

CrashCourse2

From IBF-Wiki

Based on a small project (one LED between two connectors) 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



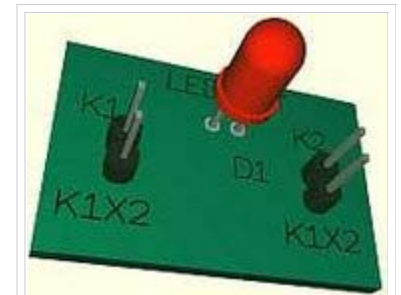
3D-view of the layout



Produce a PCB



Design and produce a frontpanel



Design a LED between two connectors in minutes!

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

Categories: First steps

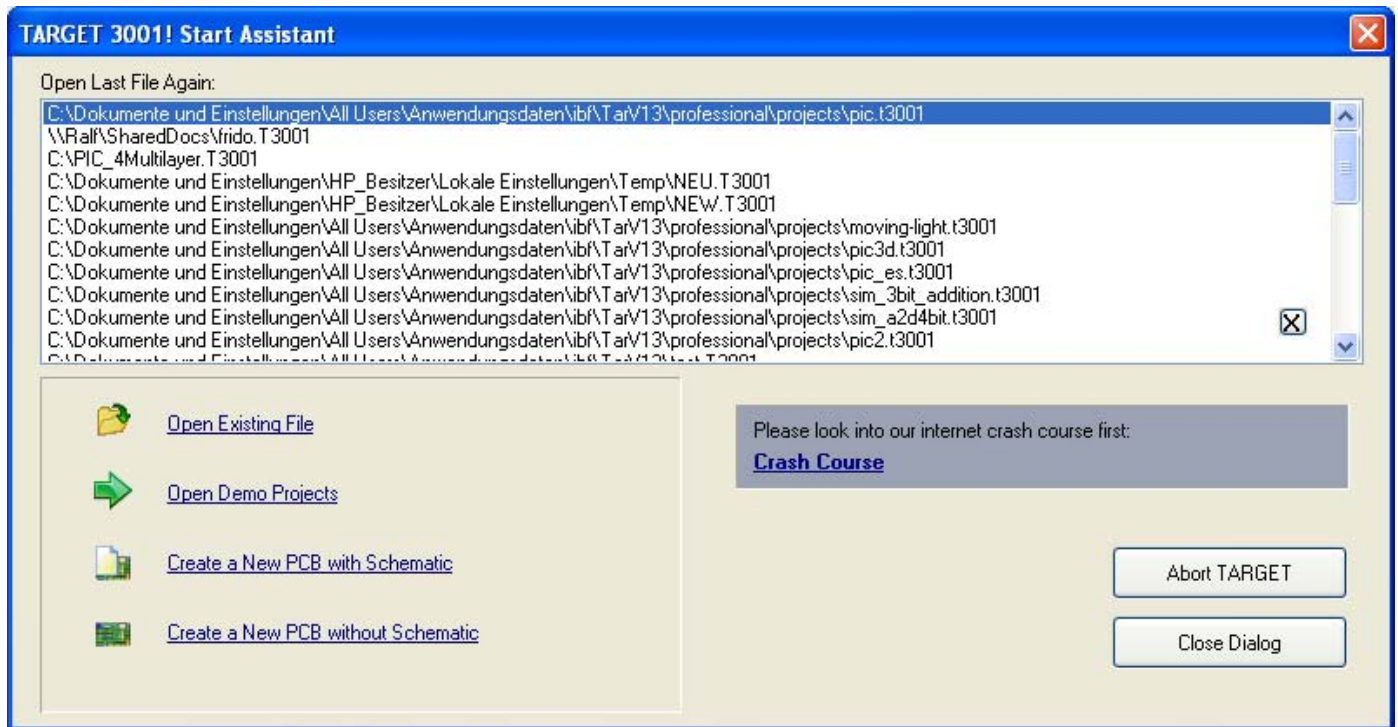
- This page was last modified 12:13, 10 May 2007.

Begin a new project

From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2

Download this article as PDF-File



Using the Start assistant you can select one of the 30 recently used projects from the list. You can open a different existing project or create a new project either with schematic or without. In the coloured field on the right you find a link to this crashcourse. In our case we decide for **Create a New PCB with Schematic**. An empty schematic page opens...

Next step

One step back

Back to overview

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

Categories: About TARGET 3001!

IBF-Intern:

- This page was last modified 13:26, 15 January 2009.
- This page has been accessed 7,146 times.

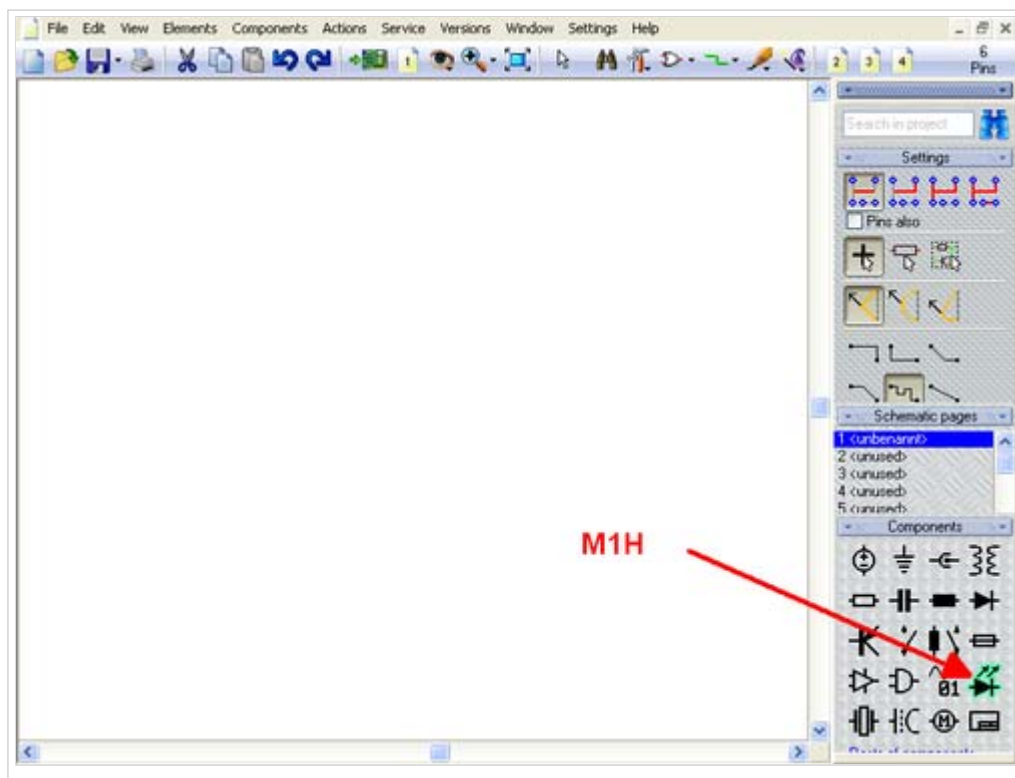
Import a component symbol to the schematic

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 sidebar. Click **M1H** upon the symbol pictogram and bring it into the schematic by drag and drop.



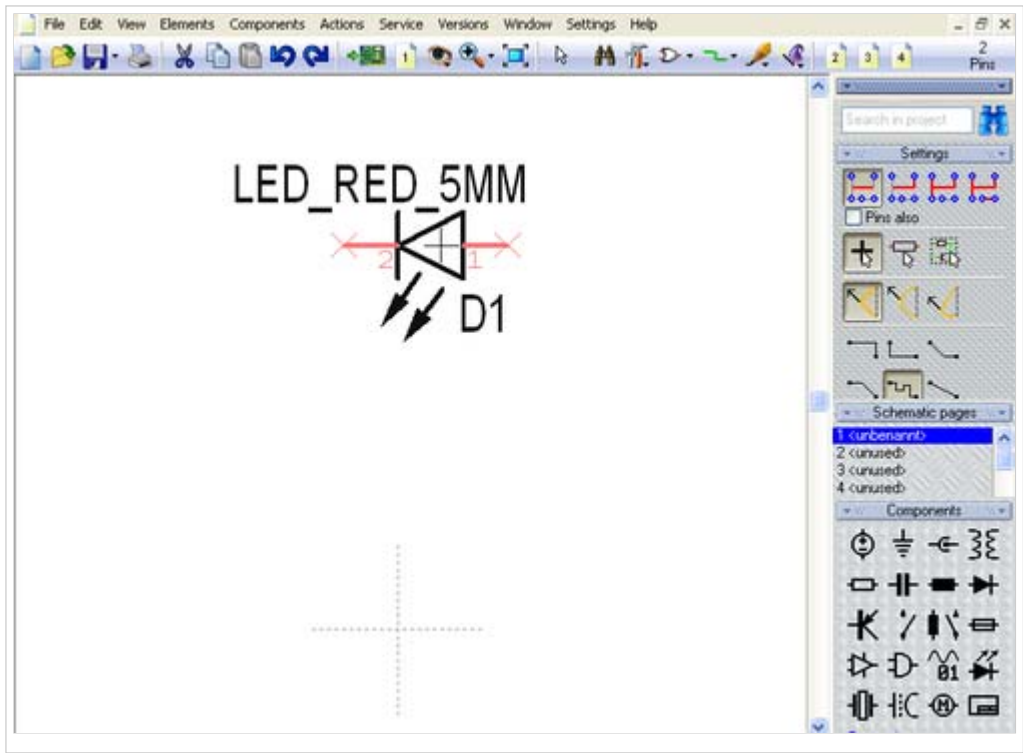
More
details?

Origin
Grid
Mousekeys
Key
combinations
Zoom

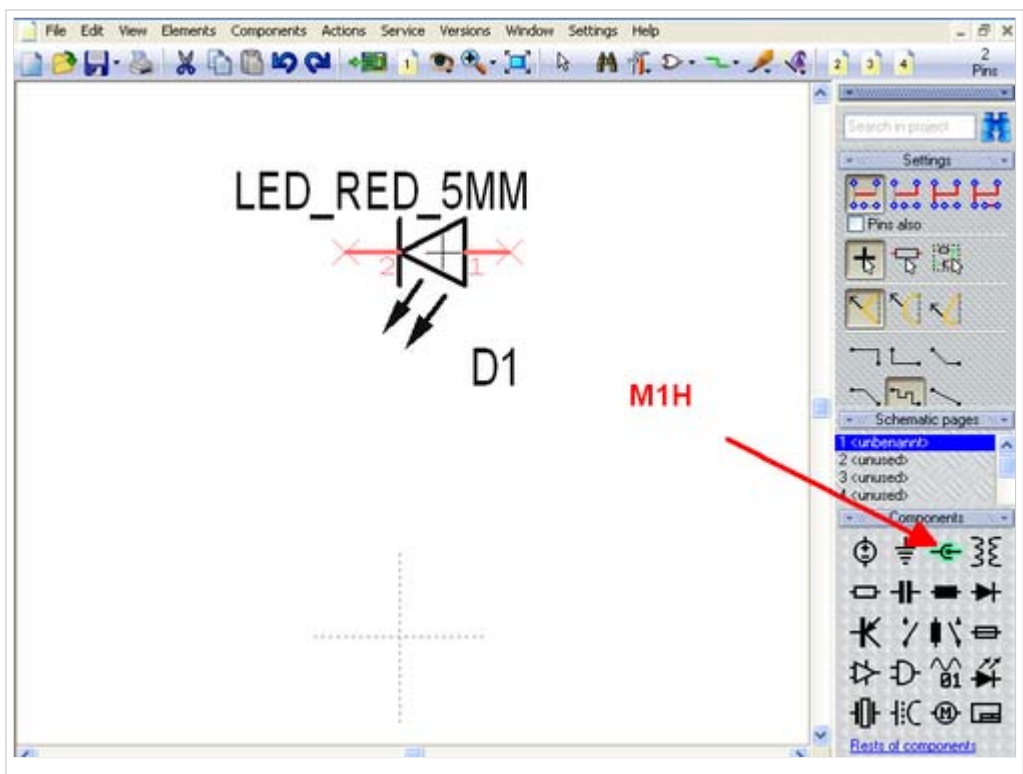
A click **M1** on the pictogram will open the library browser to import further components of this component family.

More
details?

Libraries
Drawing
frame



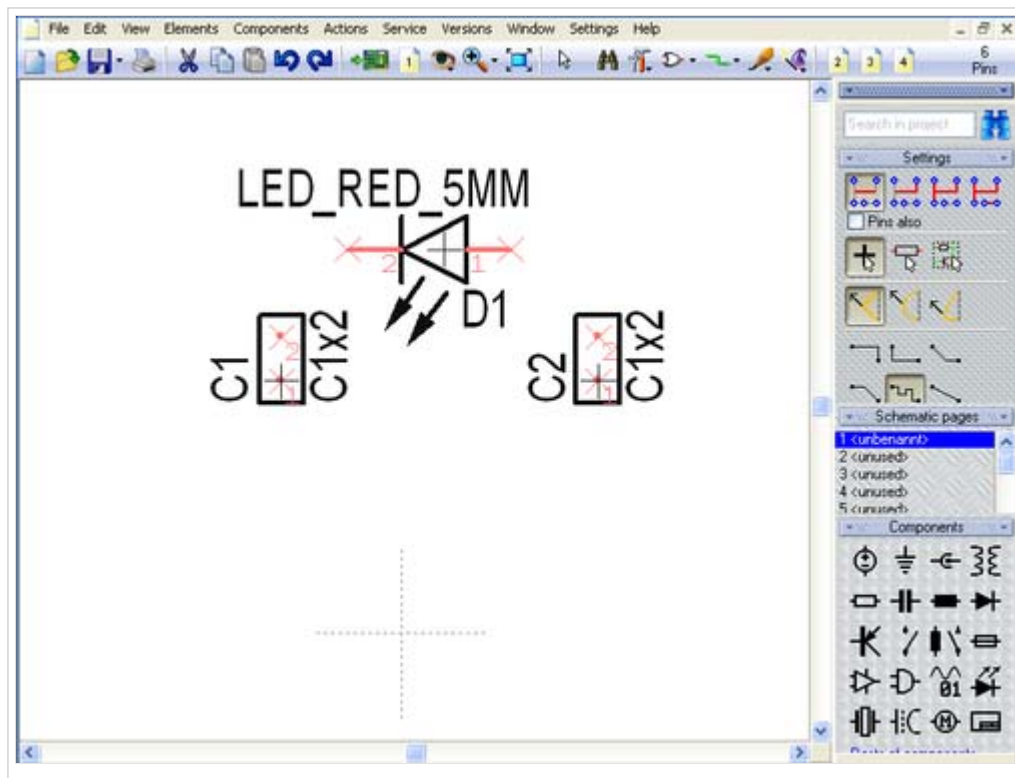
In case the component value appears too big, you might change it by M11 upon the handle cross. In the appearing dialog modify the entries. "LED_RED_5mm" stands for Component Value. "D1" stands for "Component Name".



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**.

Next step

One step back

Back to overview

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

Categories: Components/Libraries | Schematic

IBF-Intern:


- This page was last modified 13:49, 15 January 2009.
- This page has been accessed 6,775 times.

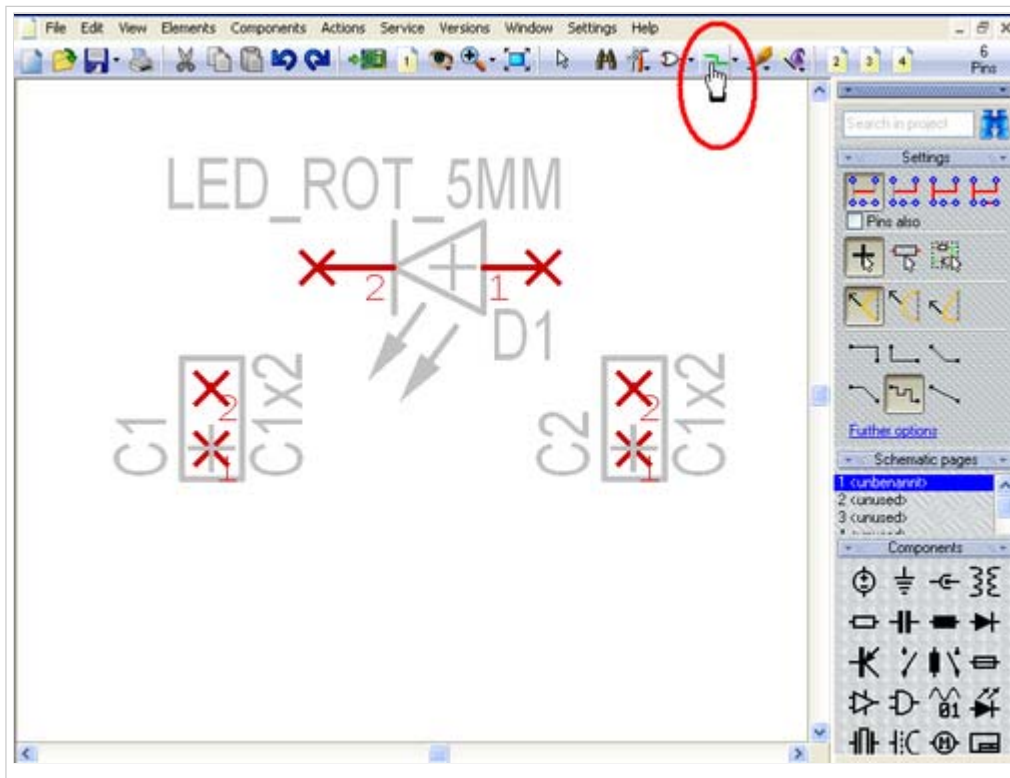
Connecting the pins

From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2

Download this article as PDF-File

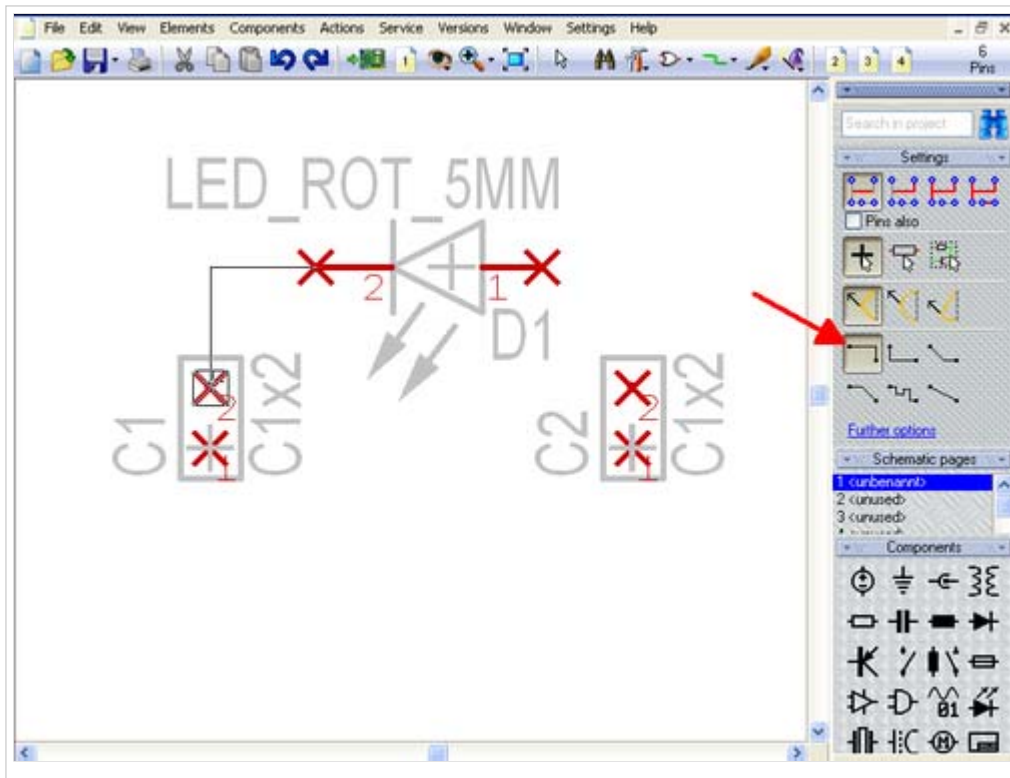
The pins of component symbols are connected by the function "Place wire" under the icon  You can either use key [2] from the keyboard to open this function.



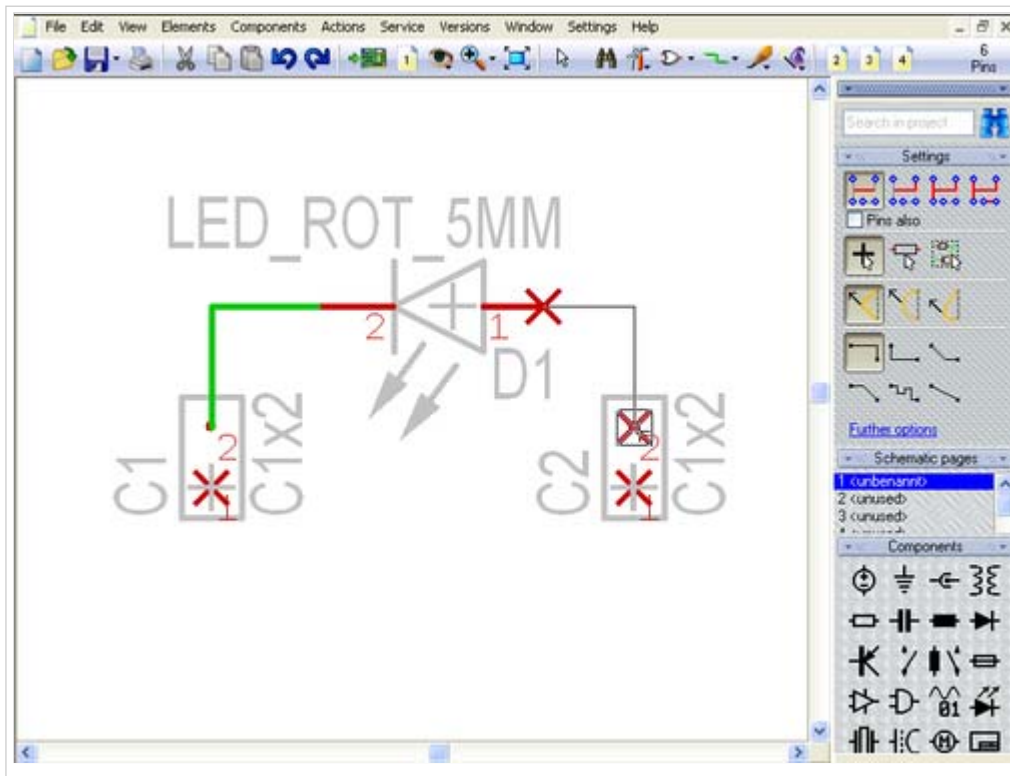
More details?

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 "space bar".



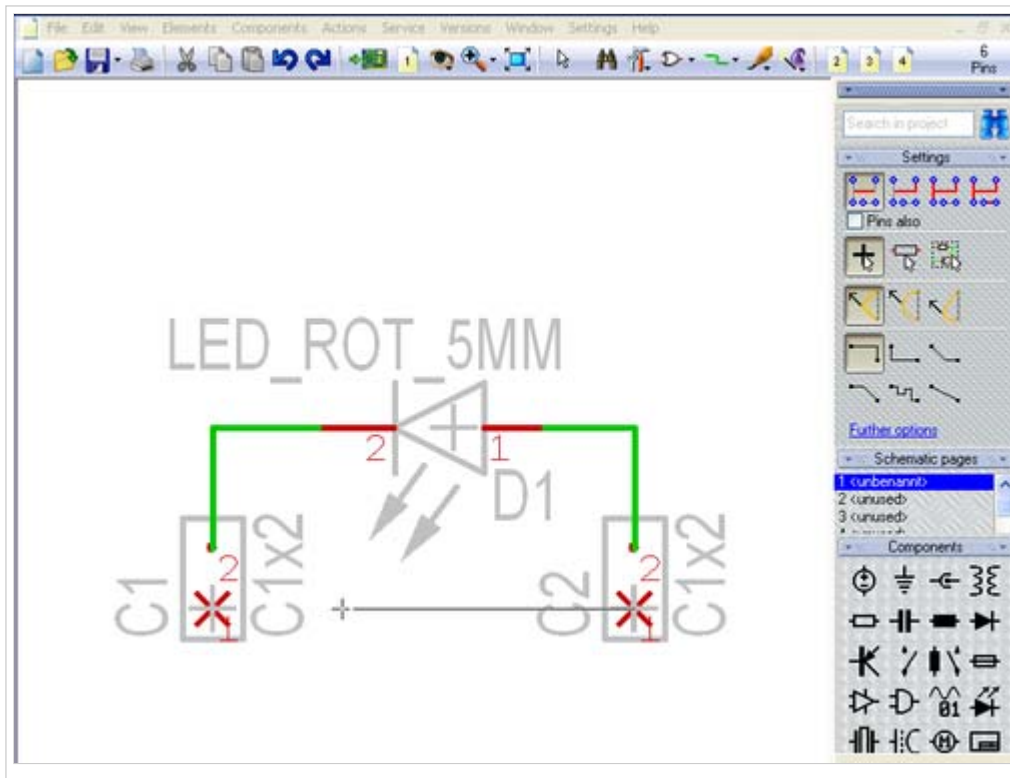
Now connect the anode of the LED...



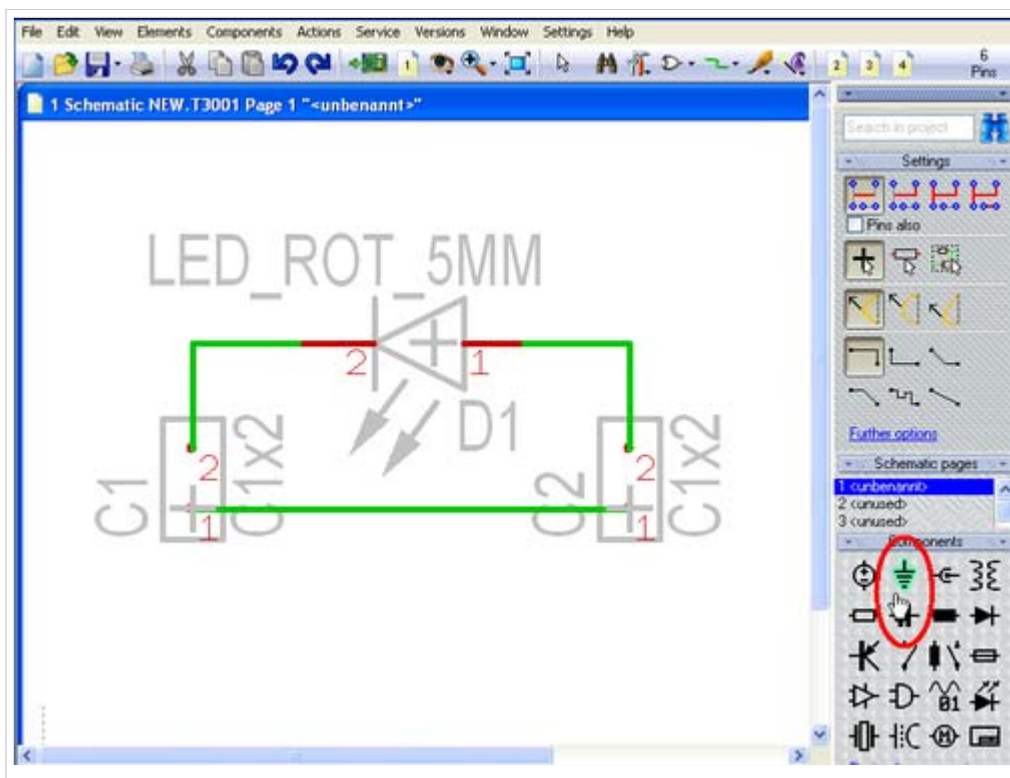
...and connect both remaining pins for ground connection.

More details?

Bending modes
Undo
Redo



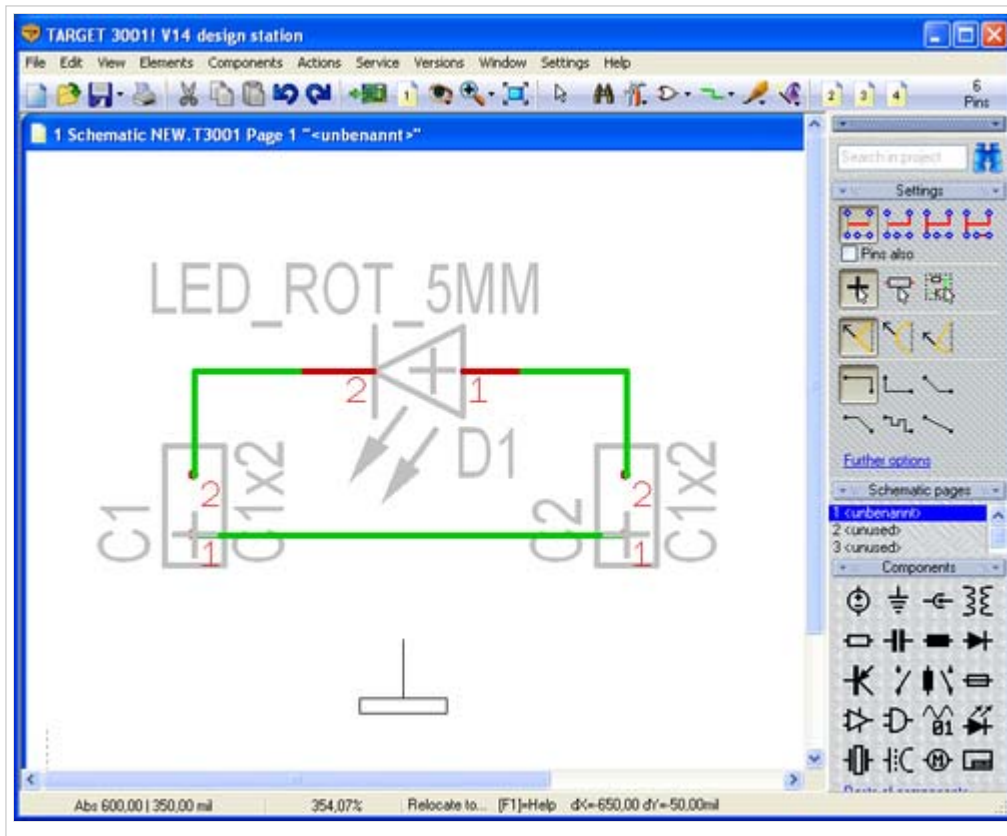
The ground signal is connected by a ground symbol. Please find it within the range of "Reference pins" in the sidebar:



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

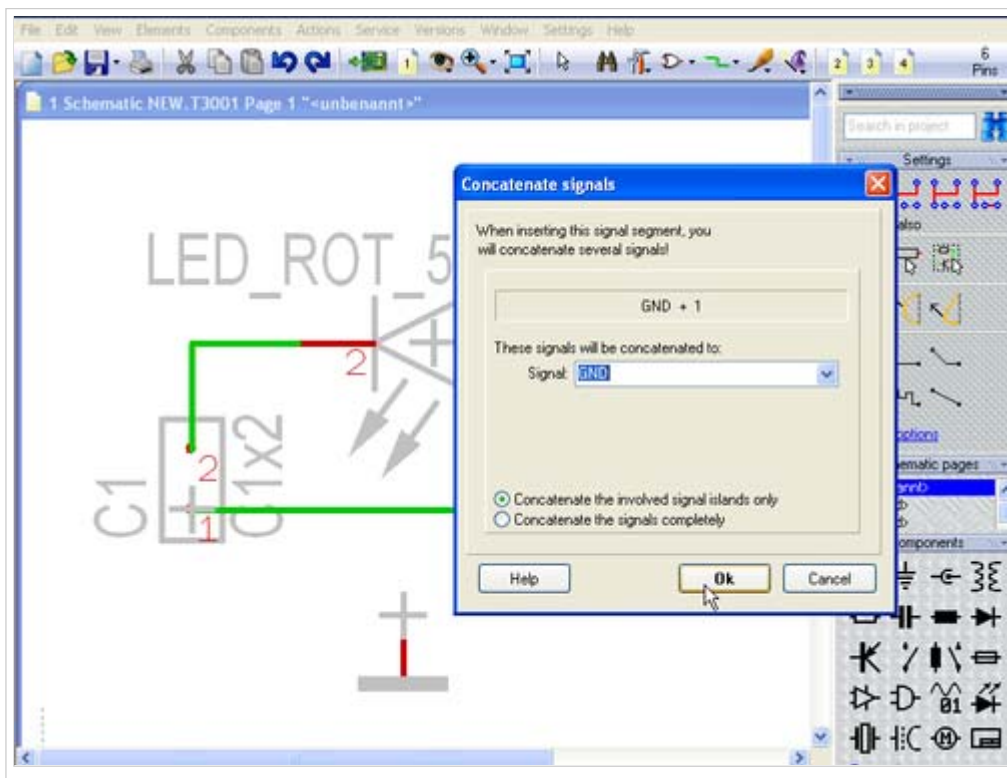
More details?

Reference
Symbol
Reorganisation

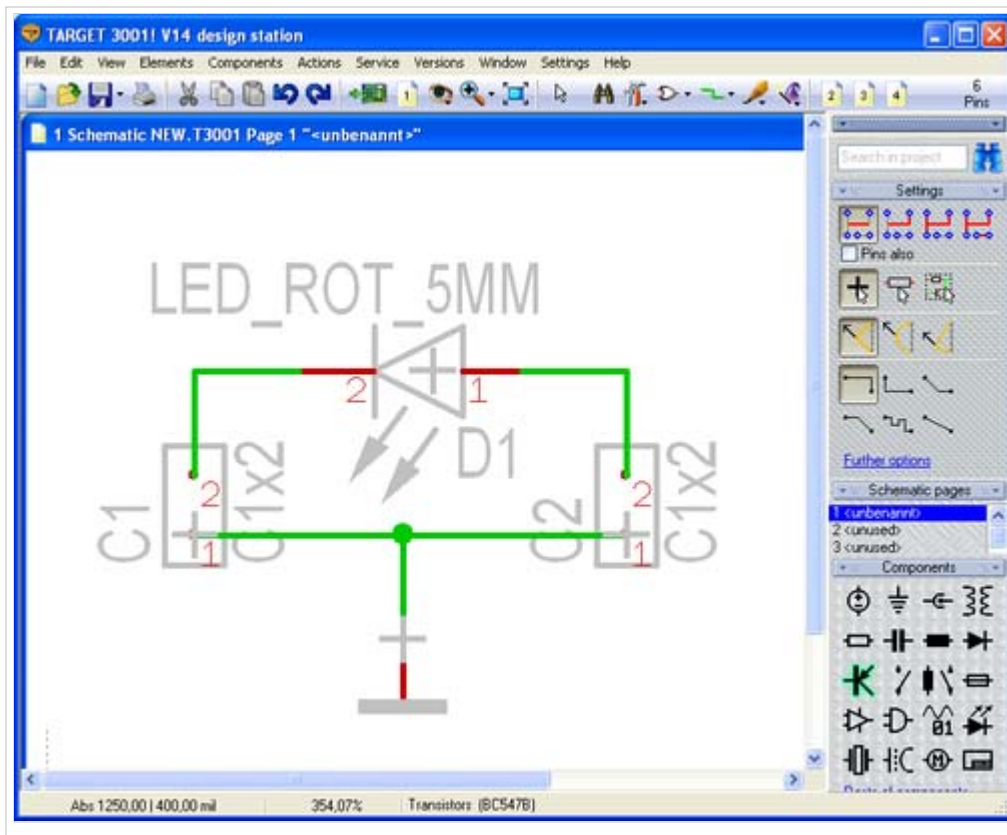


Settings /
Options

...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:



Next step

One step back

Back to overview

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

Categories: Actions | Schematic

IBF-Intern:


- This page was last modified 14:30, 25 November 2008.
- This page has been accessed 5,303 times.

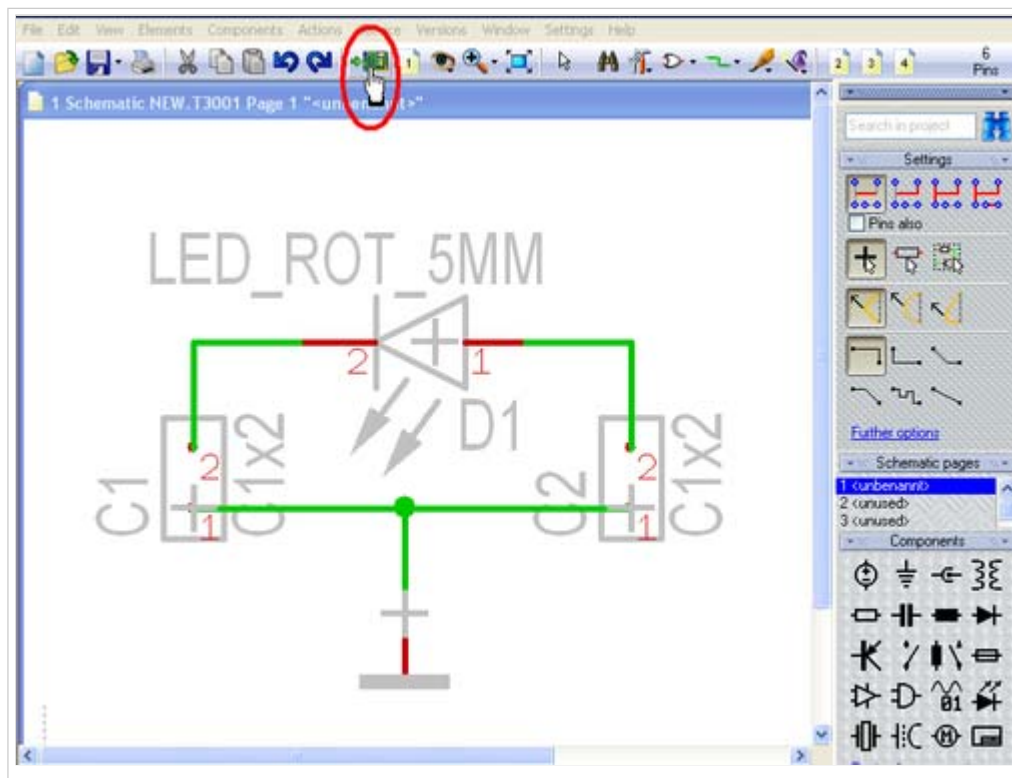
Define a PCB outline

From IBF-Wiki

This article is part of an introduction tutorial called CrashCourse2

Download this article as PDF-File

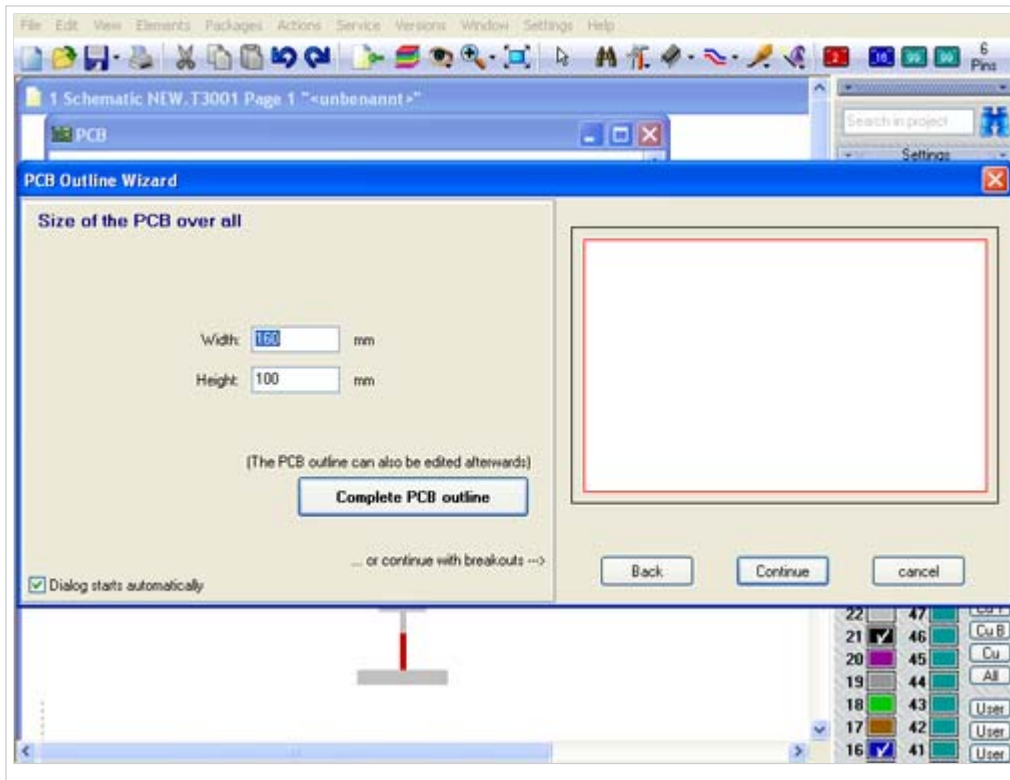
Please switch over to the PCB layout view by the icon "Go to PCB view"  (see cursor in the following image) or use the key **F3**.




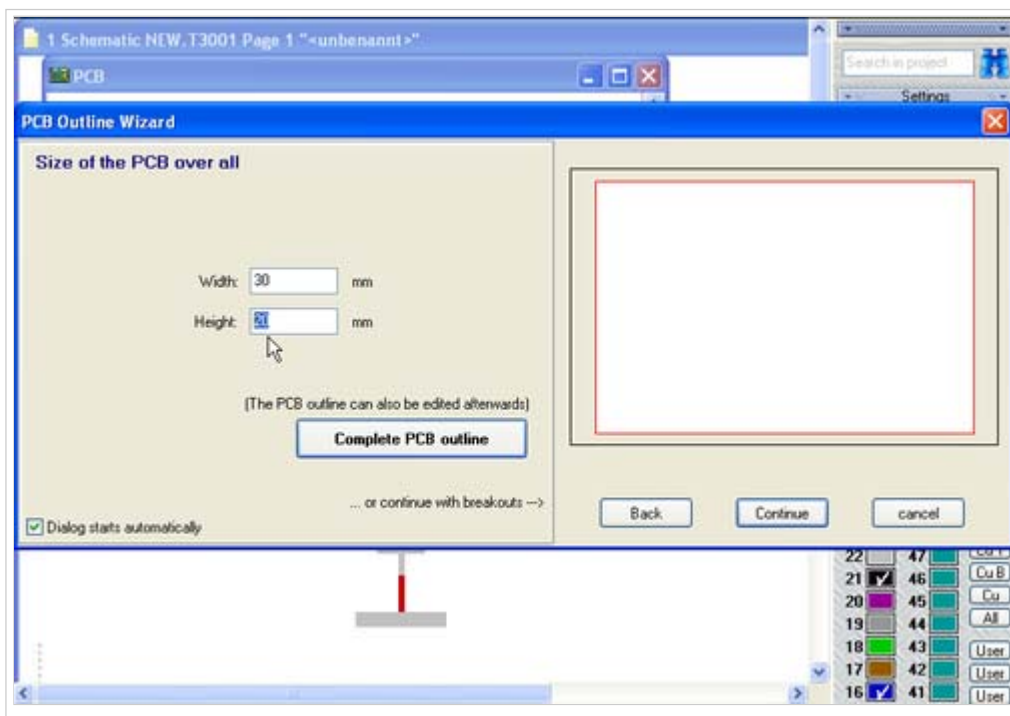
More
details?

Outline
Layer

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":

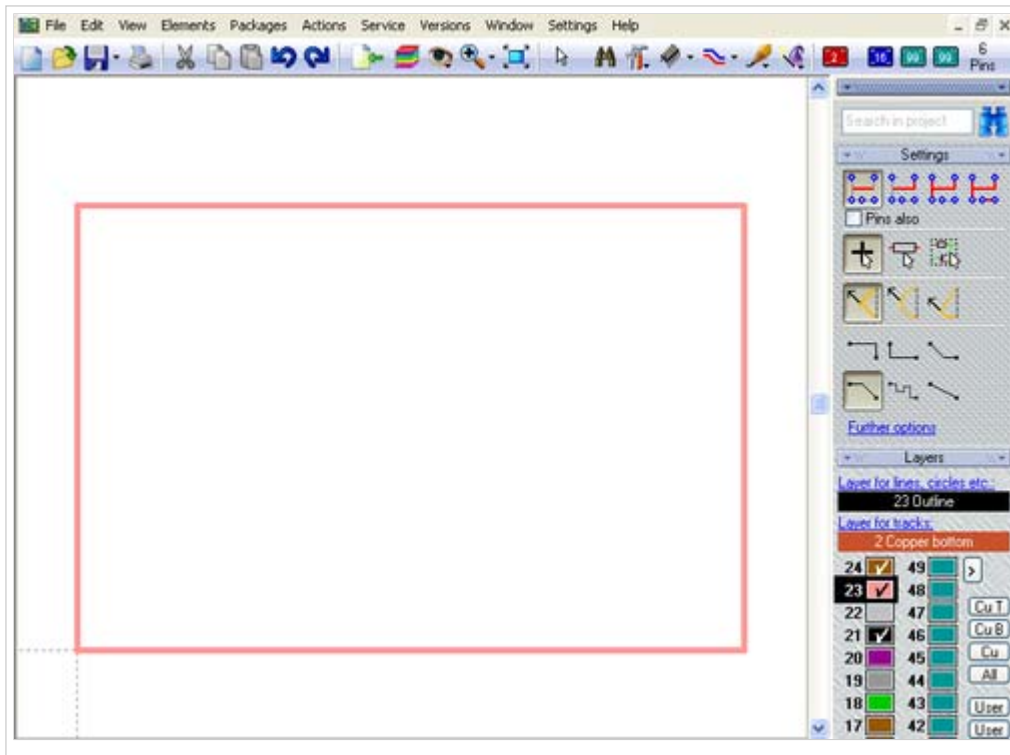


Immediately the assistant opens and we enter Width=30 and Height=20. The fact that we work in metric measurement is default. It can be adjusted to imperial measurement by the icon "Adjust View..." (=the eye).  The grid is set at the same location.



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.

More



details?

Drawing
Drag
Line
Text

Next step

One step back

Back to overview

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

Categories: Actions

IBF-Intern:

- This page was last modified 14:56, 25 November 2008.
- This page has been accessed 4,740 times.

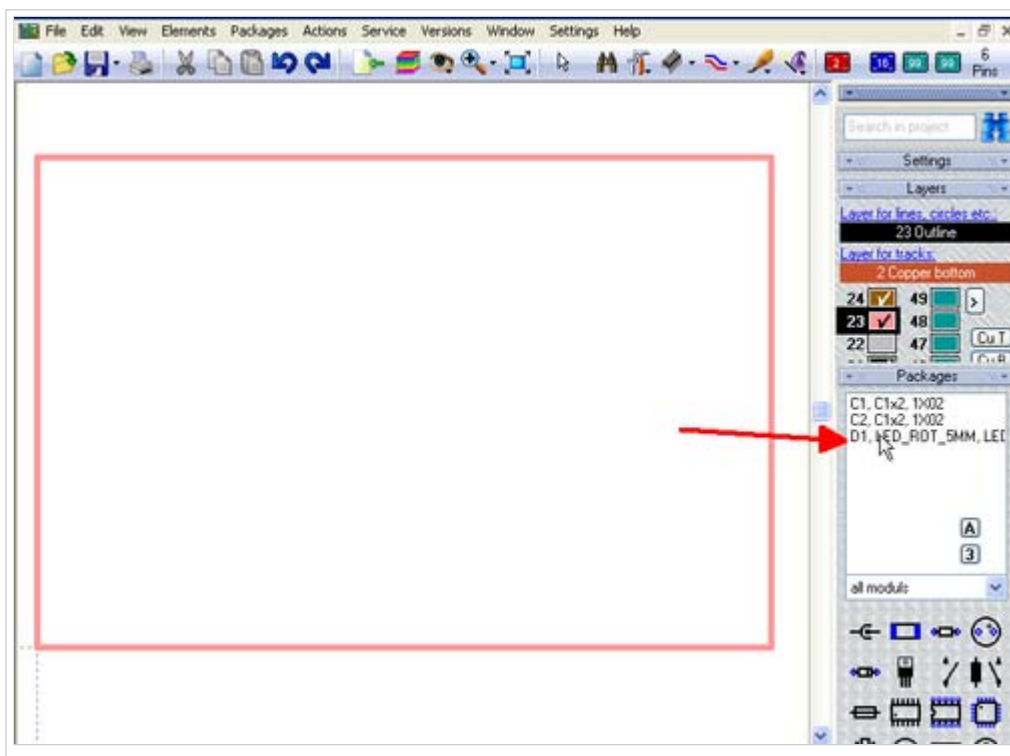
Import matching packages (footprints) to the layout

From IBF-Wiki

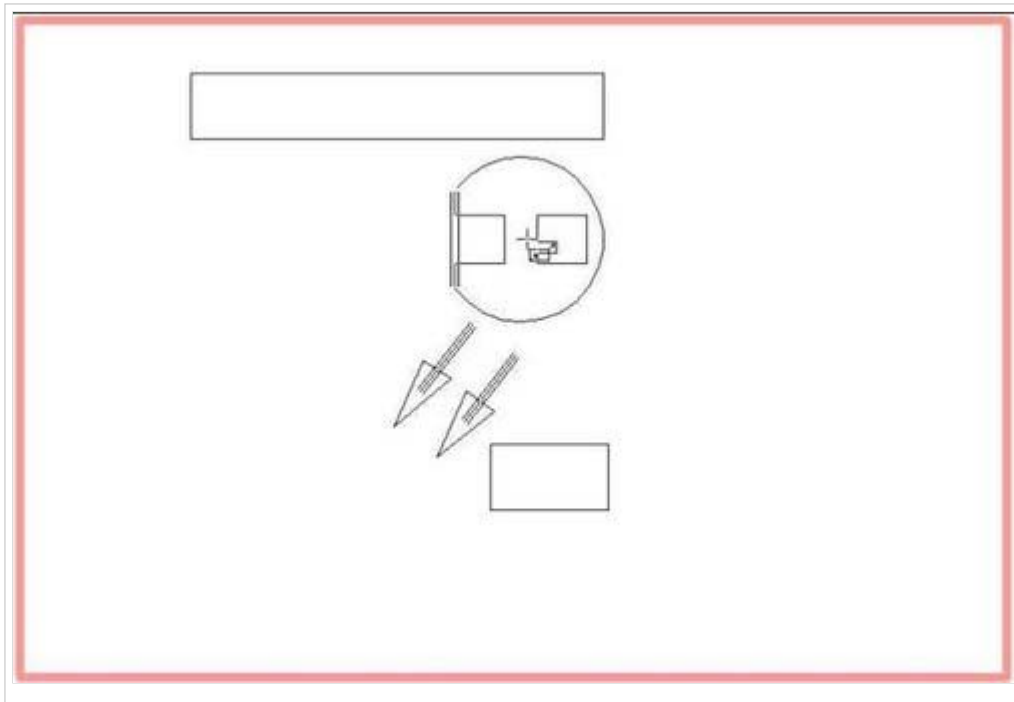
This article is part of an introduction tutorial called CrashCourse2

Download this article as PDF-File

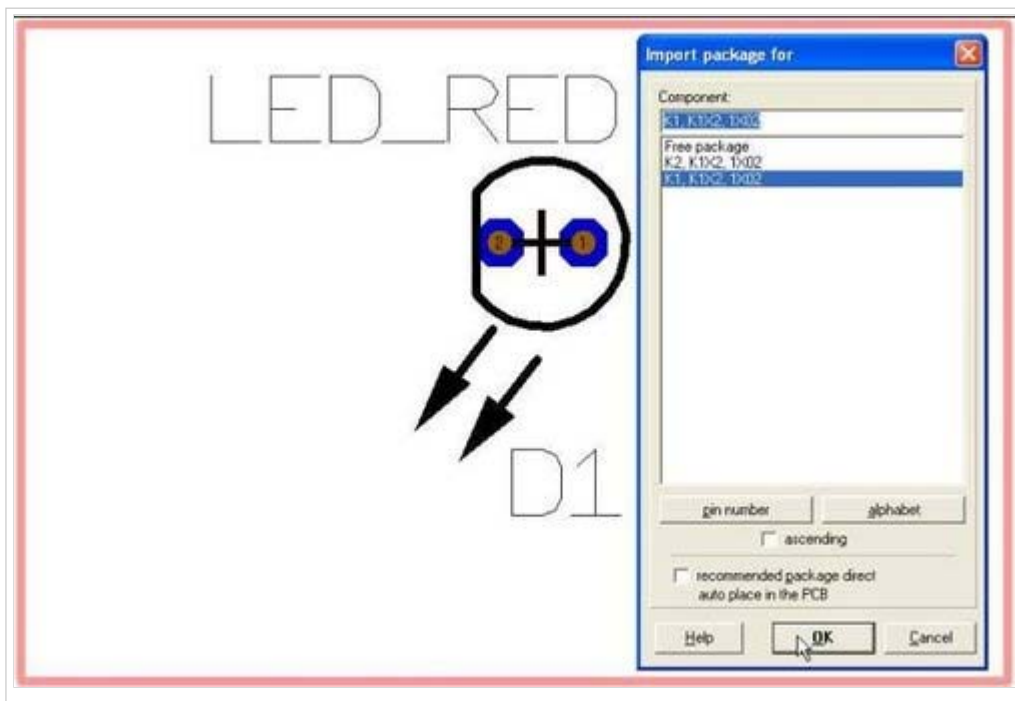
Every schematic symbol in TARGET 3001! has a package proposal. The word package in TARGET 3001! stands for "footprints" or more exact: solderable "landmarks". If you want to start with the package import, see cursor, a list of all package proposals according to the symbols used in the schematic appears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



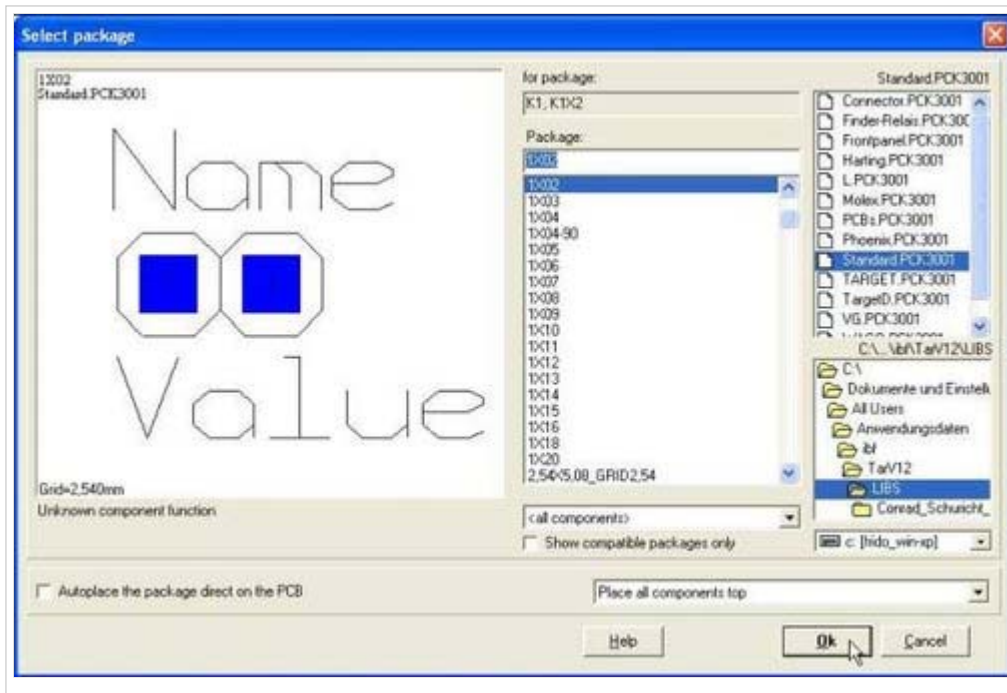
Now its shape is fixed to the cursor and allows rotation by **M2**,...



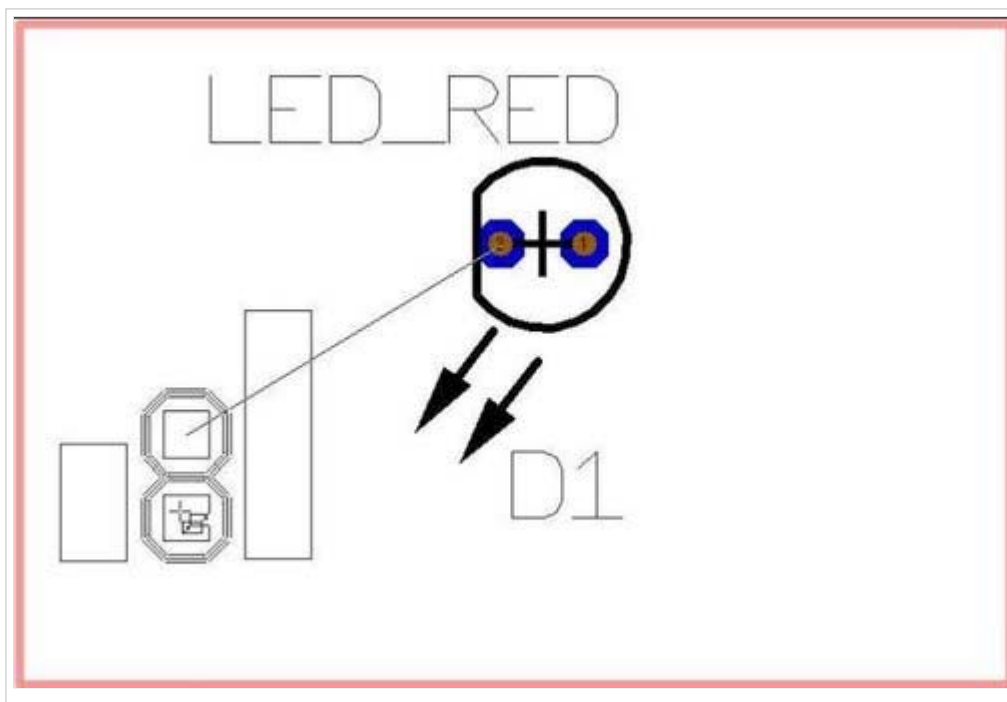
...before it is placed by releasing mousekey **M1**. Now we touch connector K1 from the sidebar list



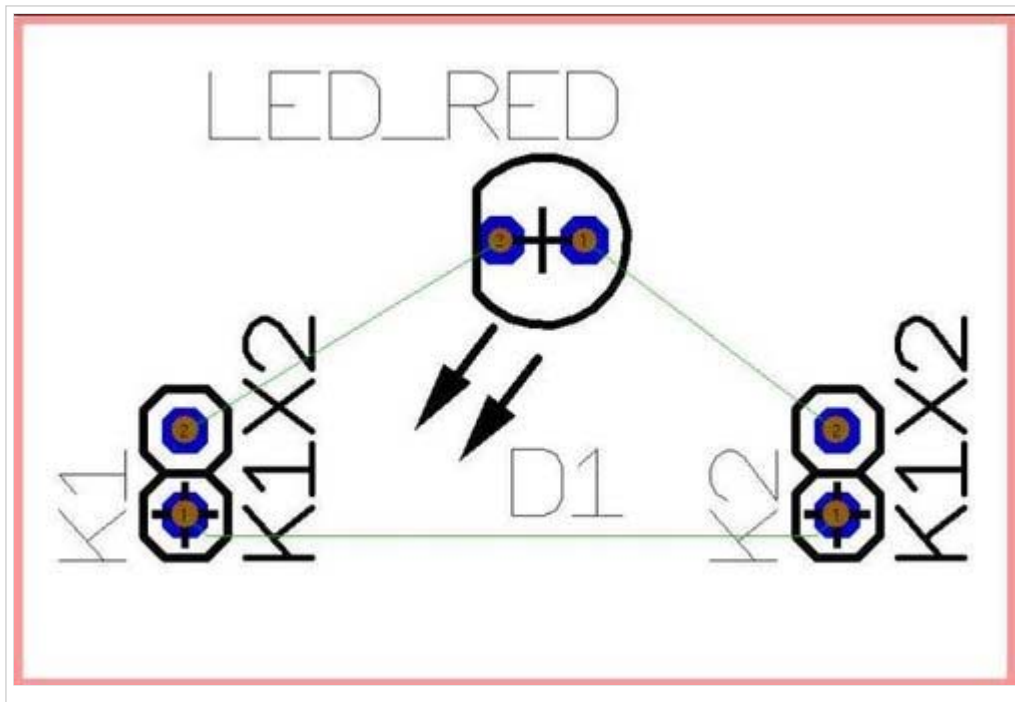
...confirm it by **[[OK]]**...



...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](#)

Retrieved from "http://server.ibfriedrich.com/wiki/ibfwikien/index.php?title=Import_matching_packages_%28footprints%29_to_the_layout"

Categories: [Layout](#)

IBF-Intern:

- This page was last modified 11:48, 26 November 2008.
- This page has been accessed 4,250 times.

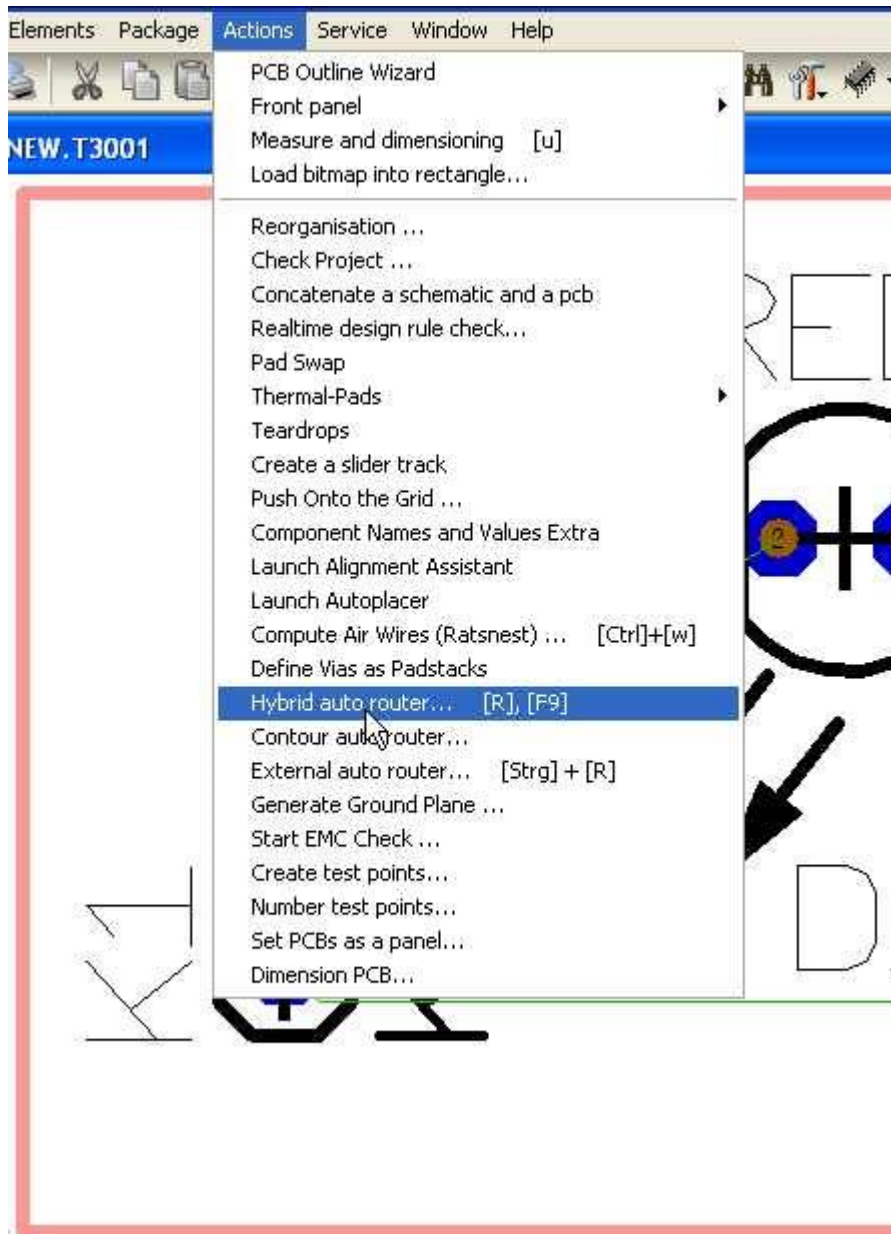
Place tracks

From IBF-Wiki

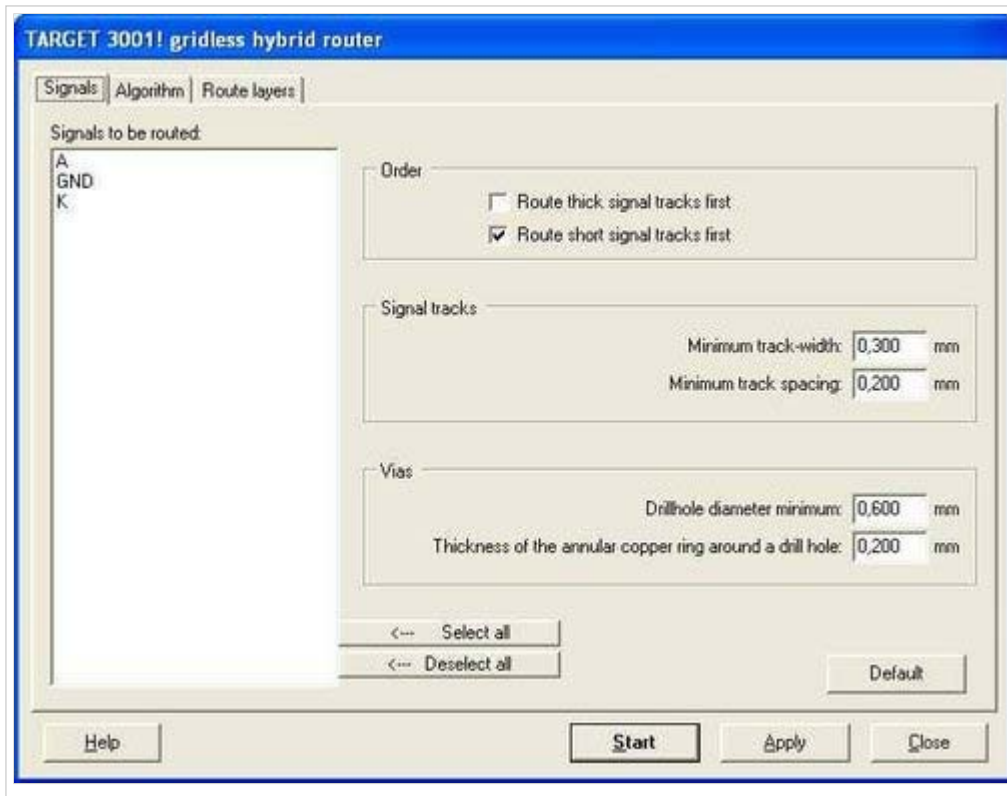
This article is part of an introduction tutorial called CrashCourse2

Download this article as PDF-File

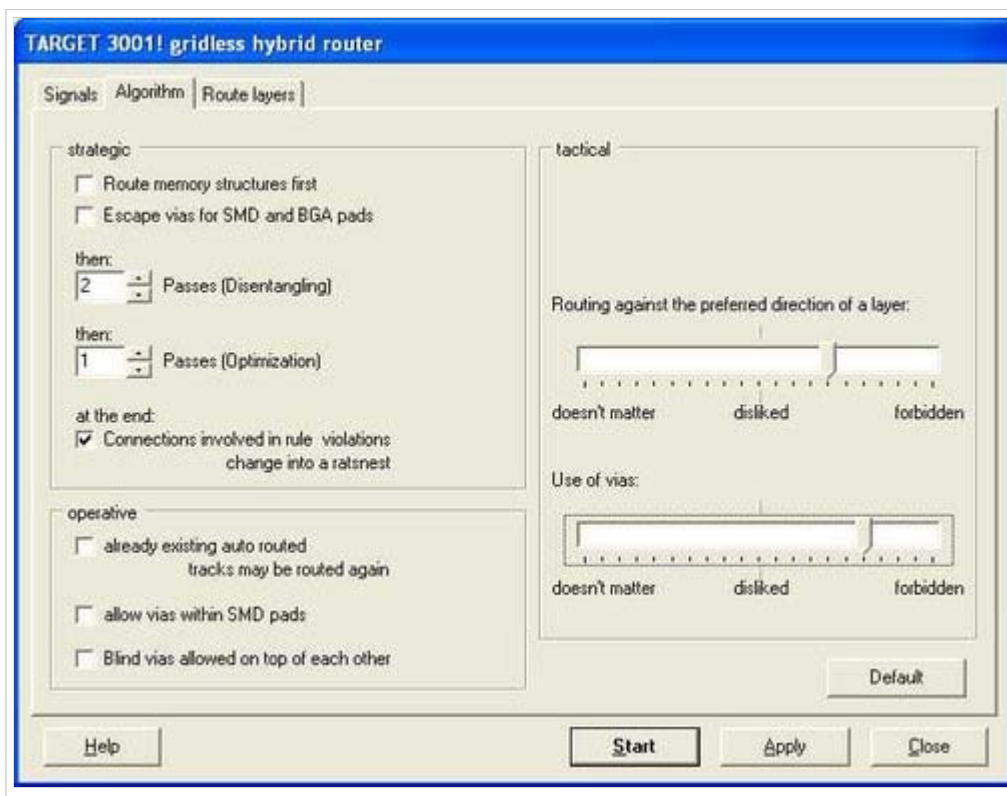
The placement of tracks can be done manually or by the use of one of the TARGET autorouters, in this case we use the "Hybrid autorouter" in layoutmenu "Actions". Also the keyboard key [r] or the function key [F9] will launch the Hybrid Autorouter.



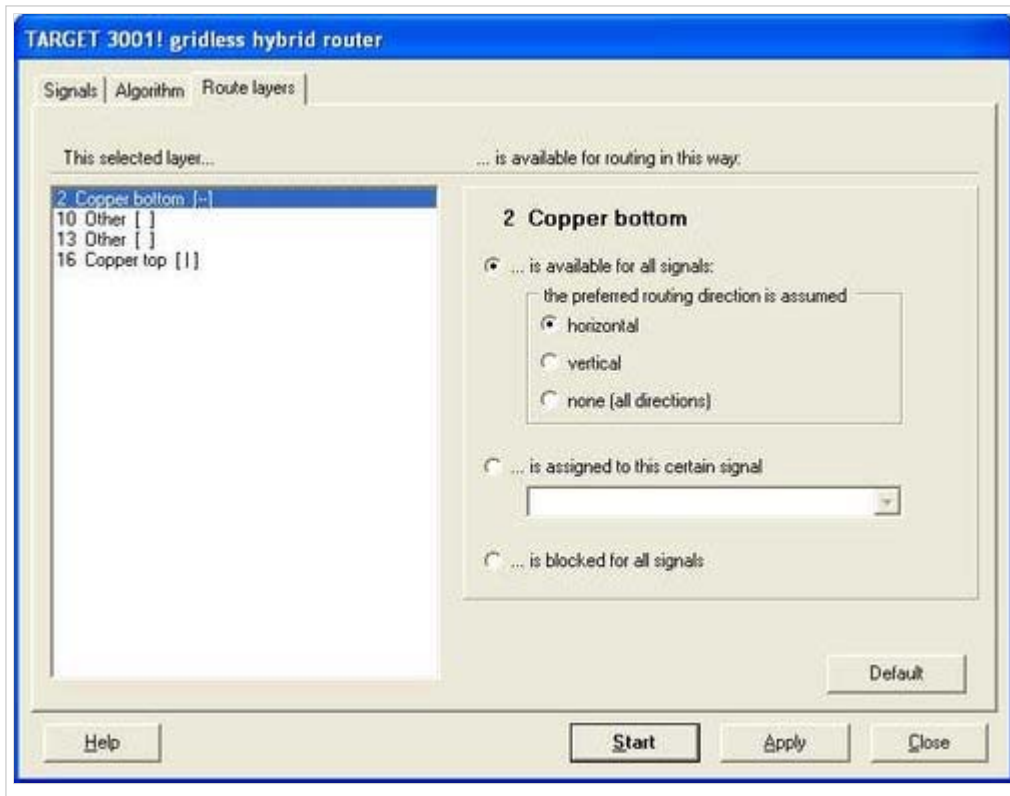
The opening dialog of the hybrid router has three sections divided by tabs. First the section "Signals".



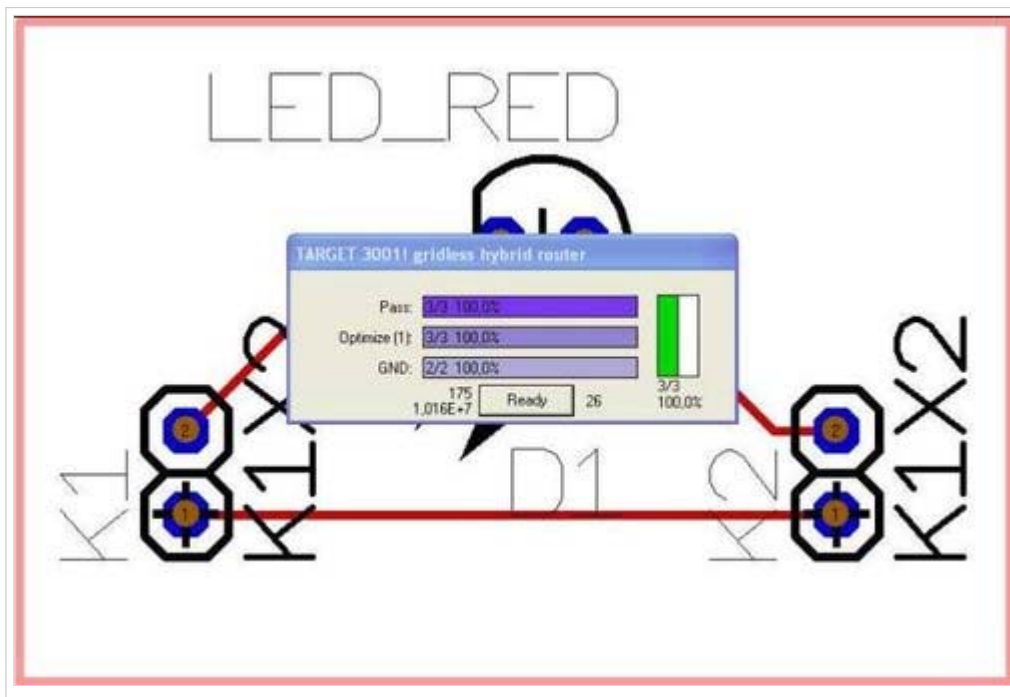
Now choose from the list one or more or all signals. If you select none, all are routed.



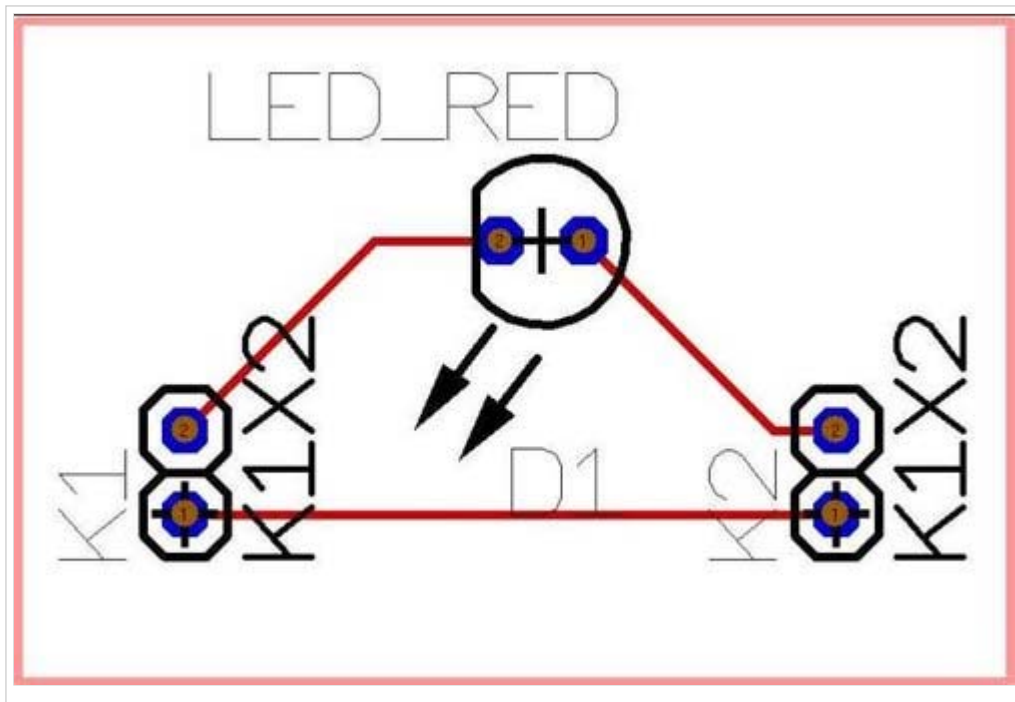
The second tab allows "strategic", "operative" and "tactical" settings. In the strategic area we leave the default settings with "two passes of disentangling" and "one pass of optimization". In the tactical area we place the slider between "disliked" and "forbidden". The third tab allows the assignment 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://server.ibfriedrich.com/wiki/ibfwikien/index.php?title=Place_tracks"

Categories: [Layout](#)

IBF-Intern:

- This page was last modified 13:53, 15 January 2009.
- This page has been accessed 4,686 times.

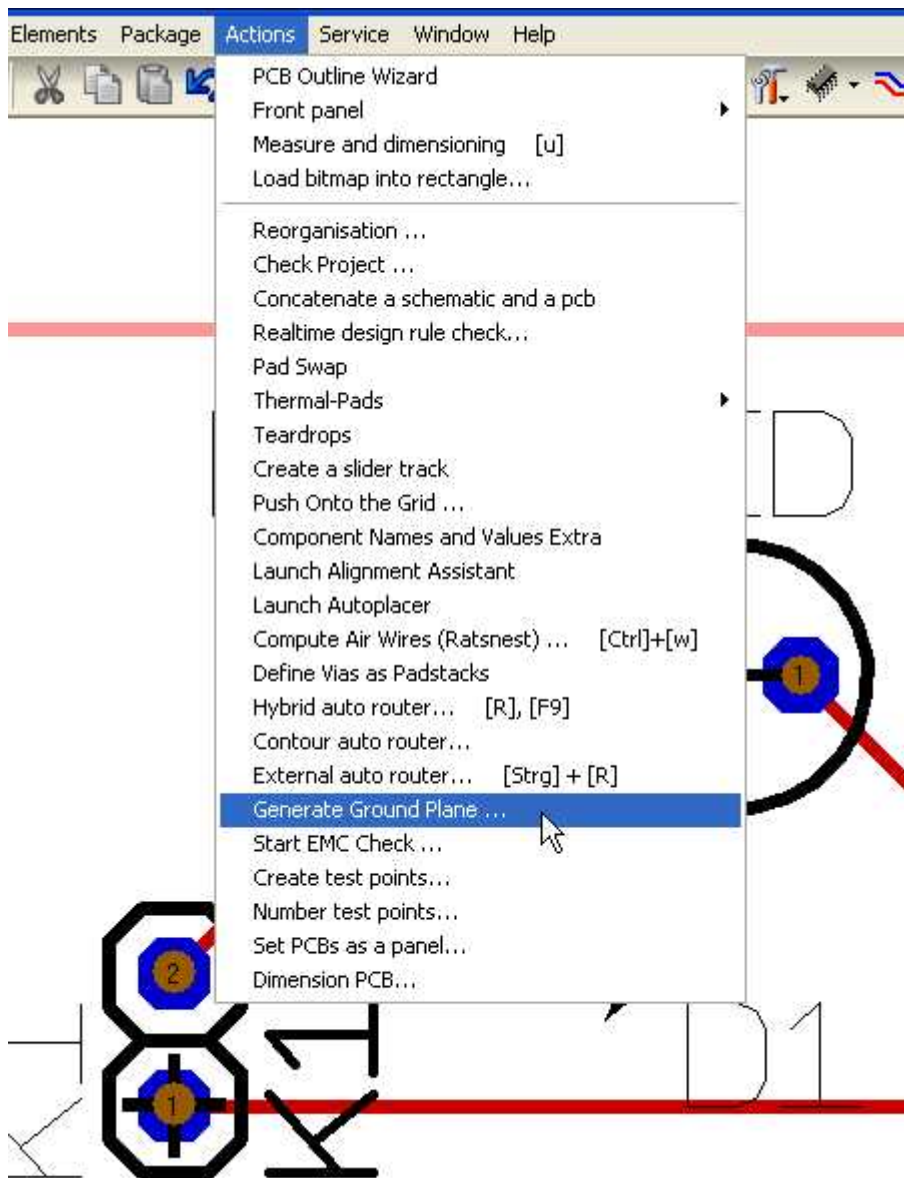
Generate a groundplane

From IBF-Wiki

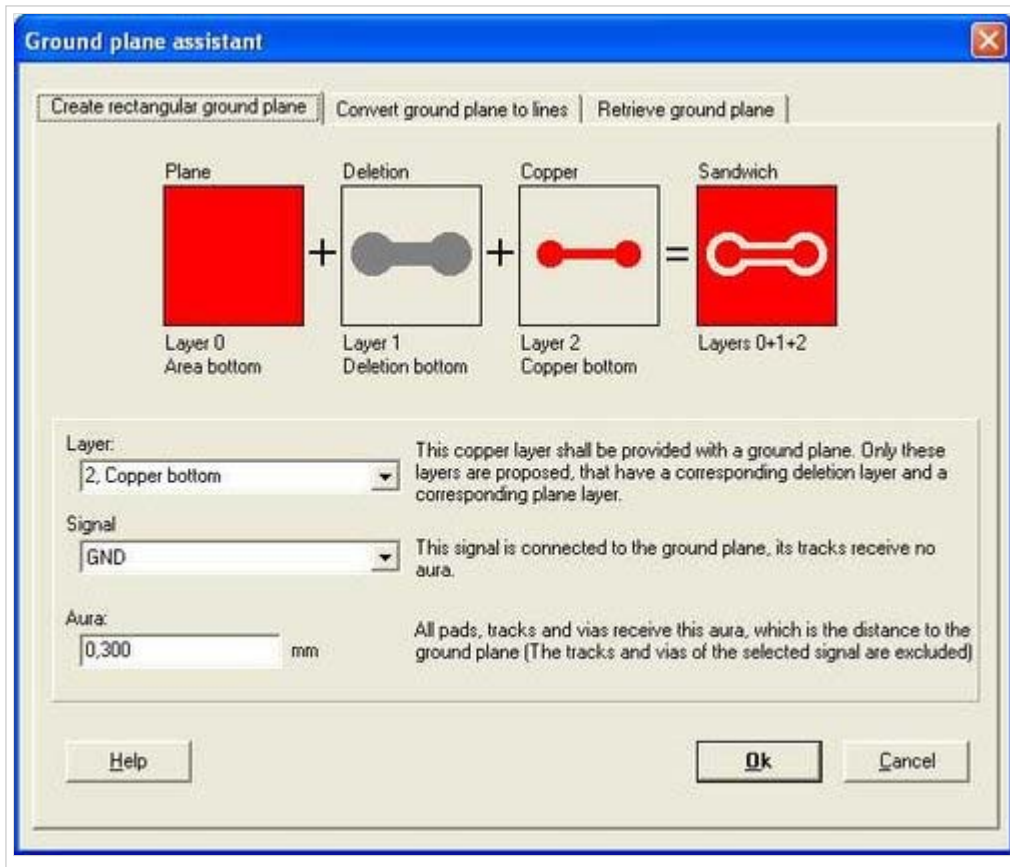
This article is part of an introduction tutorial called CrashCourse2

Download this article as PDF-File

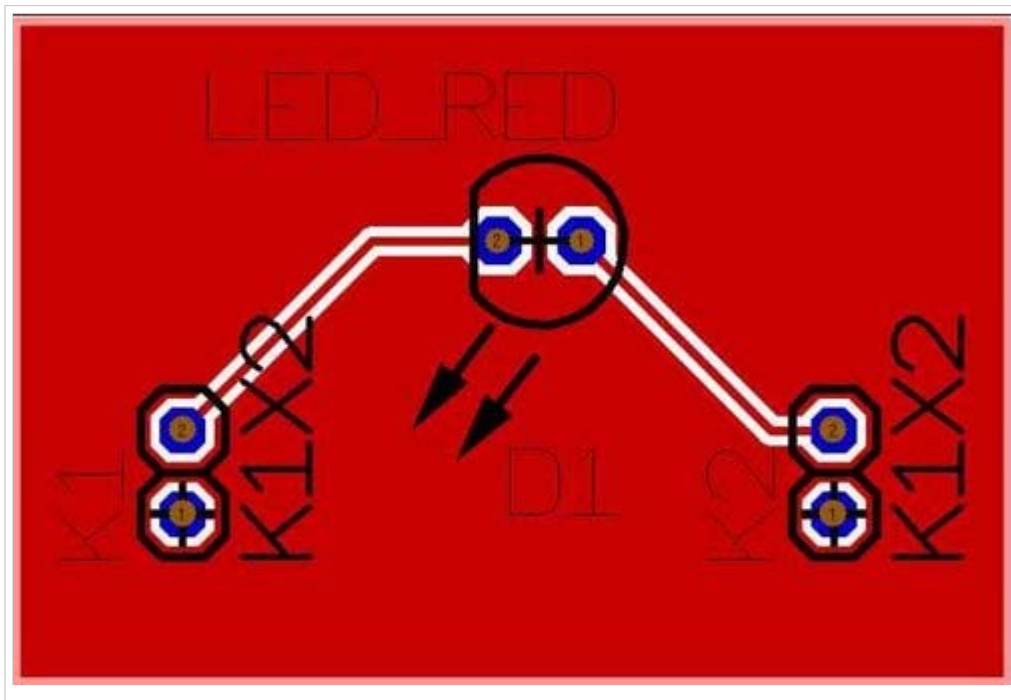
A groundplane having the same dimensions like the layout can easily be created by the use of the groundplane assistant, please see layoutmenu "**Actions**":



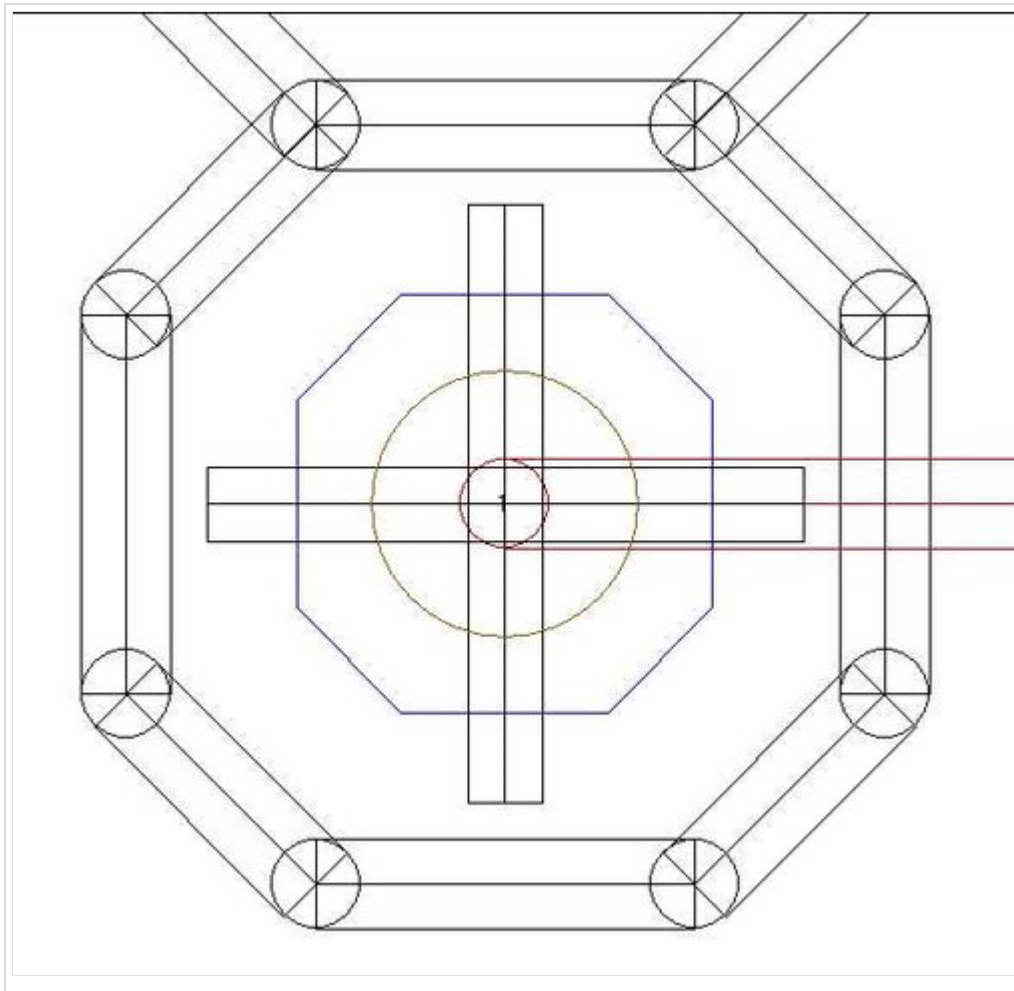
The opening dialog explains that a groundplane in TARGET 3001! is made out of a set of three layers: "copper", "area" and "deletion". The shape of the groundplane is defined on the "area" layer. The "deletion" layer defines the spacings between the groundplane and the non GND-Leading tracks. The layer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

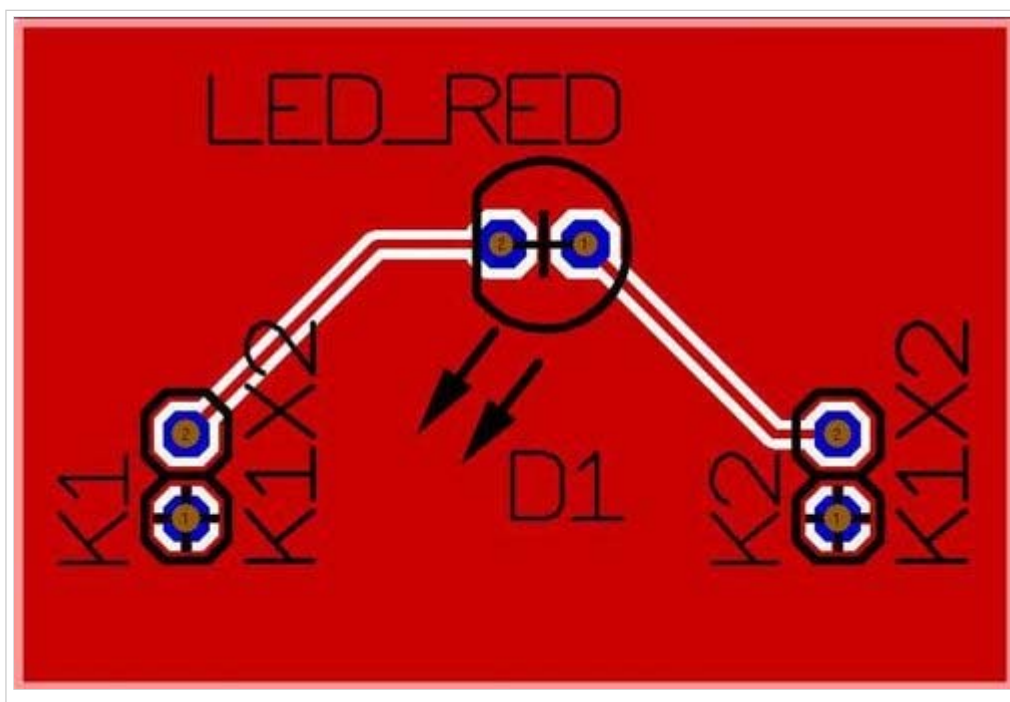


The GND leading track in the south of the layout is embedded completely into the groundplane. Use the hash key (#) for having an x-ray view of the layout. So you can see...

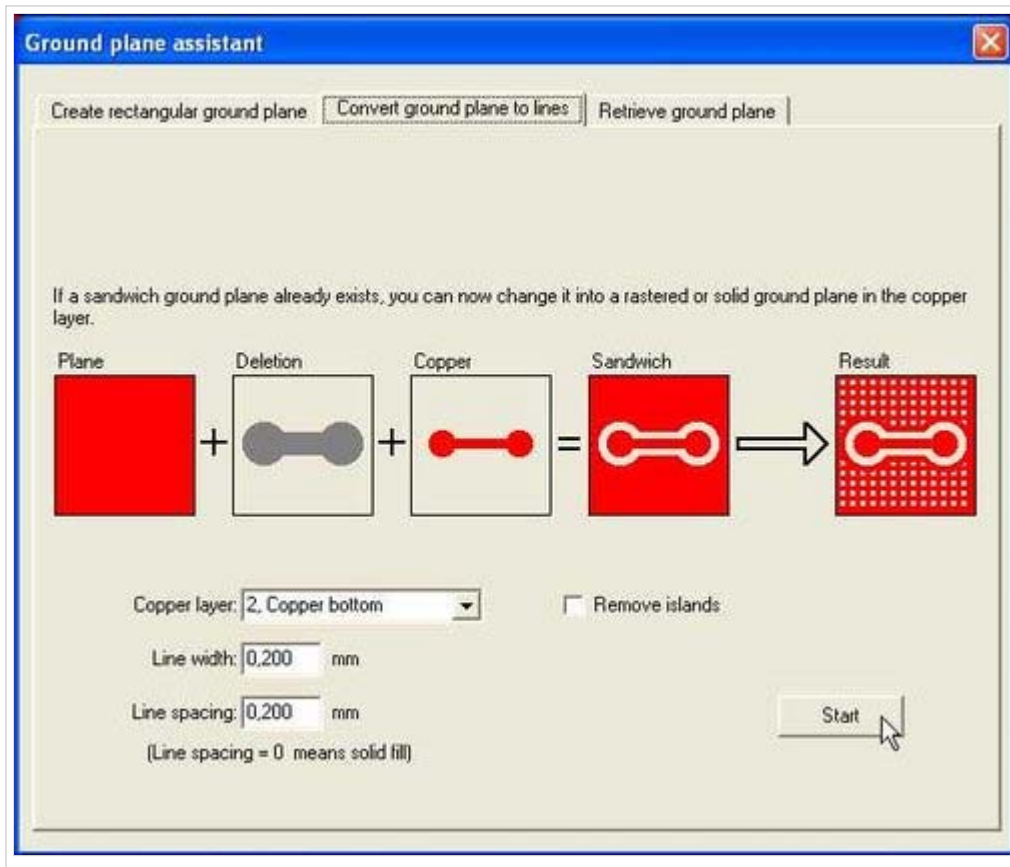


...that the pads are connected correctly.

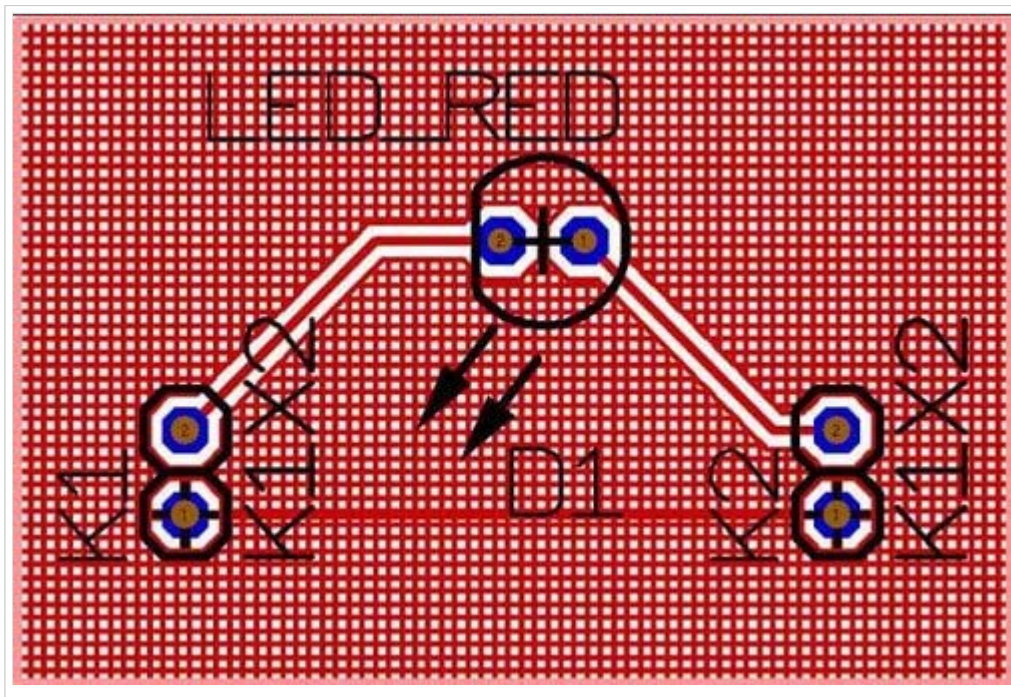
The generation of thermal pads is managed by highlighting the complete layout and choose menu "Actions/Thermal-Pads". Pads whose signal tracks are embedded to the groundplane, receive three more ligaments. Result:



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://server.ibfriedrich.com/wiki/ibfwikien/index.php?title=Generate_a_groundplane"

Categories: [Actions](#) | [Automatic functions](#)

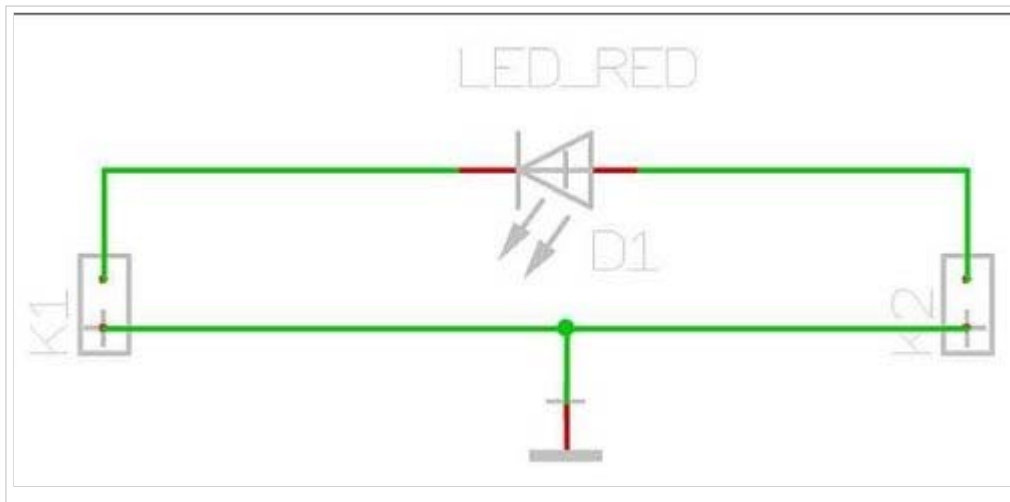
Simulate the function Part 1

From IBF-Wiki

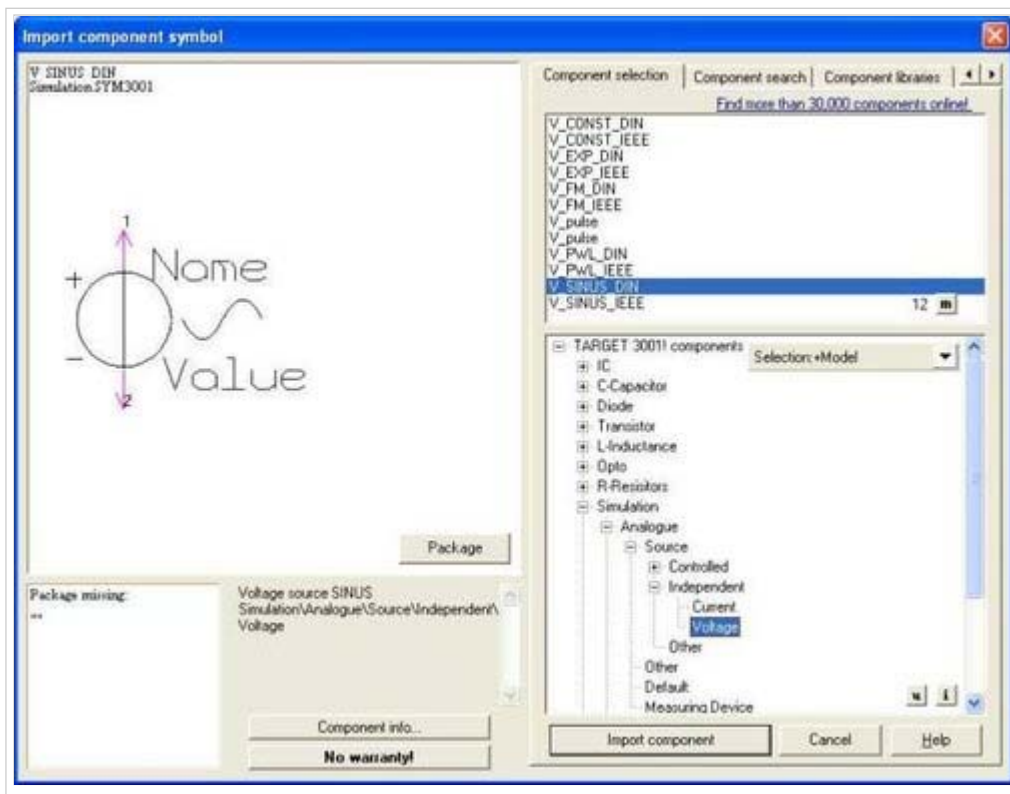
This article is part of an introduction tutorial called CrashCourse2

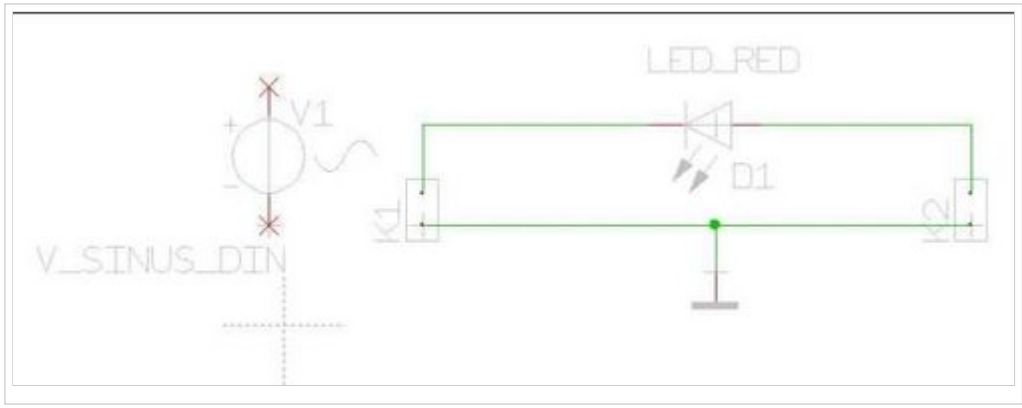
Download this article as PDF-File

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

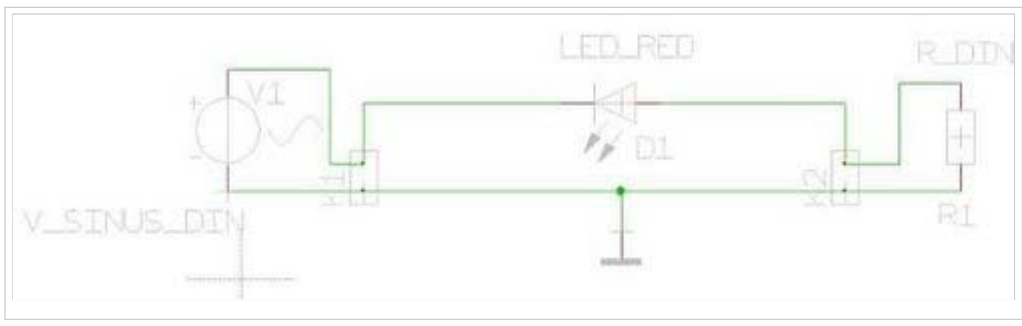
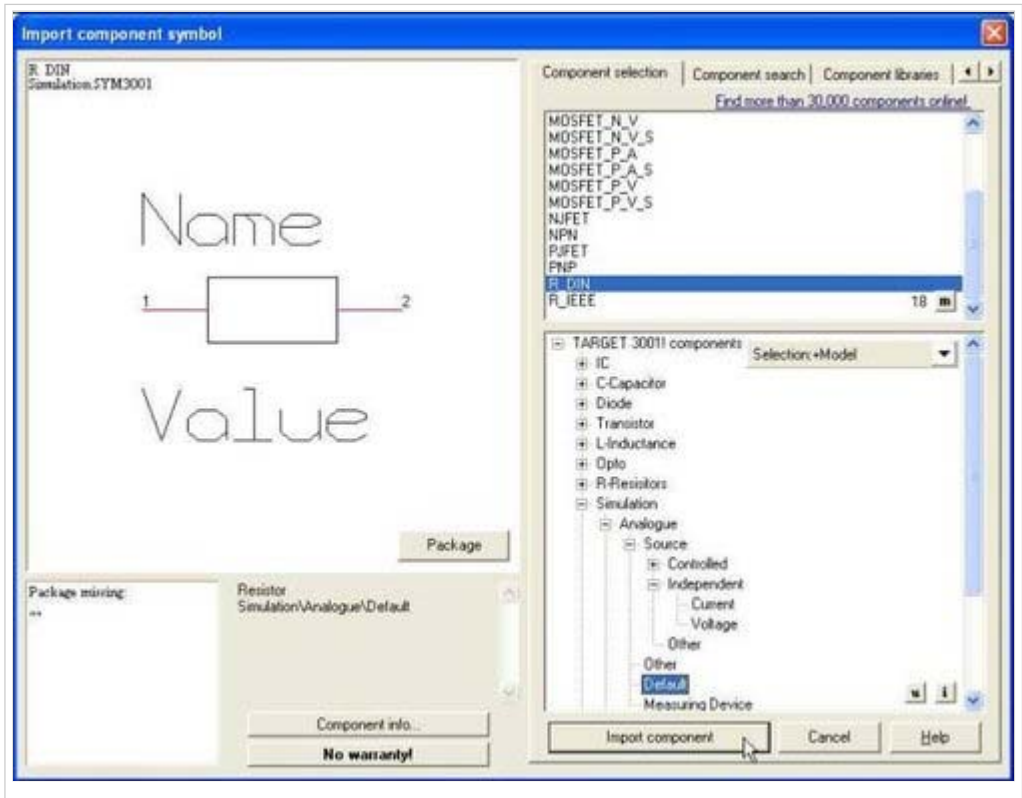


For the simulation of the LED's function we need a voltage source, and a load resistance. Those components will not be part of the layout thus don't have a package. Find such components in the tree view in the branch **Simulation**. First we import a sinus - source, then a resistor.

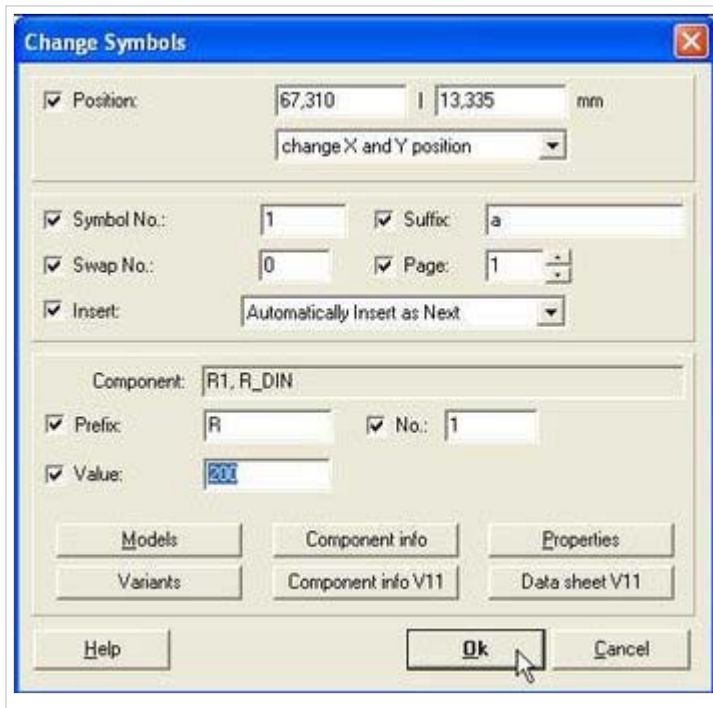




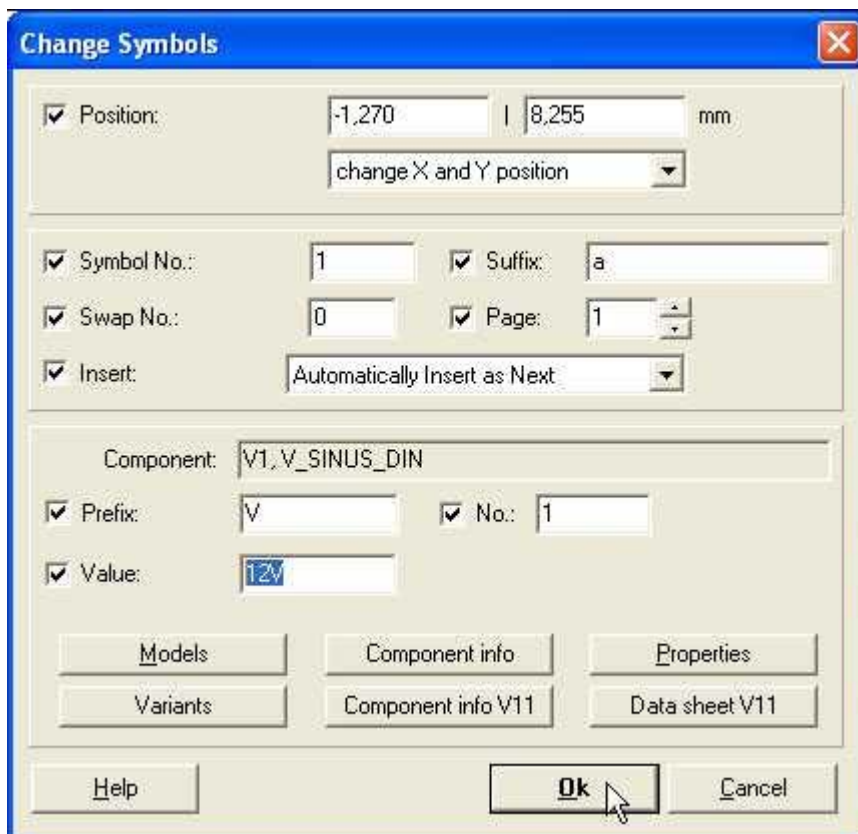
The resistor we pick from the branch Simulation/Analogue/Source/Default and connect it accordingly:



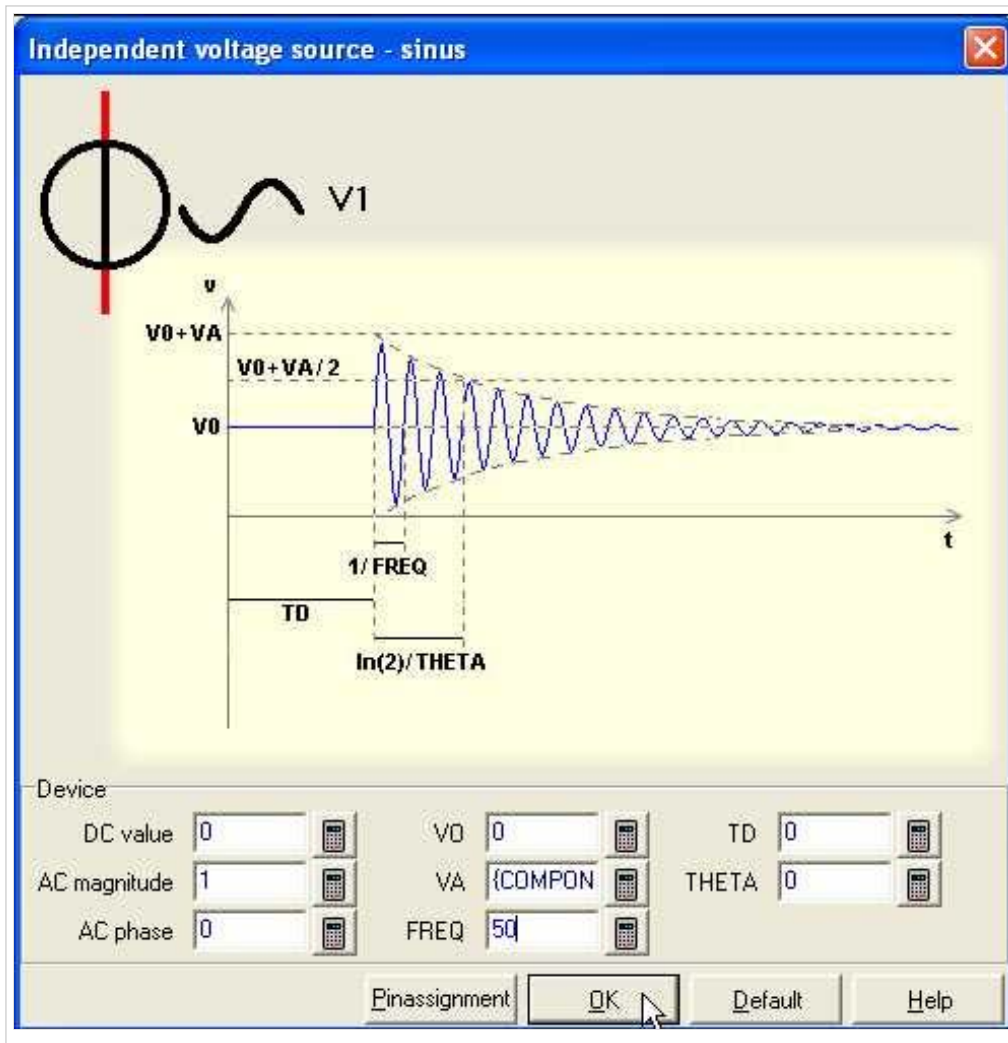
Now we set the values for source and load, first **M11** on the handle cross of the resistor:



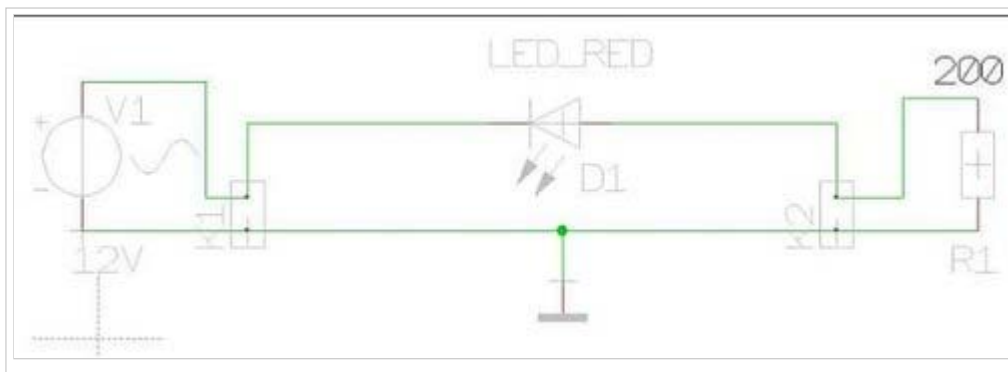
Now we enter component value "200", this stands for 200 Ohm. Press OK. Now press **M11** upon the handle cross of the sinus source:



We define its 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://server.ibfriedrich.com/wiki/ibfwikien/index.php?title=Simulate_the_function_Part_1"

Categories: Simulation

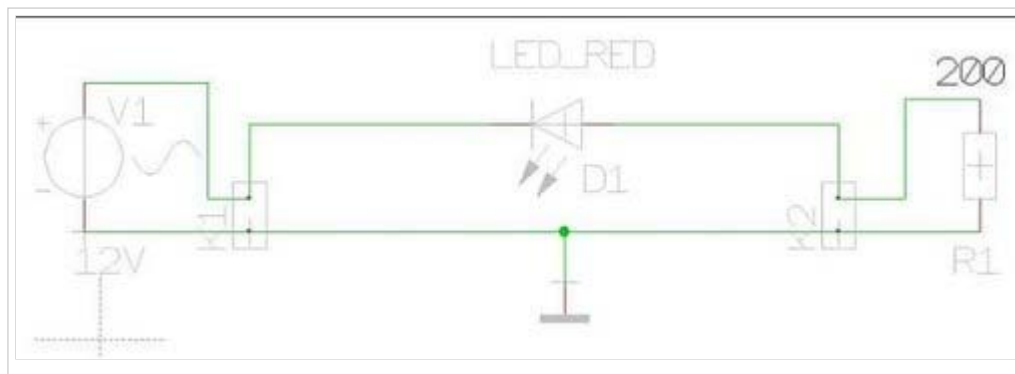
Simulate the function Part 2

From IBF-Wiki

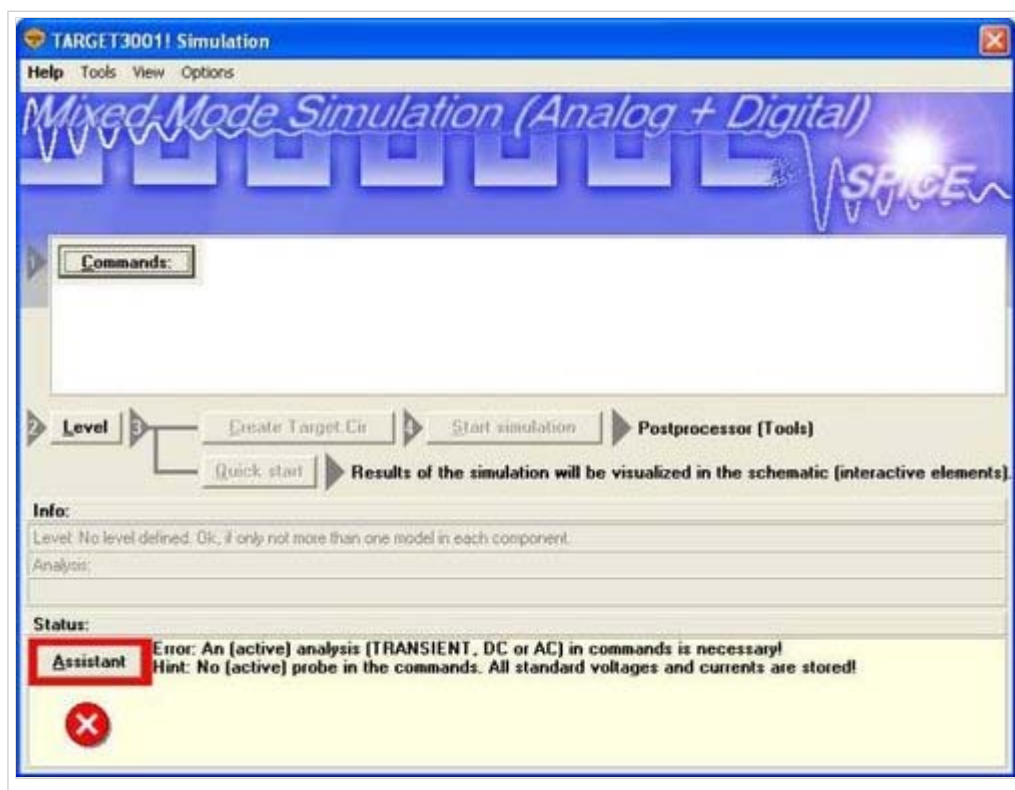
This article is part of an introduction tutorial called CrashCourse2

Download this article as PDF-File

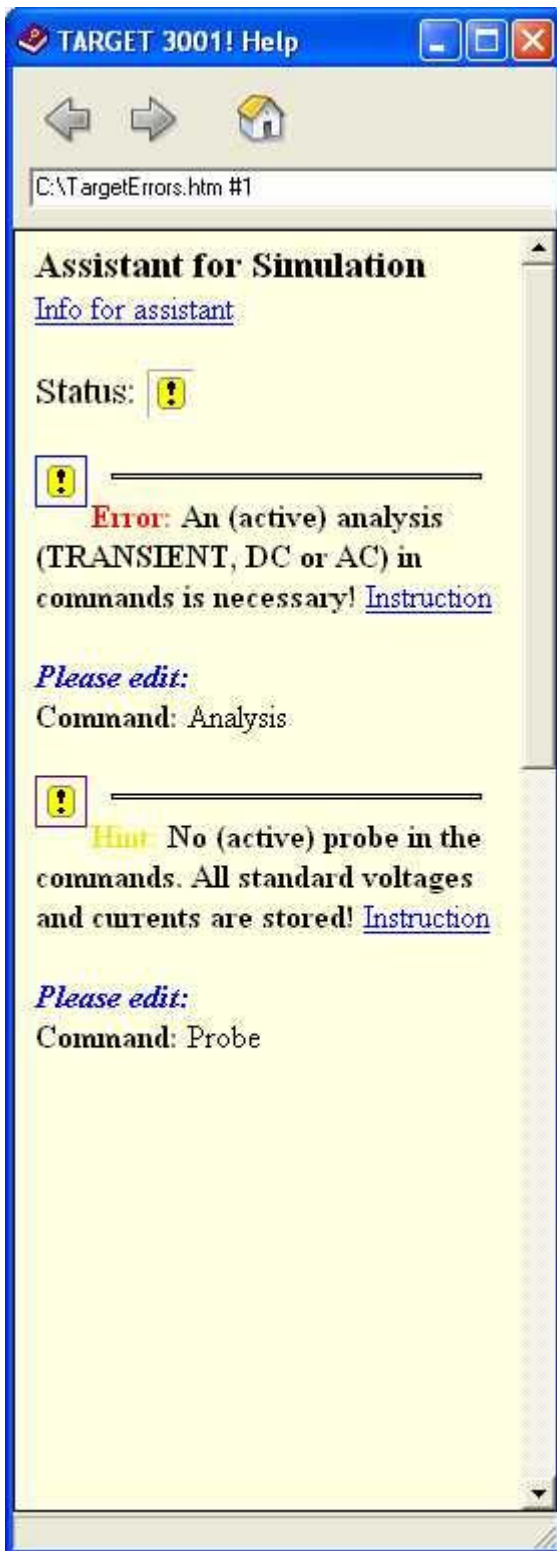
Based on the following schematic...



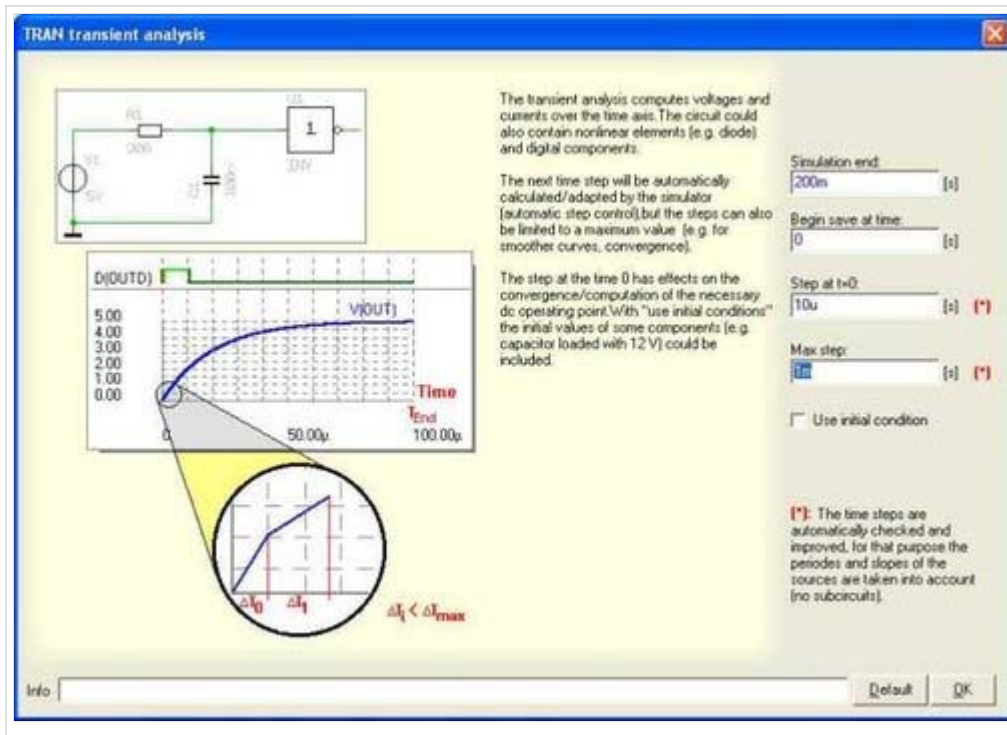
...we start the real simulation. We want to know which current flows in the LED at which tensions. First we start the simulation tool by the use of the function key [F9]. The following dialog appears:



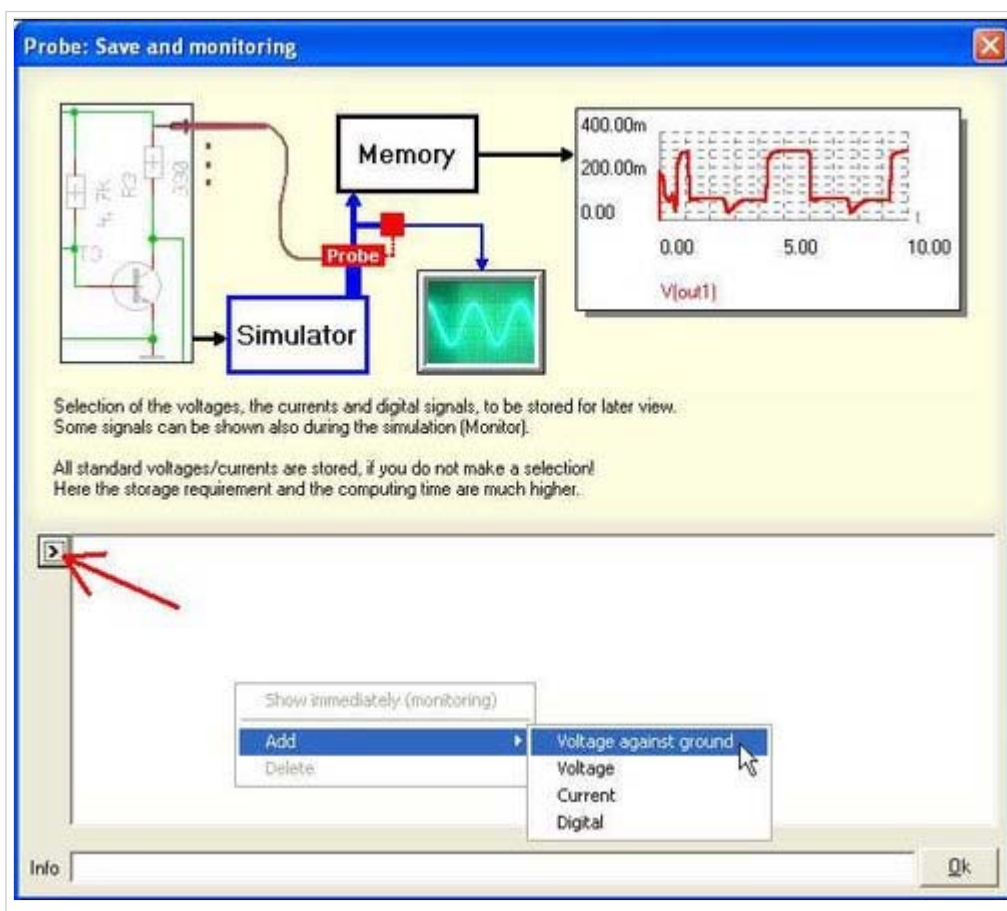
Look at the assistant. It warns in a line, that some of the components don't have a model. By M11 on this line we learn...



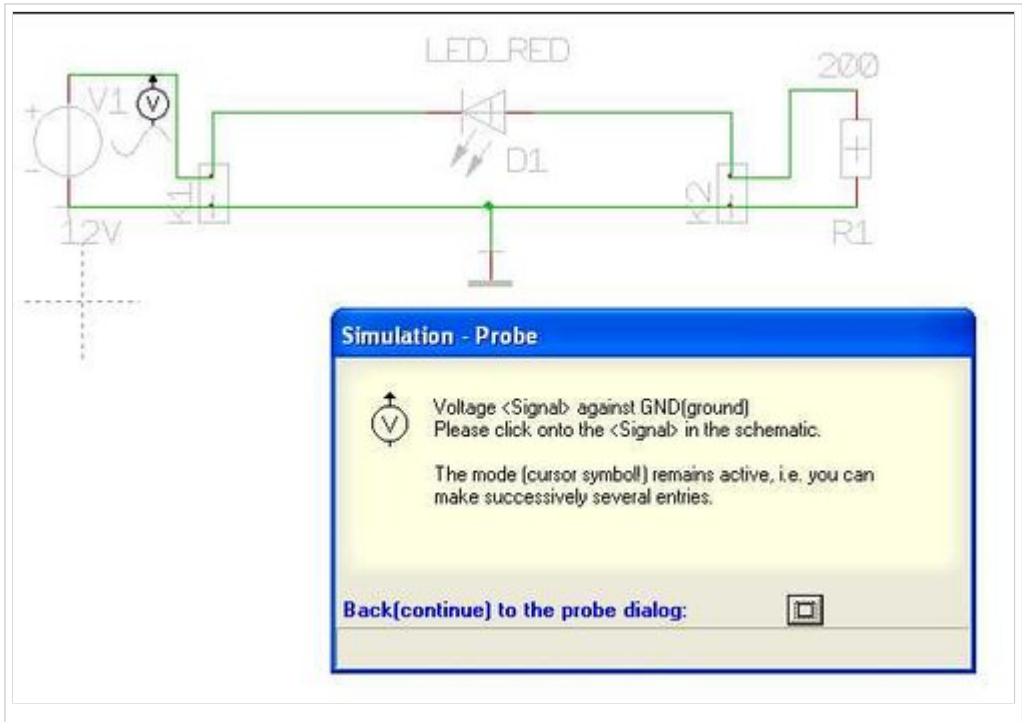
...that the connectors obviously don't have a simulation model. We ignore this message, because we don't want to simulate their function anyway. Furthermore it is said: **Error: An (active) analysis (TRANSIENT, DC or AC) in commands is necessary.** and an advice follows: **No (active) probe in the commands.**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



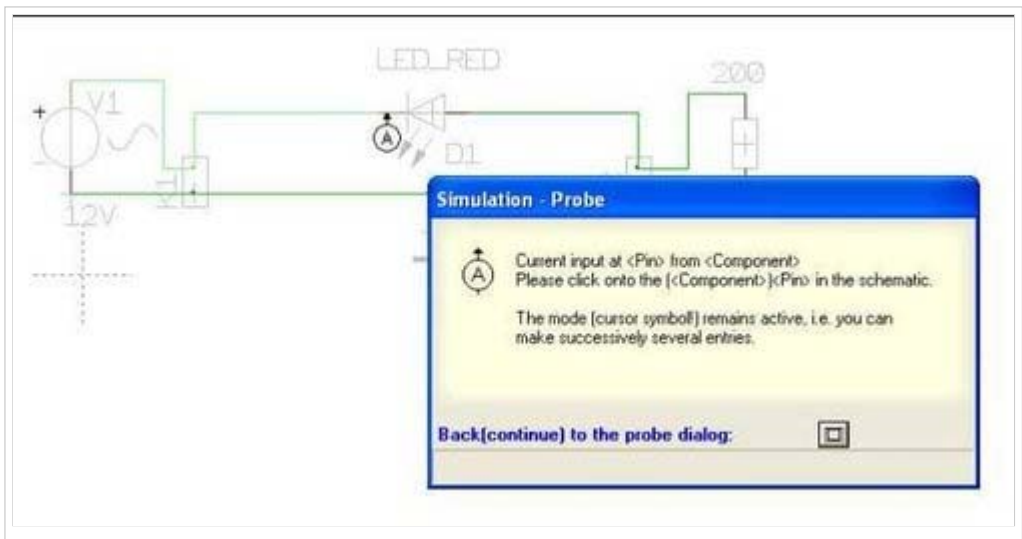
At a sinus tension with 50 (Hz) (= a period of 20ms) we choose the end of the simulation at a decuple (=200ms), so we see 10 oscillations. The step width we choose rather small, for getting a smooth curve. Here a maximum step width of 1ms. As step width for a successful definition of a initial calculation point (convergence) 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 is added, standard tensions and currents are saved).



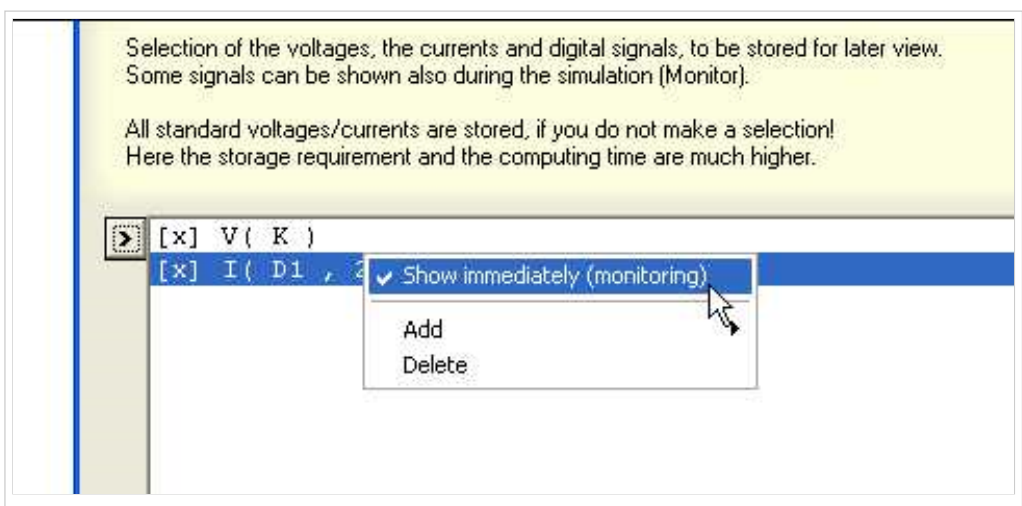
We are interested in the tension (sinus 12V)...



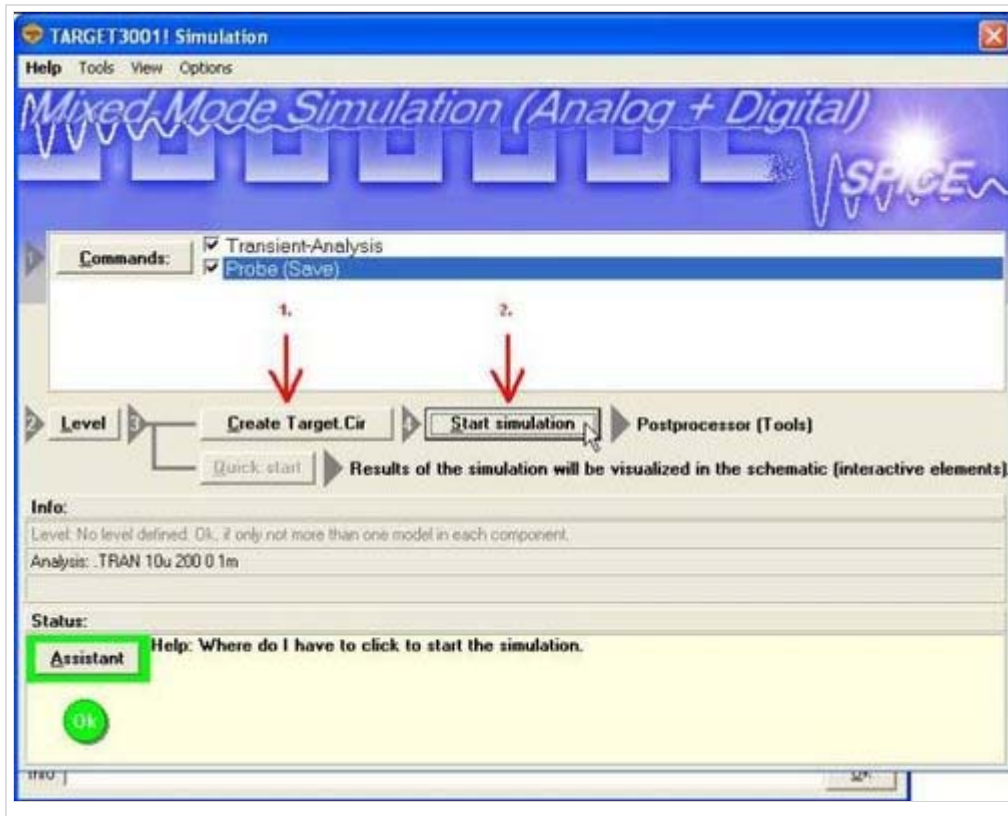
...and in the current through the diode (after selection of "current" in the probe of the main dialog). For the current **not** the signal track must be clicked **M1** but the connection pin itself.



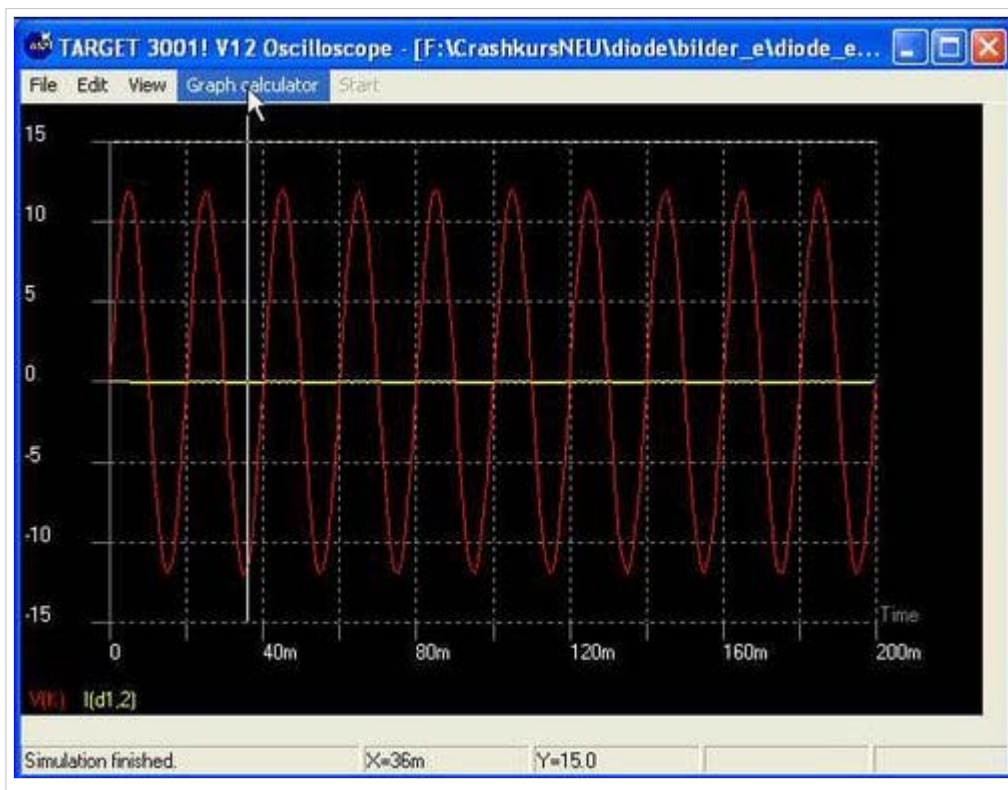
Now click the button "Back (continue) to the probe-dialog".



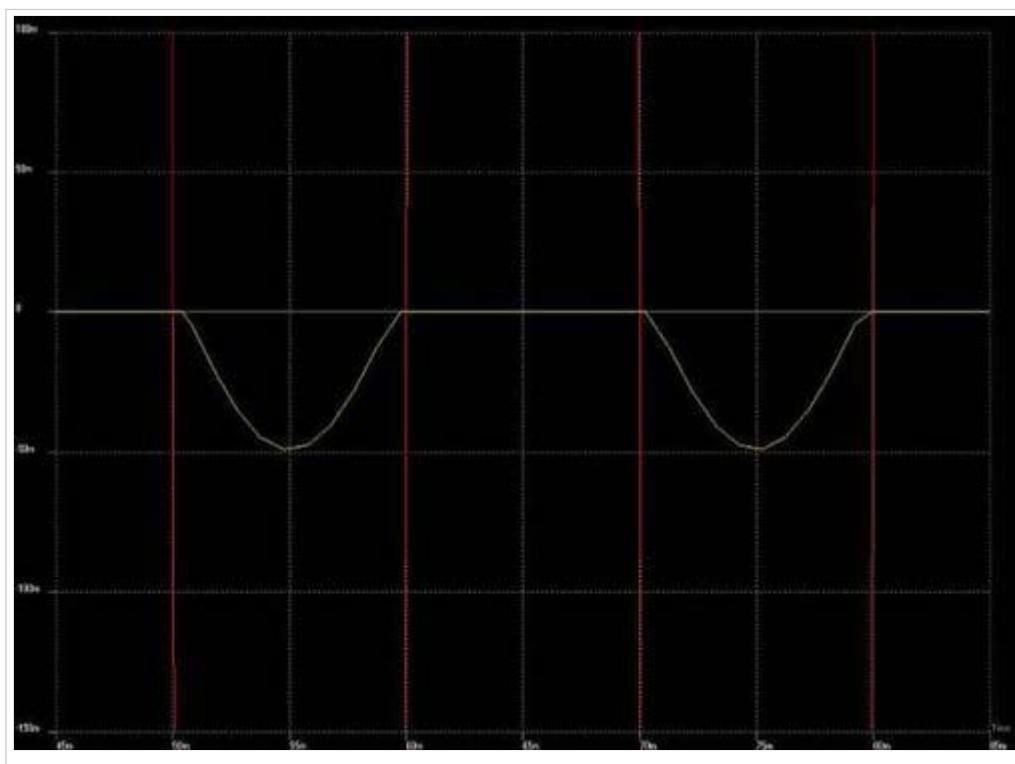
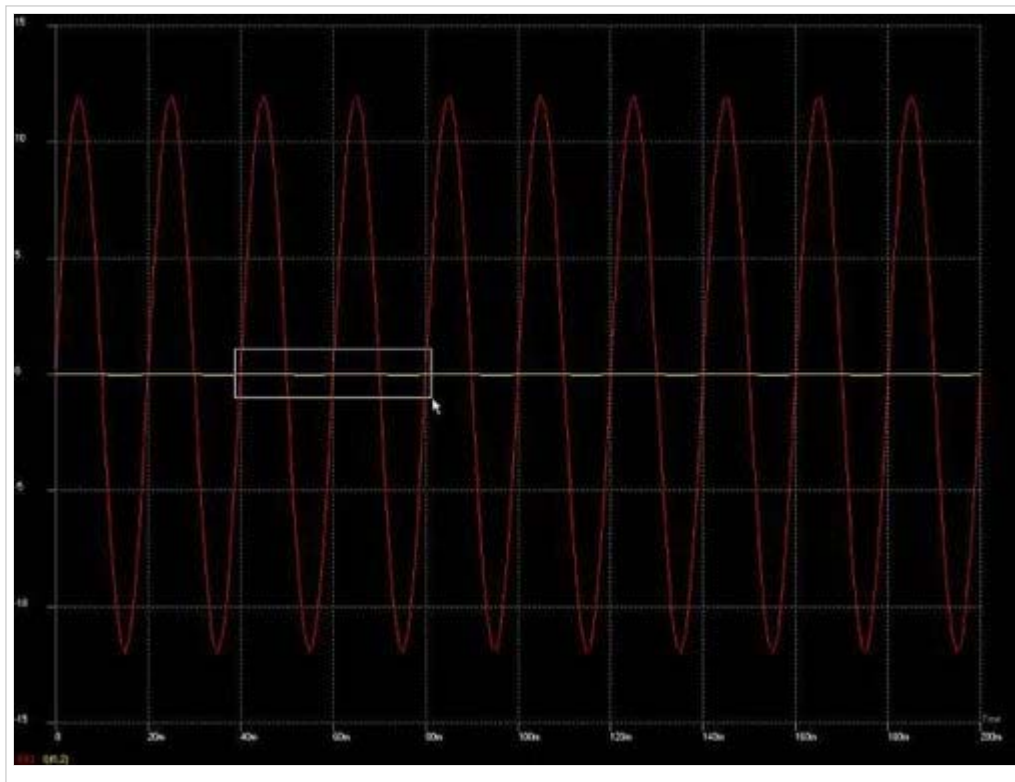
The "immediate screening" (intermediate data are displayed during the simulation process) should be activated by M2 on the corresponding list entry in the probe.



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

Retrieved from "http://server.ibfriedrich.com/wiki/ibfwikien/index.php?title=Simulate_the_function_Part_2"

Categories: Simulation

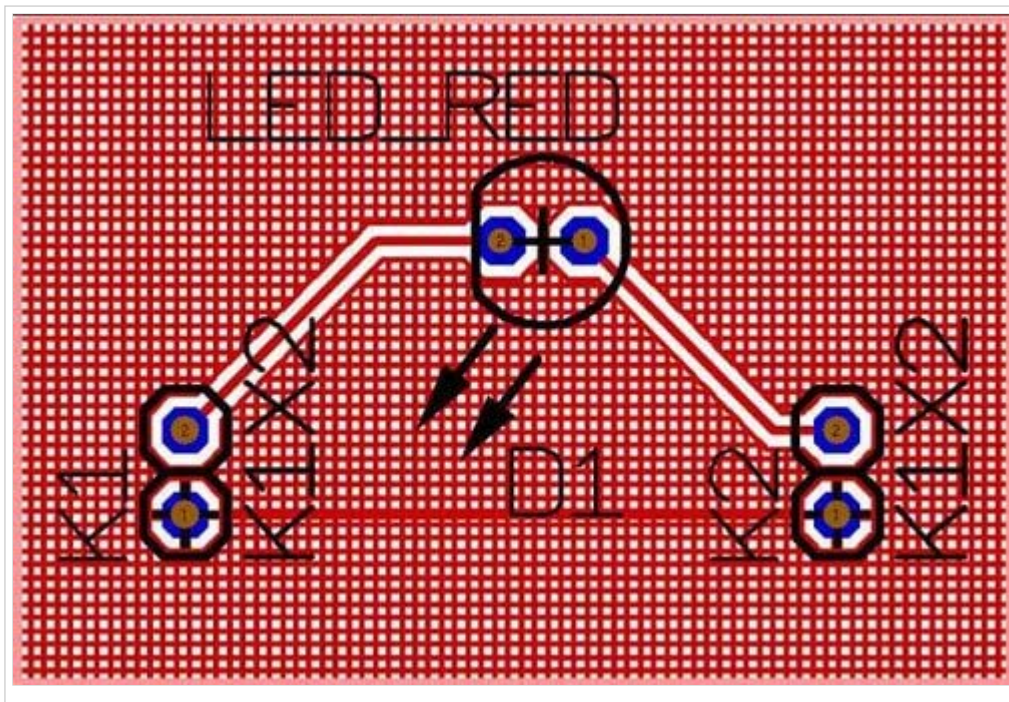
IBF-Intern:

3D-view of the layout

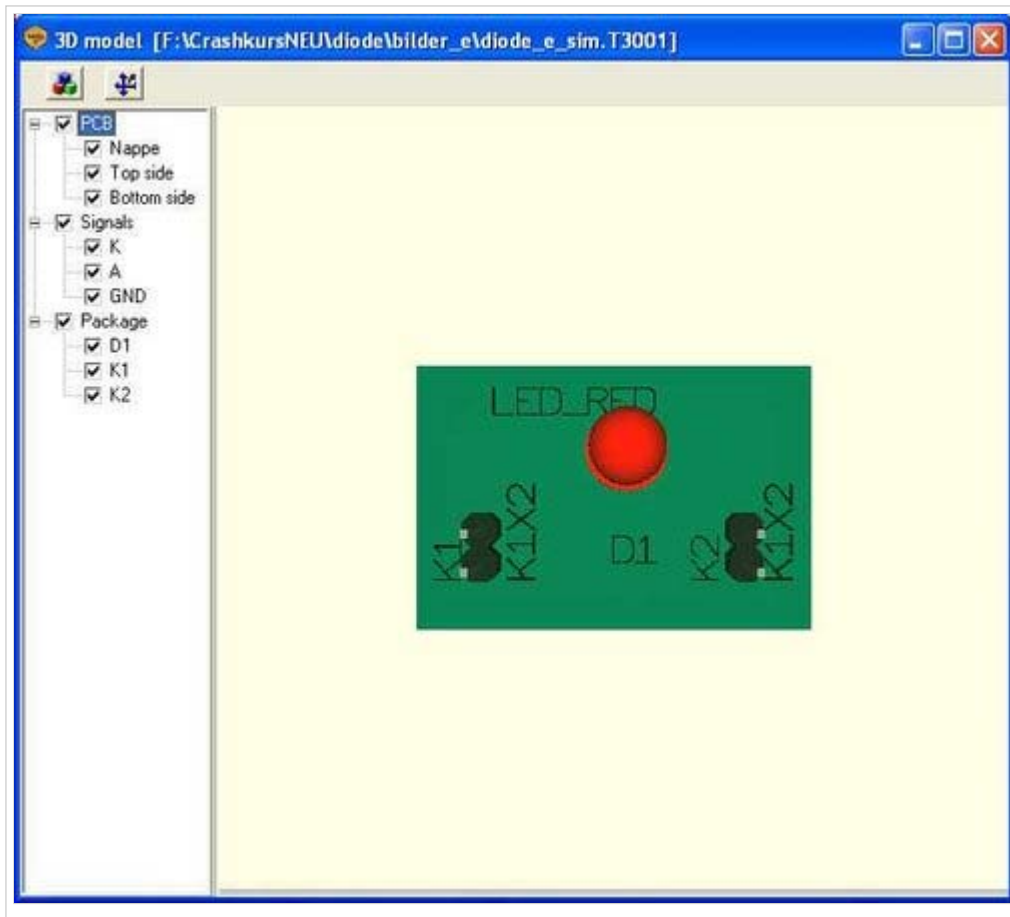
From IBF-Wiki

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

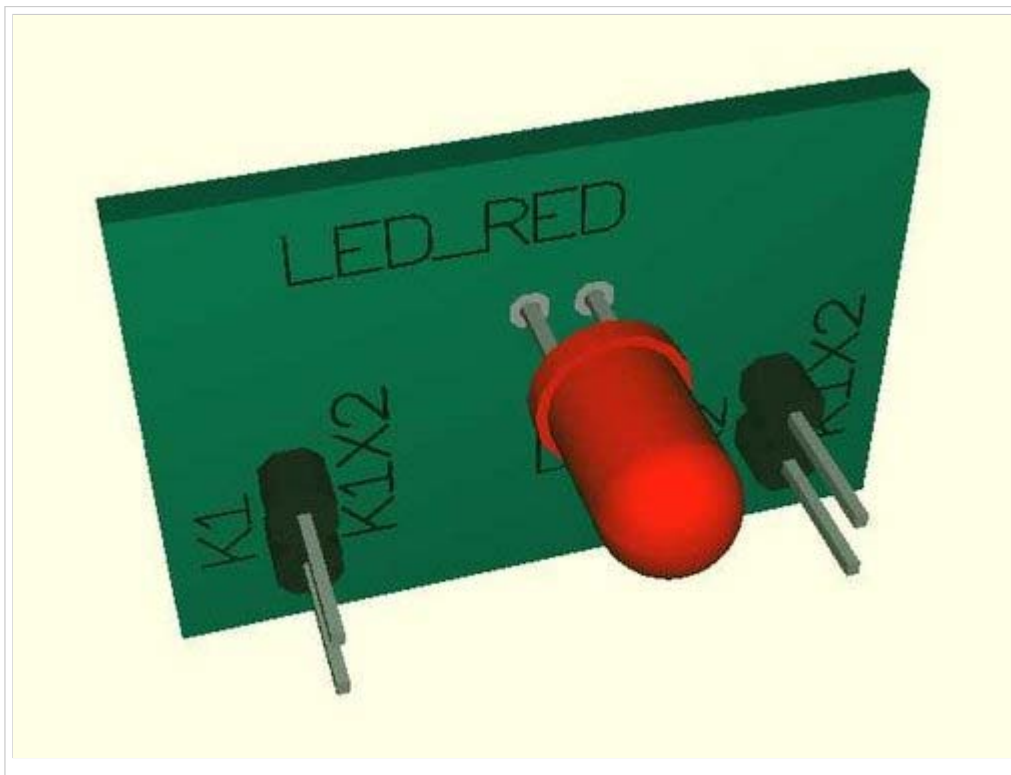
Currently the layout looks like that:



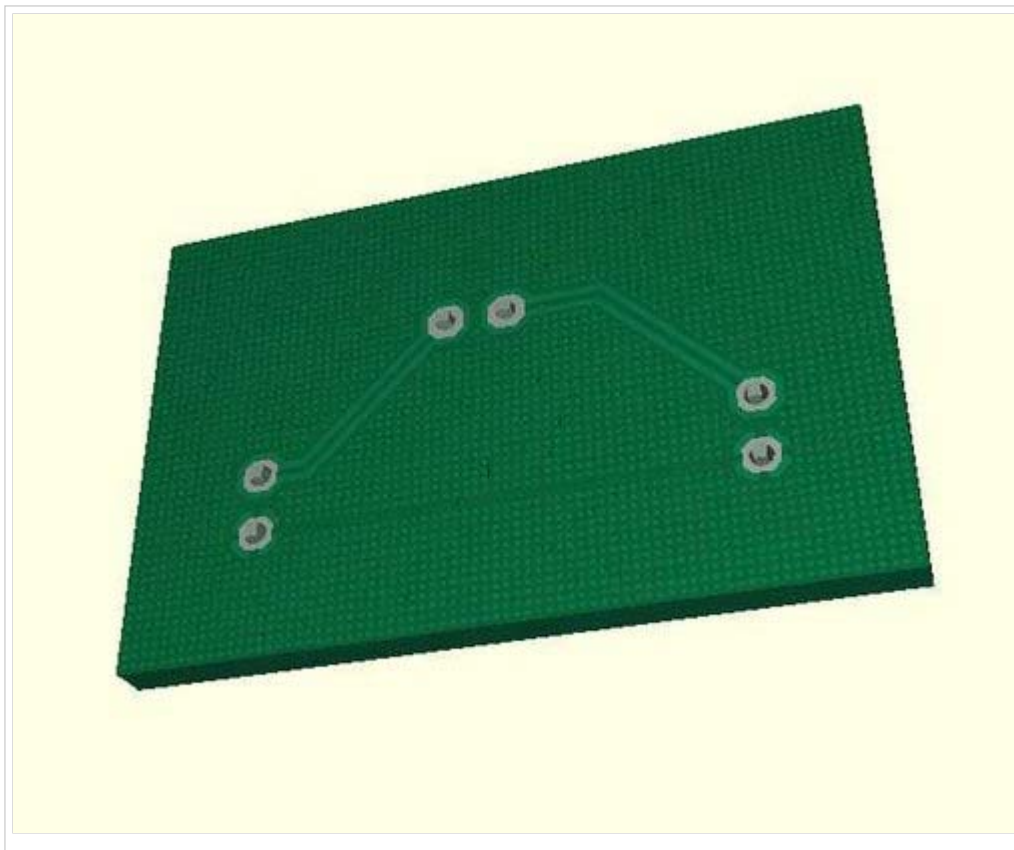
View it in 3D by menu "Actions/3D-View". After a short time of computation the result is:



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](#)

Retrieved from "http://server.ibfriedrich.com/wiki/ibfwikien/index.php?title=3D-view_of_the_layout"

Categories: [Automatic functions](#) | [Modes](#) | [About TARGET 3001!](#)

IBF-Intern:

- This page was last modified 11:55, 4 August 2008.
- This page has been accessed 4,113 times.

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...

Price Inquiry to PCB producers

Sender:

Company: My Company

Name: My Name

Street: My Street

City: My Code My City

Telephone: My Phone Telefax: My Fax

eMail: My eMailAddress

PCB price inquiry | Component price inquiry

PCB Inquiry

Dear Sirs,
Madam,

Please give me a quote for the following:

Solder stop mask: Double sided
Minimal track width: 0,20 mm
Minimal spacing:
Number of drill holes: 6 drills
Number of drill tools: 2
PCB Form: Standard form (4 corners)
Number: 1 pcs
Delivery time: 10 working days
File format: TARGET *.T3001
File: diode_e_sin.T3001

with kind regards:

My Name

Addressees (PCB producers)

- Basista Leiterplatten
- Beta-Layout GmbH
- CONRAD Leiterplattenservice
- CONTAG GmbH
- Hofmann Leiterplatten GmbH
- IBR Ringle
- LeitOn GmbH
- M & V Leiterplattenvertriebs GmbH
- mos electronic GmbH

New Edit Delete

← Please check, correct and complete these data

Send inquiries as eMail

Print inquiries as letters

Copy to clip board

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:

PCB-POOL® Calculator [PCB_diode_e_sim]

PCB

PCB type: Double sided plated through (PTH) ?

Size: 30,0 mm x 20,0 mm ?

Single area: 0,06 drf

Pool type: Prototype ?

Quantity: 1 pieces ?

Total area: 1,00 drf

Solder stop mark ?

Position print (top side) ?

Position print (bottom side) ?

Extra rout ?

E-Test ?

Over delivery at half the price ?

Delivery time: 8 WD: basic price factor 0.5 ?

Shipping to: Great Britain standard ?

Basic price PTH (1da ³)	52,82 EUR
Basic price factor 0.5	-26,41 EUR
Great Britain standard	6,00 EUR
Total net	32,41 EUR
VAT 21%	6,81 EUR
Total gross	39,22 EUR

Customer

Invoice address

Company: My Company

Name: My Name

Street: My Street

City: My Code My City

Dispatch Address (if different)

Payment by: EuroCard / MasterCard

Creditcard

Holder:

Number: 0000 - 0000 - 0000 - 0000
(will be transmitted encrypted)

valid thru: MM/YY CVN: ?

Special instructions and comments:

For call backs

Telephone: My Phone Telefax: My Fax

eMail: My eMailAddress

Continue: Gather and show data

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://server.ibfriedrich.com/wiki/ibfwikien/index.php?title=Produce_a_PCB"

Categories: Actions | Automatic functions | Production

IBF-Intern:

- This page was last modified 11:56, 4 August 2008.
- This page has been accessed 4,228 times.

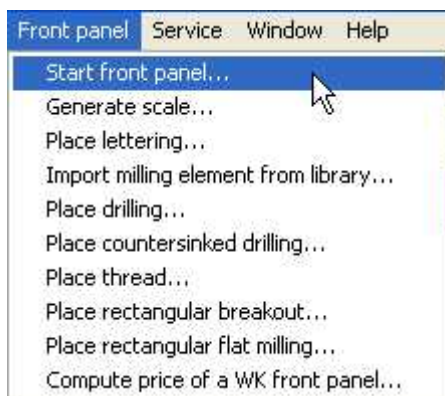
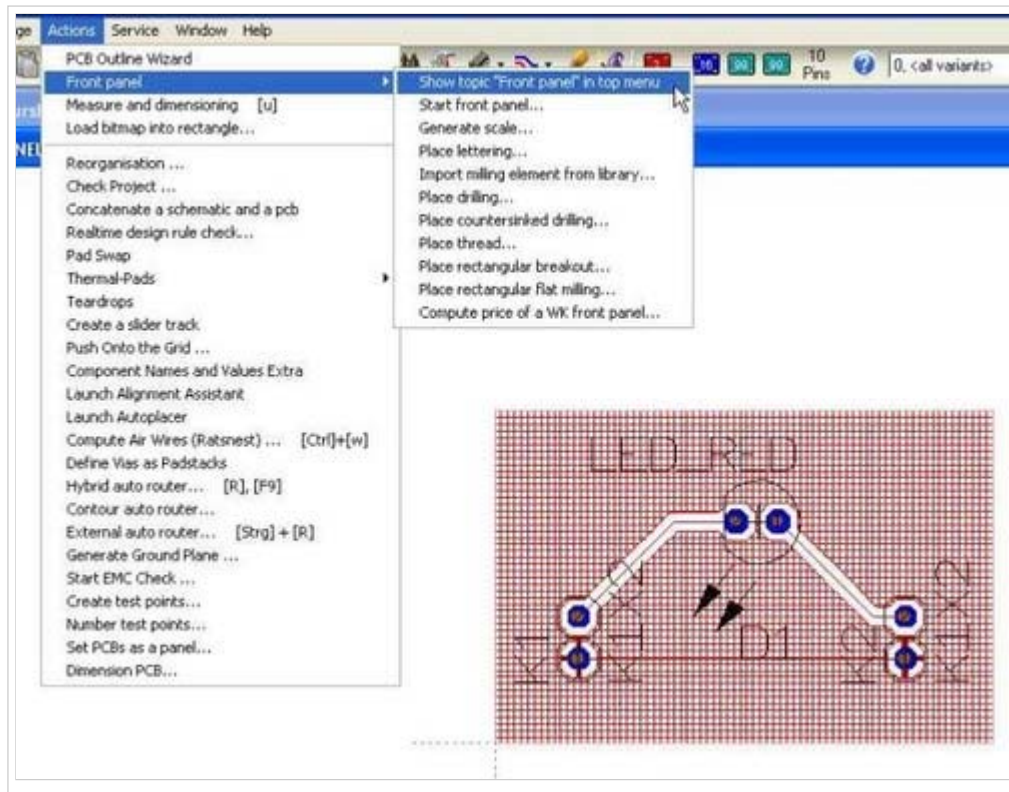
Design and produce a frontpanel

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: mm x Height: mm

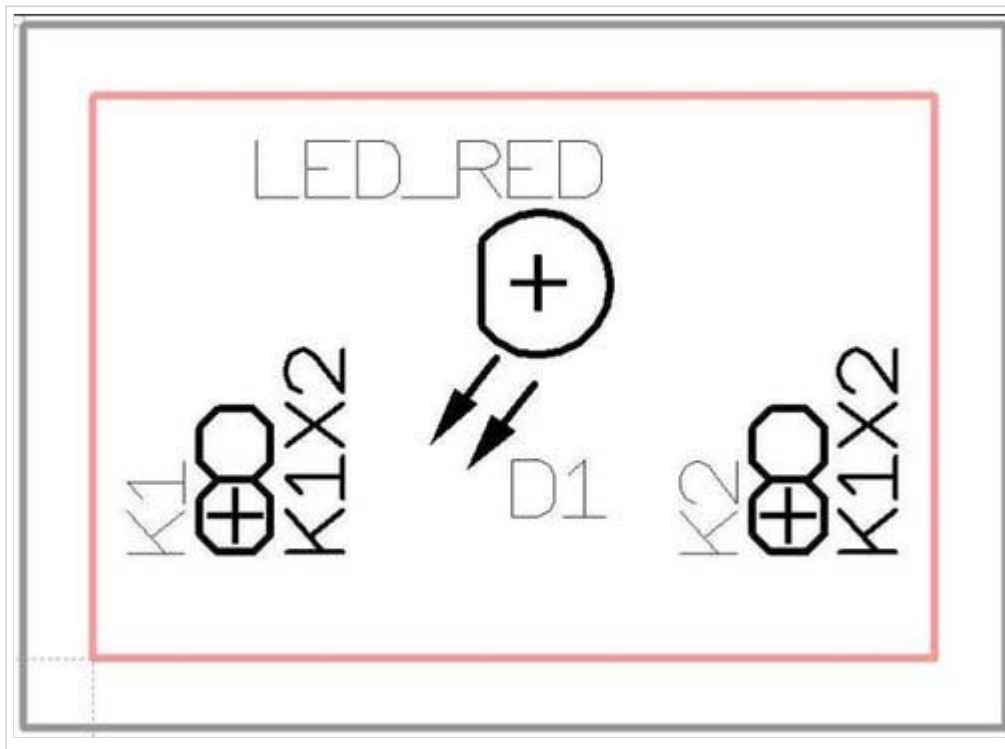
19" Frontpanel (483 mm):

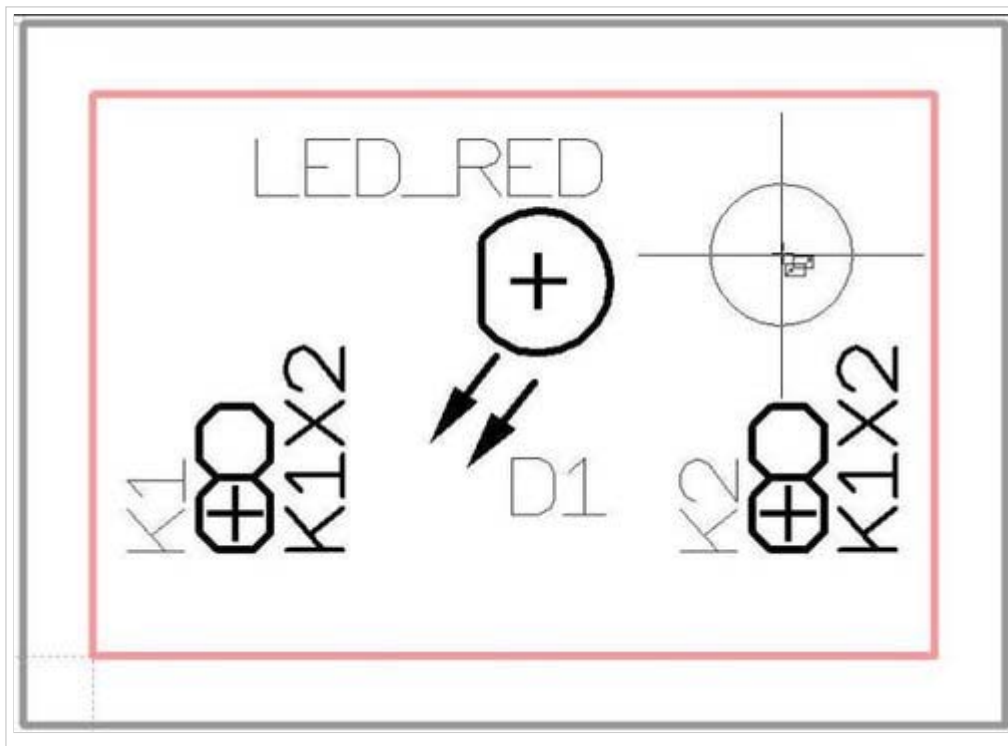
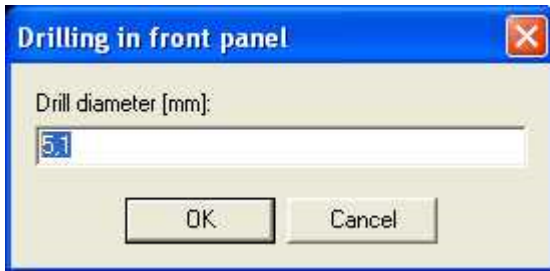
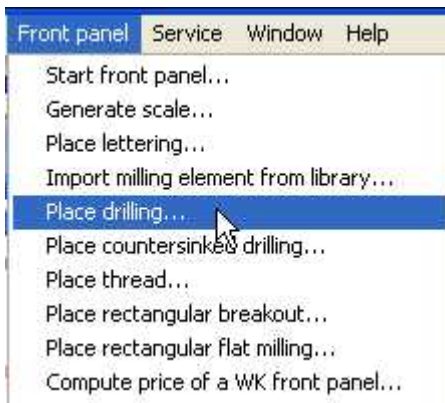
19" Segment: 3 U (128,4 mm):

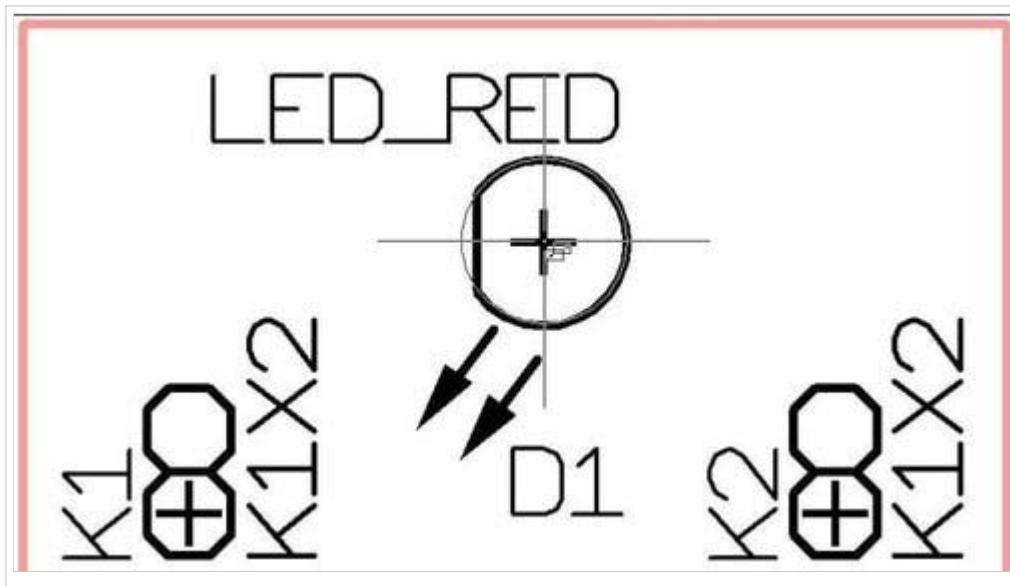
19" Segment: 6 U (261,8 mm):

The front panel can be generated besides or above the PCB:

above or over the PCB
right besides the PCB
left besides the PCB
atop besides the PCB
down besides the PCB

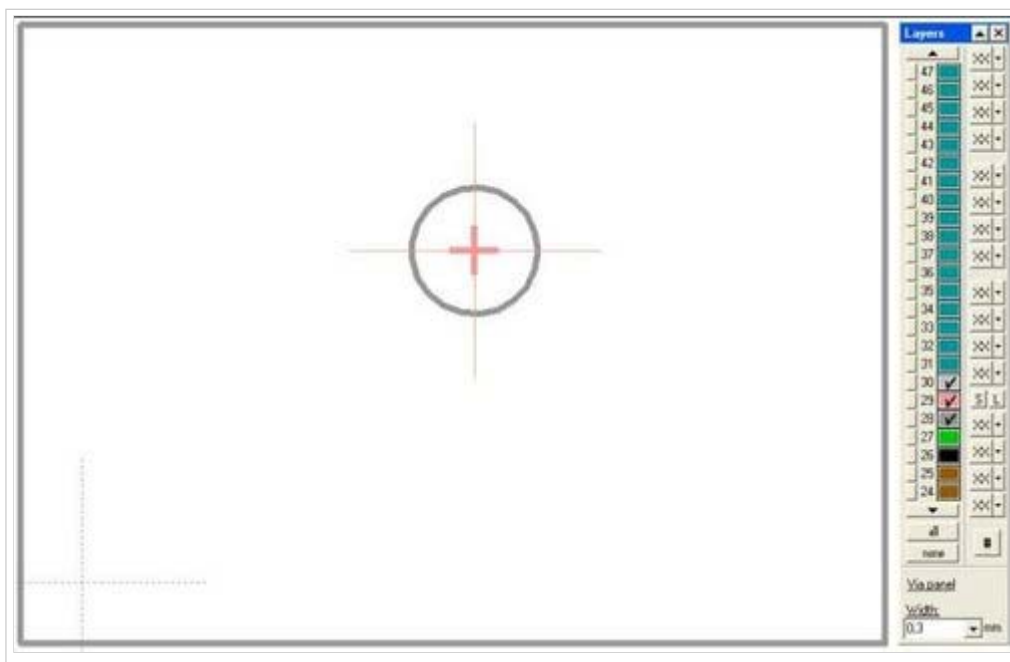


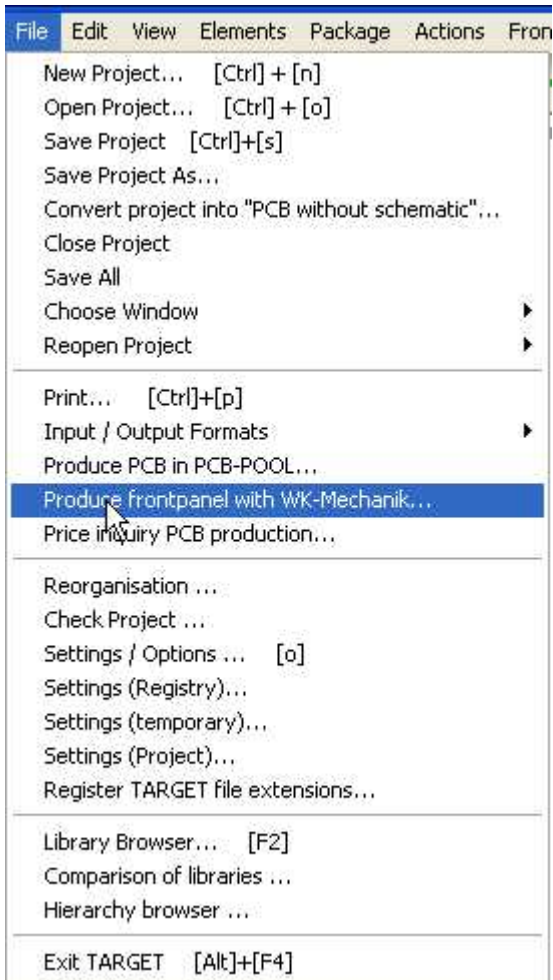




<input type="checkbox"/>	<input type="checkbox"/>	31	Other
<input checked="" type="checkbox"/>	<input type="checkbox"/>	30	Frontpanel (lettering)
<input checked="" type="checkbox"/>	<input type="checkbox"/>	29	Frontpanel (documentation)
<input checked="" type="checkbox"/>	<input type="checkbox"/>	28	Frontpanel (milling)
<input type="checkbox"/>	<input type="checkbox"/>	27	Air wire (ratsnest)
<input type="checkbox"/>	<input type="checkbox"/>	26	Pad numbers
<input type="checkbox"/>	<input type="checkbox"/>	25	Milling
<input type="checkbox"/>	<input type="checkbox"/>	24	Drill holes

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





WK Calculator [Panel diode_e_sim]

Front panel

Surface: Aluminium anodized silver

Panel thickness: 40 mil (1.0mm)

Size: 25.0 mm x 25.0 mm

Quantity: 1 pieces

Lettering: engraved

Delivery time: 5-8 working days

Shipping to: Great Britain standard

Price calculation

Order	10.00 EUR
Piece	1.20 EUR
Material	0.06 EUR
Milling	0.35 EUR
Threads	0.00 EUR
Countersinks	0.00 EUR
Engraving	0.00 EUR
Front panel	11.61 EUR
Shipping (Great Britain standard 1kg)	20.00 EUR
Total net	31.61 EUR
VAT 21%	6.64 EUR
Total gross	38.25 EUR

Customer

Invoice address

Dispatch Address (if different)

Company: My Company

Name: My Name

Street: My Street

City: My Code My City GB-

Payment by: EuroCard / MasterCard

Creditcard

Holder:

Number: 0000 -0000 -0000 -0000
(will be transmitted encrypted)

valid thru MM/YY CVN:

Special instructions and comments:

For call backs

Telephone: My Phone Telefax: My Fax

eMail: My eMailAddress

Continue: Gather and show data

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

This is the end