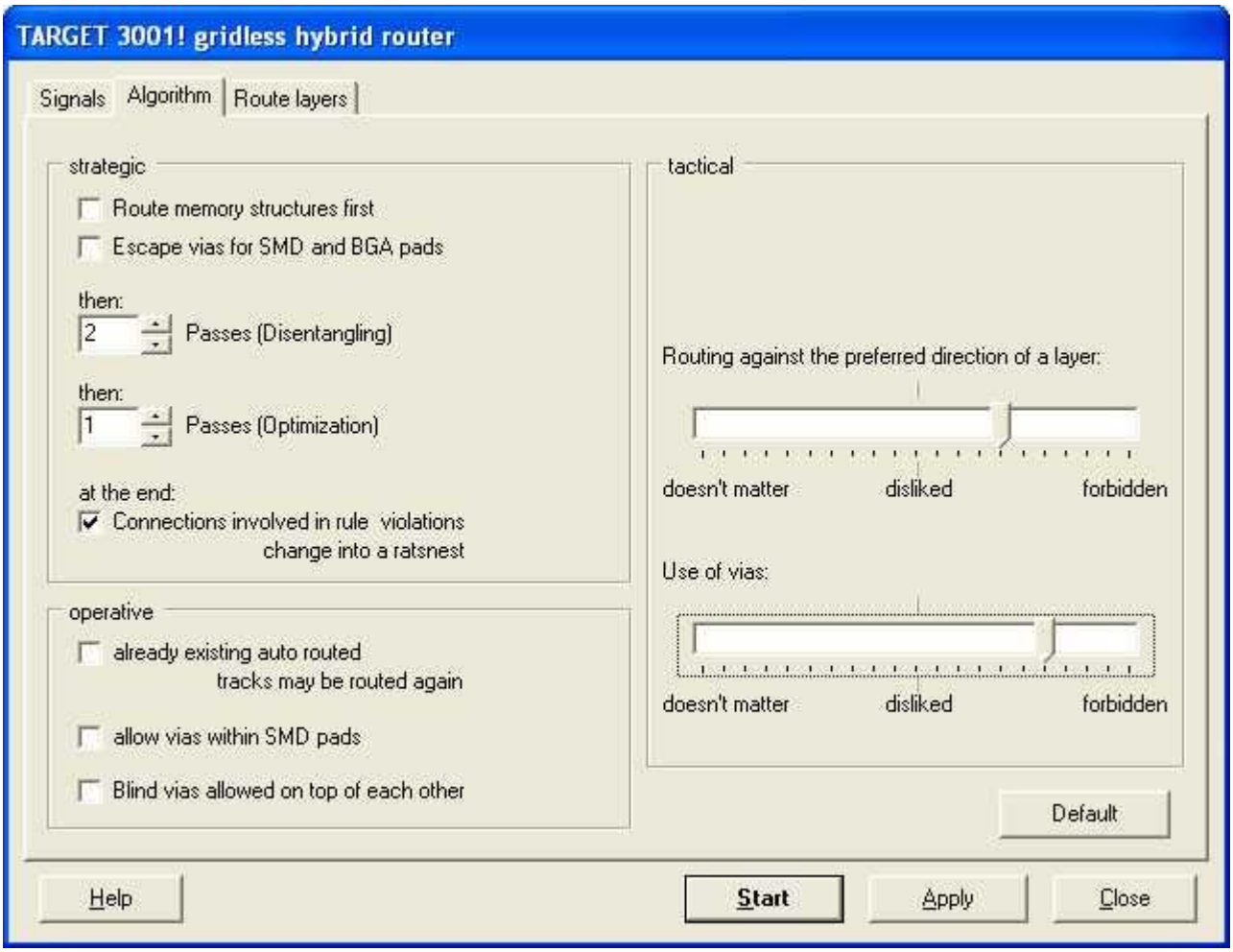


The opening dialog of the hybrid router has three sections divided by tabs. First Der Dialog des Hybridrouters beinhaltet drei Bereiche, die mit Tabs voneinander abgegrenzt sind. First the section "Signals".



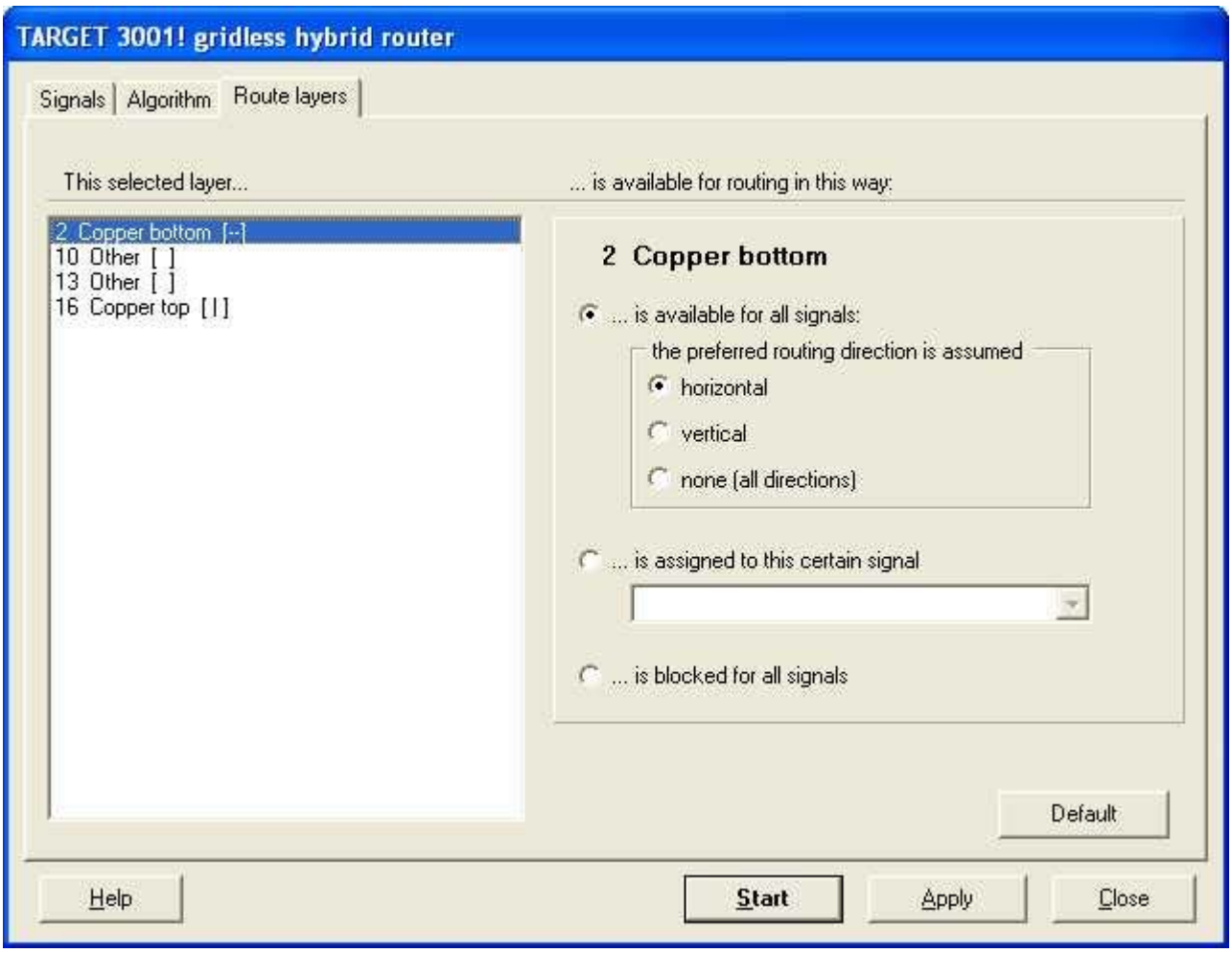


Now choose from the list one or more or all signals. Iv you select none, all are routed.

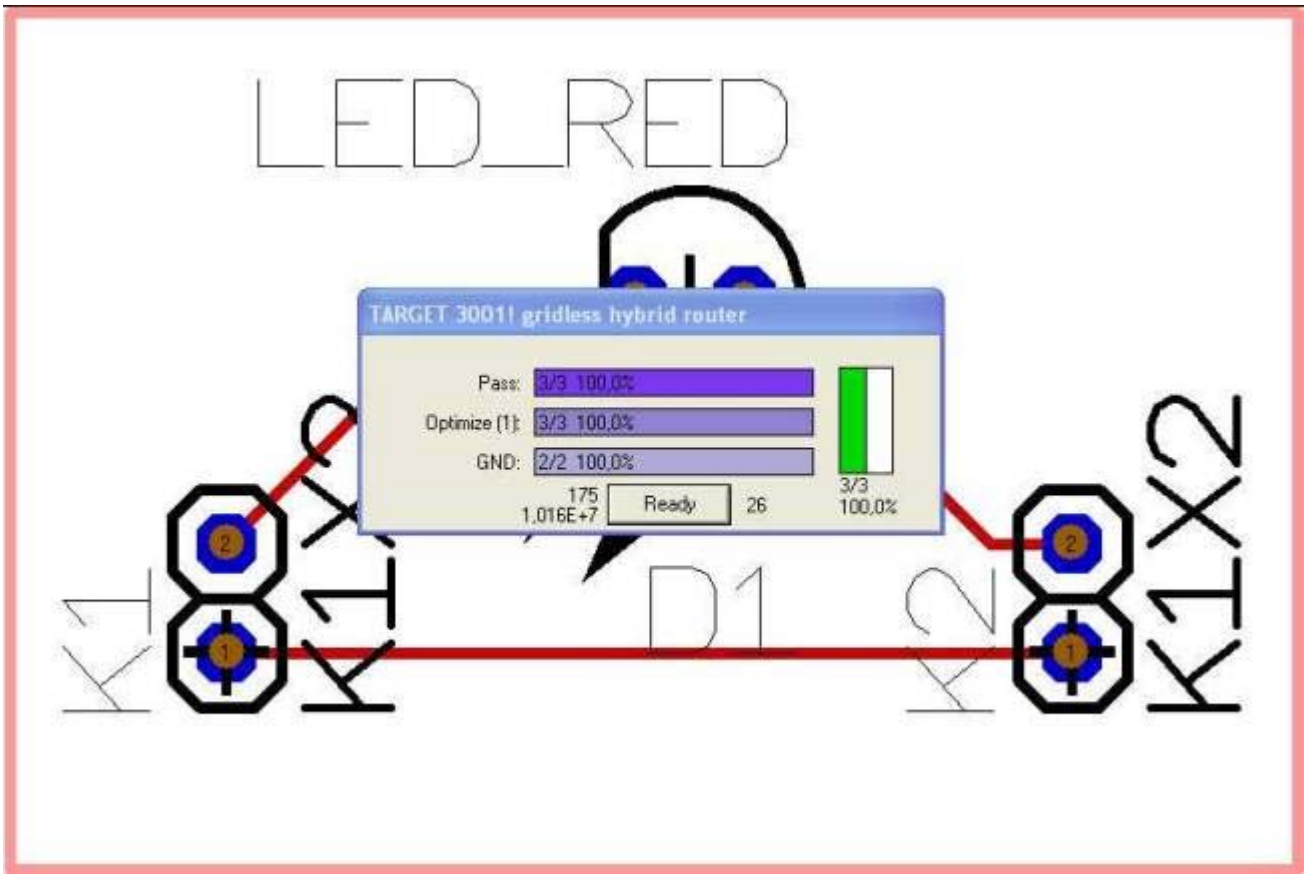


The second tab allows "strategic", "operative" and "tactical" settings. In the strategic area we leave the default settings with "two passes of disentangeling" and "one pass of optimization". In the tactical area we place the slider between "disliked" and "forbidden". The third tab allows the assignment 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

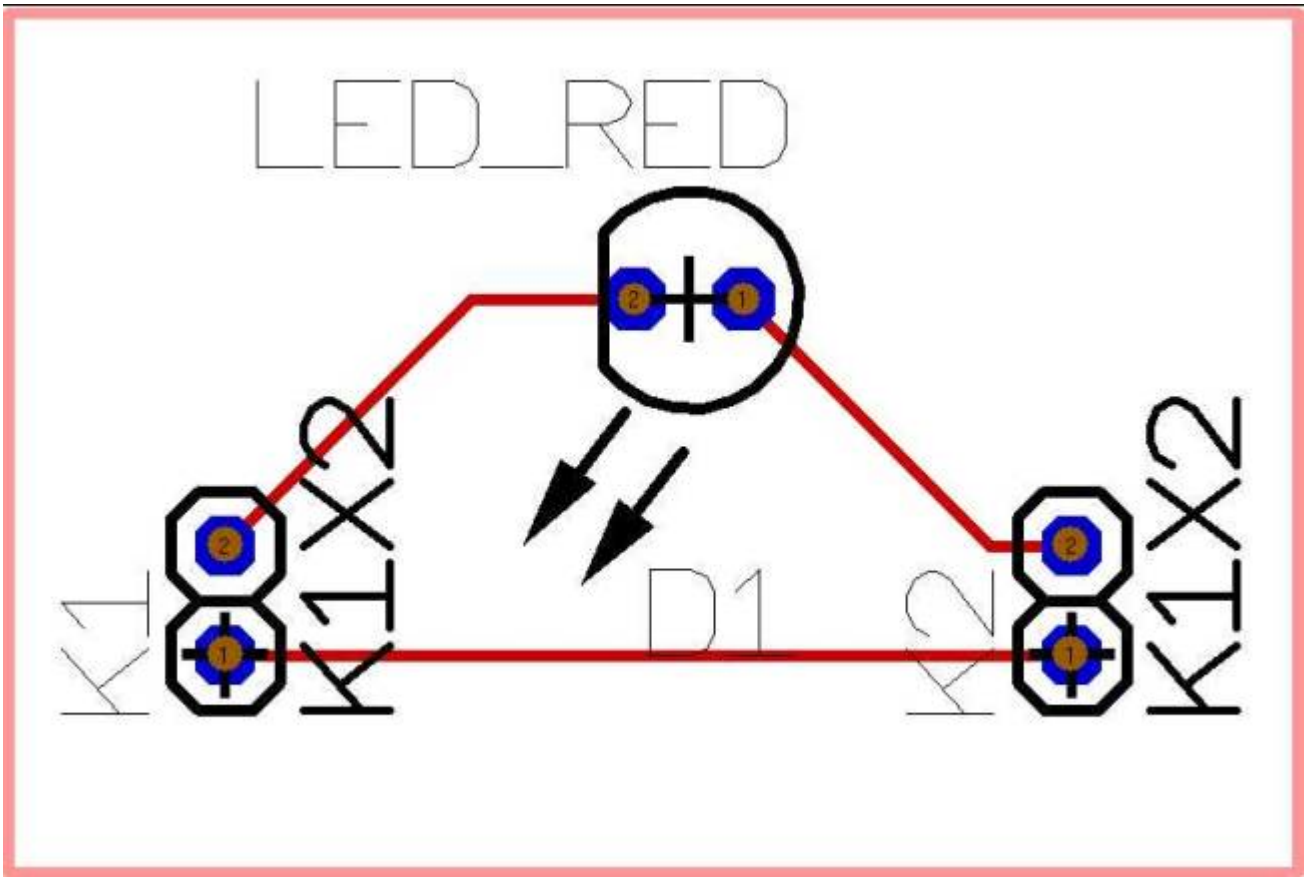
each layer line at the left to set their availability.



Now watch the router at his work...



...and expect the following result:



Next Step
One step back

Back to the overview

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Place_tracks"

Categories: Layout

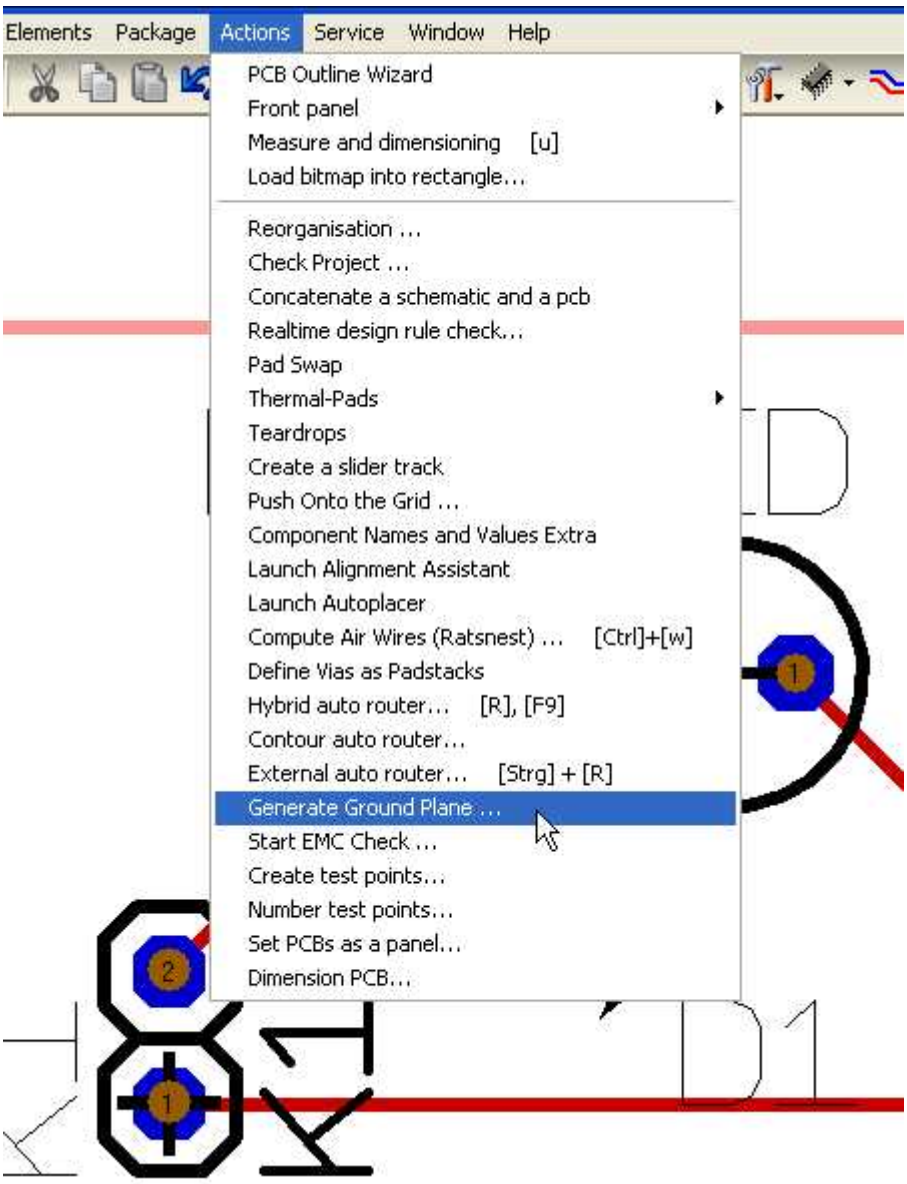
- This page was last modified 10:08, 30 August 2007.

Generate a groundplane

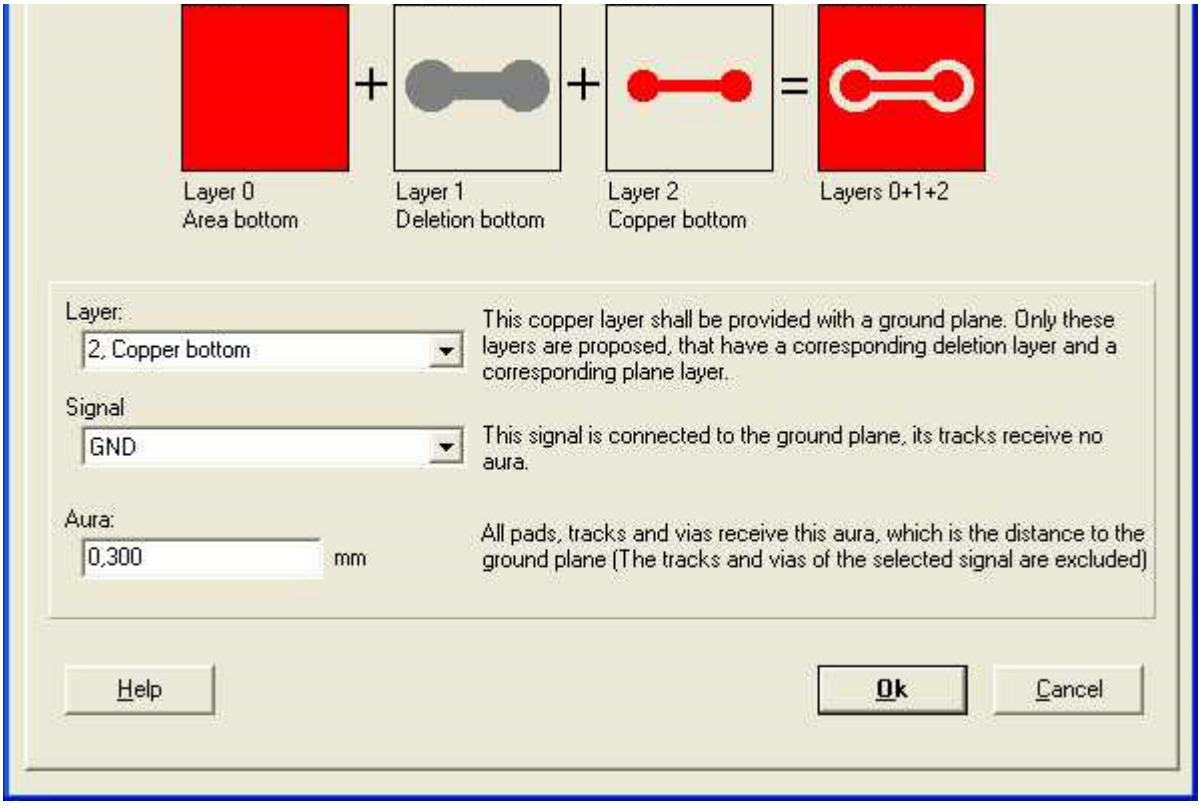
From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

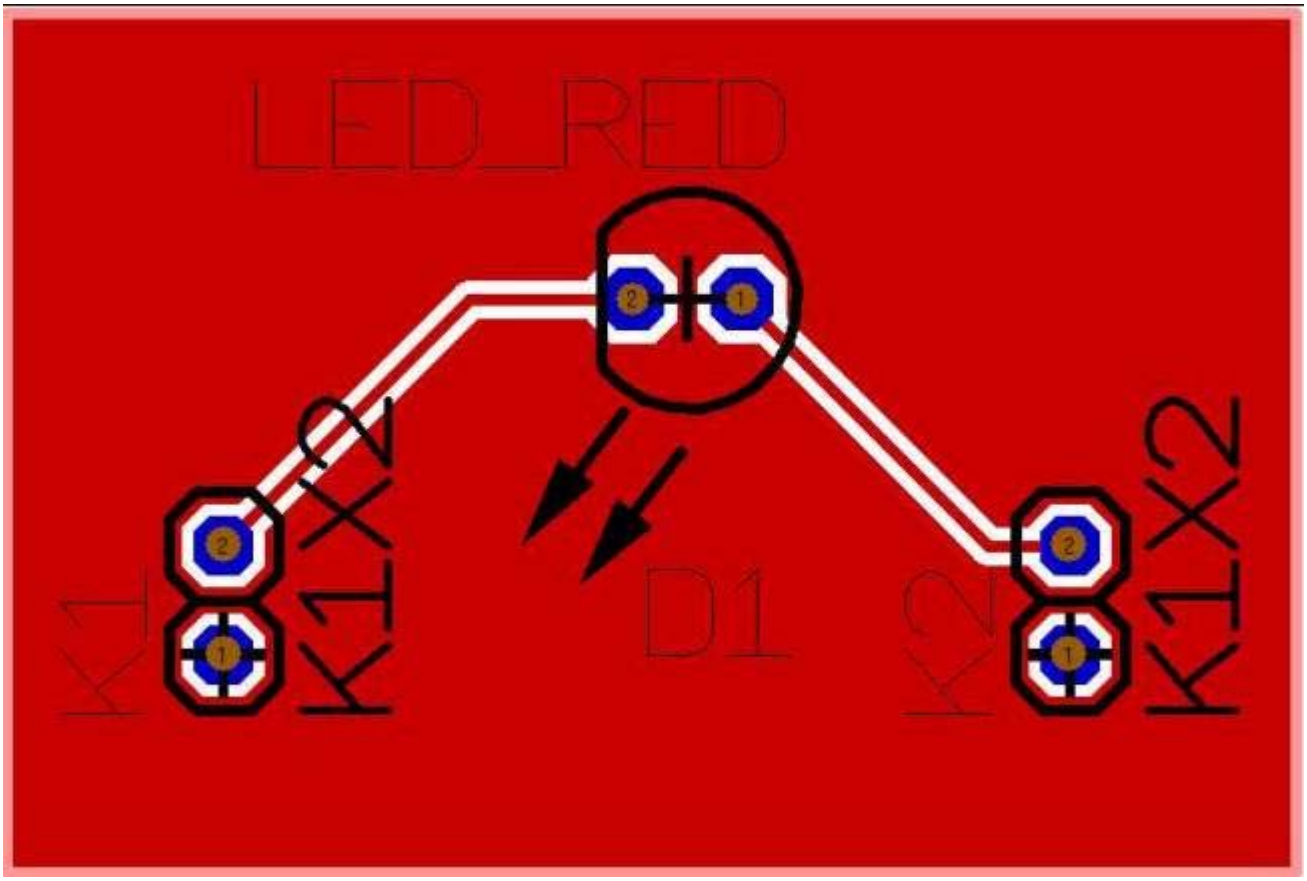
A groundplane having the same dimensions like the layout can easily be created by the use of the groundplane assistant, please see layoutmenu **"Actions"**:



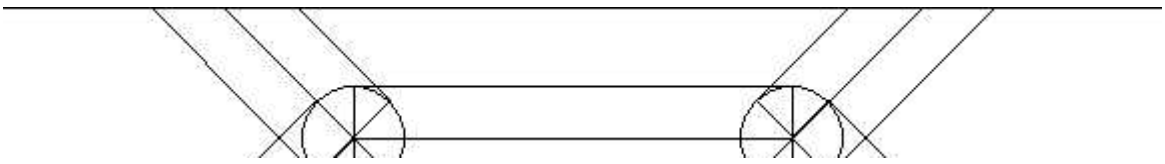
The opening dialog explains that a groundplane in TARGET 3001! is made out of a set of three layers: "copper", "area" and "deletion". The shape of the groundplane is defined on the "area" layer. The "deletion" layer defines the spacings between the groundplane and the non GND-Leading tracks. The layer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

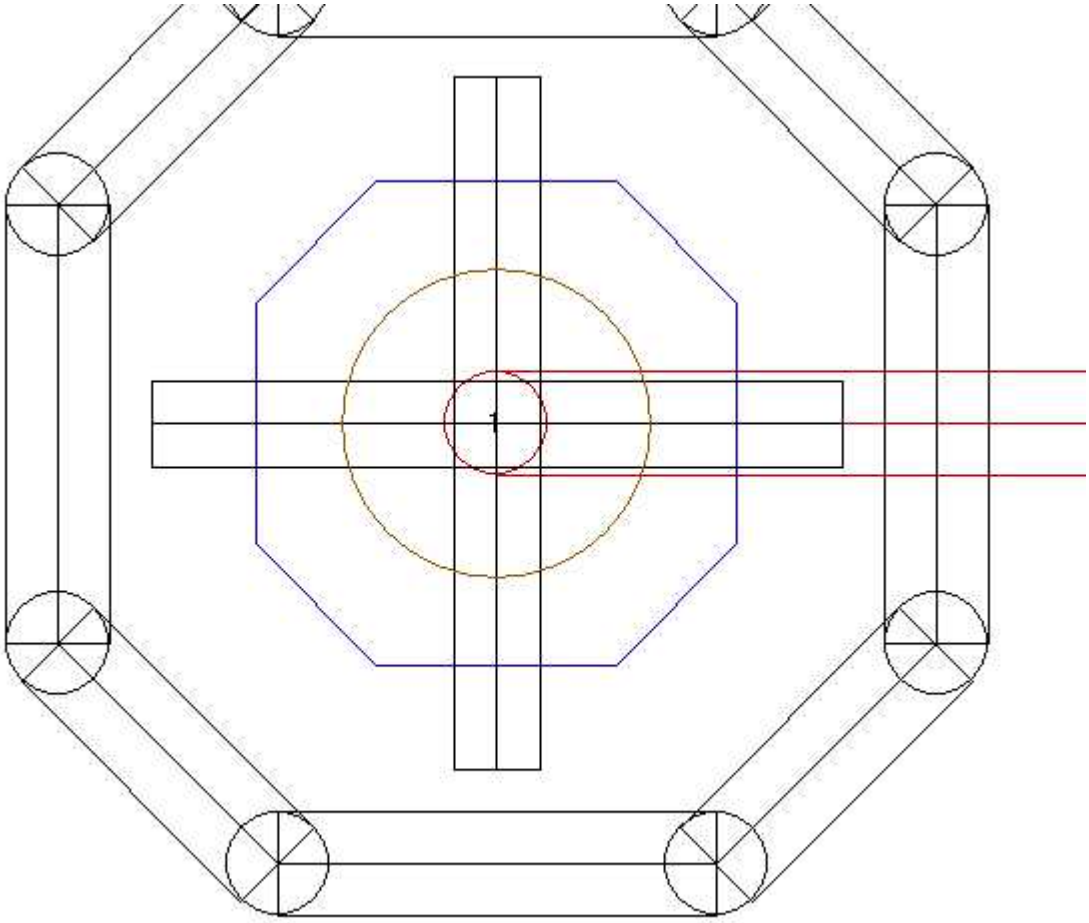


Please confirm the Standard settings and reach the following:



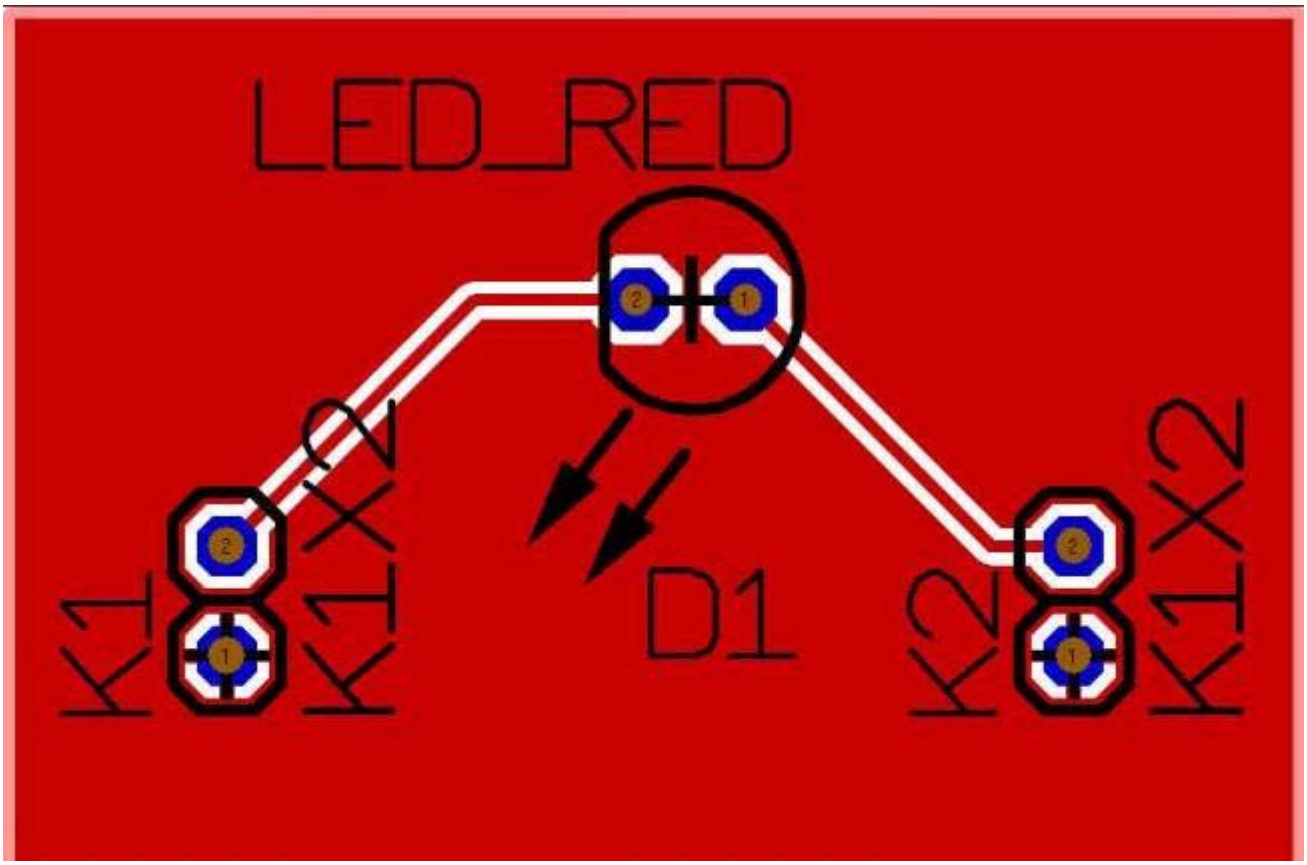
The GND leading track in the south of the layout is embedded completely into the groundplane. Use the hash key (#) for having an x-ray view of the layout. So you can see...



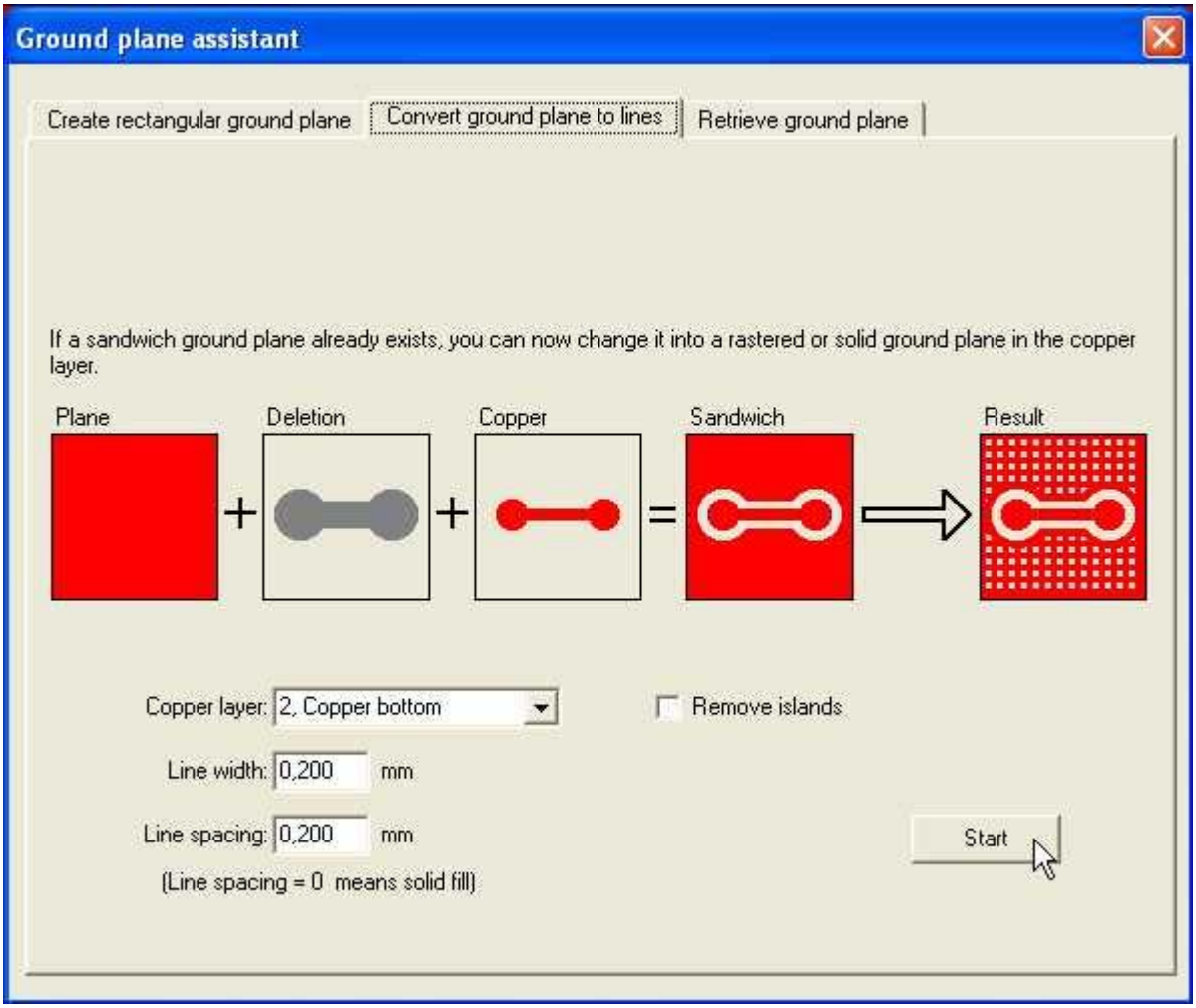


...that the pads are connected correctly.

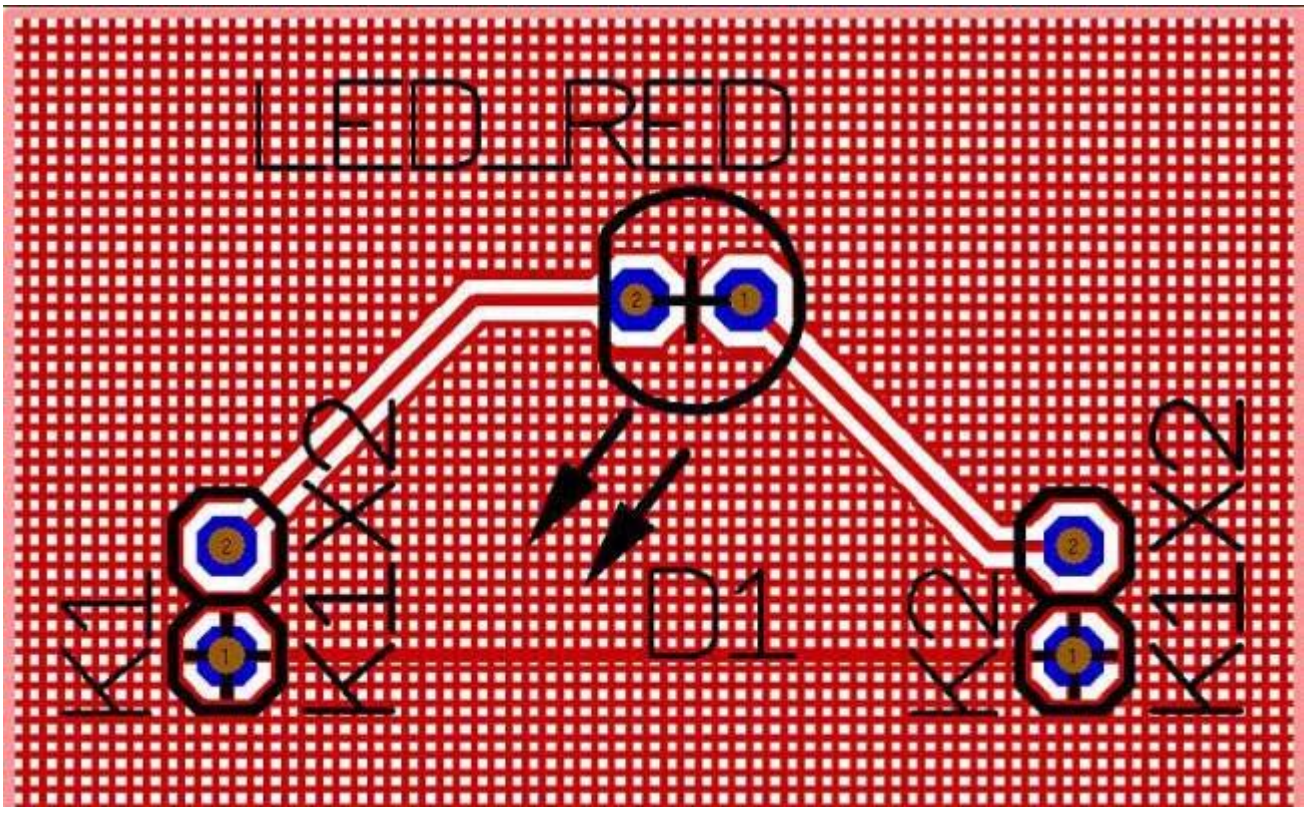
The generation of thermal pads is managed by highlighting the complete layout and choose menu "Actions/Thermal-Pads". Pads whose signal tracks are embedded to the groundplane, receive three more ligaments. Result:



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



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



Next step
One step back

Back to the overview

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Generate_a_groundplane"

Categories: Actions | Automatic functions

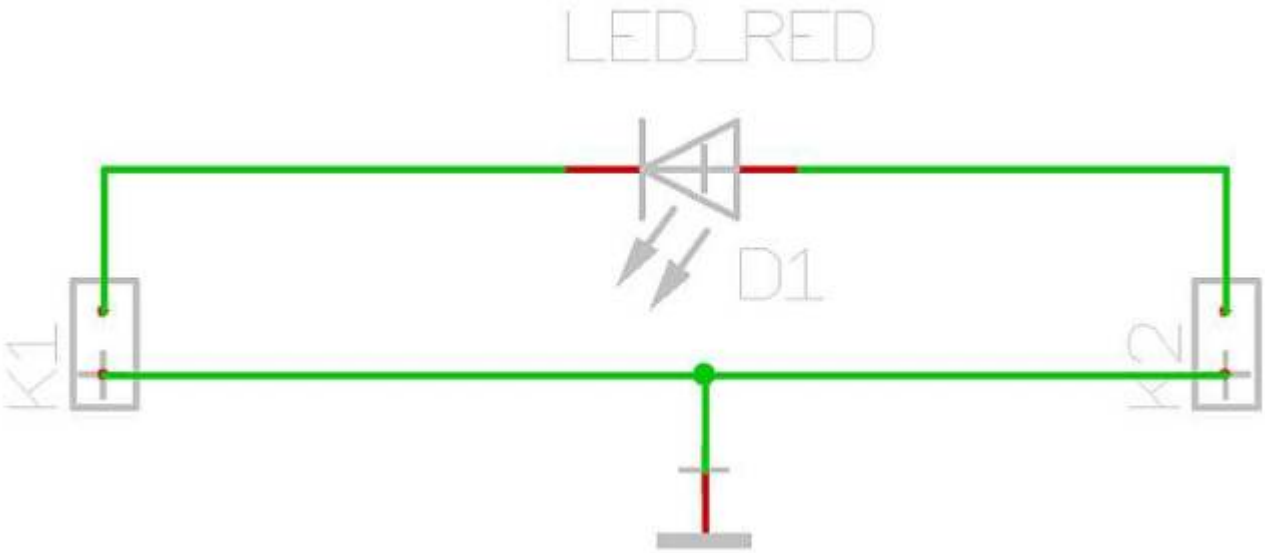
- This page was last modified 10:12, 30 August 2007.

Simulate the function Part 1

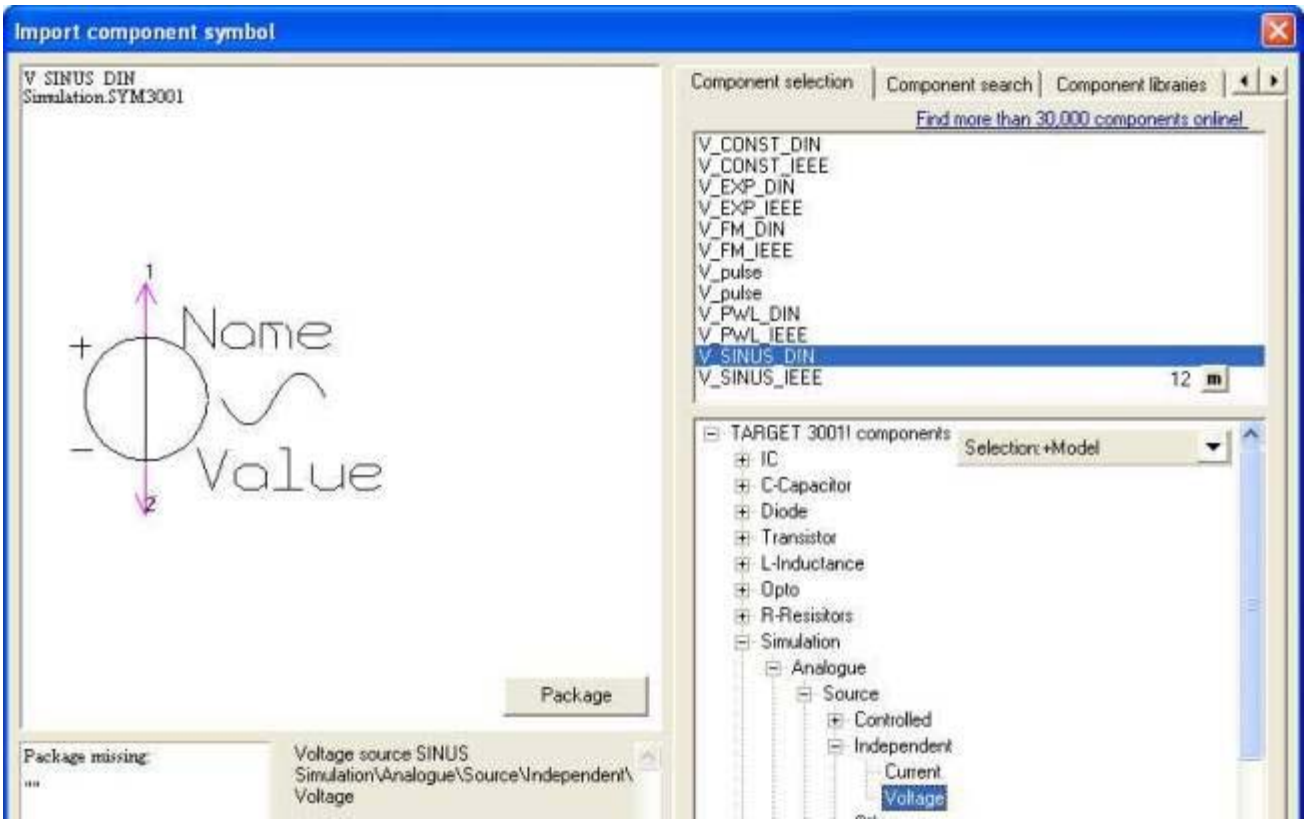
From IBF-Wiki

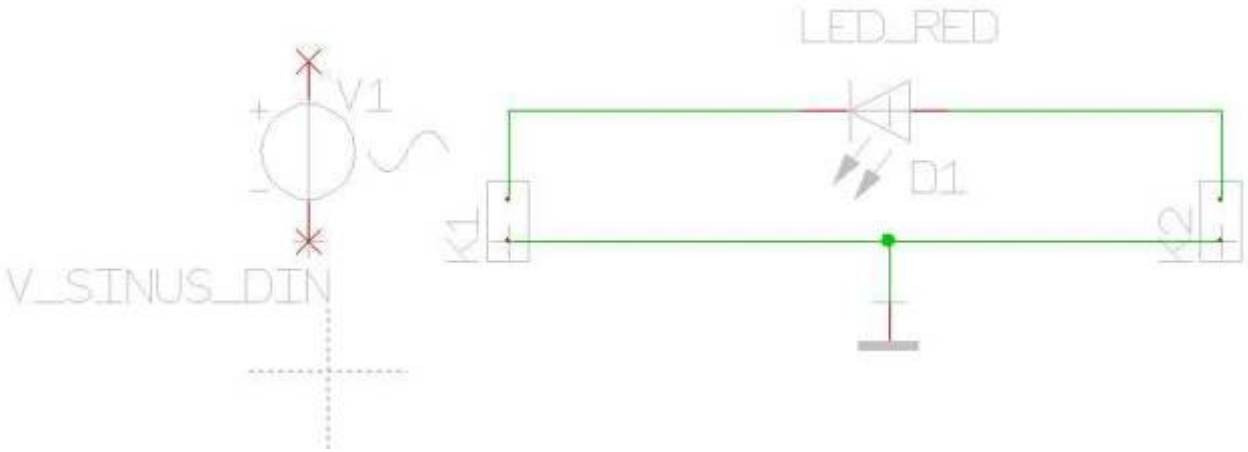
This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

The simulation in TARGET 3001! is a matter of the schematic. So we switch over to it and see our well known picture:

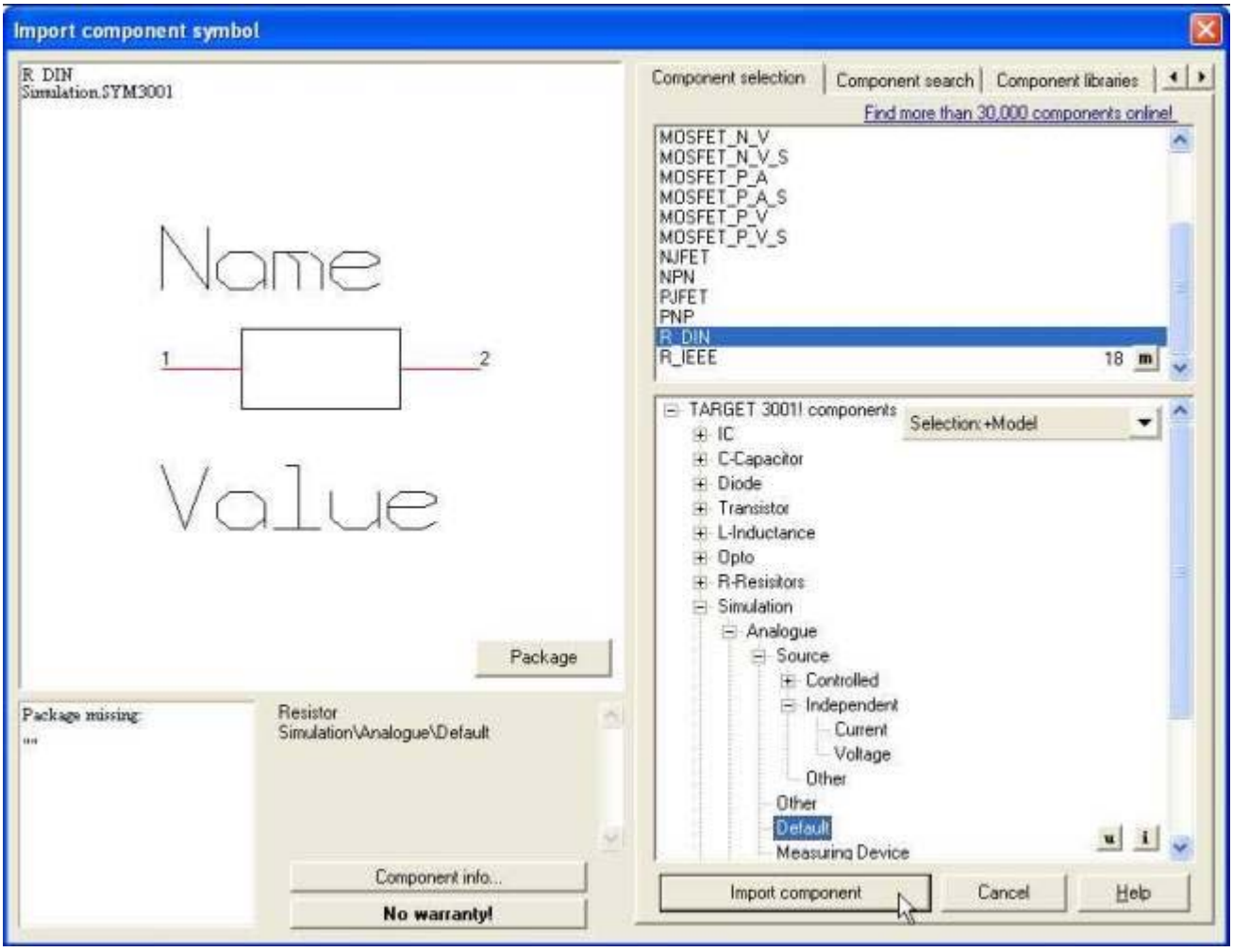


For the simulation of the LED's function we need a voltage source, and a load resistance. Those components will not be part of the layout thus don't have a package. Find such components in the tree view in the branch **Simulation**. First we import a sinus - source, then a resistor.

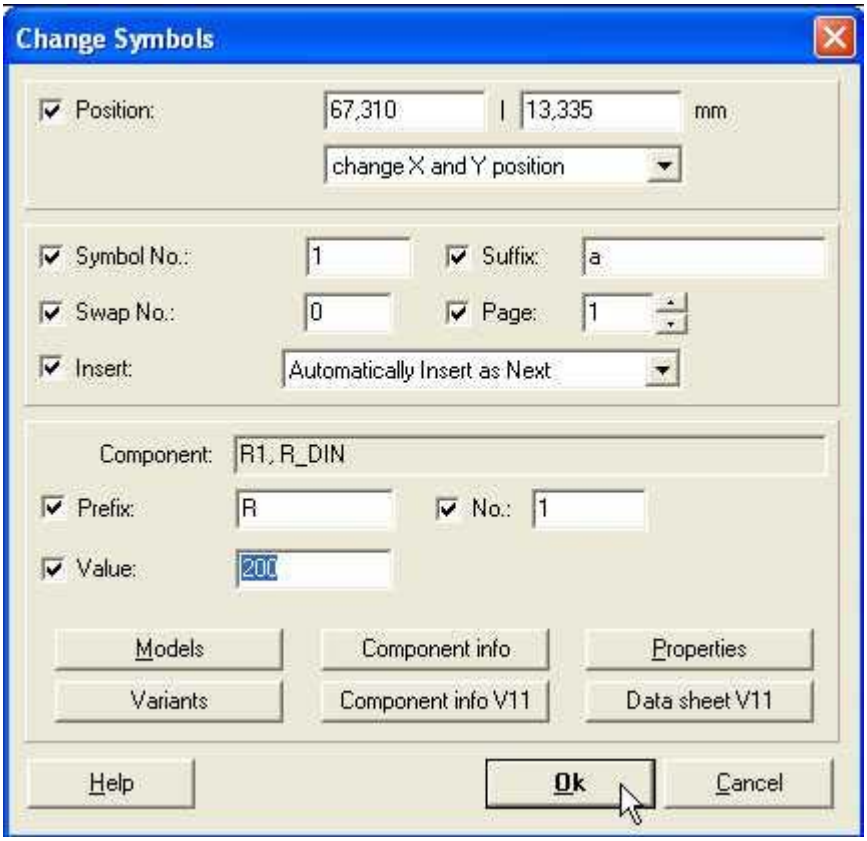




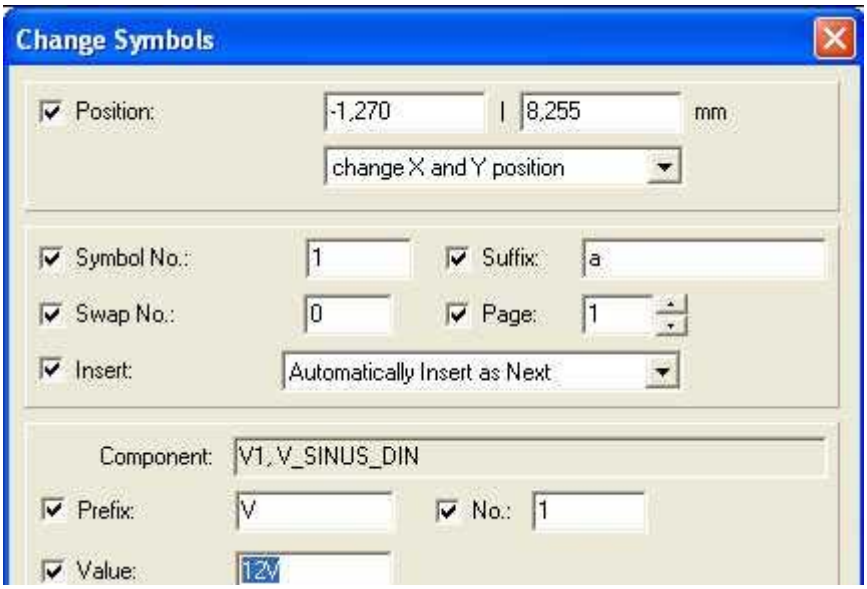
The resistor we pick from the branch Simulation/Analog/Source/Default and connect it accordingly:

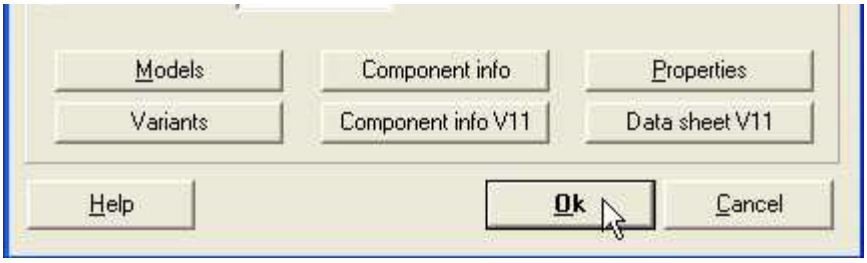


Now we set the values for source and load, first **M11** on the handle cross of the resistor:

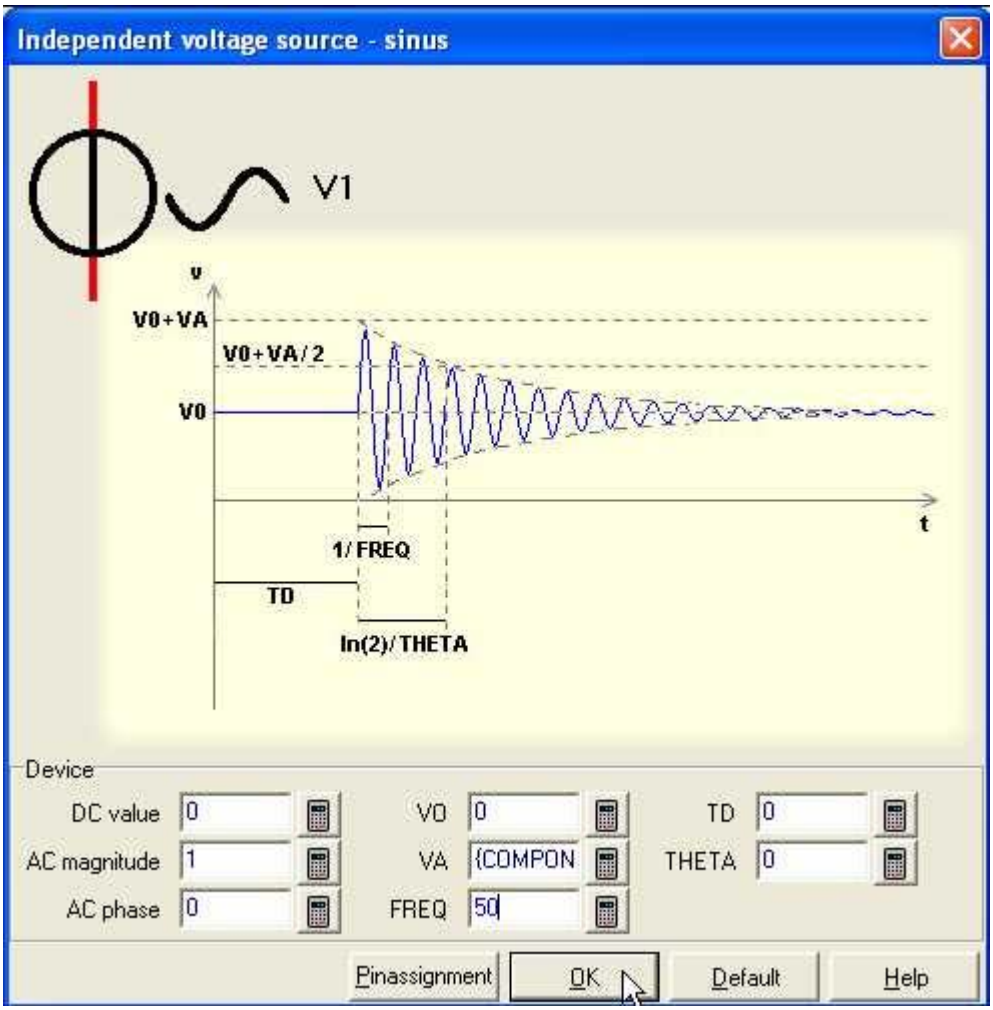


Now we enter component value "200", this stands for 200 Ohm. Press OK. Now press **M11** upon the handle cross of the sinus source:

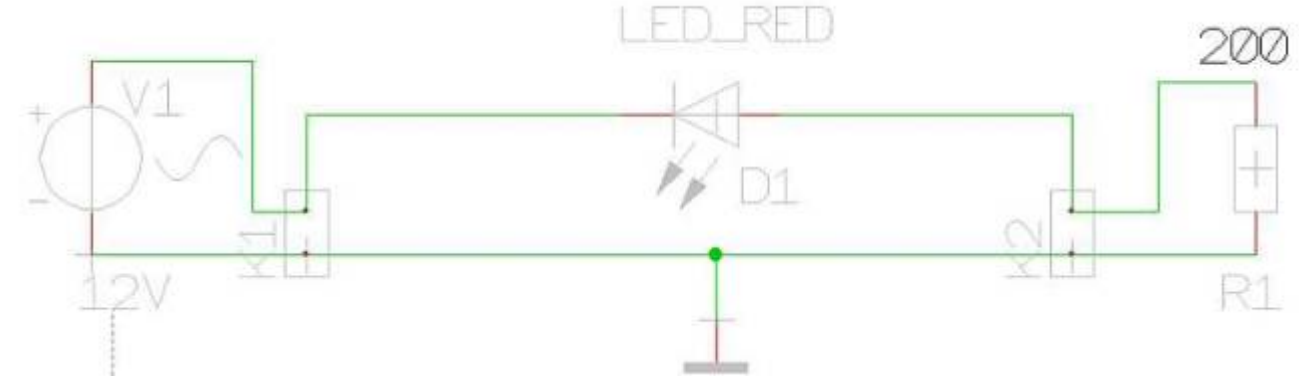




We define it's component value by 12V. By the use of the "Models" button in the same dialog we set the frequency. In the following dialog press button "Edit". Afterwards a setting of the parameters can be done:



At "FREQ" enter value 50. Now the preliminaries are done. We confirm all dialogs and the schematic might look like this:



Next step
One Step back

Back to the overview

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Simulate_the_function_Part_1"

Categories: Simulation

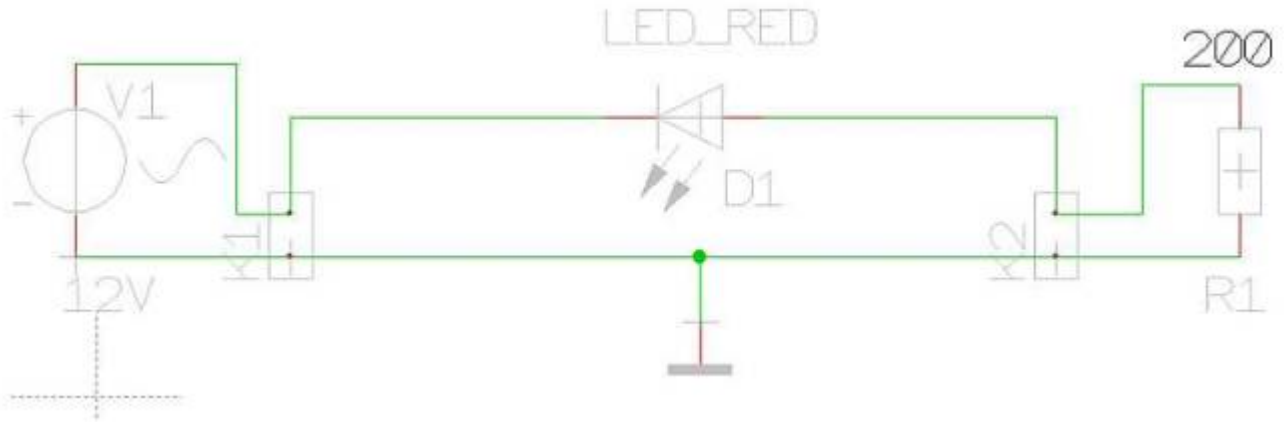
- This page was last modified 10:21, 30 August 2007.

Simulate the function Part 2

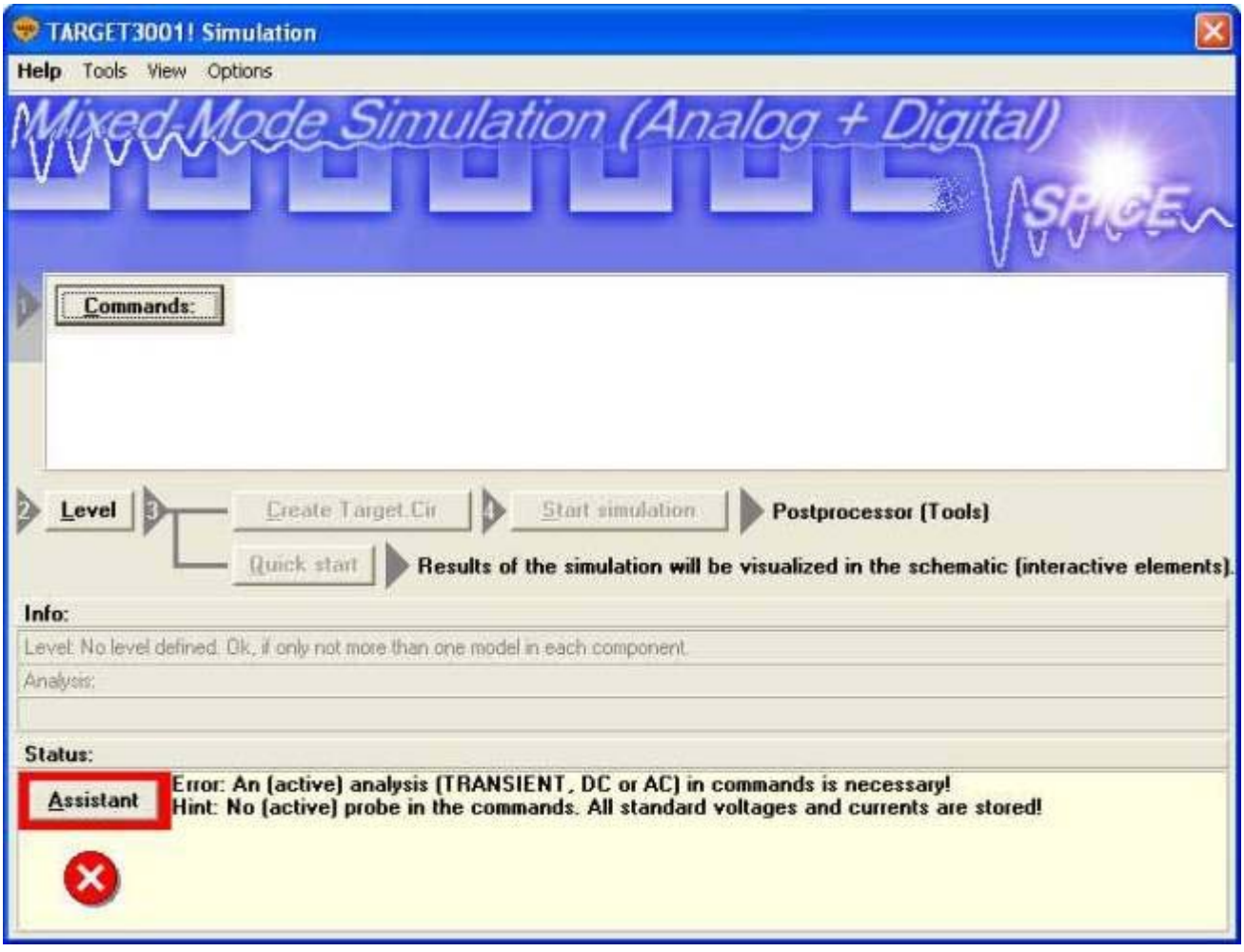
From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

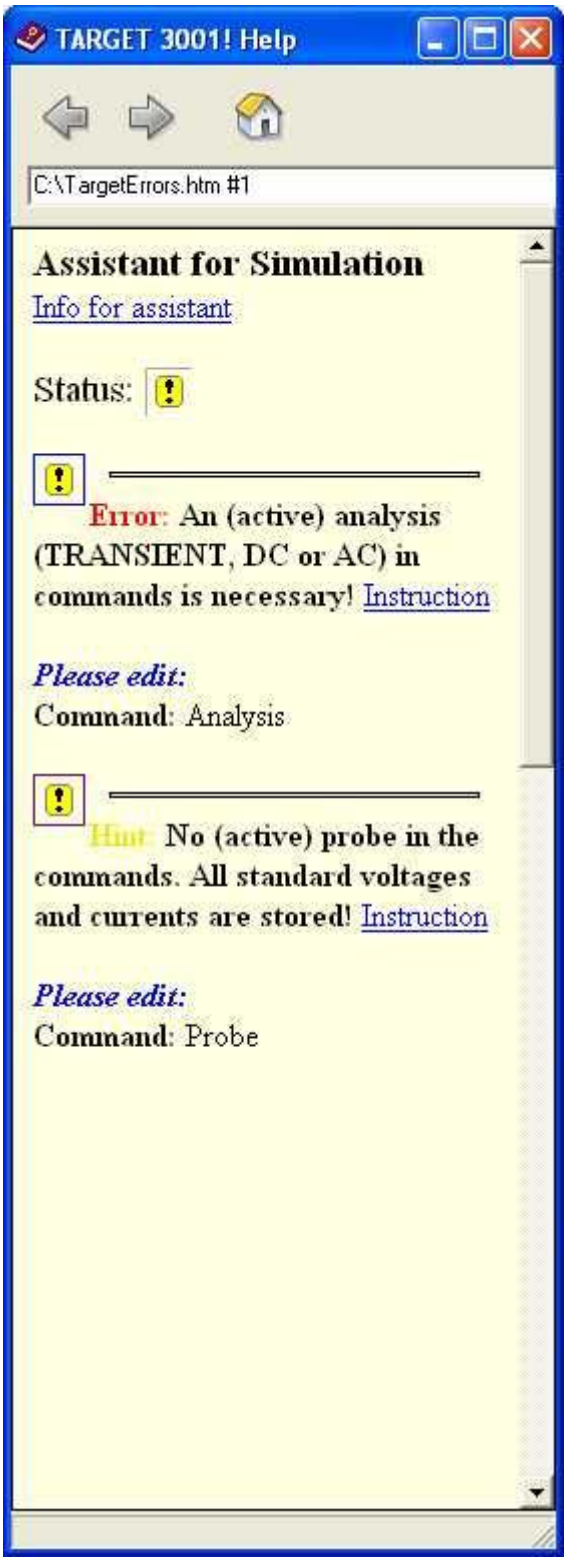
Based on the following schematic...



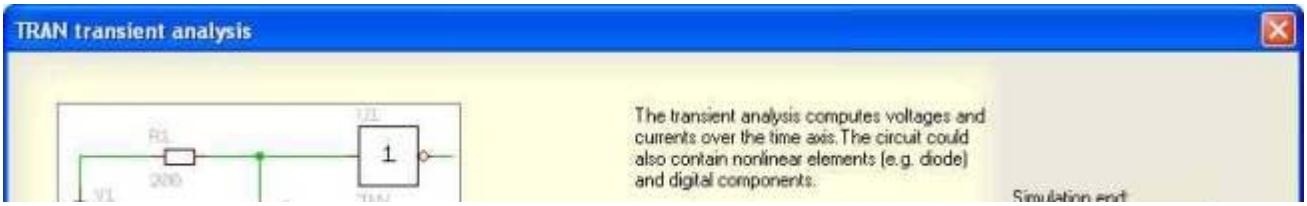
...we start the real simulation. We want to know which current flows in the LED at which tensions. First we start the simulation tool by the use of the function key **[F9]**. The following dialog appears:

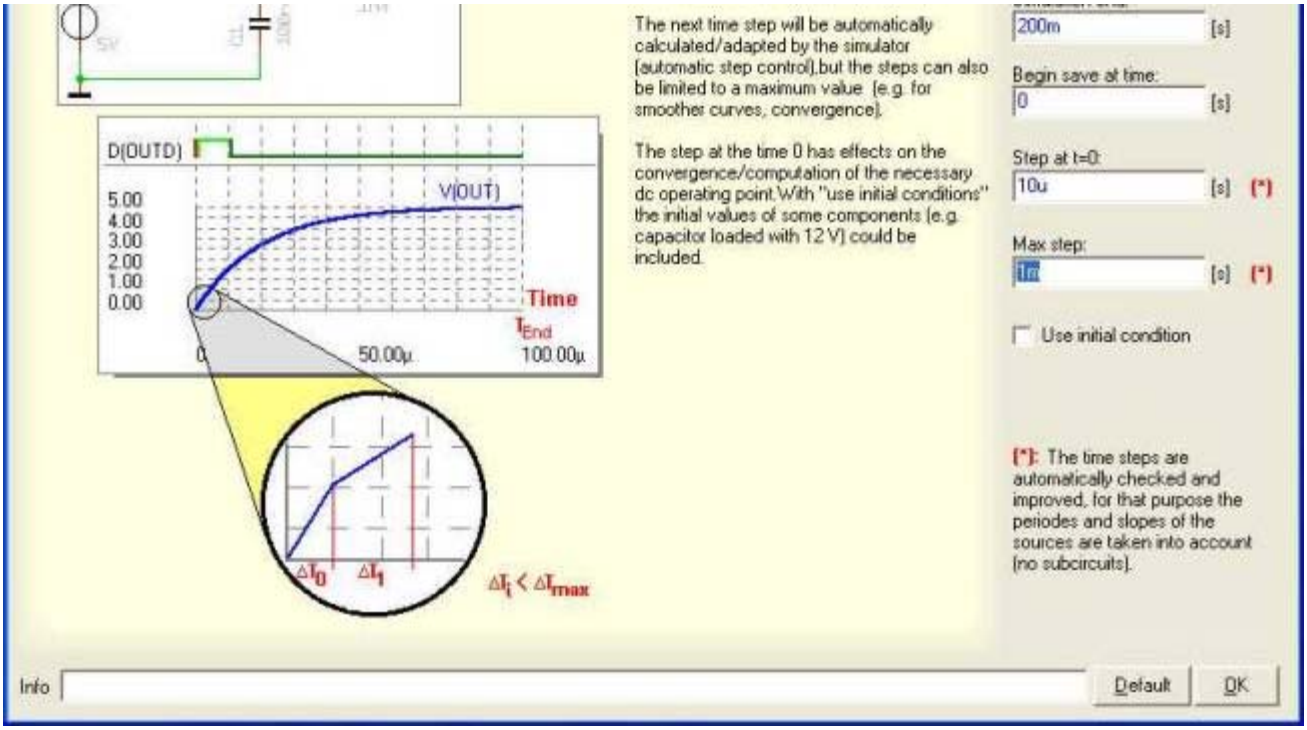


Look at the assistant. It warns in a line, that some of the components don't have a model. By M11 on this line we learn...

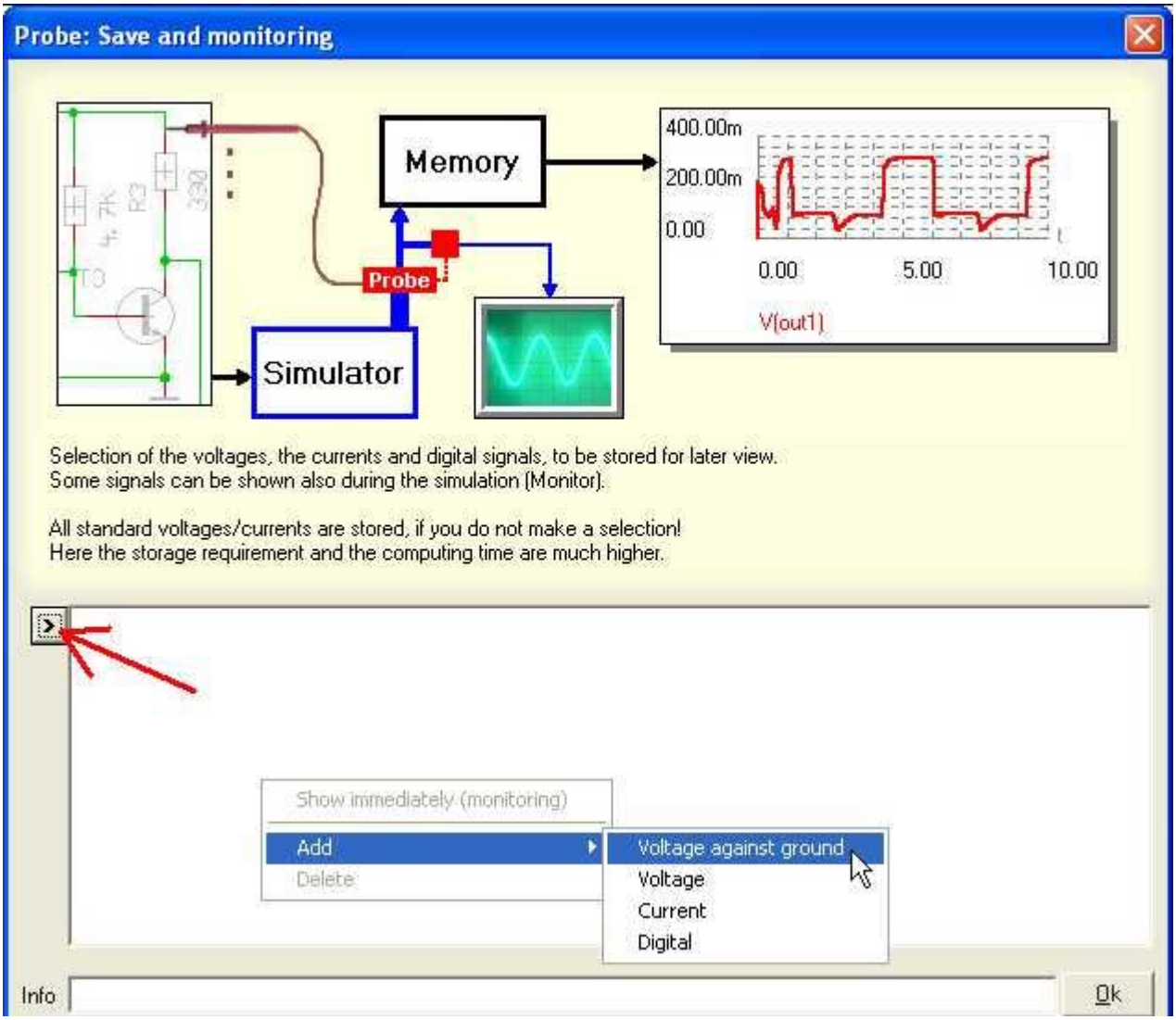


...that the connectors obviously don't have a simulation model. We ignore this message, because we don't want to simulate their function anyway. Furthermore it is said: **Error: An (active) analysis (TRANSIENT, DC or AC) in commands is neccessary.** and an advice follows: **No (active) probe in the commands.** Thus we click M1 on the commands button. Under "Add" we choose "Transient-Analysis". The following picture appears:

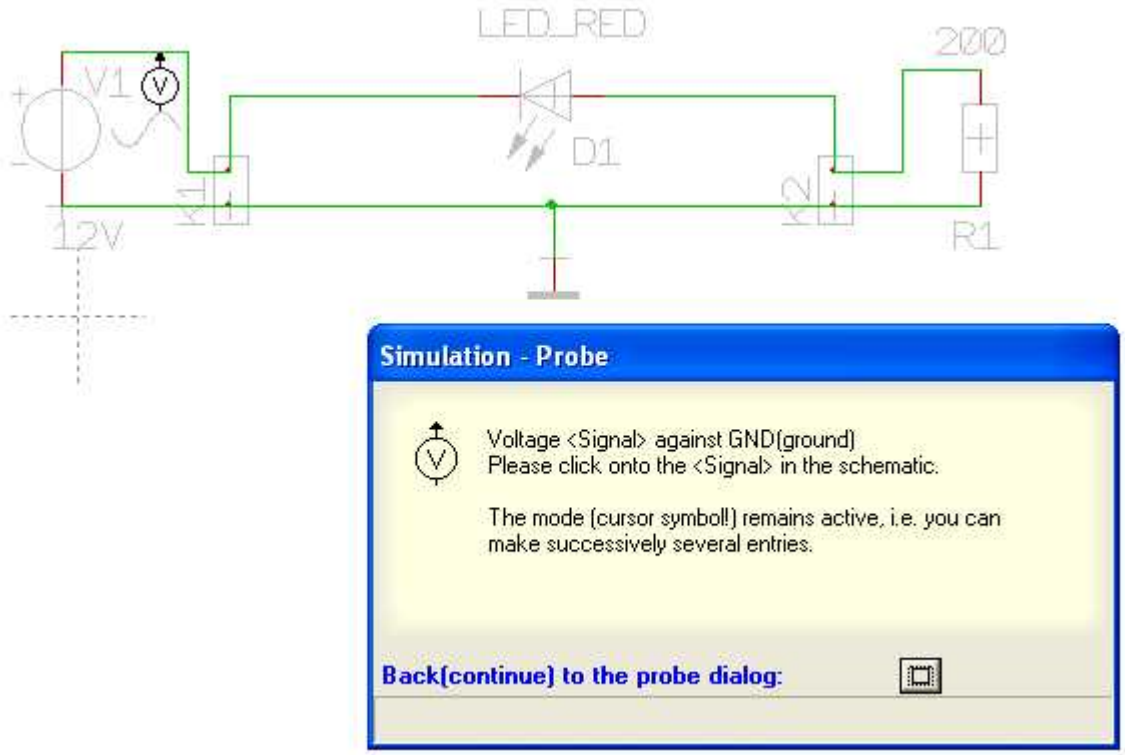




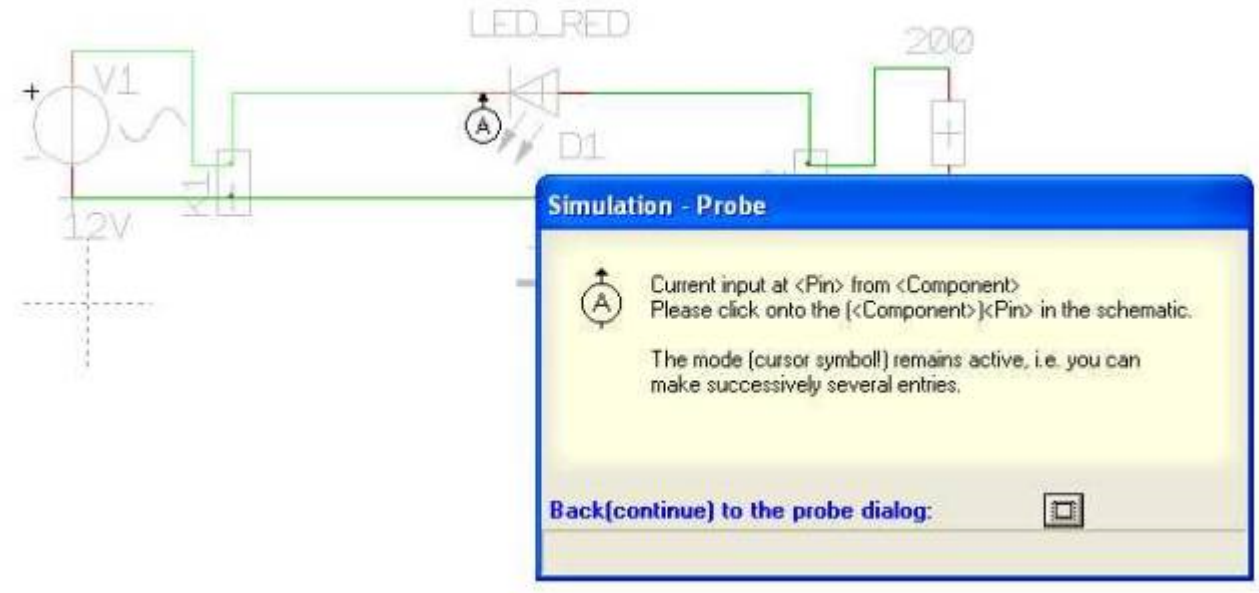
At a sinus tension with 50 (Hz) (= a period of 20ms) we choose the end of the simulation at a decuple (=200ms), so we see 10 oscillations. The step width we choose rather small, for getting a smoothe curve. Here a maximum step width of 1ms. As step width for a successful definition of a initial calculation point (convergency) we choose 10µs. Press OK. In the "Commands" button we now add: Probe. The probe-dialog opens. Here we define tensions and currents which shall be saved for simulation (if no probe ia added, standard tensions and currents are saved).



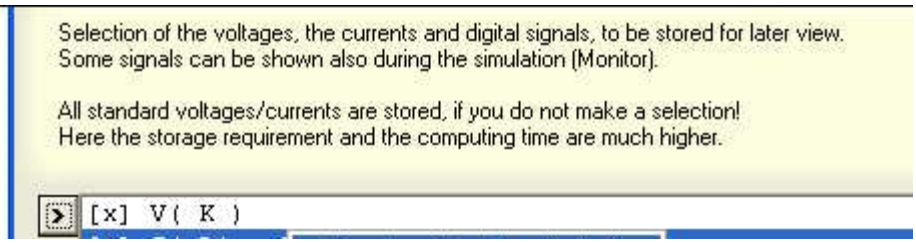
We are interested in the tension (sinus 12V)...



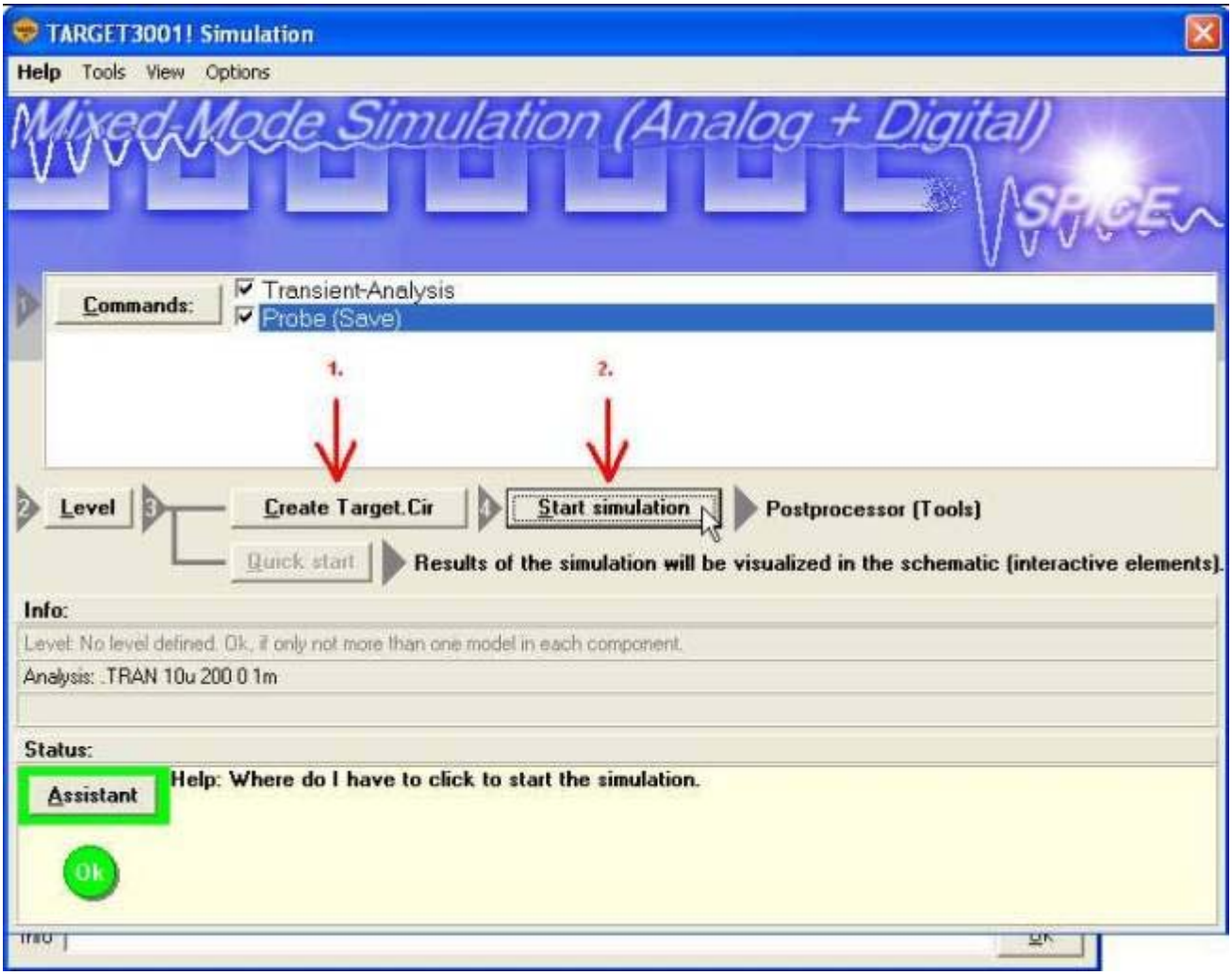
...and in the current through the diode (after selection of "current" in the probe of the main dialog). For the current **not** the signal track must be clicked **M1** but the connection pin itself.



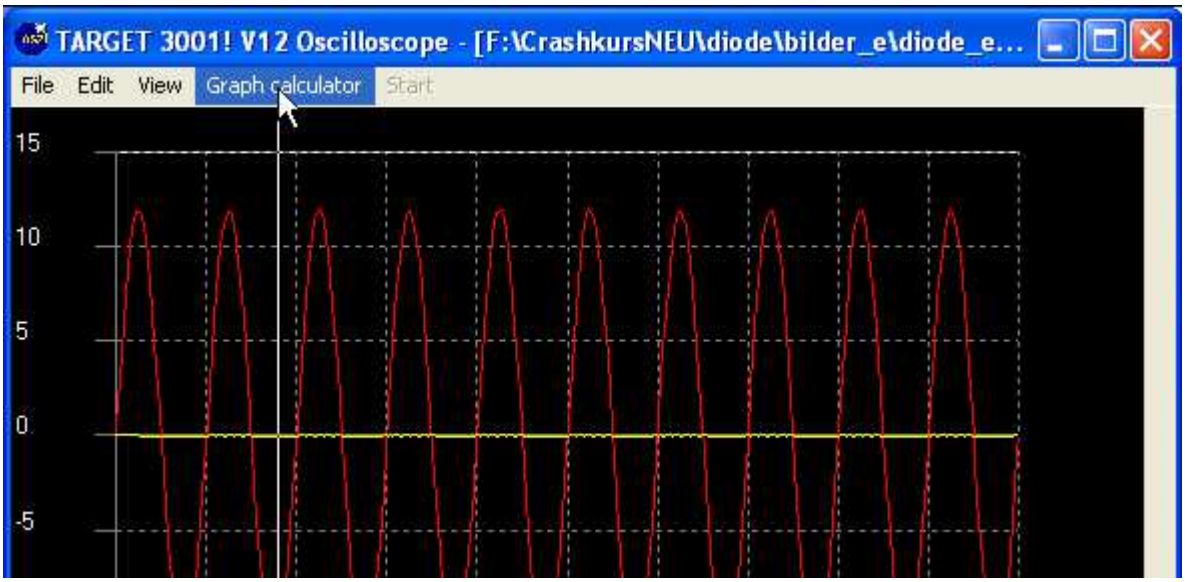
Now click the button "Back (continue) to the probe-dialog".

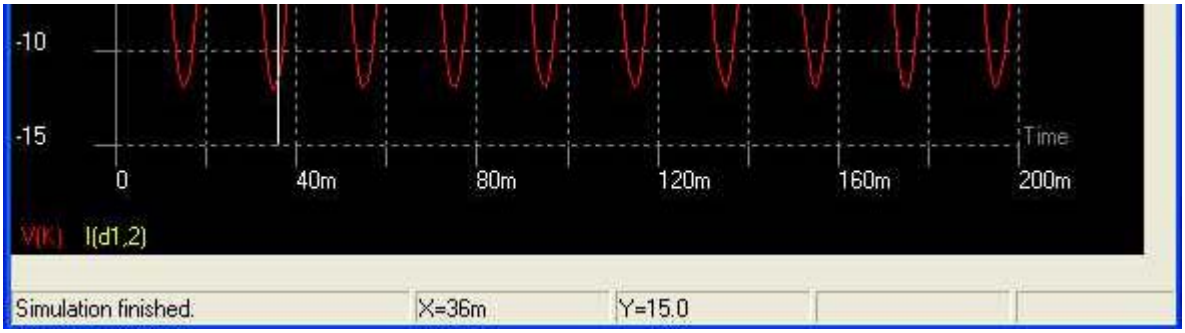


The "immediate screening" (intermediate data are displayed during the simulation process) should be activated by M2 on the corresponding list entry in the probe.

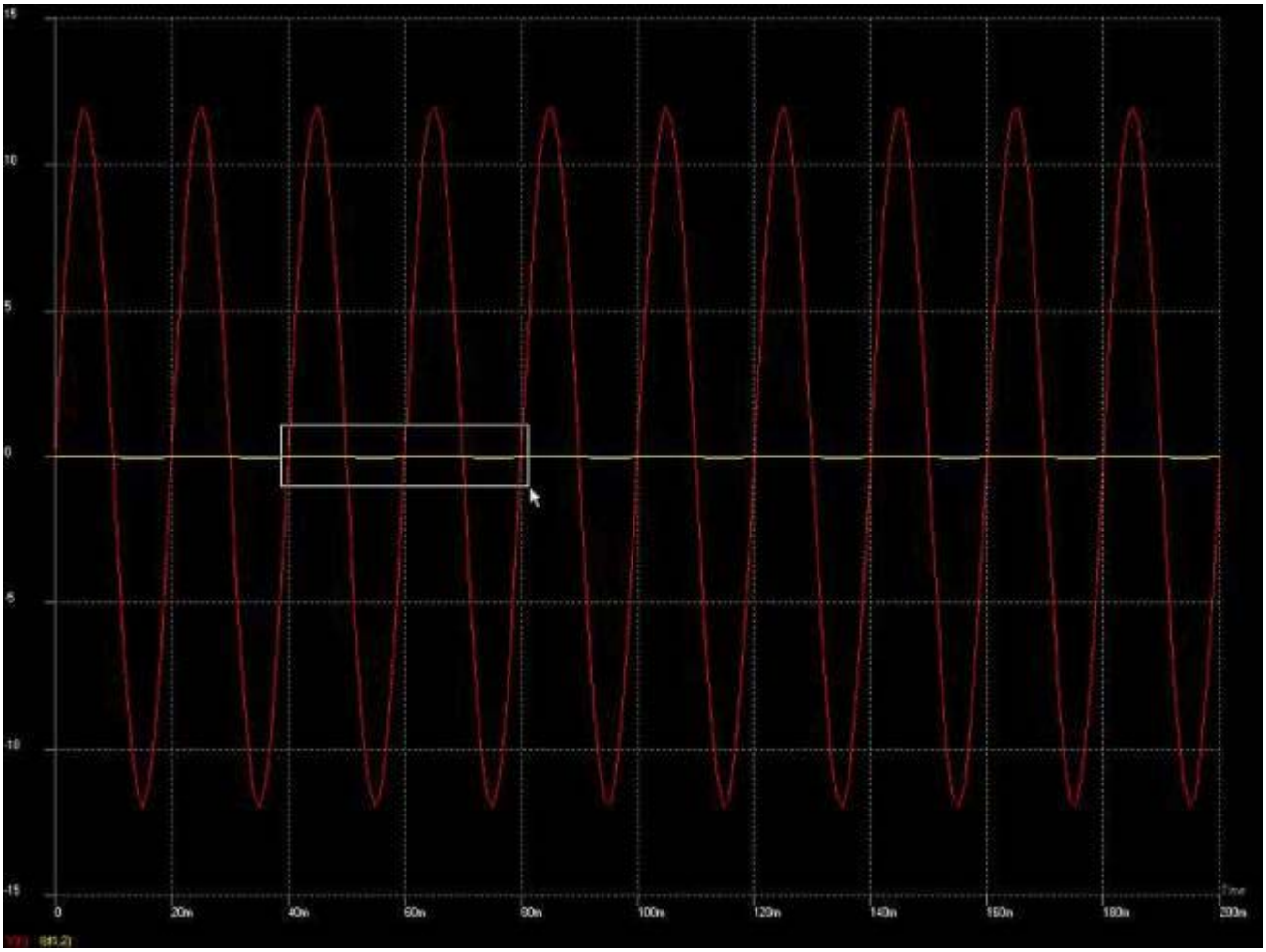


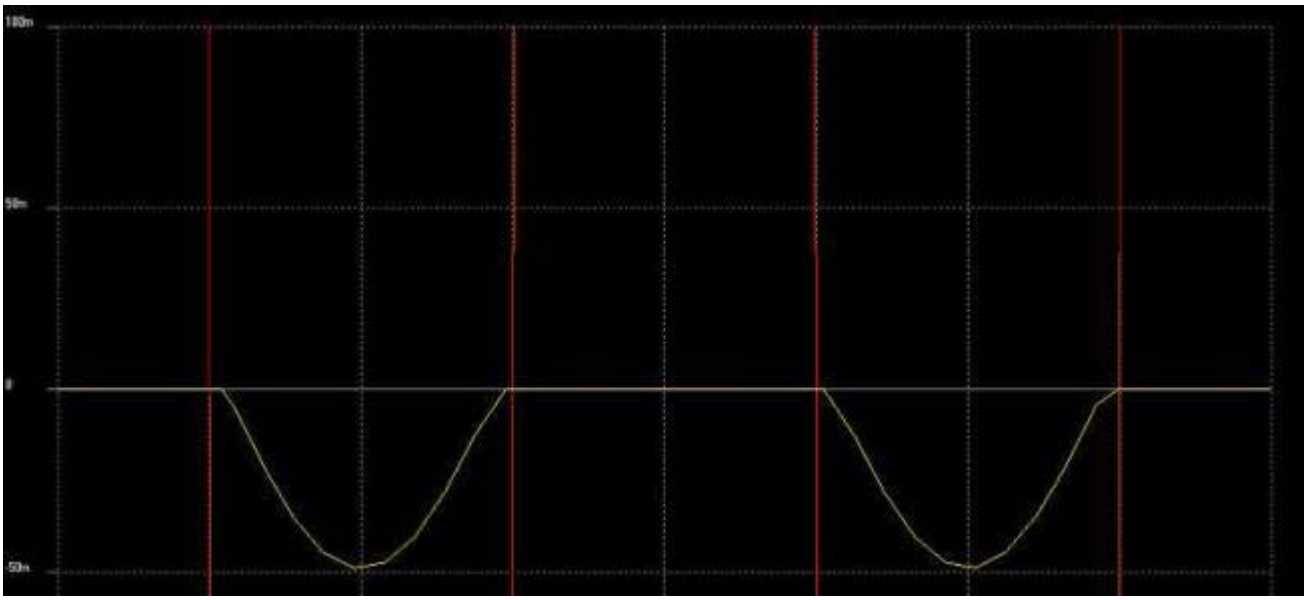
In menu "Graph calculator" you can choose different colors for tension and current and receive:

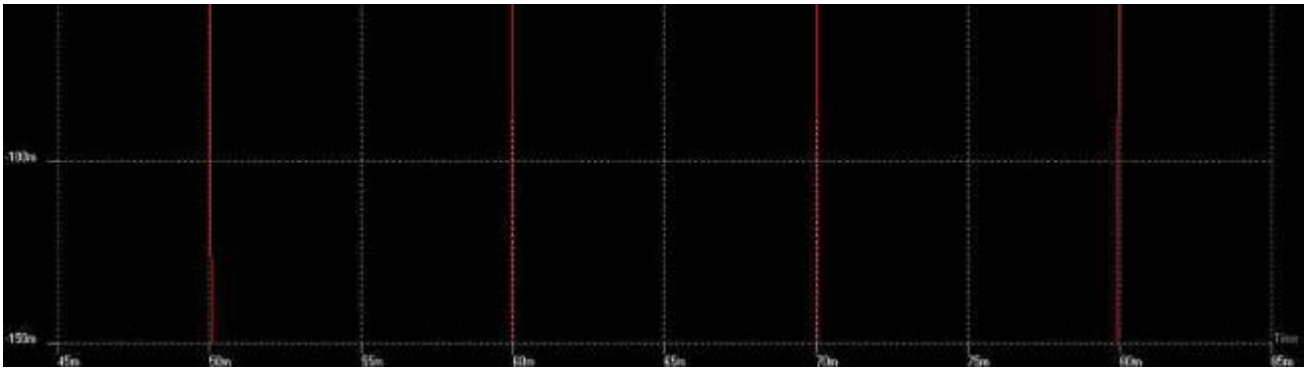




If we now zoom in and view a section of the graph, we can inspect the course of the tension as well as the current:







Next step
One step back

Back to the overview

Categories: Simulation

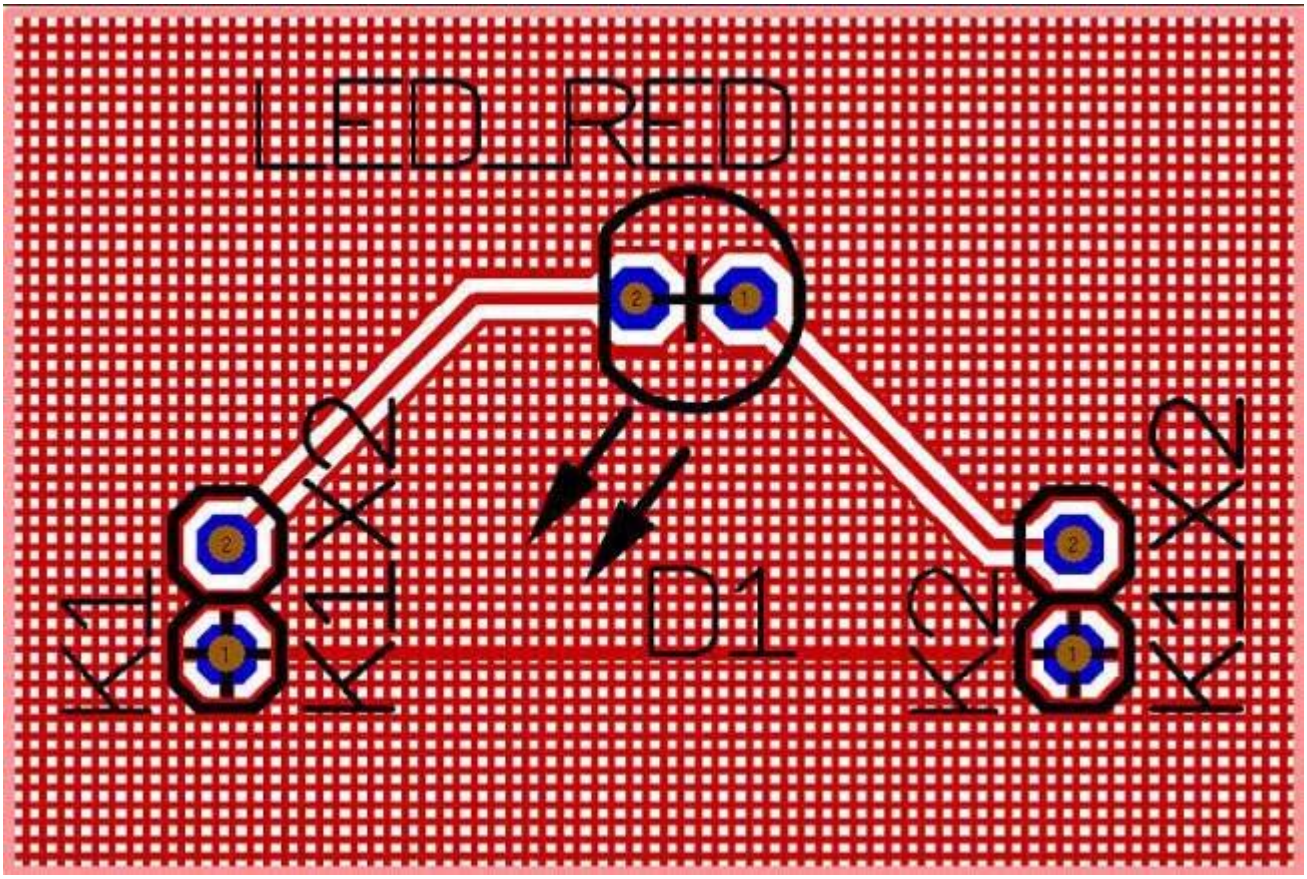
- This page was last modified 10:45, 30 August 2007.

3D-view of the layout

From IBF-Wiki

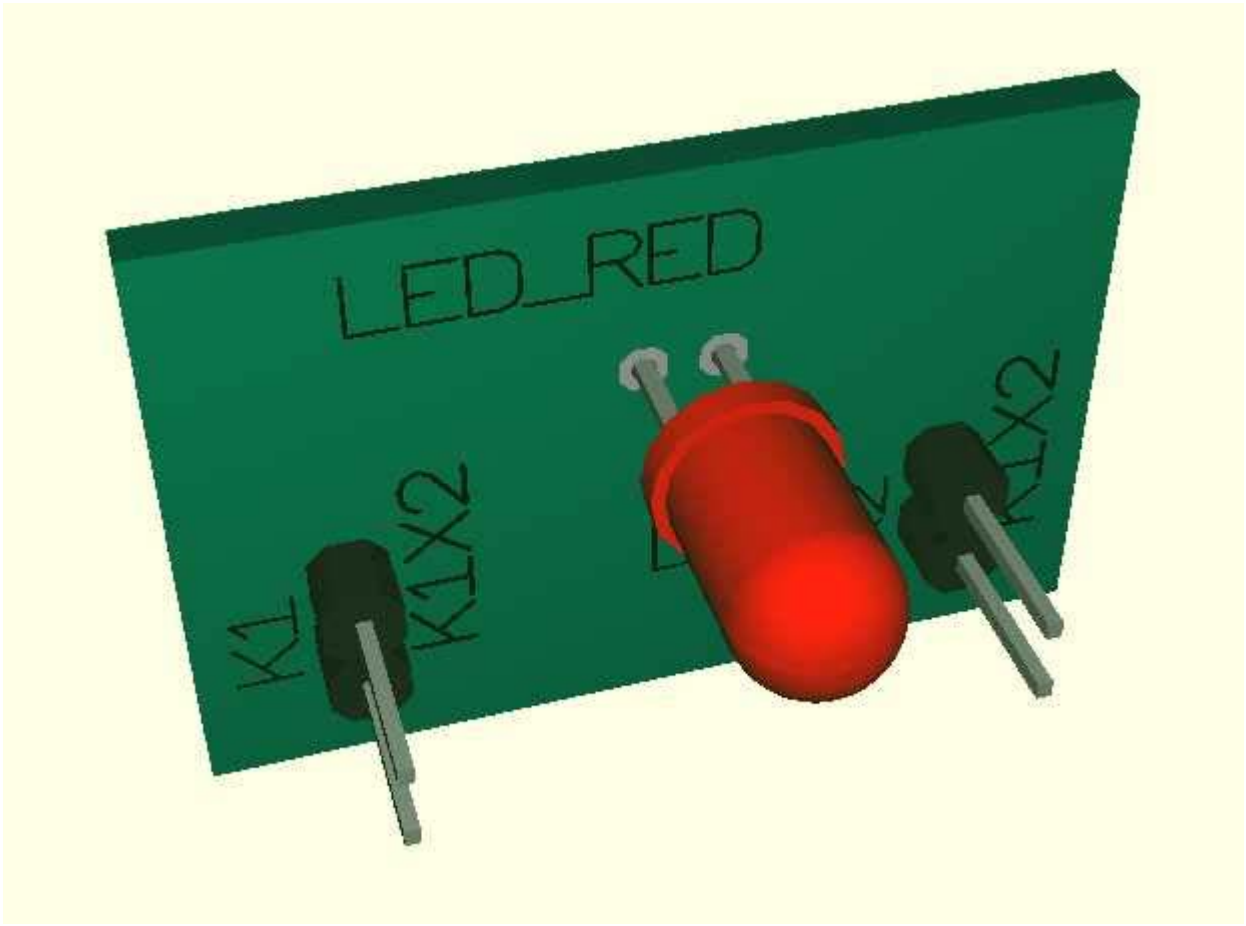
This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

Currently the layout looks like that:

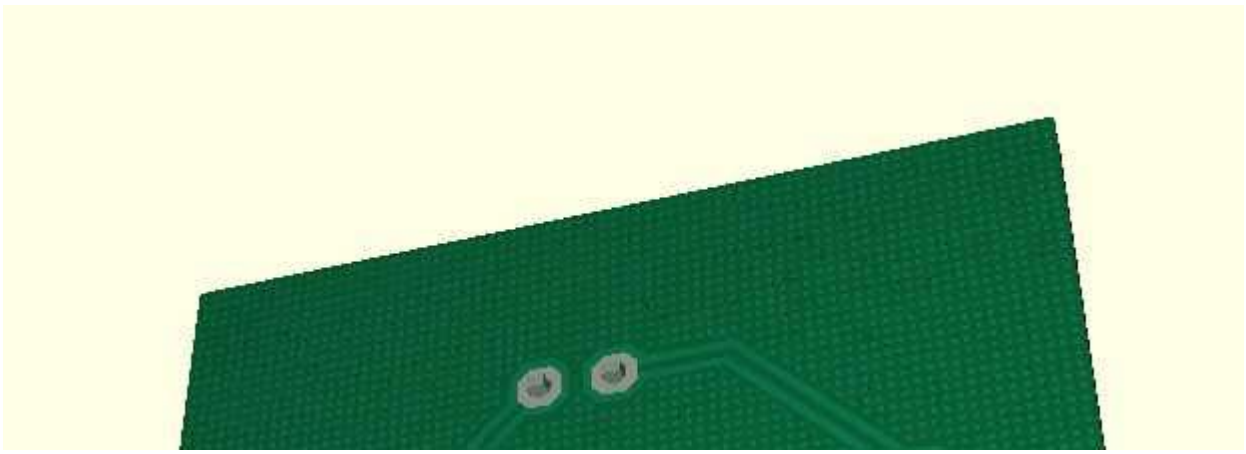


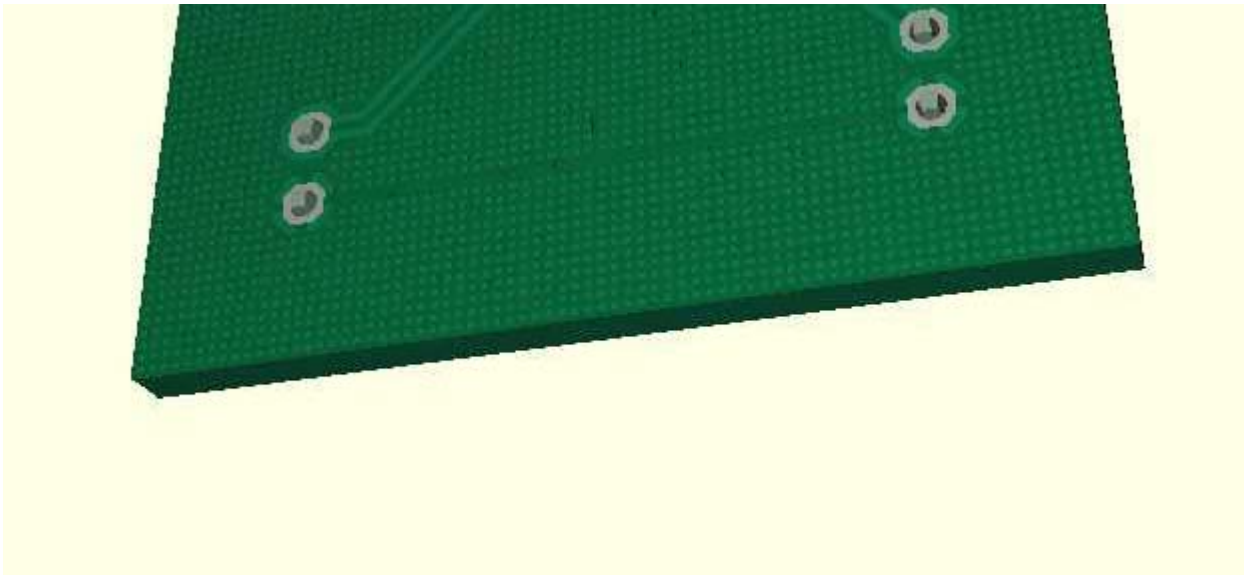
View it in 3D by menu "Actions/3D-View". After a short time of computation the result is:

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



See the grid structure on the bottom side as well as the Thermal Pads and all the other things...





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 the small hooks. With a **M1** click onto the names you can mark the parts in blue color on the right side in the 3D view.

Please note that a solid (not gridded) groundplane currently is not displayed in 3D View. But the computation time before displaying 3D is much shorter in that case.

Next step
One step back

Back to the overview

Categories: Automatic functions | Modes | About TARGET 3001!

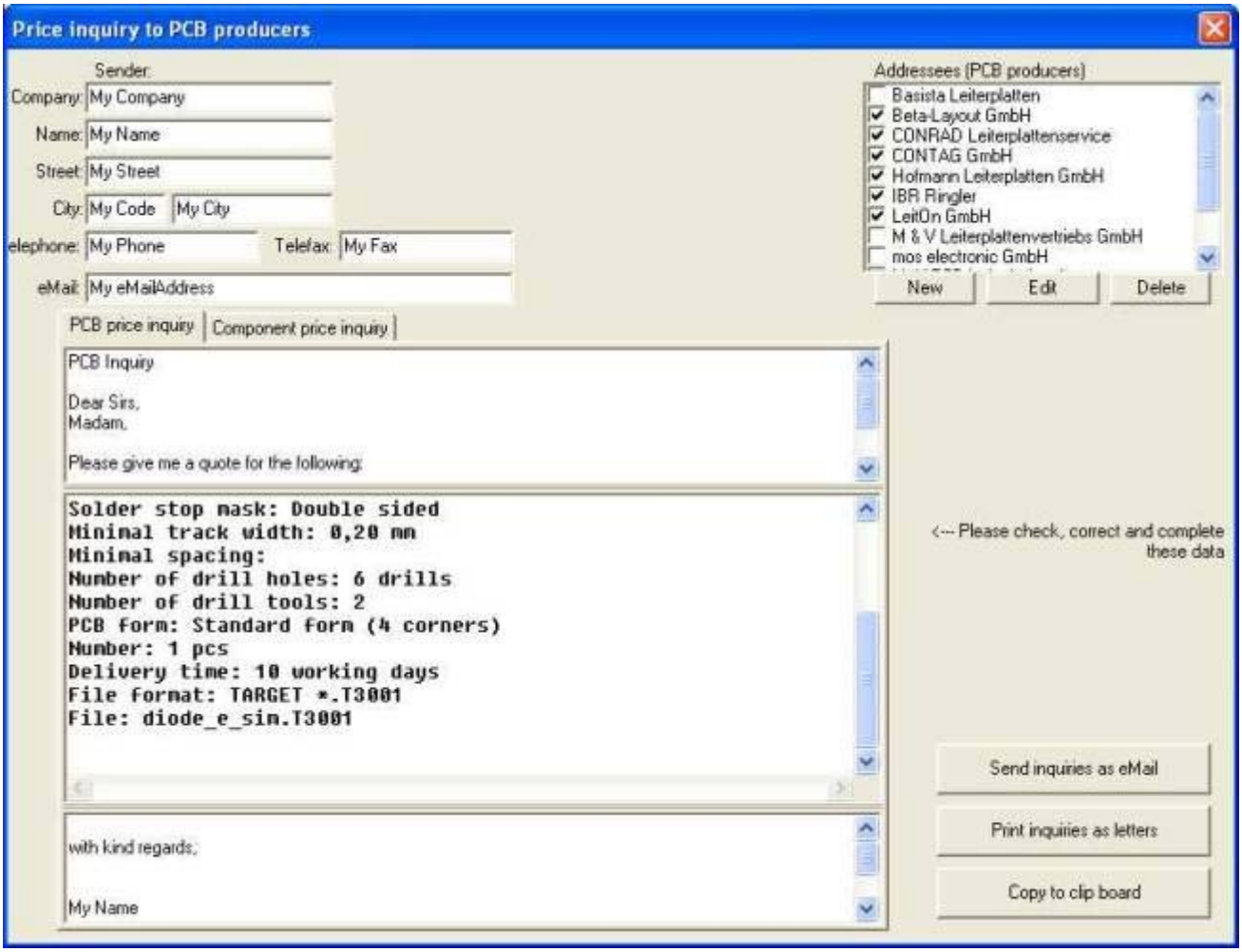
- This page was last modified 10:49, 30 August 2007.

Produce a PCB

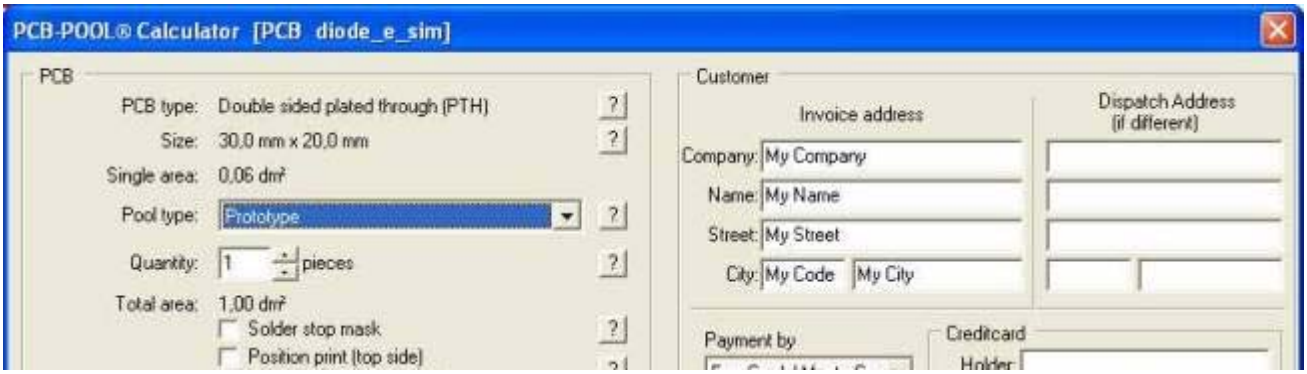
From IBF-Wiki

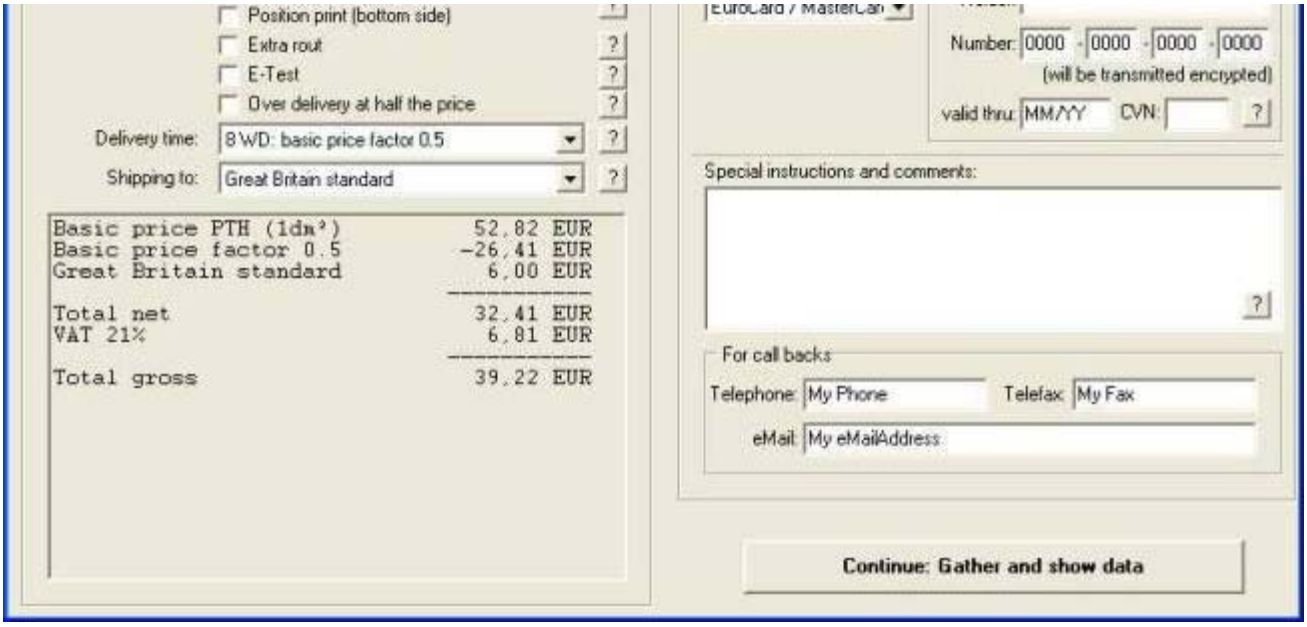
This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

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 enquiries 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 (R) calculator. See Menu "File/Produce OCB in PCB-Pool(R)". PCB-Pool (R) is a German pcb manufacturer who directly accepts TARGET-files:





A prototype (1 piece) of this project costs EUR 39,22 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 projectfile before 9.00 am to PCB-Pool (R), your PCB is dispatched the next working day. This is a unique cooperation between TARGET 3001! and PCB-Pool (R). In case you're really in a hurry...

Next step
One step back

Back to overview

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Produce_a_PCB"

Categories: Actions | Automatic functions | Production

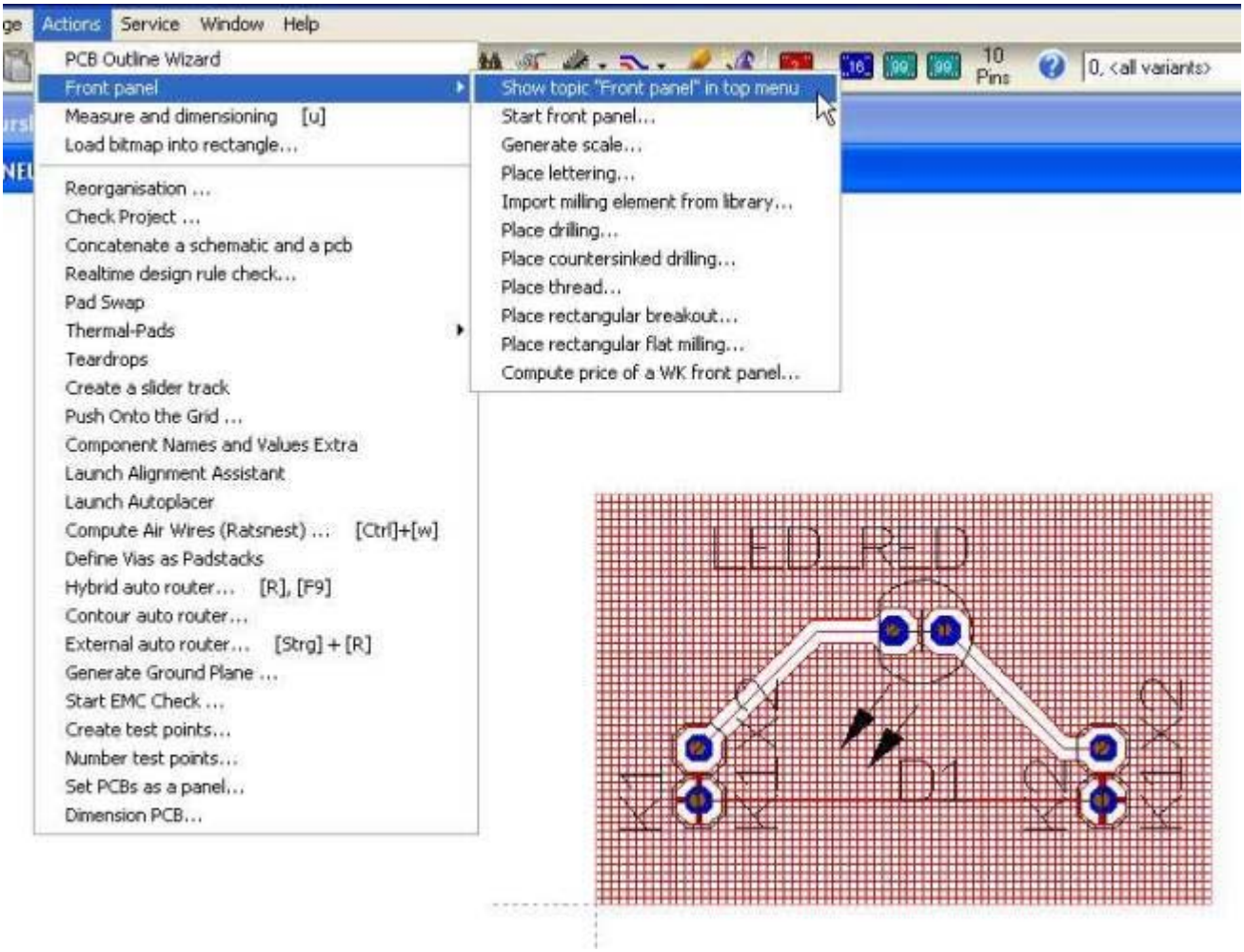
- This page was last modified 10:53, 30 August 2007.

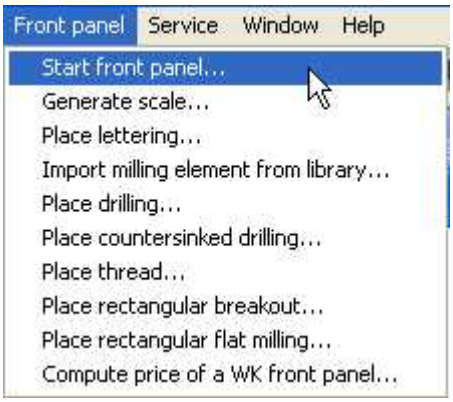
Design and produce a frontpanel

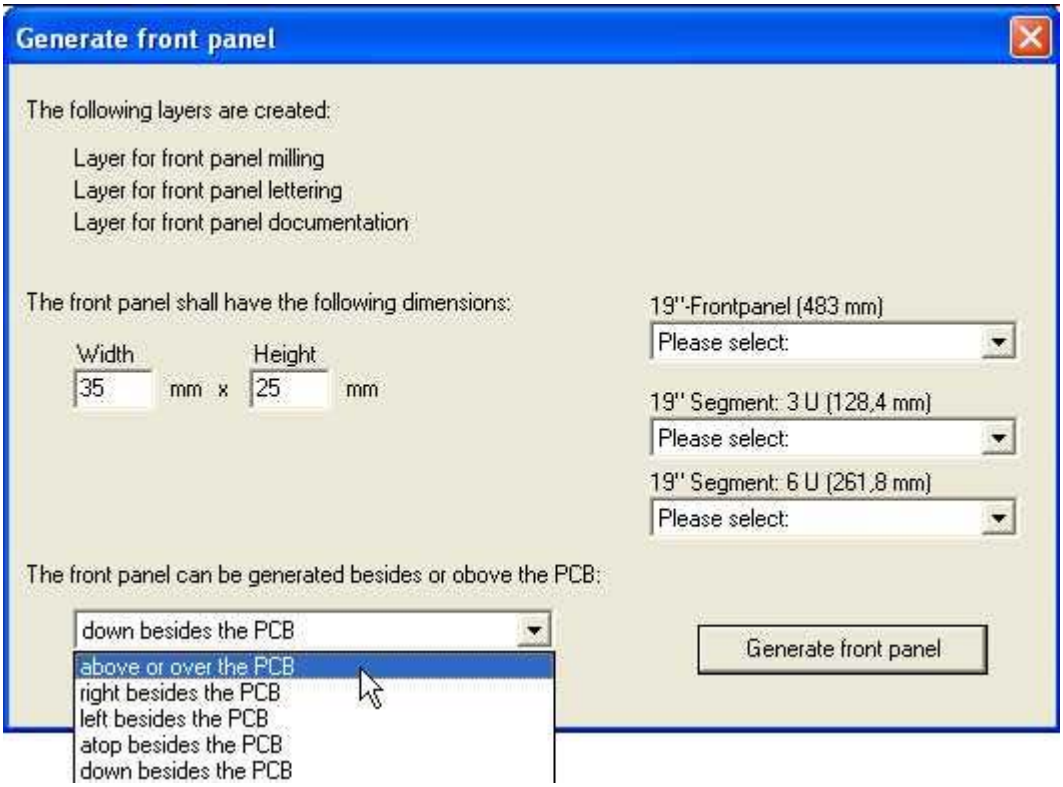
From IBF-Wiki

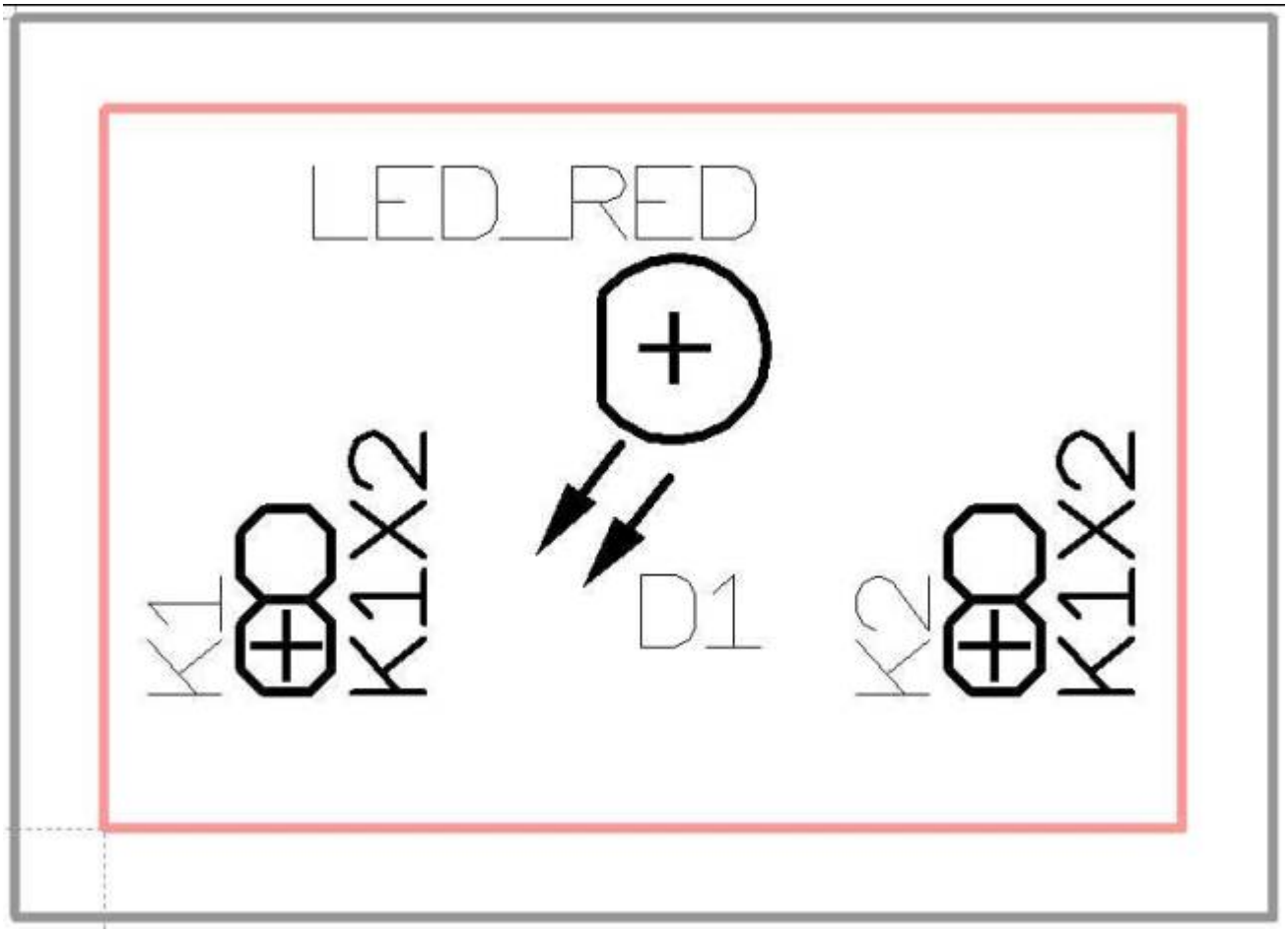
This article is part of an introduction tutorial called CrashCourse2
Download this article as PDF-File

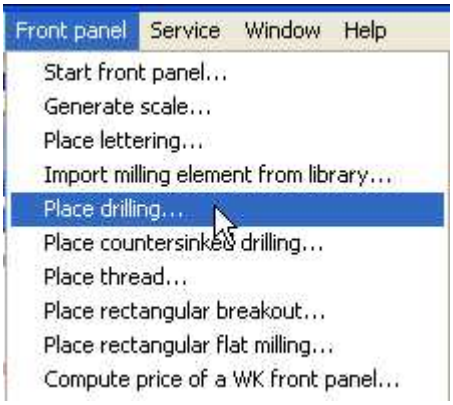
This process is that easy, that the images should be sufficient...

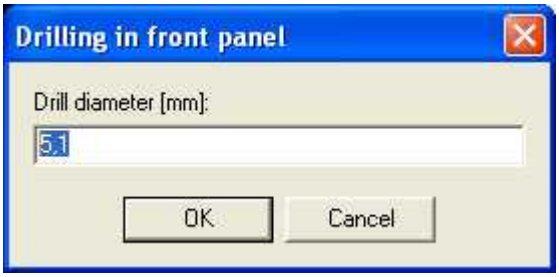


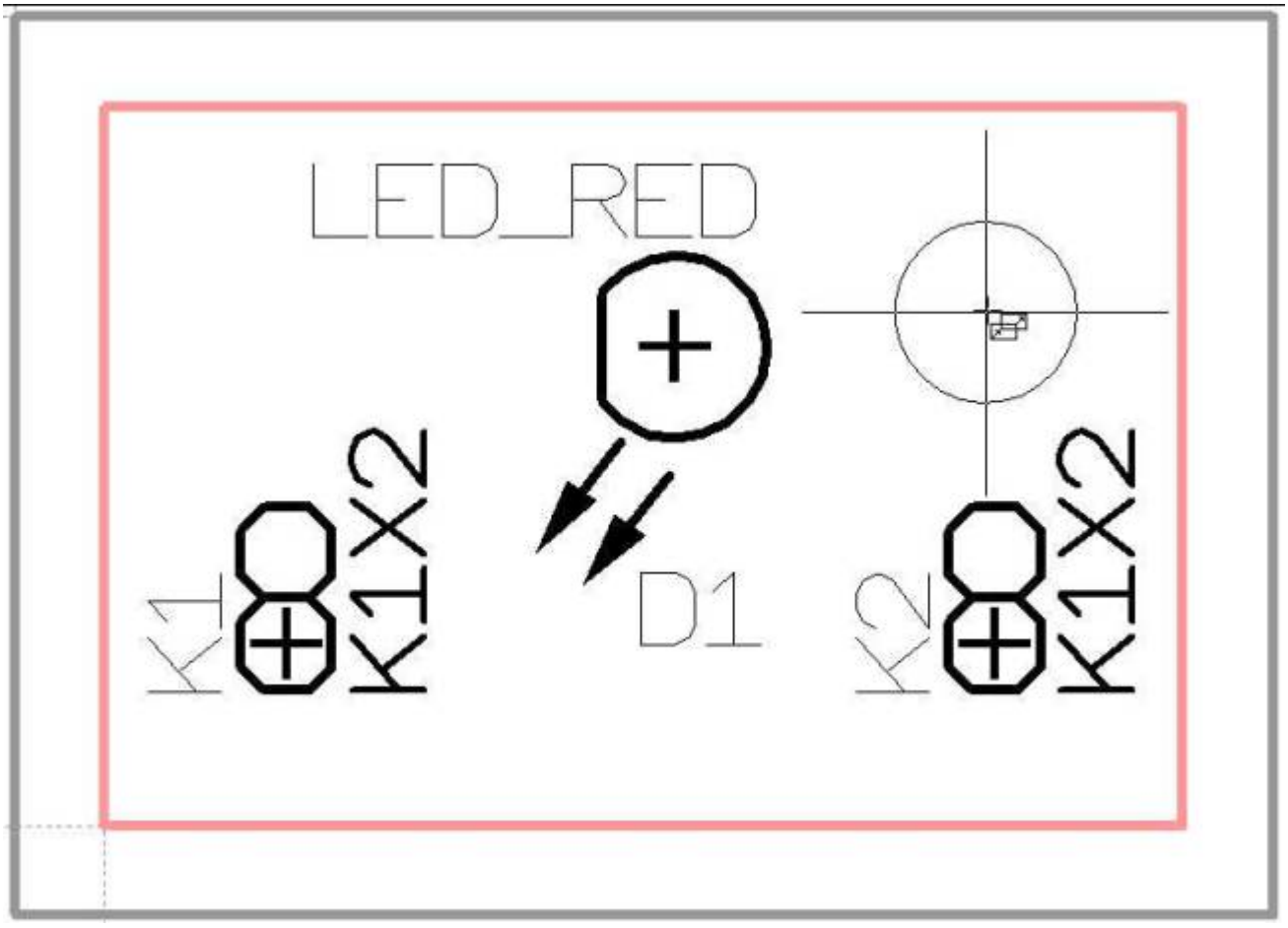


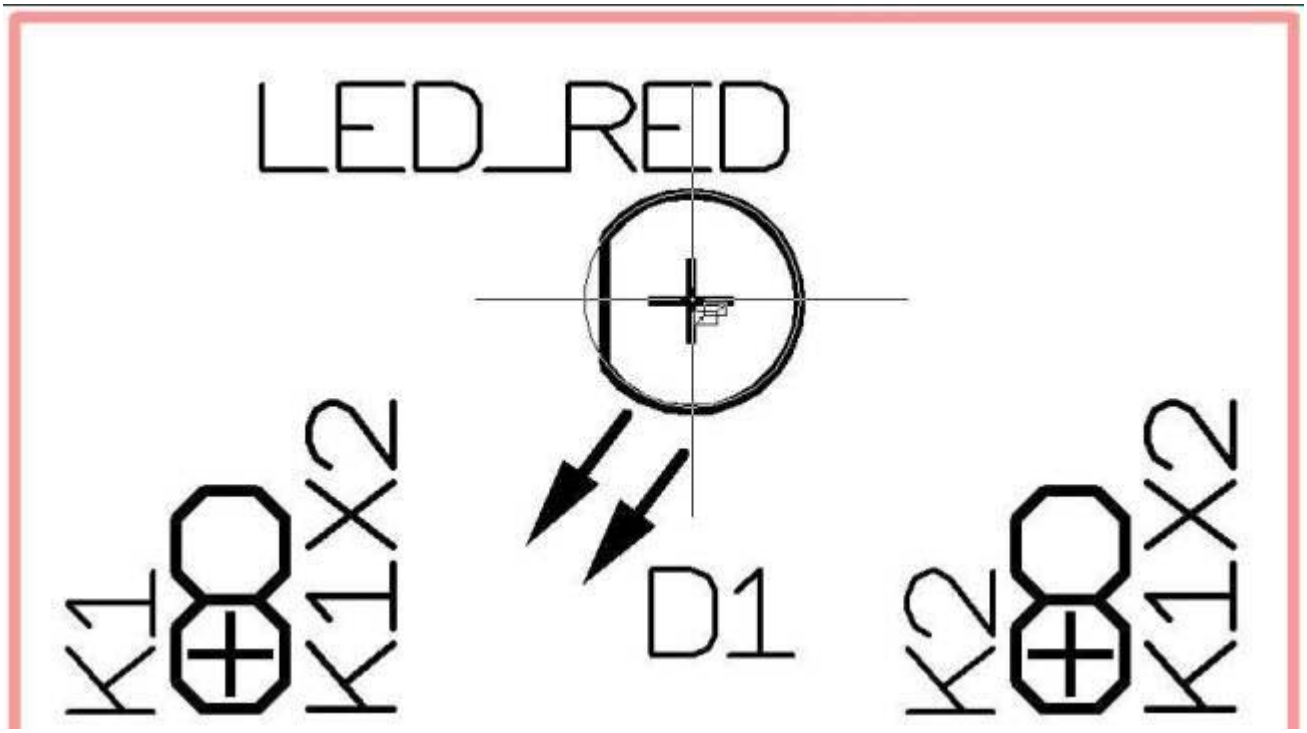

Generate front panel
The following layers are created:
Layer for front panel milling
Layer for front panel lettering
Layer for front panel documentation
The front panel shall have the following dimensions:
Width
35
mm x
Height
25
mm
19''-Frontpanel (483 mm)
Please select:
19'' Segment: 3 U (128,4 mm)
Please select:
19'' Segment: 6 U (261,8 mm)
Please select:
The front panel can be generated besides or above the PCB:
down besides the PCB
above or over the PCB
right besides the PCB
left besides the PCB
atop besides the PCB
down besides the PCB
Generate front panel


LED_RED
D1
K1X2
K1X2

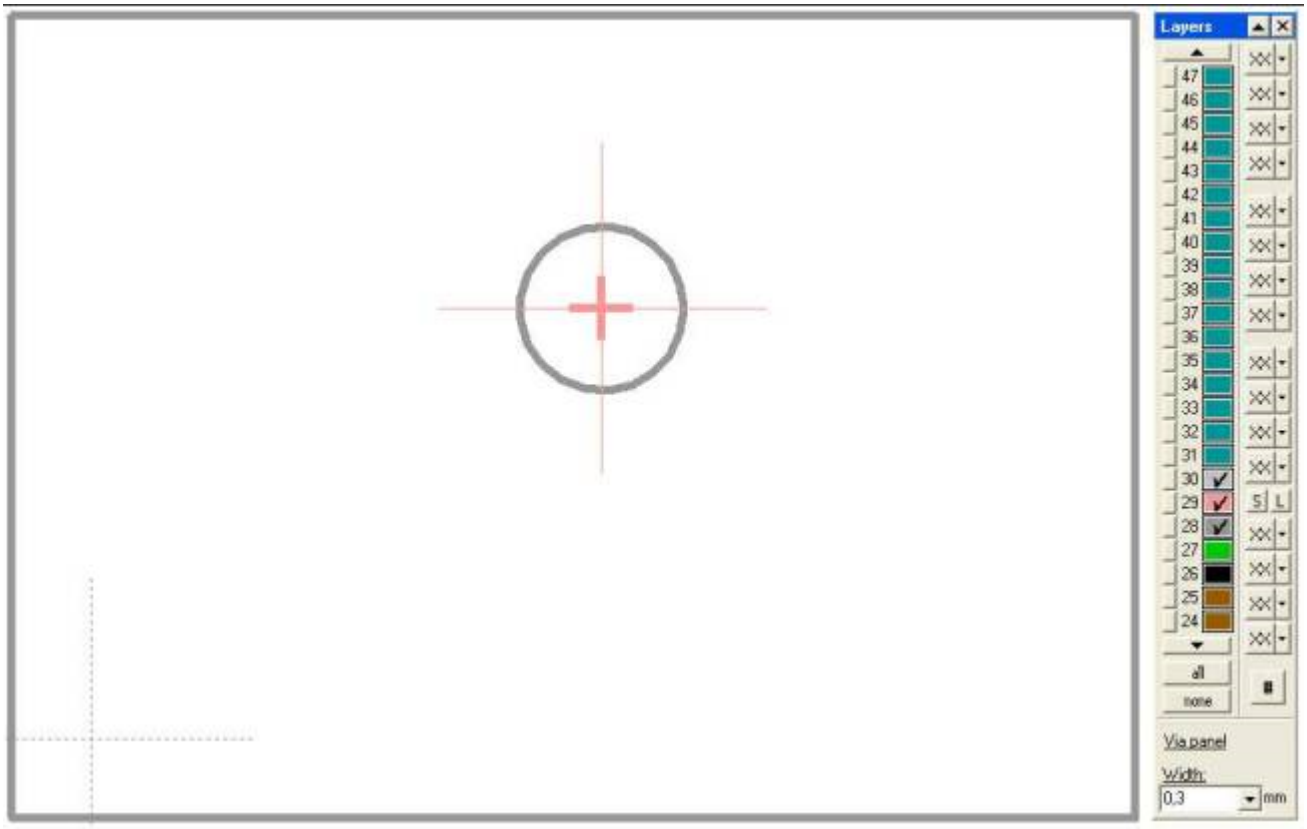

Front panel
Service
Window
Help
Start front panel...
Generate scale...
Place lettering...
Import milling element from library...
Place drilling...
Place countersinked drilling...
Place thread...
Place rectangular breakout...
Place rectangular flat milling...
Compute price of a WK front panel...

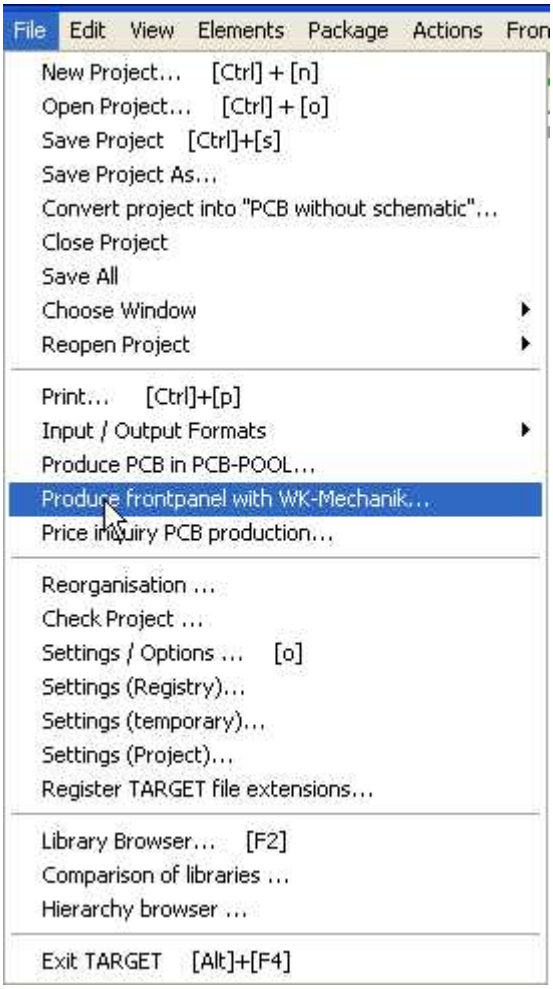

Drilling in front panel
Drill diameter [mm]:
5.1
OK
Cancel

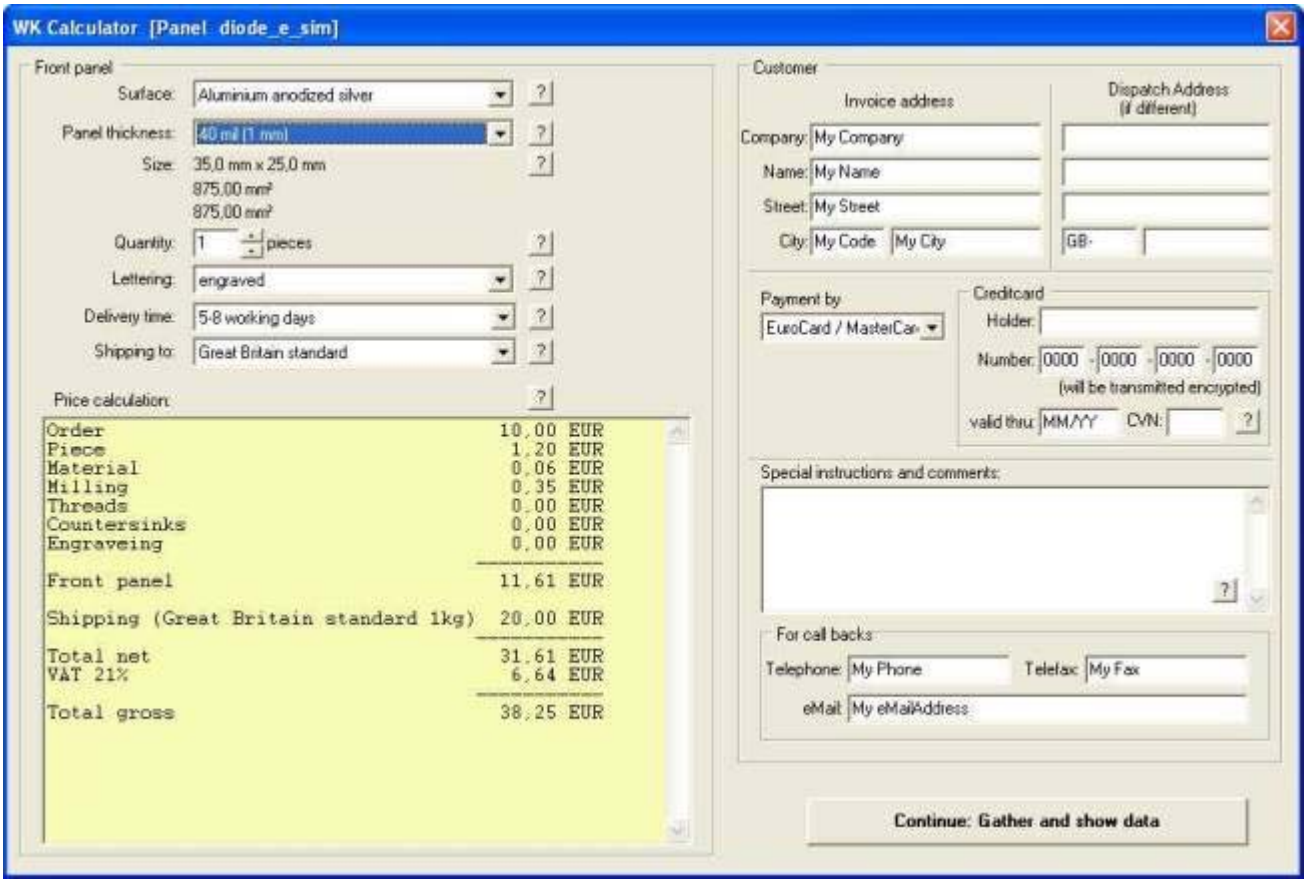

LED_RED
D1
K1X2
K1X2



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







The frontpanel data are generated in DXF for production at any other frontpanel producer.

This is the end
One step back

Back to the overview

Retrieved from "http://ibfriedrich.dyndns.org/wiki/ibfwikien/index.php?title=Design_and_produce_a_frontpanel"

Categories: Automatic functions | Actions

- This page was last modified 10:56, 30 August 2007.