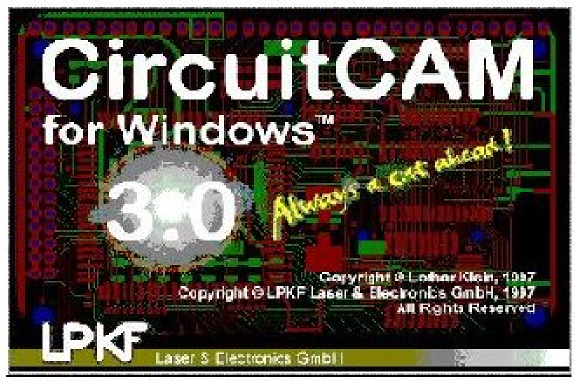
Introduction Protel, CircuitCam & BoardMaster



by Sascha Reinhardt, Marco Wilzbach

Technische Informatik, University of Heidelberg

5th November 1999

Abstract

This documentation originated from the project work about LVDS in the time of August 23rd 1999 till 5th November 1999. It describes the programs used to create a printed circuit board. The programs have the following functions:

- *Protel* is used to construct a wiring schematic and to turn it into a real layout.
- *CircuitCam* prepares the layout from Protel for the LPKF Plotter.
- *BoardMaster* is the controller software for the LPKF-Fräsbohrplotter, which is available at the IHEP for the rapid prototyping of PCB's.

We had to learn the program Protel by doing, without a documentation. Thus we weren't able to use all existing features of the software, plainly by not knowing of their existence. Only project relevant functions were used in the period of our LVDS design. This shouldn't be a too big of a restriction, because we hope to show you the most common.

CircuitCAM and BoardMaster were shipped with the LPKF plotter, so you should find plenty of documentation. Those programs are rather voluminous (as Protel), so our intention is not to give a complete documentation, but to summarize the steps we had to take to get our PCB's. That is why we used a processing circle of one of our PCB's to show exemplary the necessary steps involved. With the LPKF we got excellent support from Ralf Achenbach, head of the ASIC test laboratory. This wouldn't have been possible without him.

Further pieces of information can be obtained by the authors. Their email is:

- Sascha Reinhardt sascha.reinhardt@urz.uni-heidelberg.de
- Marco Wilzbach PCB@Wilzbach.de

This PDF document will be available on the WWW:
http://www.wilzbach.de/PCB_prototyping.html

Contents

Chapter 1

Protel

1.1 Creating a new design

In order to create a new design in Protel you have to choose the Menu item `File->New Design...` A window opens, where the directory, the name and a password can be provided (1.1).

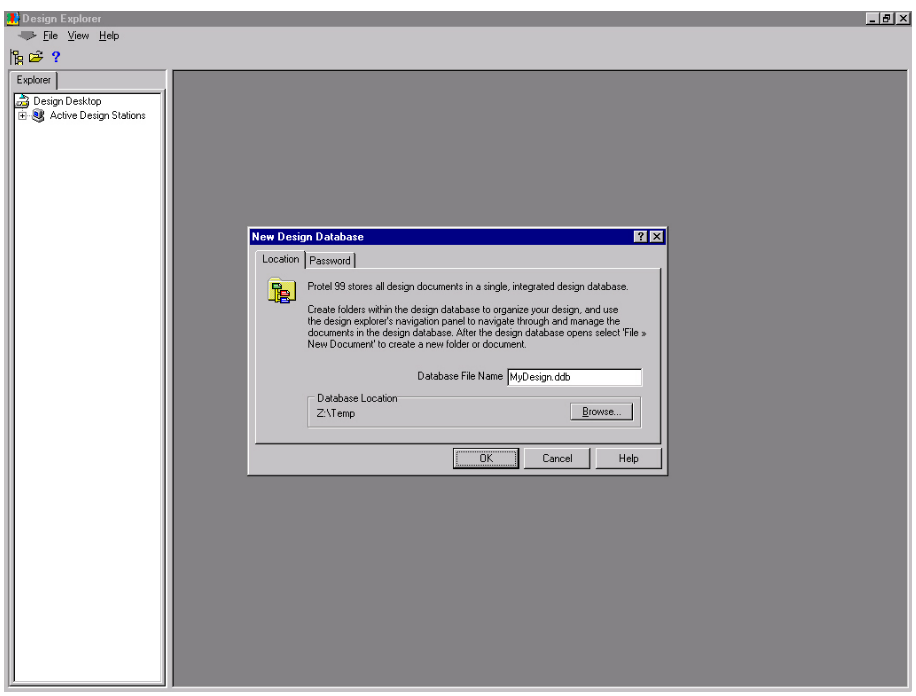


Figure 1.1: `New Design Database`

1.1.1 Wiring schematic

To add a wiring schematic document you have to choose `File->New...`. Then there appears a list of available documents. Choose the 'Schematic Document'. In the list you will also find all the other documents you will need later (1.2). The new Schematic Document should apear in the 'design files window'. You can rename it in the internal explorer and also open it (if it isn't already open).

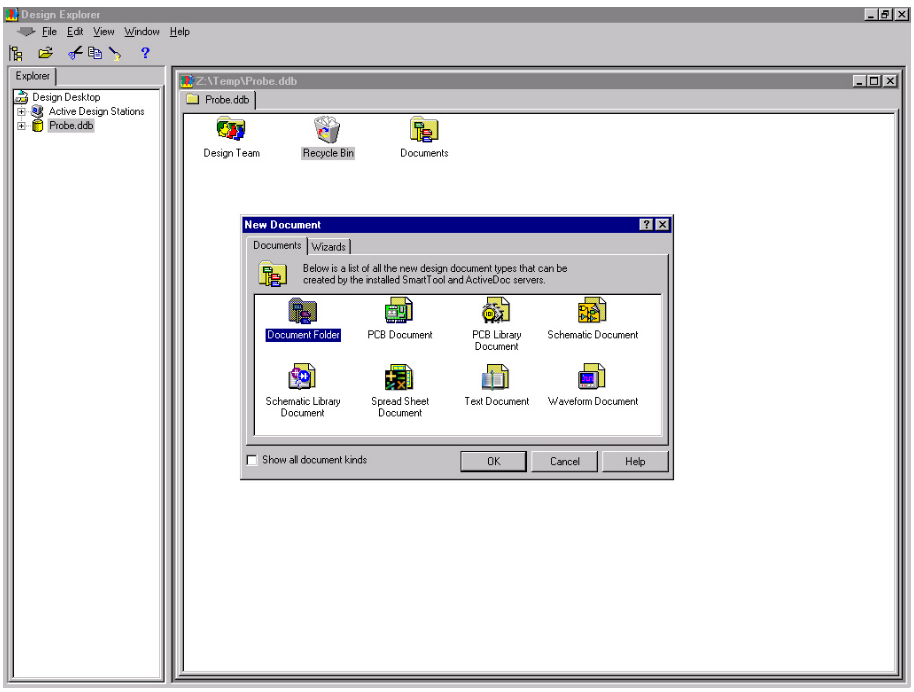


Figure 1.2: New Document

1.1.2 Creating own components

Components for the wiring schematics are created in a `Schematic Library Document` or are loaded from existing libraries. You will get the `Schematic Library Document` as simple as the `Schematic Document`.

With the tools you can create every component you will need. Once you have selected a tool you can change the preferences of it by pressing *TAB* . E.g. you do intend to create a component with pins on both sides of the component. You will have to swap the orientation of the pins on one side, to have the name of the pins all in/out of your component. It also can happen that the automatic pin numbering does not provide the correct counter. E.g. you delete a pin, and with the next pin the number is now too high. To solve that problem you have to press *TAB* and then to modify the number.

To deactivate the current tool you have to press the right button of your mouse.

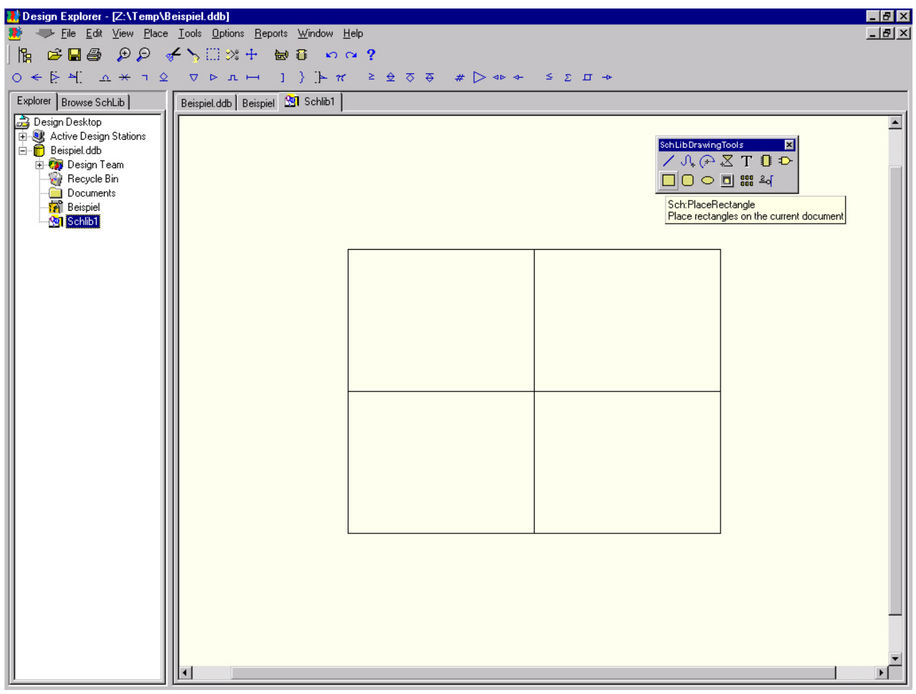


Figure 1.3: Surface of the `Schematic Library Document`

Once a component is finished, you should choose `Browser->Schlib Description`. A menu opens where you can change the name of the component. An important point is the footprint data (1.4). You will later need a corresponding 'real' part, which you place in the layout, with a footprint corresponding to your schematic component. Otherwise the program is not able to translate schematics into layout. You should choose simple and obvious names, for your own sake.

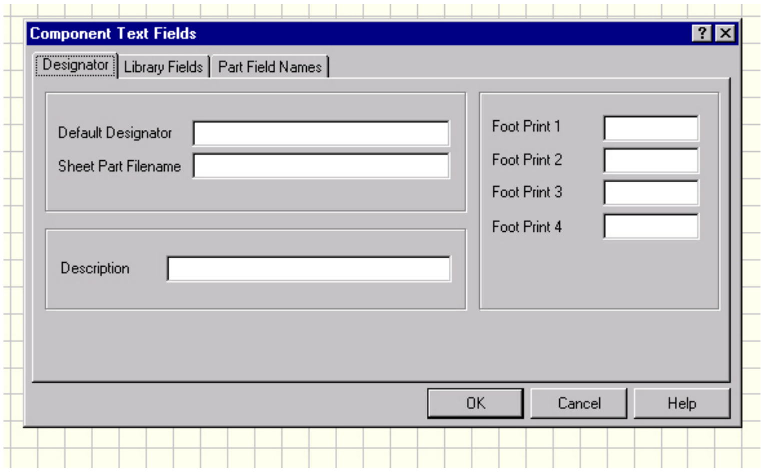


Figure 1.4: Description window

1.1.3 Components on the wiring schematic

You have to switch to the `Explorer->Schematic Document` . Aside the `Explorer` you will see the `Browser Schlib` . Select it and choose `Libraries` . With `Add/Remove` you will add/remove the desired library, e.g. your own library or prefabricated libraries. To add your own, you have to choose your complete design document as a library (1.5).

The take along libraries of Protel are located `\Protel99\Library\-Sch\`.

A library with standard components has the name `Miscelloneous Devices.ddb` Once you clicked OK, you will be able to mark the added library in the Browse window. If you select a library you will get a list with available components. Just select the desired component by a click and drag it to the schematic window. A left click places the component onto your document. To deselect right click. Now you can choose a component by clicking on it. Every single component needs its own unique name.

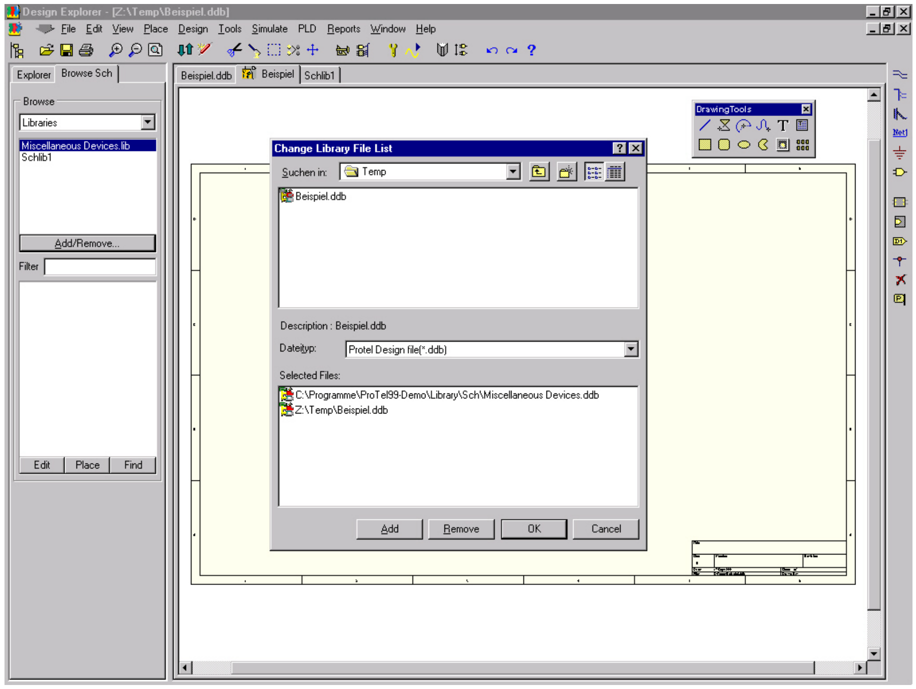


Figure 1.5: Selection of libraries

1.1.4 Connections with wires, busses and ports

There are three ways connecting components.

- You simply link the pins with the tool `Wires` (1.6).

   Tool `Wires`

  You use the tool as follows: With a left click you can set the starting points and the fixed points of the wires (Similary to polygon drawing in a graphics program). With a right click you end the wire drawing mode, and you can start a different connection. A second right click deactivates the wiring tool. The starting/end point should be the end of a pin or a different wire.

- With the tool `Bus` you can use a representation of many wires by one thick wire. You can place it like the tool `Wires`. This helps you to clearly arrange your connections. At the bus you can place `BusEntrys` which then can be connected with pins by wires. To let the program know which two `BusEntrys` are linked, you have to label them. You can label the adjacent wires or the `BusEntry` itself. The corresponding tool is `NetLabel`. Just place the same label at the two wires and they are connected (1.7).

 Tool `Bus`

 Tool `NetLabel`

By clicking twice onto a NetLabel you will get a menu for changing the preferences of the Label. If you use numbers with the labels, they will automatically be incremented by one. The orientation of a BusEntry (or any object) can be changed by pressing the keys `x or y` when having it dragged (hold left click) or in the placing process.

- With `Port` you can connect pins. If you add a port to a pin you will be able to name it. So all ports with the same name are connected to each other. This option is preferably used with power supply and ground. (1.8).

   Tool `Port`

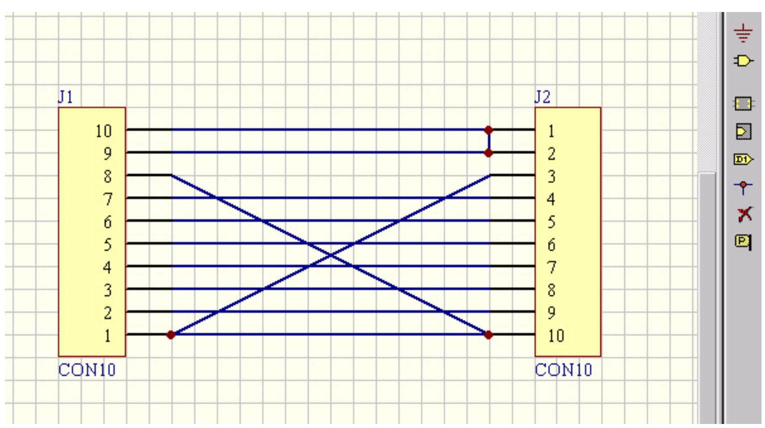


Figure 1.6: Connection with Wires

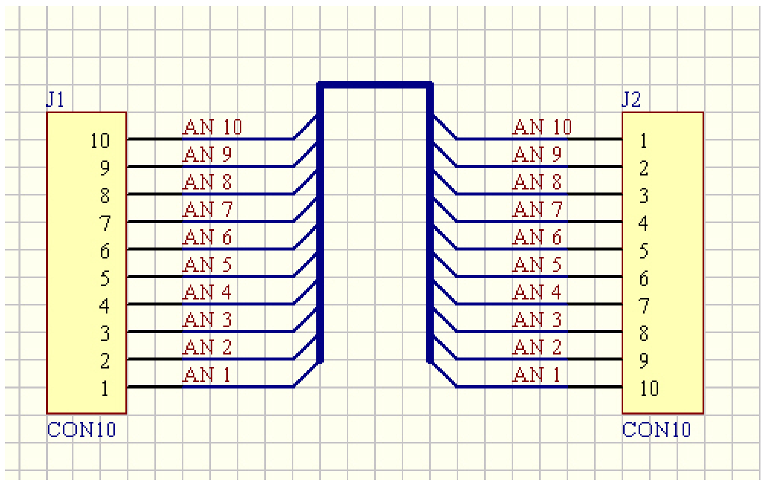


Figure 1.7: Connection with a Bus

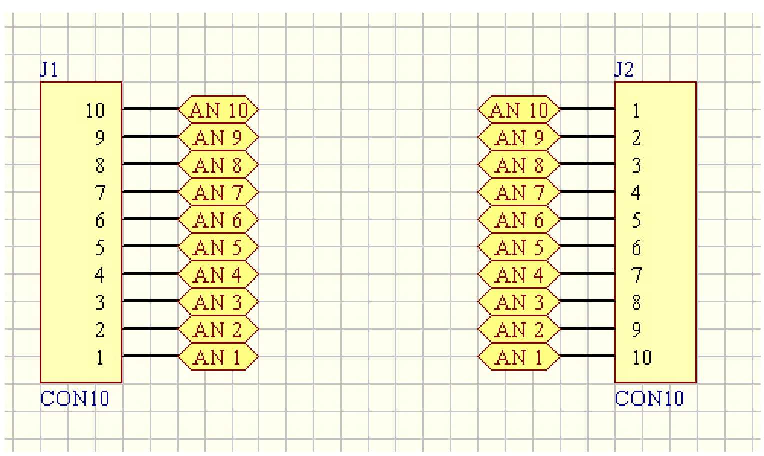


Figure 1.8: Connection with Ports

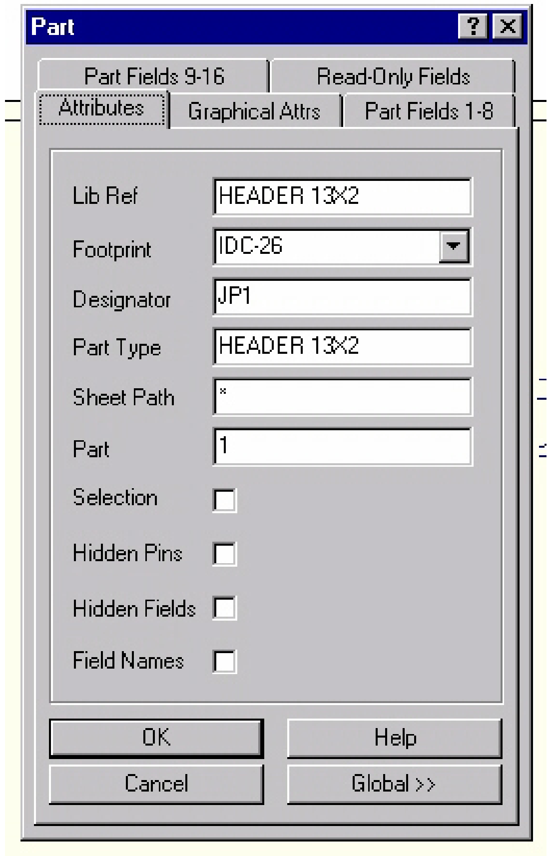


Figure 1.9: Showing preferences of a component

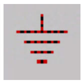
1.1.5 Footprints

Each component needs a footprint for further use (some standard already have one) (1.9).

The meaning of the footprints is to have a link between schematic components and real PCB components. In the footprint library's the dimensions are given.

1.1.6 Power supply and ground

The power supply and the ground level are inserted with `PowerPort` on the schematic. In the preferences card you can change the look with `Style`. The option `Net` you can change the netname, e.g. GND or V-CC. Here the name plays no role and is arbitrary. If you use different power supplies you have to place the desired quantity and to name them differently, in order that they stay seperatly in the layout.

 Tool `PowerPort`

1.1.7 ERC

When you have finished your wiring schematic you can let it check by the program. You can use the electrical rule check `Tools -> ERC...` to find errors, e.g. if a component has no name or is not connected.

1.2 From the wiring schematic to the PCB

The wiring schematic is something abstract and generally has nothing to do with the placement in reality. E.g. you have crossing wires on your schematic, but this wouldn't do good on the PCB until it is desired.

1.2.1 Creation of the Printed Circuit Board (PCB)

You simply have to open the window `New Document`. There you can choose `PCB Document`. But if you already have an expectation to your circuit board, you should choose the `Wizard`, which you also find in `New Document`. There you can specify all desired options for your PCB, step by step, e.g. the dimensions, number of layers.

1.2.2 Creating your own footprints

Choose in `New Document` the File `PCB Library Document`. There you can add new components with their footprints by pressing `Add`. By doing this a `Component wizard` assists you. (1.11).

You can choose the desired type. The next step is to enter the dimensions, which usually are given in the data sheet of your component. Does the wizard not contain your type of component you have

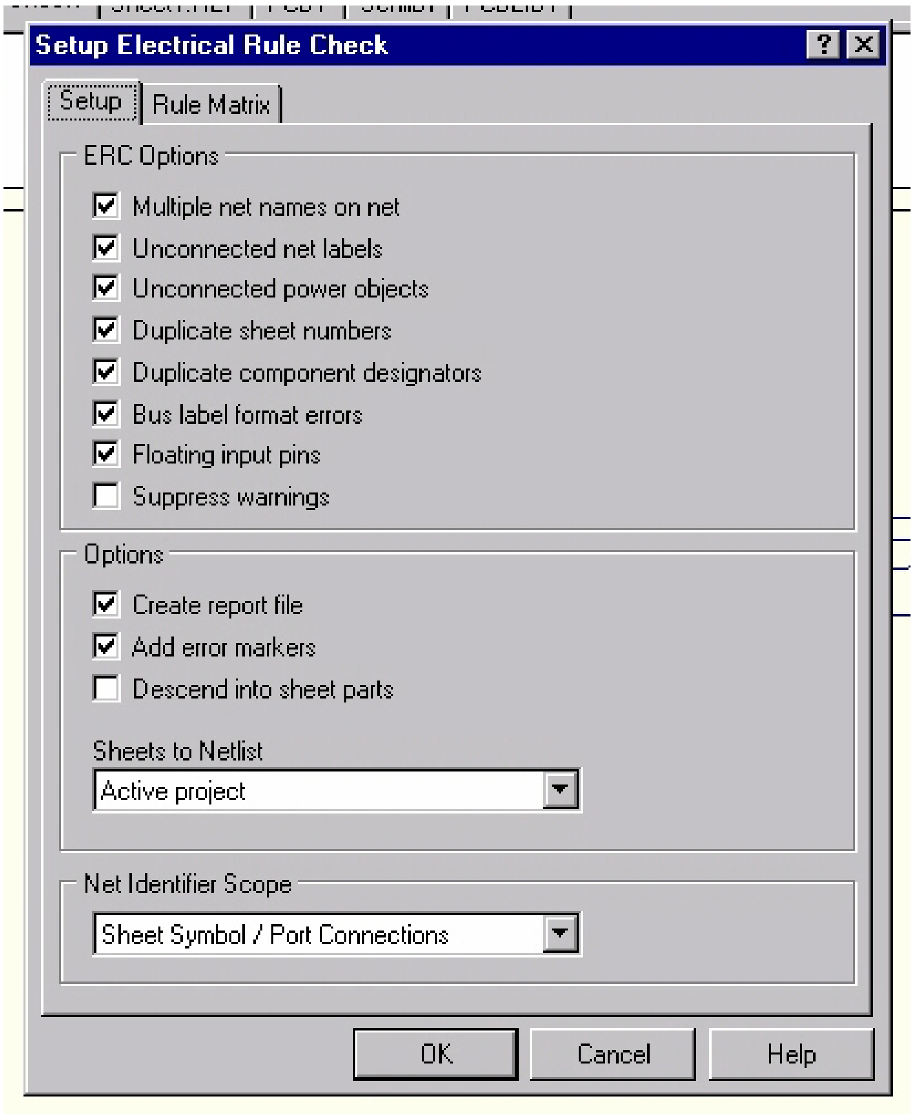


Figure 1.10: `ERC`

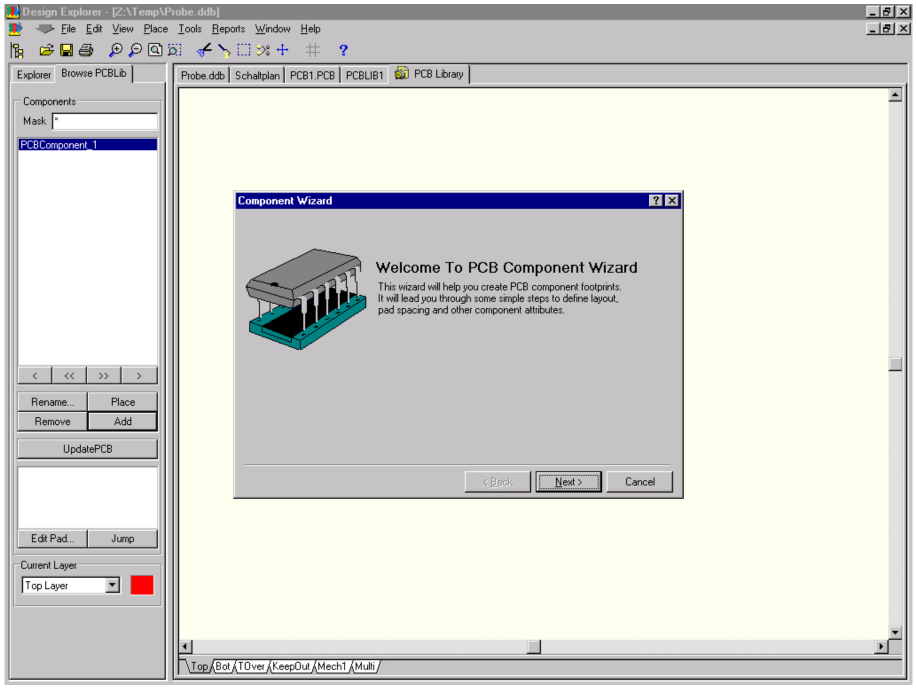


Figure 1.11: Wizard for generating new components

to create it manually. This works similar as in the wiring schematic. The names of your footprints have to correspond to the names of the schematic pins, to let the program know how it has to arrange all the connections.

1.2.3 Updating the PCB

If all components in the schematic do have a corresponding component in a PCB library, then you can let create the PCB. To do this, you have to open the finished wiring schematic and to choose in the menu `Design` the option `Update PCB...`. If there are more than one PCB files you have to choose the right one to be updated.

If there are no errors, then all components and their connections are on the PCB. The connections are displayed as thin lines, which directly link the right (hopefully) footprints. If you created your PCB with the wizard, then your components are usually outside of the boundary line of your PCB.

1.2.4 DRC

The `Design Rule Check (DRC)` is a support for the creation and editing of a PCB. The rules for the DRC can be edited in the menu `Design -> Rules`. There you can specify among other things the minimal clearance between two traces and the minimal/maximum width of your traces.

After the DRC you will get a report. Did you violate one of the rules the problem will be described.

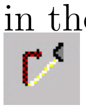
1.2.5 Autoroute and Autoplace

Autoplace: In the menu `Tools -> Autoplace...` you can also use an automated placement of your components. With few components it is usually easier to place them by hand.

Autoroute: In the menu `Auto Rout` are several options for autorouting. The simpliest is `All`. This one connects all wires automatically. Important wires should be drawed by hand.

1.2.6 Drawing wires

Traces can be drawn with the tool `ManualRouter`. You just click on a starting point, then on the next junction, analogously to the wires in the schematic.

 Tool `ManualRouter`

Once a trace has been drawn, it changes its colour (e.g. red to yellow). A little help is that all traces according to the same net are highlighted by the same colour (yellow) when selected.

1.2.7 Pads, Vias and other things

A `Pad` is an area on which SMD components are placed. Usually these pads are created automatically in the placing process (when you place a component). You shouldn't choose to small pads, so that you can easy solder.

 Tool `Pads`

`Vias` are drilled holes which are plated. They connect multiple layers of your PCB. You should use as few as possible of them, because plating is a very ugly procedure (with our IHEP plotter).

 Tool `Vias`

Another useful tool is `Polygon Plane`. Once you finished the drawing of your PCB you can add a plane on your PCB, which is poured over it. You usually connect the place to ground or power. The advantage of the plane is, that you reduce the empty space between the traces, thus taking care of the Universal cutter (a tool of the LPKF plotter, with a weak durability), and to get it done faster for the plotter.

If you are working with high speed, you should activate the option `Remove dead copper`. This removes the areas of copper, which weren't in a location to be connected to the selected net. That is because isolated areas are likely to work as small antennas (EM), thus disturbing other lines.

 Tool `Polygon Plane`

1.3 The way from Protel to CircuitCAM

Is your PCB finished and without bugs, you can start exporting your data to CircuitCAM, which prepares the data for the plotter. CircuitCAM is the program for the LPKF plotter, and was shipped with it, thus having a giant documentation.

1.3.1 Setup Printer and the Aperture Library

Once you have reached the state of an error free PCB, you can start setting up the printer.

In the menu `File->Setup Printer` you get a selection of output devices. You choose `Protel Gerber RS274`. Now you can select

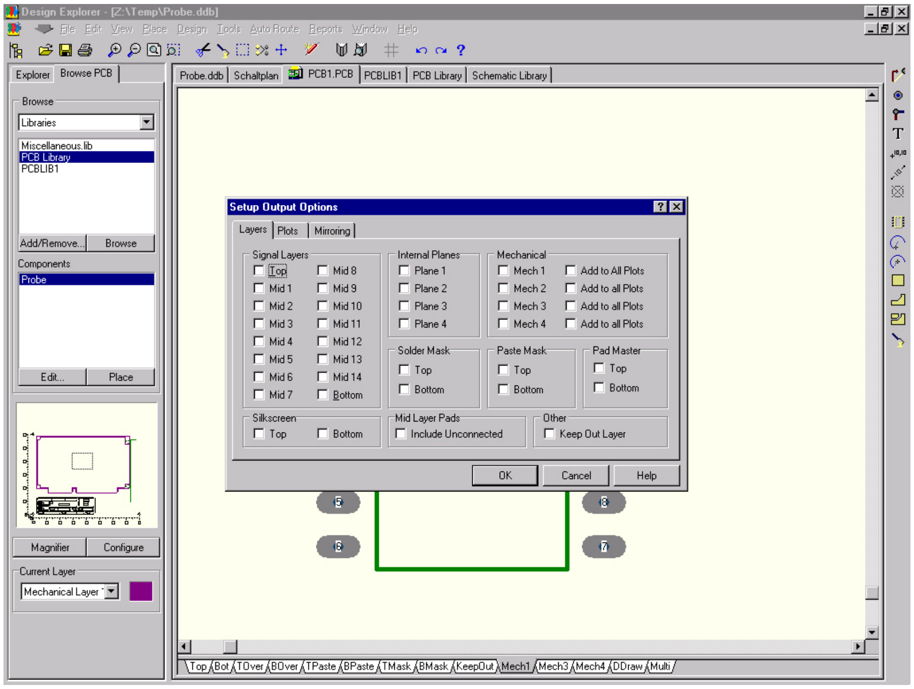


Figure 1.12: Selecton of the layers

`Print, Options, Layers`. First of all you should go to the Layers, where you mark your desired layers, usually *Top, Bottom, and Internal Plane 1 for the drilling*. The other layers should be deselected (1.12). Press OK.

Now you choose the `Options` where you first select `Aperture Library`, to create the right apertures for your board. Selecting `Create List from PCB` normally should do it. If you are working with the LPKF plotter you should in advance deselect the `Flash Pad Shapes`, which are only for an optical PCB processing (with photo varnish). (1.13).

Then you close the options.

After you pressed print you have to specify a directory within your Protel document, where the files should be stored (it is a good idea to have an special directory created before the exporting step, e.g. 'plotter-files').

When Protel is finished plotting the data you get two files for each

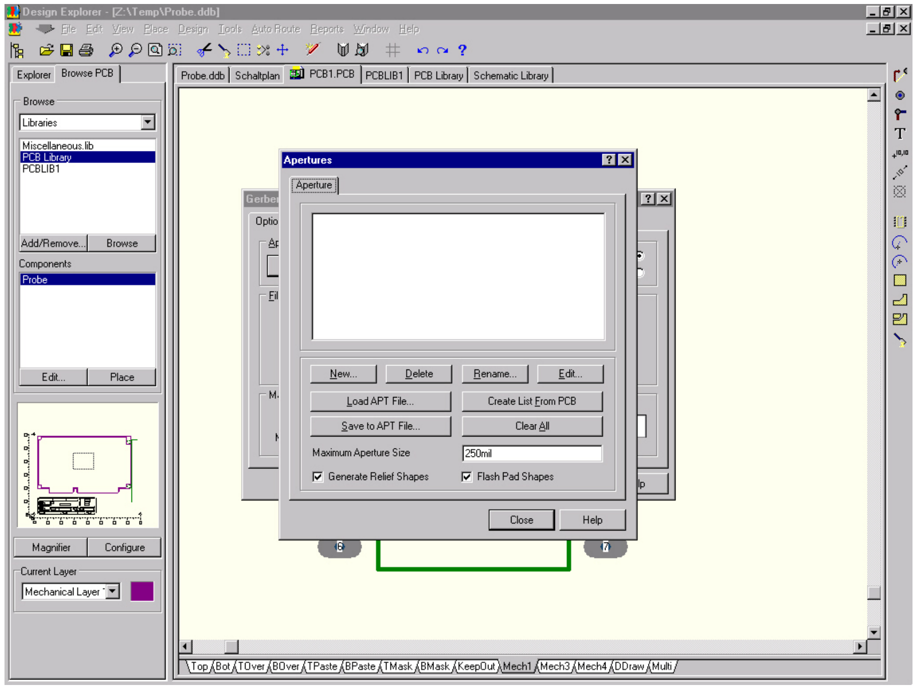


Figure 1.13: Creating Apertures

layer. One aperture file (xxx.Ayy) and a gerber file (xxx.Gyy).

1.3.2 Exporting the files

The plotted files finally have to be exported out of the Protel archive into the real file system. For that you mark all the created files (in the plotting process) in the Protel `Explorer` and then press the right mouse button. In the menu you choose `Export...`. Choose the directory in the real file system and press OK. Now you could profit from a 'plotter-files' directory, since you could export the whole directory.

Chapter 2

CircuitCAM

2.1 Foreword

CircuitCam is used for preparing the PCB raw data generated by Protel into a format, which the LPKF plotter understands. In CircuitCAM you get the final possibility to change something of your layout, even if that generally should be done in Protel. But minor things are OK. Later in BoardMaster you can't change the data. To come to our resulting BoardMaster data, we need several steps to take.

2.2 Import of Protel files

Usually when you generate your output files in Protel, you get two files for each layer, which are belonging to each other. E.g. if you generated data for top layer, bottom layer and drills, then you should have six files. There are two different kinds:

- Aperture files contain the data for the different tools you will need in order to get the used structures worked out. E.g. different drills, or milling tools of varying size. These definitions are needed *before* you can import the so called 'gerber' files. The file names have following structure: xxx.Ayy; where xxx is the filename in Protel. yy can stand for TL (Top layer), BL (Bottom layer), P1 (Plated 1), possible are also other endings, depending on the selected layers which were exported in Protel.

- Gerber files contain the contour data for the milling tools and the hole positions for the drills. The file names have following structure: xxx.Gyy

Bare in mind, that your first step is to import the aperture file and *then* the corresponding gerber file.

2.2.1 Import of the apertures

To import the aperture file you have to choose `Datei->Importieren` (2.1).

Figure 2.1: Menu Import

Once you selected your directory with the exported protel files, you will get a list with the aperture and the gerber files (2.2).

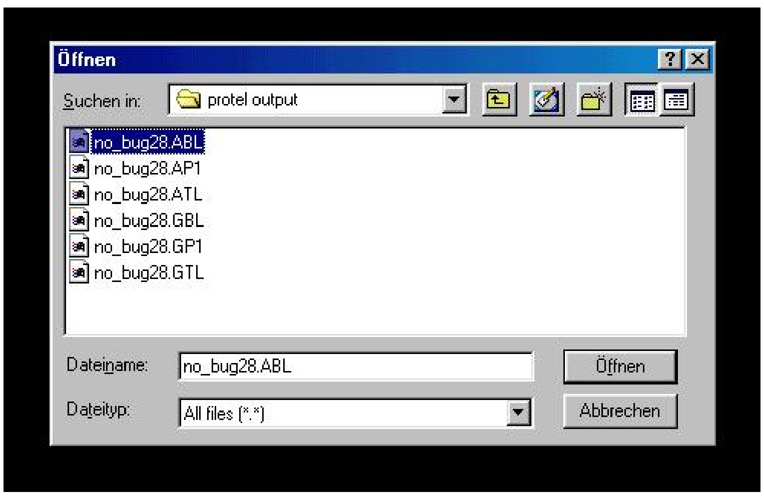


Figure 2.2: List of files

Load the aperture file of your choice. Then you will get to another window, where you have to name the aperture definitions file, to allow the program to identify different aperture files. As name you should use the ending Ayy of your current file, so it is easier for you to arrange the gerber files later (2.3).

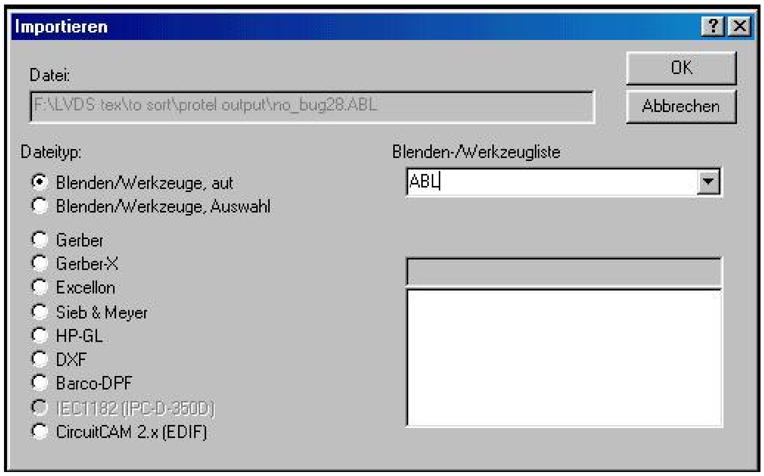


Figure 2.3: Name aperture file

For the `Dateityp` select `Blenden/Werkzeuge, aut`. Press OK. Then you should get the message: xy apertures recognized (2.4).

Figure 2.4: Recognized message

Now you could import the other aperture files repeating the steps above, or you can go along to import you corresponding gerber file.

2.2.2 Import of the gerber data

Proceed like in importing the aperture files. Select `Datei->Importieren`, then select a gerber file, whose aperture file you already imported (2.5).

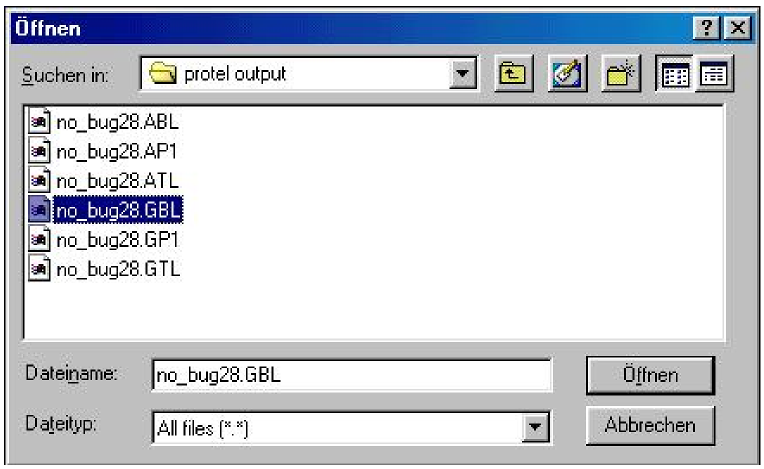


Figure 2.5: Select gerber file

Then you get to the main import menu (2.6).

Gerber-X should already be selected. Then select your corresponding aperture file. If you took my advice with importing the apertures you just have to enter Ayy, where yy stands for the ending from your gerber file xxx.Gyy. Also select the right layer. E.g. if yy=TL then you should select Top layer, or yy=BL then select Bottom layer. Protel usually uses different endings yy for each layer, so that you can identify your exported layers clearly.

The result should look like (2.7), if you started with importing the bottom layer.

Do the same with the remaining layers. The final result should look like (2.8).

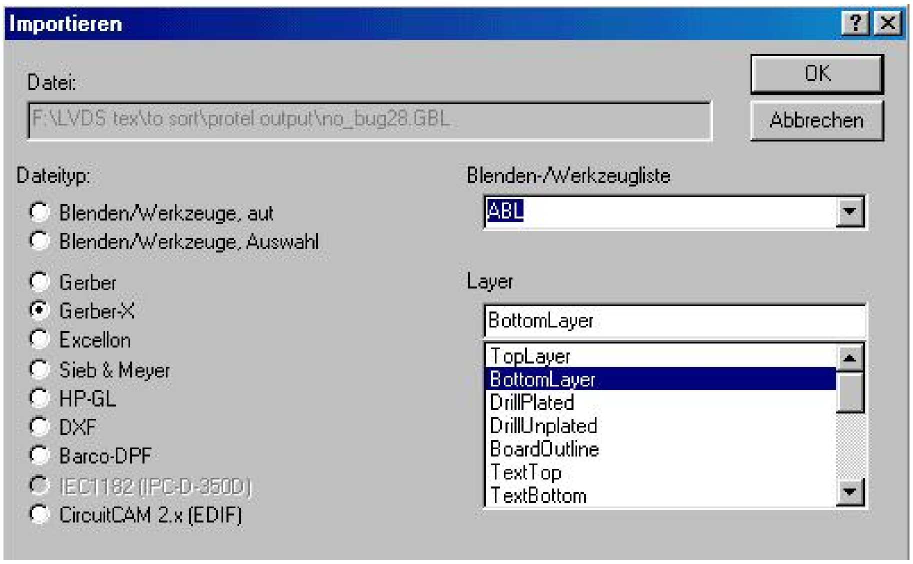


Figure 2.6: Menu Import gerber

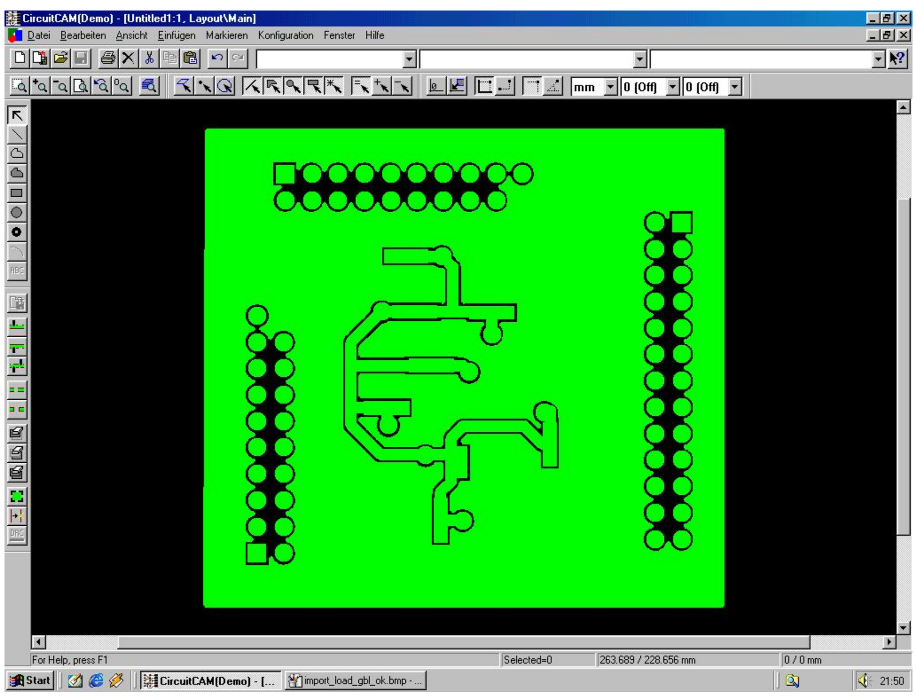


Figure 2.7: Result of importing bottom gerber

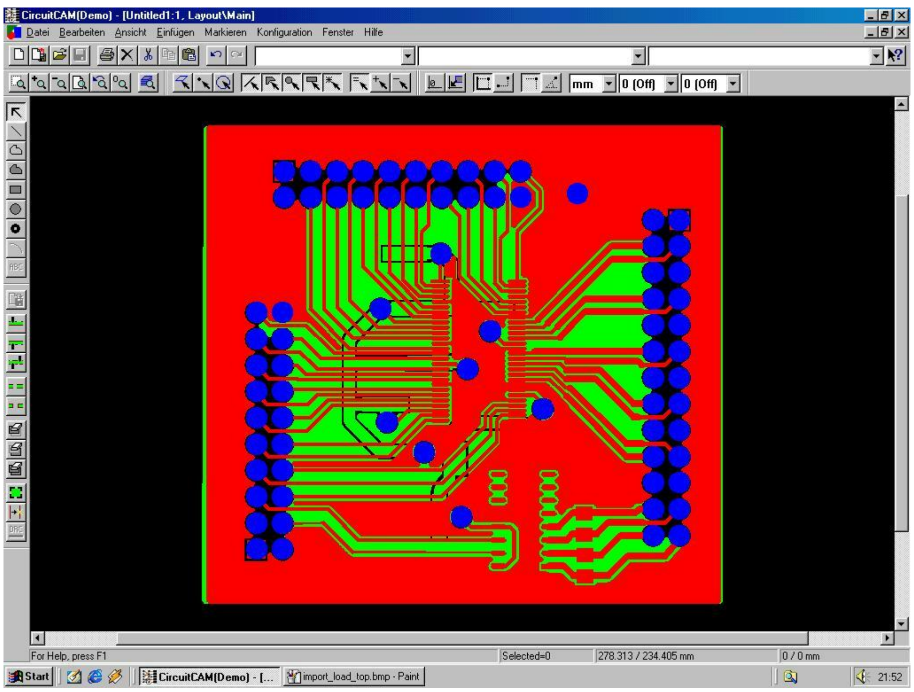


Figure 2.8: Final result

2.2.3 Errors with pad sizes

Sometimes it may happen, that a pad size was misinterpreted by the program. E.g. your pads of your chip are too big (lucky if you realize that in time). You could change these apertures by hand in Protel, but usually you don't know them in advance. If you have only a few do it directly by hand. If there are more, change the aperture definition. So click on one of these faulty pads. You will see a name like D12 in the third column of the programs main window. This is the aperture you have to edit. (The layer has to be markable in order you can mark the pad. (2.14, 2.15)) Then select in the menu `Konfiguration->Format->Gerber Blendenliste...` (2.9).

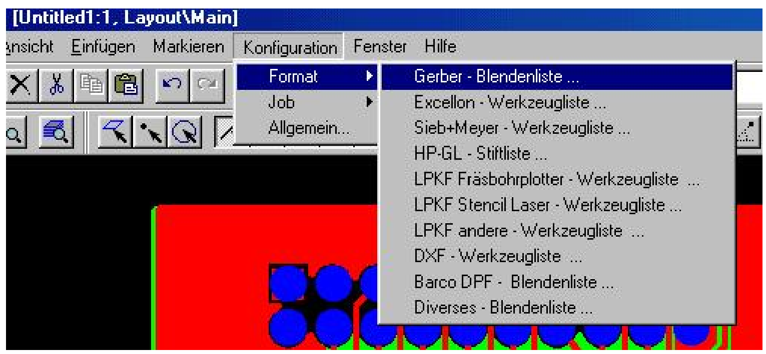


Figure 2.9: Loading Gerber Apertures

Then select the aperture file, corresponding to the layer where you encountered the error (2.10).

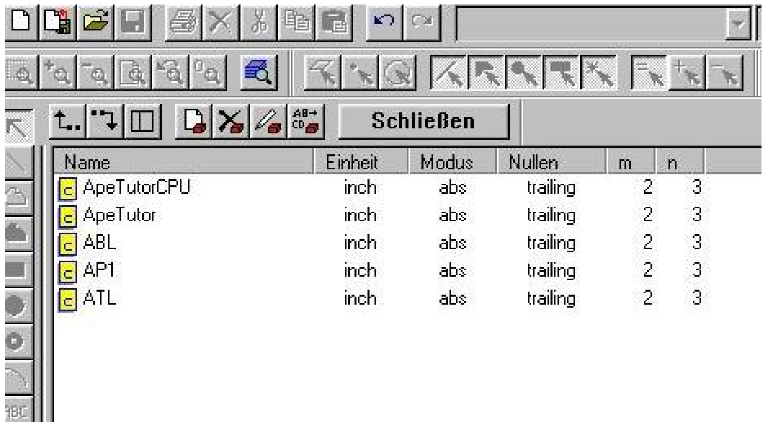


Figure 2.10: List of different aperture files

Then you get a list with all existing apertures (2.11).

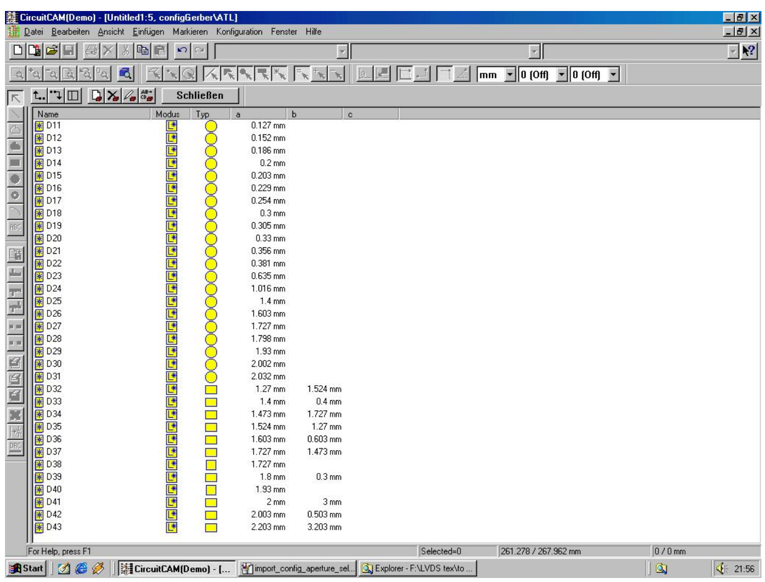


Figure 2.11: List of apertures

Double click the faulty aperture. Then you will be able to edit it (2.12). You can delete it first, or you just can change its size.

Once you finished editing your aperture, save the changes, and then go back to the main window (2.13).

Now the faulty pad(s) should be modified.

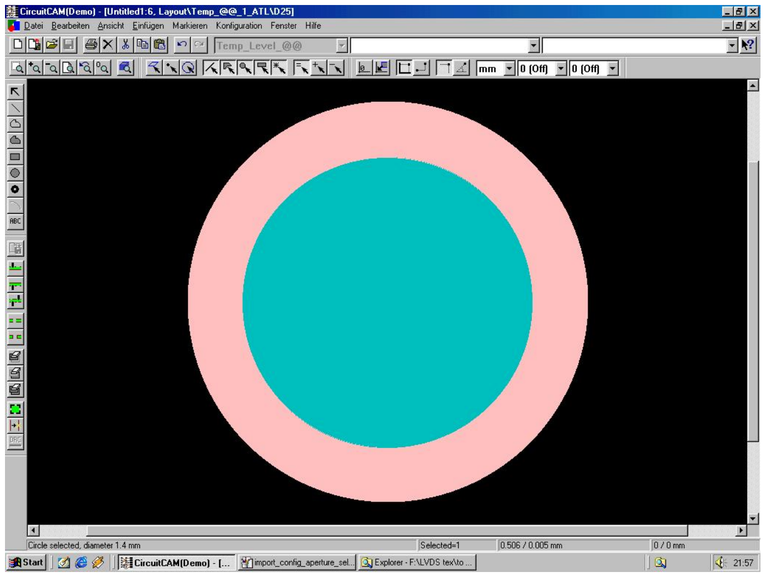


Figure 2.12: Edit aperture

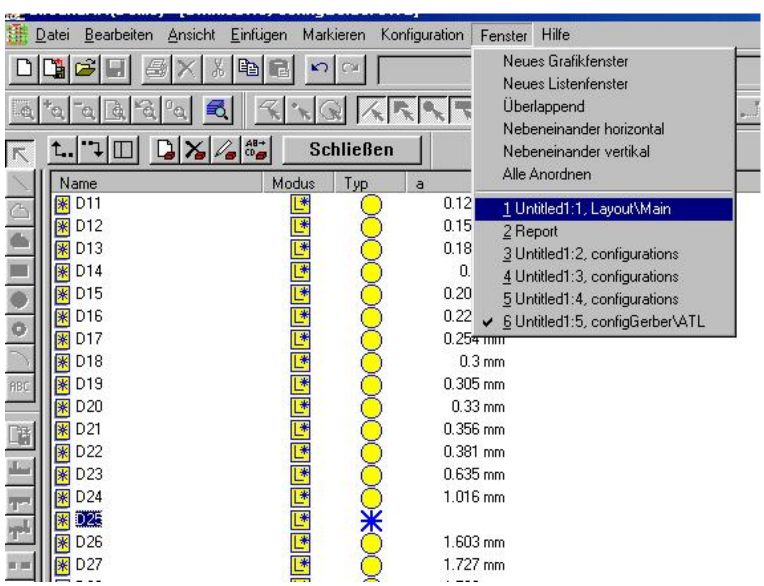


Figure 2.13: Change window

2.3 Setting the drill size

Since protel doesn't know the drills available with the LPKF plotter, you have to set them in CircuitCAM.

2.3.1 Selecting the right layers

First of all you have to turn of all layers, except the drills. To get to the layer selection menu press the `Layers` button (2.14)

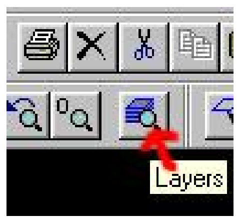


Figure 2.14: Select the right layers

Then you will get to a screen, where you can select/deselect your layers. (2.15).

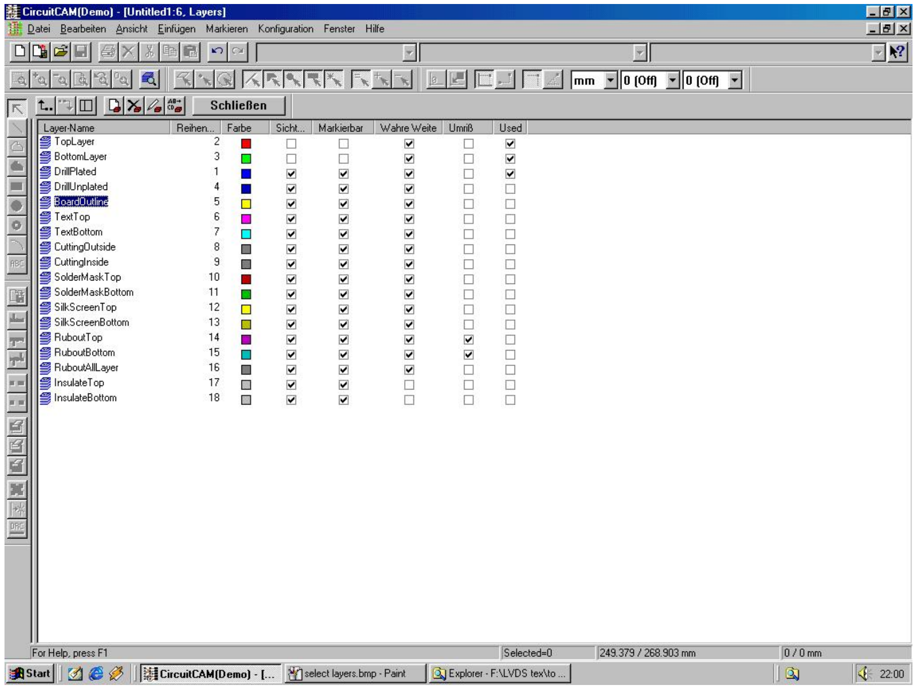


Figure 2.15: Select the right layers

There you should deselect all layers you don't need. In the end only layers you don't have and the drills (plated/unplated) may be selected. In order that you will be able to mark the drills you also have to select `Markierbar`. It is always good to have that option on.

2.3.2 Adjusting the hole size

You have to start with marking the holes you want to adjust. In the top of the CircuitCAM window are three columns (2.16).

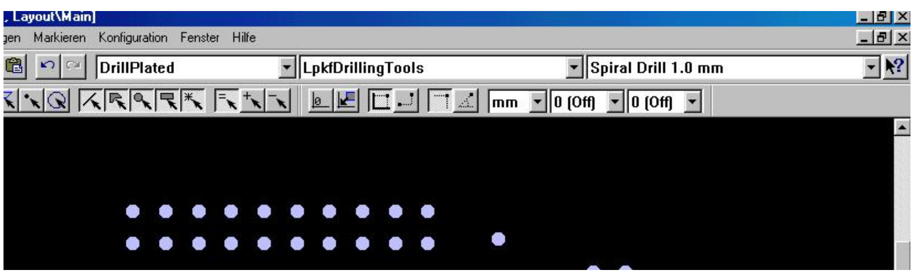


Figure 2.16: Adjusting hole size

- In the first column you can select either the holes should be plated or unplated.
- In the second column you have to choose the LPKF drilling tools.
- In the third column you can choose the drill's diameter.

2.3.3 Select layers

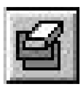
To finish the drills you finally have to select the other layers again, which were deselected earlier (2.14, 2.15).

2.4 Rubout layers

Now you have to prepare your PCB for the contour routing, which defines the line for the cutting out. You have to draw a rectangle, which covers up your PCB. All within the rectangle will be safe. Later, the contour router will go along the outline of your rectangle, without cutting out this line.

Do you need to leave a margin around your PCB you simply have to draw the rectangle bigger.

To do the above described press the tool `Rubout All Layers`


Tool `Rubout All Layers`

Then draw your rectangle, or several rectangles for polygons if needed. The result should look like 2.17.

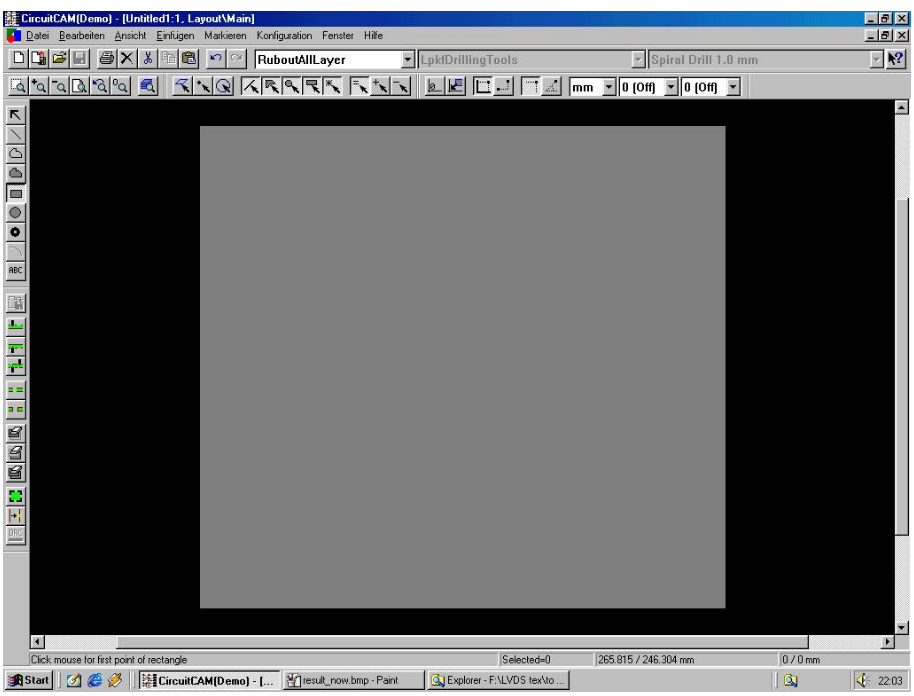


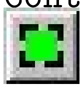
Figure 2.17: Rubout All Layer

2.5 Contour Routing

With the contour routing you define the line, where the LPKF plotter cuts out your PCB.

2.5.1 Setting the contour

You start with defining a line, where to cut out, by pressing the `Contour Tool`.

 Tool `Contour Routing`

Then you will get a window where you should select the following: (2.18)

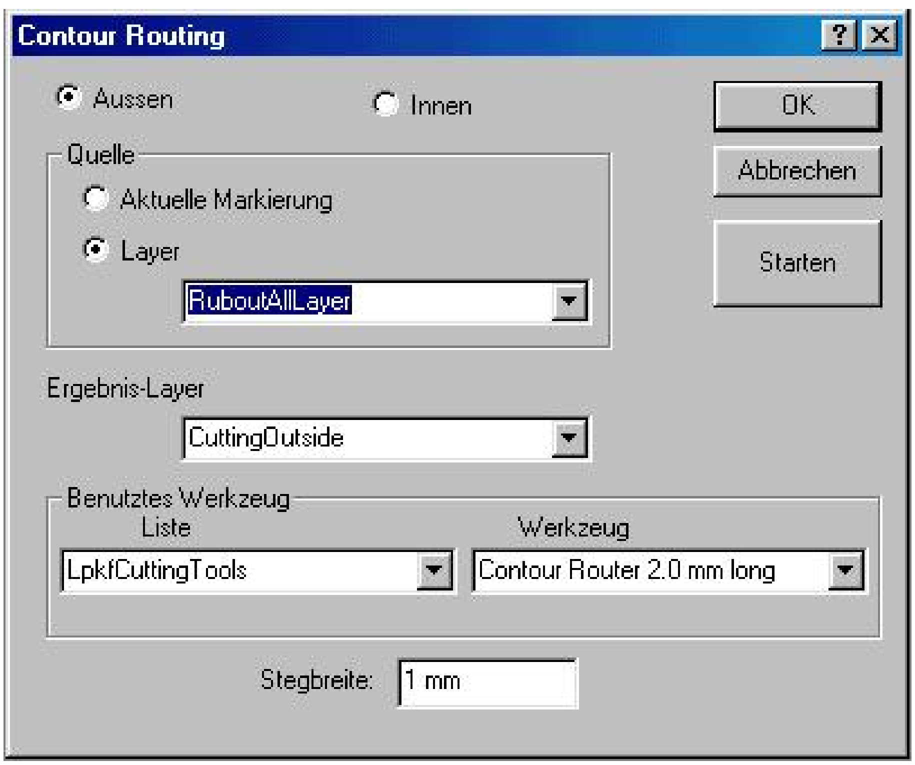


Figure 2.18: Contour Routing menu

- `Aussen` so the contour router tool of the LPKF will cut outside of the rectangle defined in the `Rubout All Layers`.

- For the layer select `Rubout All Layer`, which was defined earlier.
- For the resulting layer (`Ergebnis Layer`) you should select `Cutting Outside`
- For the used tool (`Benutztes Werkzeug`) select `LPKFCutting tools`.
- For the tool (`Werkzeug`) use `Contour Router 2.0mm long` or `Contour Router 1.0mm long`
- The `Stegbreite` should be 1mm or 2mm. `Stegbreite` defines the width of the gap you insert later.

The result should look like (2.19)

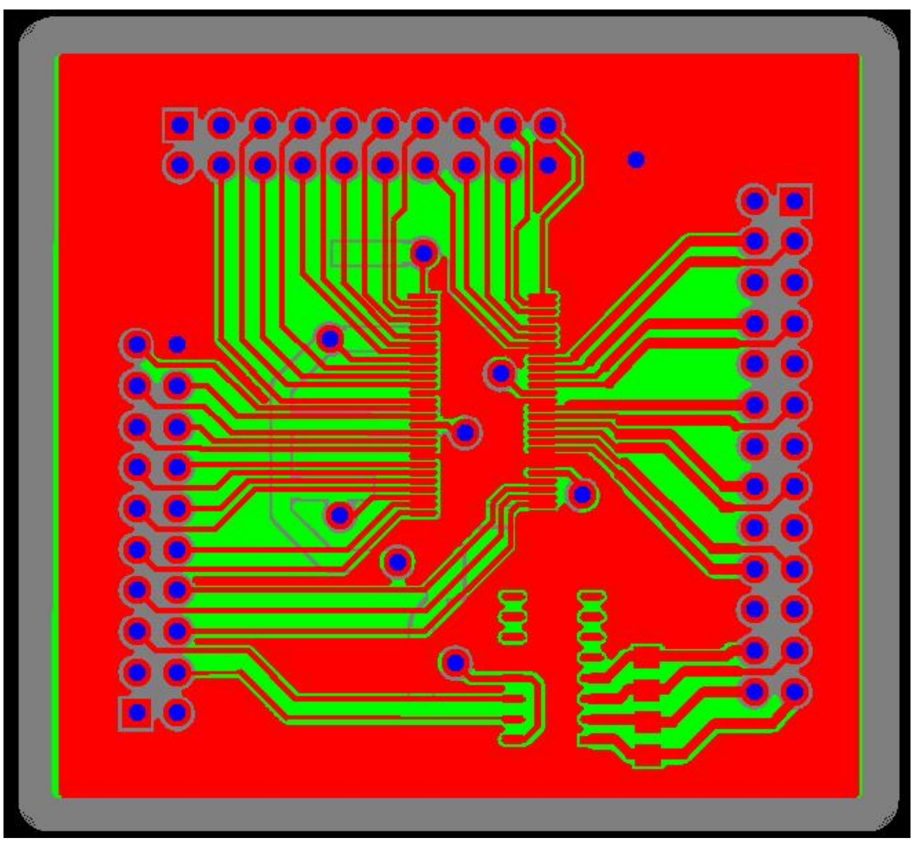
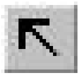


Figure 2.19: Result of Contour Routing

2.5.2 Insert Gap

In order the cut out will work, it must not be done all around, there must be left several gaps, so that your PCB won't fall out. And as you probably have to dispense, the PCB has to stay fixed.

First you start marking the grey border around your PCB with the arrow tool.



If you had a rectangle then you see eight squares, which mark the border (2.20). You can step through the squares by pressing + or -, which is sometimes easier than selecting them with the mouse.

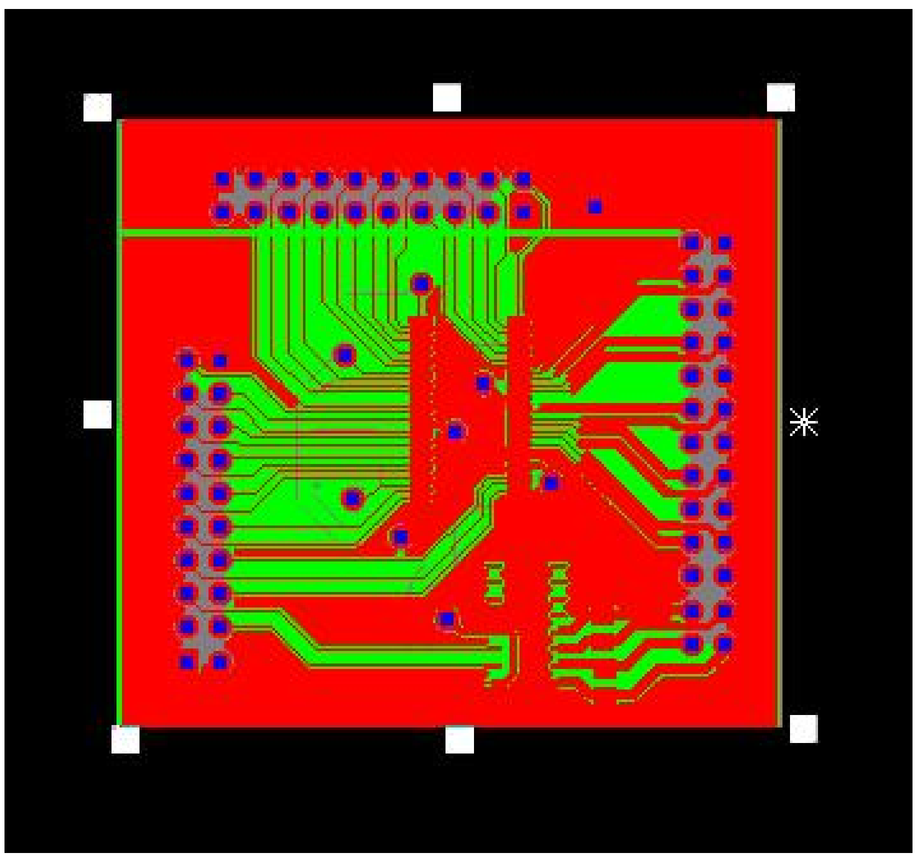
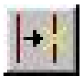


Figure 2.20: Marked outline

Usually it is a good idea to insert the gaps in the middle of each side of the rectangle. But you also can choose other points where to insert the gaps, but always insert them in a way that your PCB still is firmly linked with the rest of the board.

To insert a gap mark one of the squares then press the tool gap 

Do that with all squares you wish. The result should look like 2.21.

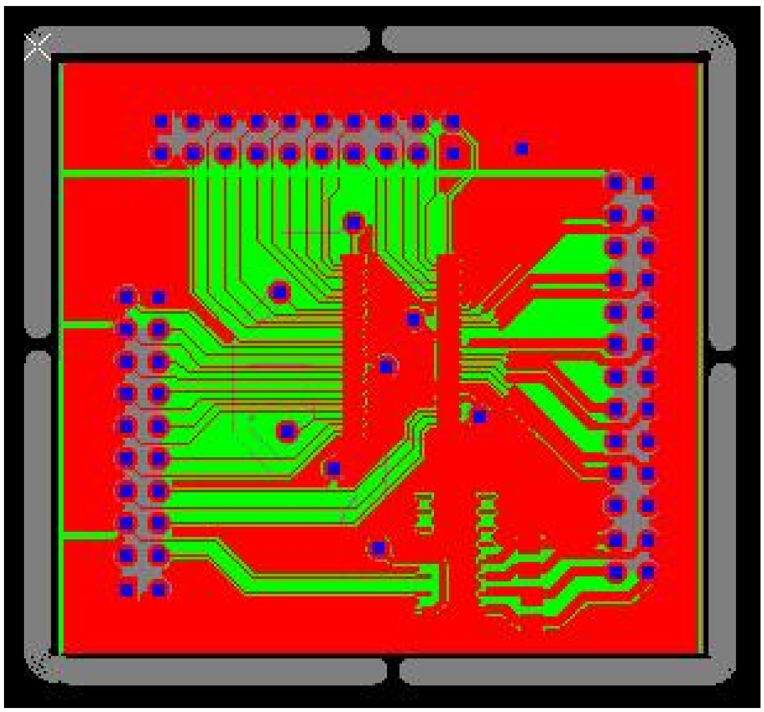


Figure 2.21: PCB with inserted gaps

2.6 Insulation of layers

Now we have to calculate the tracks for the different tools of the LPKF plotter. This is called 'insulate'.

2.6.1 Insulate Bottom

Select `Bearbeiten->Isolieren` (2.22).

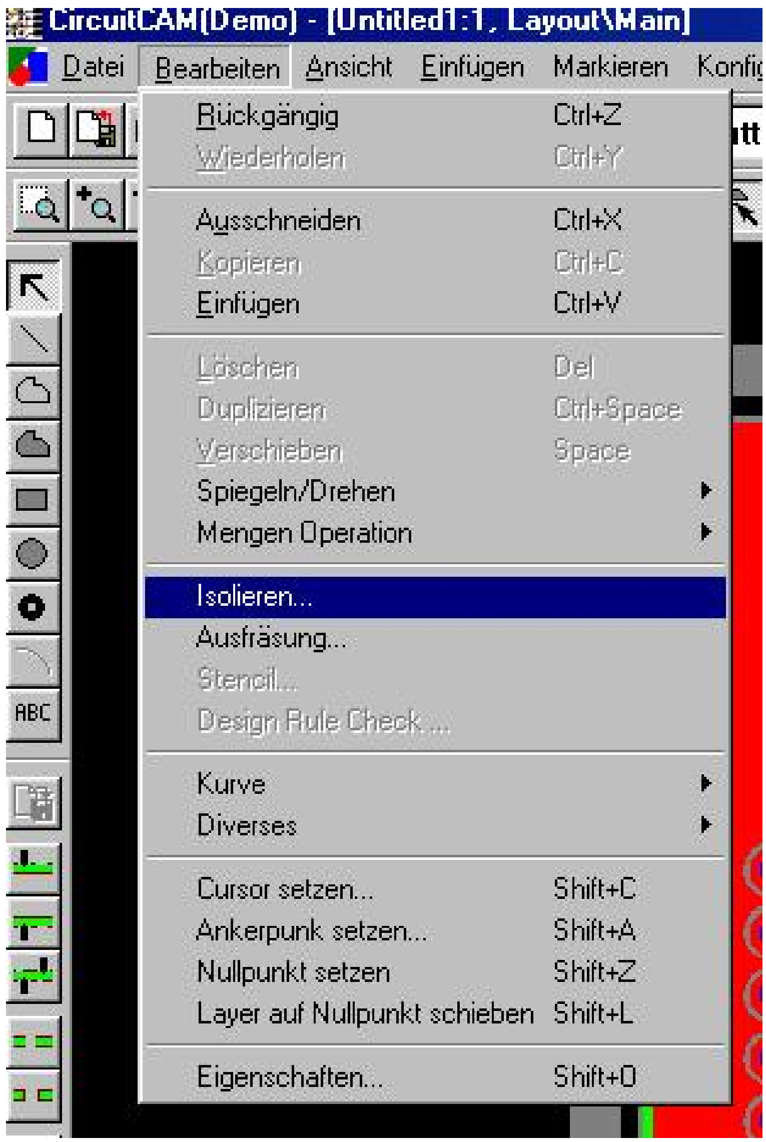


Figure 2.22: Menu Insulate

Then you get to another window where you have to select the job and the tools available. Go to the card `Job` (2.23). There you have to select *InsulateDefaultBottom* for the `Bottom Layer`. Then go to the card `Start` (2.24) Select `Lötseite(Bottom)`. For the standard tool select `Universal Cutter 0.2mm`, and for the bigger tool select `Double Edged Cutter 1.0mm`. These are the default tools of the LPKF plotter. Then press `Start`. Now the insulation paths for the bottom layer are calculated.

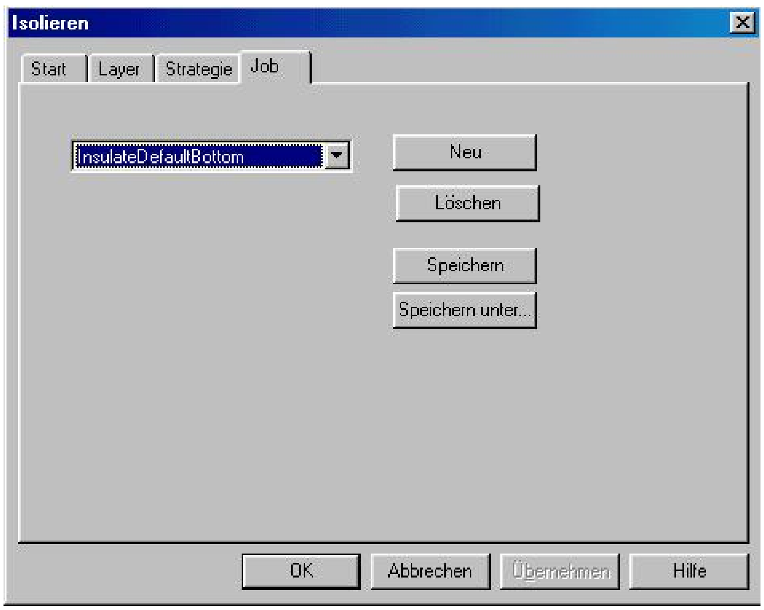


Figure 2.23: Window Insulate: Job

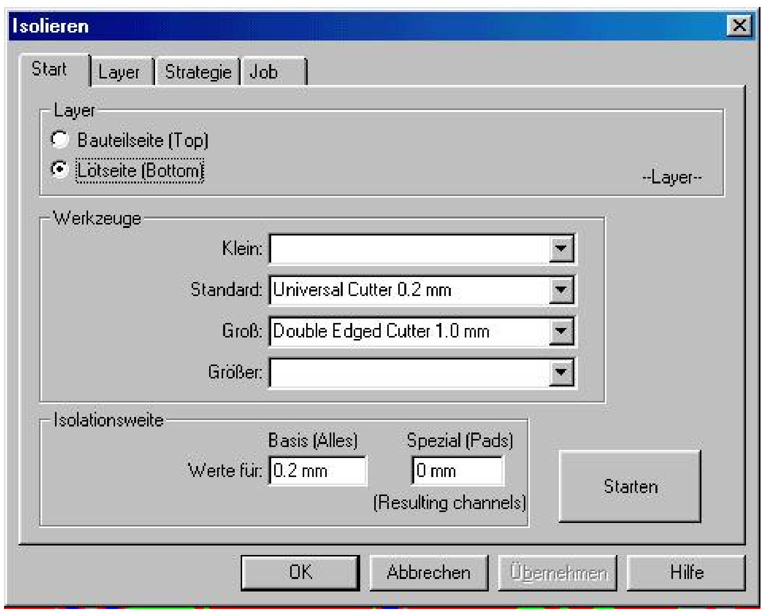


Figure 2.24: Window Insulate: Start

2.6.2 Insulate Top

In the menu select `Bearbeiten->Isolieren`. In the `Job` card select `InsulateDefaultTop`. On the `Start` card you select the same tools as you selected for insulating the bottom layer. For the layer you choose `Bauteilseite(Top)`. Then press start.

When the calculation is done you have insulated the top and the bottom layer. If you switch off the bottom layer you should be able to see a result like 2.25.

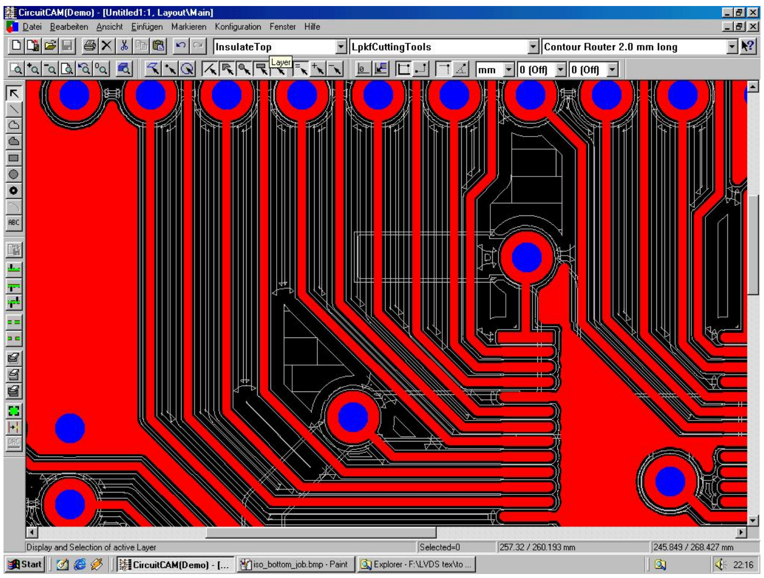
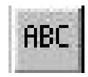


Figure 2.25: Possible result of insulation

2.7 Text on your PCB

Now to another pleasing point of this documentation: Inserting text. Surely you have the need for text on your PCB (we had!), such as the date or the version of your PCB and nevertheless of your name. For some people this may be the most important point. You can insert text by pressing the tool `ABC`. 

Then another windows opens where you can specify the layer on which you want your text placed (2.26).

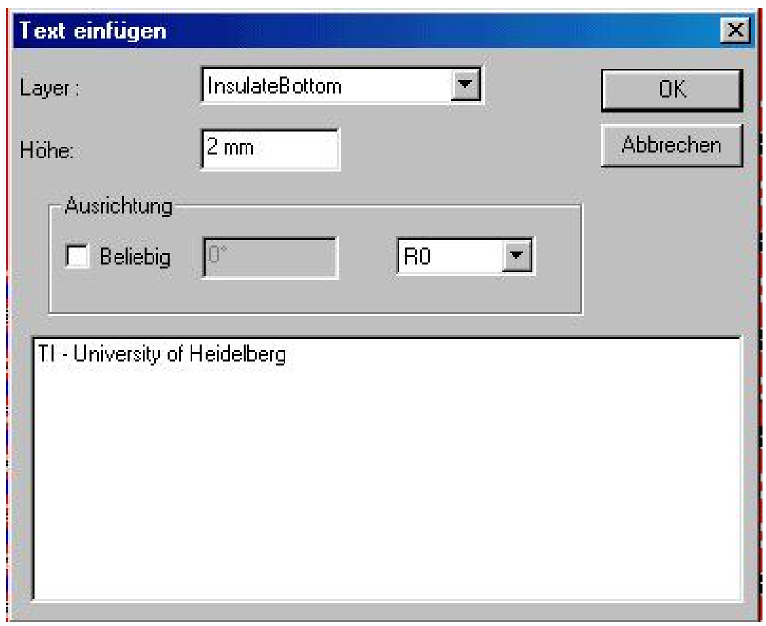


Figure 2.26: Window for inserting text

This can be `InsulateTop` for top layer or `InsulateBottom` for bottom layer. When pressing OK you will be able to move the text with your mouse. Position your text and click the left button of your mouse.

The result should look like 2.27.

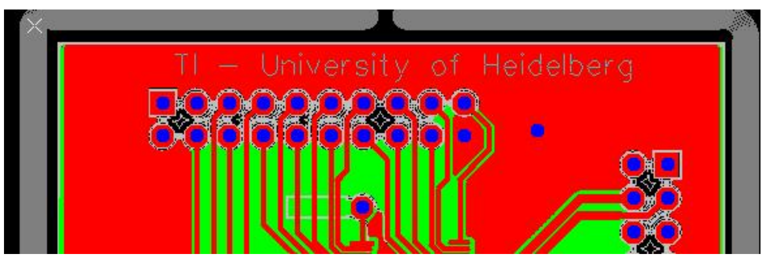


Figure 2.27: Inserted text visible

It may happen that other layers cover your text once you placed it. But when you were able to drag it around, and when it was visible before you placed it, it should be alright. This is then just a graphics problem, or better: a problem of the displayed layers.

2.8 Exporting the data

First of all you should save you file in the CircuitCAM format, by selecting `Datei->Speichern unter`.

Then select `Datei->Exportieren->LPKF Fräßbohrplotter` (2.28). The file should be saved in the same directory as you saved the CircuitCAM file and has the name xxx.LMD, where xxx stands for the name you gave your CircuitCAM file. We had some difficulties with that point, because the exported file sometimes wasn't in the directory we expected it to be. If that happens you can help yourself by using the 'Windows 9x/NT explorer', and then search for all *.LMD files.

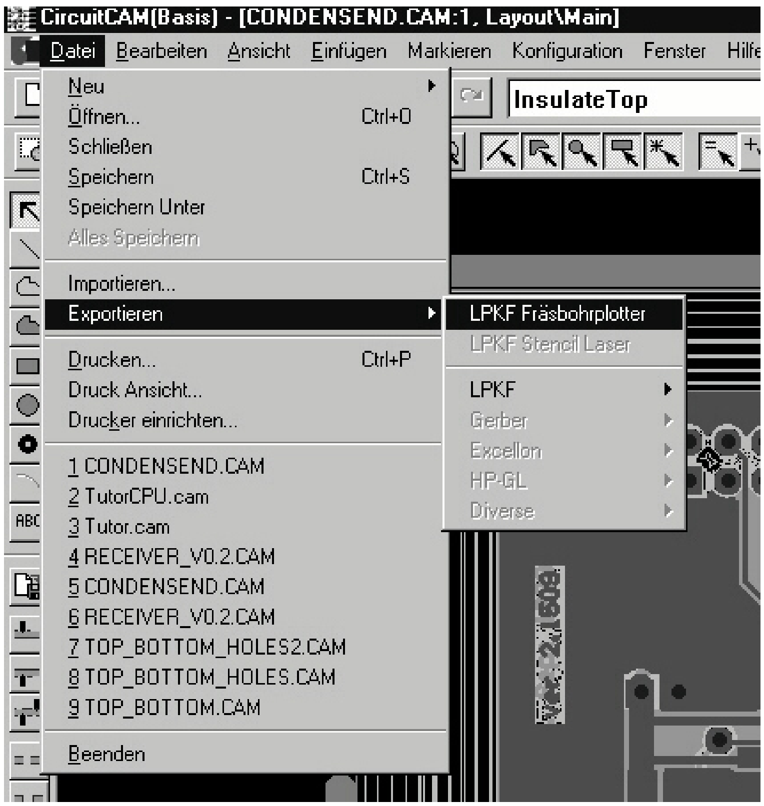


Figure 2.28: Export the data

2.9 Design rules for the LPKF

Due to the fact that the LPKF plotter has certain poduction restrictions, you have to keep several rules in mind:

- *Tracks* should be at least 4mil width, the min insulation width has to be 8mil. In special cases, such as bond pads for chip on board layouts you can use 4mil. This restriction is caused by the week durability of the MicroCutter.
- *Plated holes* can be of sizes from 0.5mm to 1.4mm in 0.1mm steps. The pad where the vacuum needle is placed on must be about 0.4mm bigger than the drilled hole. The produced vias will be 0.3mm smaller than the drilled hole.
- *Unplated holes* can range from 0.5mm to 1.5mm in diameter. Hole sized bigger than 1.5mm will be milled out by the 1mm ContourCutter.

Further pieces of information can be obtained from Ralf Achenbach.

Chapter 3

Boardmaster

3.1 Foreword

Now we finally reach our last step in the production of your PCB: **drilling and milling**.

This is a step where you usually need help for the first time. Even this documentation can't replace the support of an experienced person, such as Ralf Achenbach. He is the head of the ASIC test laboratory. Many thanks to him.

BoardMaster is the controller software for the 'LPKF-Fräßbohrplotter'. You have to take several steps to manufacture your PCB.

The first step is to prepare the copper-board and the LPKF-Fräßbohrplotter (from now shortly: LPKF) and setting up all the tools during the process. When this is done, you have finished about 50% of your total job. The preparation is as important as the milling/drilling of your PCB, because if you make an error with the preparation, you surely can dump your PCB later.

The next 35% is the milling/drilling and the final 15% of your work is the dispensing, vakuum and heating of your PCB.

We didn't intend to make a full dokumentation, because it already exists, since BoardMaster is shipped with the LPKF, but it is a very bulky one.

In this documentation we try to go on chronologically.

One last hint: Save your file after several steps, and after each step, where drilling and milling jobs were done.

3.2 Preparing the board

You have to start with preparing your copper-board. Its size is DIN A4. They only differ in thickness of the board ($1.0mm$ and $1.5mm$) and of their copper coating ($18\mu m$ and $36\mu m$). You also can choose between boards coated with copper on one side and on two sides.

Choose one and stick it to a wooden board of the same size with adhesive tape. (The wooden boards are in the same drawer as the copper boards.)

Then drill a hole of $3mm$ diameter into the board in the center of each short side. The holes should be about $1cm$ away from the edge of the board (figure 3.1).

When you have done that, you can remove the adhesive tape.

3.3 Preparing the LPKF plotter

3.3.1 Mounting your board on the LPKF

Move the plotter head to the 'Pause' position by `Verfahre nach -> Pause`.

With the LPKF you will find two pins of $2.95mm$ diameter. Put them into the drilled holes of your copper and wood boards.

On the table of the LPKF you see two orange plastic stripes. (Let the left one on the plotter, otherwise you could change the setup. The left pin should be in touch with a pin, which is connected on the table.) Remove the right and fix it to the right of your boards. Then fix the rest on the plotter.

When you attached it well, use some adhesive tape at the margin of your board to fix it to the table of the plotter (figure 3.1).

Then load the standard tools library (asic.tol). You can do that

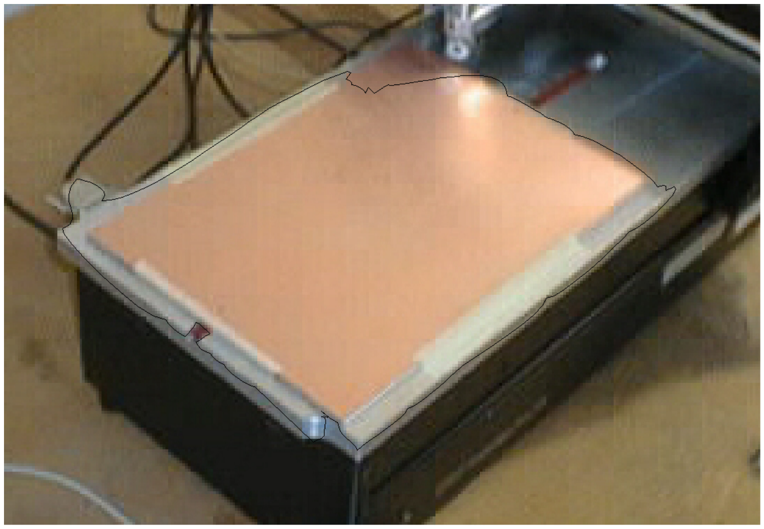


Figure 3.1: Ready copper-board

by: `Konfiguration->Bibliothek->Bohr Fräßkopf` in the directory C:/LPKF30/DATA/ASIC.TOL

3.3.2 Setting the workspace

Now you have to define the workspace of the plotter head.

Therefore move the head to your minimum coordinates. Imagine a two dimensional coordinate system, with your copper board as the first quadrant. The min position is very close to the origin of the system. You have to pay attention, that you leave enough distance to the edges ($\approx 2cm$), that the plotter isn't able to mill/drill in the vincinity of your left pin, which fixes the board to the table of the LPKF.

Then choose in the menu `Konfiguration-> Material-> Setzte MIN XY`

Then move the head to the maximum position. Select `Konfiguration-> Material-> Setzte MAX XY`. Bear in mind that also the right pin shouldn't lie within the area you defined with setting the two diametrical corner points of a rectangle (figure 3.2).

On this area you will be able to mill/drill.

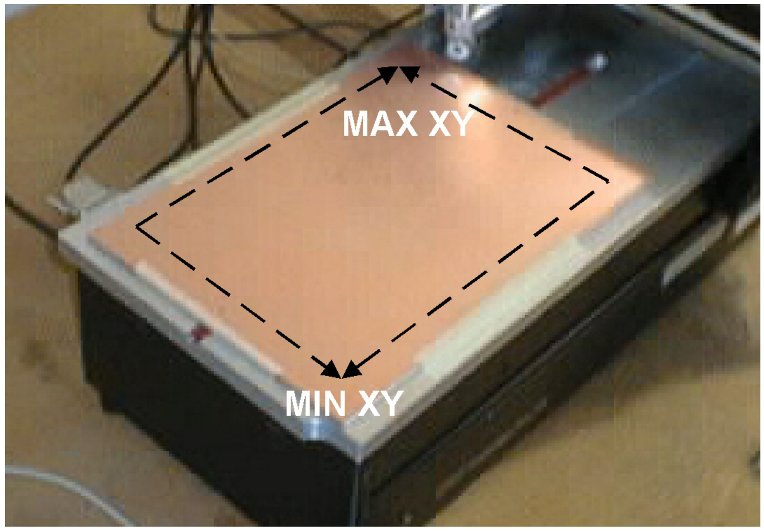


Figure 3.2: Set MIN XY and MAY XY

3.4 Prepare the milling/drilling

Now you prepare the plotter for the milling and drilling process. To be able to test the right adjustment, you need test structures for the cutter and for the drills, to be able to know if you have the right settings (e.g. height). Then you will have to set the order of the different phases.

3.4.1 Importing

Test structures

Usually you need the Universal Cutter 0.2mm for the standard milling job. So import a test structure for it. Choose the menu `Datei-> Importieren-> LMD` and then the file
C:/LPKF30/DATA/struk0_2.lmd

Make several copies of that structure, at least three.

Then import the test structures for your plated hole. If you have different drill sizes, just import the smallest drill.

You can do that in the menu `Datei->Importieren->LMD` and the file C:/LPKF30/DATA/1_0.lmd for a 1mm diameter drill (0_6.lmd for a 0.6mm drill and so on).

Your PCB

In CircuitCAM you exported a file(xxx.LMD). Import that one by choosing the menu `Datei->Importieren->LMD`.

Since the LPKF can't dispense on the whole work area you defined above, you have to place the plated holes within the dispensing area.

To display the dispense area click on the tool column and select a vacuum cartridge. Then you will see a white outline. Within that area you will be able to dispense. Within the area you defined as your workspace you will be able to mill/drill.
In the following you can have a look on the icons and the menu of Boardmaster:

In figure 3.3 you can see the toolbar of BoardMaster.

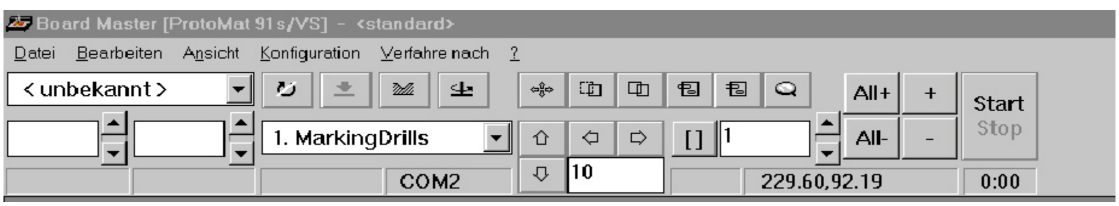
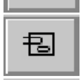
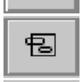
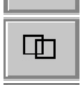


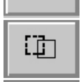
Figure 3.3: The toolbar in Boardmaster

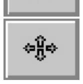
 Magnifier

 Selection only in the area

 Selection the whole objects in the area

 Copy a object

 Move a object

 Move plotter to marked place

Removing structures

If you have too many structures you can remove them by choosing the menu `Bearbeiten-> Plazierung`. Then select the file you intend to delete, and press the delete button.

3.4.2 Setting phases

Usually the phases for the whole milling/drilling process are set. With the phases, you define the order of the different work steps. You also select the side (top, bottom) on which a process should be executed.

If the phases are not set, select the menu `Konfiguration-> Phasen` and choose a process similar to the following (table 3.1):

Overview of the phases		
number	process	side
1.	Marking Drills	TOP
2.	Drilling	TOP
3.	Milling Top	TOP
4.	Milling Bottom	BOTTOM
5.	Cutting	BOTTOM
6.	Dispensing	TOP
7.	Vakuum	TOP

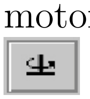
Table 3.1: Overview of the phases

3.5 Setting the right height

This section is a very important one.

You always will need this modus operandi. **After each replacement of a milling tool you have to adjust the height**. This is why you should have loaded enough test structures.

You always should have activated the automatic starting of the motor.

 Automatic starting of the motor

3.5.1 Universal Cutter

Select Milling Top, and the Universal Cutter 0.2mm. Then insert the tool. Turn the wheel as long as you can see the Cutter from aside the tool-head(figure **??**).

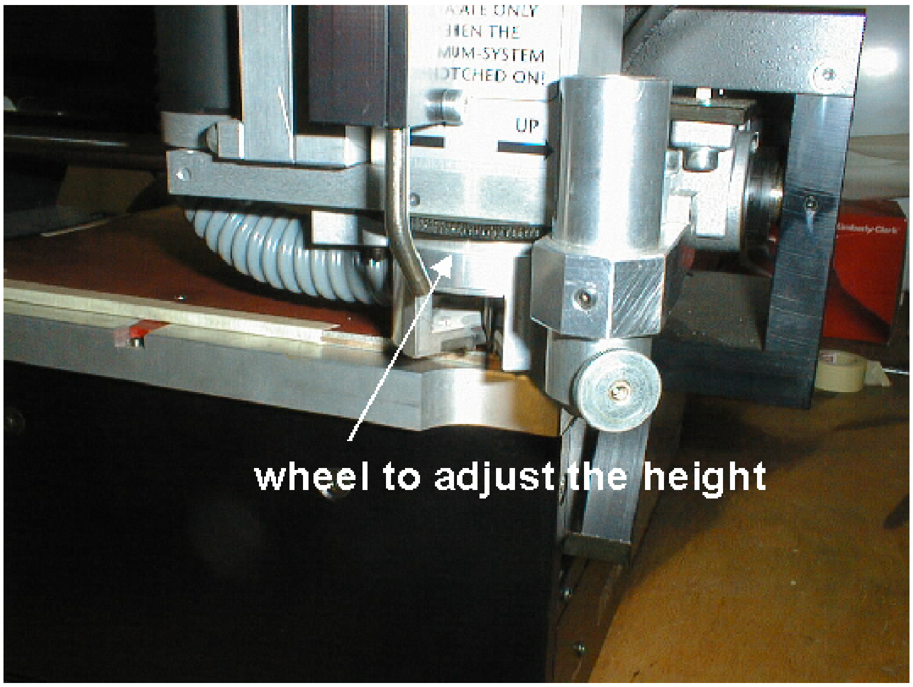


Figure 3.4: tool-head

PRESS ALL- to deselect the rest of your board. Mark about 1/4 to 1/3 of a test structure (this is enough to be able to see the right setting). Then press the plus in the menu.

Buttons for select, de-select and start

Press 'Start'. Now the plotter should work in the area of the test-structure.

When it stopped, move the tool-head a bit aside, so that you are able to see the milled structure. You can use a magnifying glass.

There are now three possibilities:

- The copper was removed completely. The Universal Cutter is set too low. So turn the wheel up. Now mark the next 1/4 or 1/3 of your test structure, and press start, ... and so on.
- The copper wasn't removed, nothing to see of a removement. That means that the Universal Cutter is set to high. Turn the wheel down. Mark the next 1/4 or 1/3 of your test structure, and press start, ... and so on.
- You see a structure of copper and removed copper by turns. Now there are three possibilities:
  - If the copper structures are wider than the milled structures, then the U.Cutter is still set a bit too high. So turn the wheel down (now in small steps, about 4 clicks or less).
  - If the copper structures are smaller than the milled structures, then the U.Cutter is still set a bit too low. So turn up the wheel (in small steps, about 4 clicks or less).
  - The width of the copper structures and the milled structures is the same. So you got the right height. Proceed as explained later.

3.6 Working procedures

To finish the PCB we have several steps to do.

3.6.1 Marking Drills

You already set the order of the phases. Now you are going to mark the drills.

Therefore use the Universal Cutter 0.2*mm*. Then set the right height as explained before (section 3.5). Mark all the drills by pressing All+, then press start.

3.6.2 Drilling

Now use the right drill. Adjust the height, that the drill will get through the copper board, but NOT through the wooden board. Press All+ (if the drills are not marked any longer), and press Start.

3.6.3 Milling Top

Now we are milling the top of your PCB. Insert the right tool. Set the height (section 3.5). Press All+, mark the test structures and press - (so the test structures won't be milled). Press Start.

Repeat the last step with all tools you need for milling the top.

3.6.4 Milling Bottom

Turn around your board.

Relative position

Before you begin with milling, you have to check, if the position of the tool-head has changed. If it has changed, the two layers will not be on top of each other.

To check the position, insert a tool, then zoom in to a small hole. (To be able to see the holes on the bottom layer you have to adjust the phases. How you do that was explained, when setting the phases. After this step cancel the changes.)

Then manually move the plotter head to that hole. Press down the tool-head with your hand. If the tool you inserted fits into the drilled hole, the relative positions of the top and bottom layer should be alright.

If NOT, you have to adjust the home position of the plotter. Select `Konfiguration -> Plotter ...`. Then select `Entsichern`. Remember the old values of the Homeposition. Now adjust the values. Select `Sichern` and then press OK. Repeat the step above until you get the right position.

Usually you don't have to do that, because the plotter should be well setup.

Milling

Now do the same as in Milling Top. **Then set the old homeposition**.

3.6.5 Cutting

The program will tell you which tool to insert (Contour Router $2mm$ or $1mm$, depending on your cutout).

Adjust the height in a way, that the copper board will be cut, but the wooden board must not be cut through.

Press All+, then press Start.

3.6.6 Dispense

Now we are getting to a rather annoying point: the dispensing.

Preparing the wooden board

First of all you have to prepare the wooden board. You have to cut all the pieces out, which are under a plated hole. That is because the paste will be pressed through the holes, so the board would obstruct the flow of the paste.

But try to keep some wooden board between the areas, where are no plated holes, so the wooden board is able to hold the copper board on the same level, because the LPKF-tool-head always presses with a certain force onto the copper board.

Rub off

Then you have to rub off cautiously both copper layers with emery.

Preparing the paste

You have to prepare the paste in two syringes by pressing them from one into the other several times. It will be more liquid then.

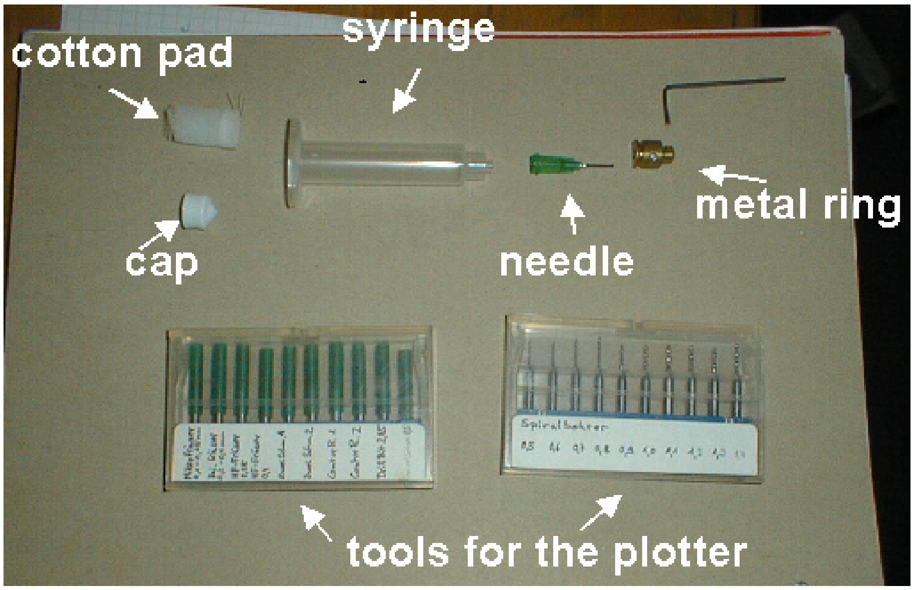


Figure 3.5: Tools for the plotter

Prepare the syringe

You should use a new syringe. Put in a white cap on the one side of the syringe and press it to the bottom of it.

Then fill up the syringe by pressing the paste through the tip of the syringe.

Then put on the metal ring over the needle. Screw on tight. Then screw on the needle onto the syringe.

Insert it into the LPKF-head and screw it on tight, and then attach the hose to syringe.

Adjusting pressure

Now you have to set the pressure. Start with a pressure of $1 bar$ and a vacuum of $0.05 bar$.

Figure 3.6: Station for pressure, vacuum and light

Activate the LPKF-head manually. Do that until the paste will come out of the needle and all the air will be removed from the syringe.

Then again use your test holes. Mark two of them and press +, then press start. Then check if the paste was pressed through the hole and was coming out on the other side. If not, you have to increase the pressure.

If the paste is dropped between two holes you have to increase the vacuum.

Repeat the procedure of adjusting pressure and vacuum, until

- the paste is pressed through the hole, and produces a small stub on the other side and
- the paste doesn't drop between two holes.

When you have a small stub on the other side, you can be sure, that the two copper layers are linked by the paste.

Then mark all (except the test holes) and press start.

Check the paste

Then remove the copper board, but watch out NOT to smear the paste stubs. Check if all holes have little stubs on the bottom layer. If not then you can repeat dispensing one (or even three) time(s) with the holes, which weren't alright. But this should be done within 15 to 20 minutes. Otherwise the paste will dry.

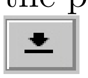
If you have many plated holes where dispensing/vacuum would take longer than half an hour you should do the dispensing/vacuum in several independent steps.

3.6.7 Vacuum

Now we have to remove the paste in the centers of the holes. We don't remove it completely, because a thin layer sticks to the wall of the hole, and this connects the two copper layers.

Now use another new syringe. Insert a cotton pad. Screw on a new needle, with the metal ring. Then attach it to the LPKF-head, and the hose to the syringe.

Switch off the pressure. Now go to a part of the copper board, where you have only copper (without structures and holes). Set down the plotter head onto the copper abd adjust the vacuum to $-0.3bar$.

 Set down head

Then press All+, and press Start.

Then the needle will suck out the paste in the center of the holes.

If the paste wasn't sucked out on the first turn, repeat the vacuum, until the paste is sucked out well.

3.6.8 Baking

Now remove the wooden board, and put the copper board into the oven. Pay attention that you don't smear the paste.

Then heat the oven to $160°C$ and set the timer to half an hour.

Now wait until your pizza is crispy an the cheese is golden.

List of Figures